CEDRUS

Manual

Copyright Cubus AG, Zurich

cubus
ENGINEERING SOFTWARE

Vol. A Introduction A-1

A 1 Overview A-1

A 1.1 Introduction A-1

A 1.2 Documentation A-2

A 1.2.1 CEDRUS: Specific Topics A-2

A 1.2.2 Shared Program Modules A-2

A 1.2.3 Context-specific Help A-2

A 1.2.4 Document Format and Document Access A-3

Supported PDF-Viewer and PdfViewer.ini-file A-3

A 1.2.5 Printed manuals A-3

A 1.3 Program Installation A-3

Installation A-3

A 1.4 Slab Analysis Procedure A-5

Tab sheet 'CAD/BIM' A-5

Tab sheet 'Structure' A-5

Tab sheet 'Punching' A-7

Tab sheet 'PT module' A-7

Tab sheet 'Loads' A-7

Tab sheet 'FE mesh' A-8

Tab sheet 'Results' A-9

A 1.5 Procedure for a Building Analysis A-10

A 1.5.1 Geometrical Building Model A-10

A 1.5.2 Geometric building model from BIM data A-11

A 1.5.3 Stack of Slabs Model A-12

A 1.6 Program Modules A-14

A 1.6.1 Base Module A-14

Structure A-14

Loads A-14

Basic Results A-14

Punching shear A-15

Dimensioning of the reinforcement A-15

Displacements in the cracked state A-15

Crack analysis A-16

A 1.6.2 Buildings Module - Option G A-16

A 1.6.3 Buildings Module - Option H A-16

Top-down analysis of horizontal loads A-16

Dimensioning of walls and verification of masonry walls A-17

Dimensioning and verification of reinforced concrete columns A-17

Creation of space frame models for STATIK A-17

A 1.6.4 Prestressing Module - Option V A-18

A 1.6.5 Dynamics Module - Option D A-20

A 1.6.6 Reinforcement and Ultimate Load - Options R + T A-20

A 1.6.7 Externally Bonded Reinforcement - Option K A-21

A 1.6.8 Walls - Option S A-22

A 1.7 CubusViewer A-23

A 1.8 CubusExplorer A-24

A 2 Introductory Examples A-25

A 2.1 Introduction A-25

A 2.1.1 Presentation Conventions for the Examples A-25

A 2.2 Introductory Example SLAB A-26

A 2.2.1 Problem Description A-26

A 2.2.2 Program Start A-26

A 2.2.3 Beginning an Analysis A-27
A 2.2.4 The tab sheets A-29
A 2.2.5 The tab sheet 'Structure' A-29
Outline, openings A-29
Materials A-32
Material zones A-33
Downstand beam A-35
Supports A-35
Selecting objects and the right mouse button A-39
A 2.2.6 The tab sheet 'Punching' A-41
A 2.2.7 The tab sheet 'Loads' A-42
The dialogue with the List of Actions A-44
The dialogue for load import A-44
A 2.2.8 The tab sheet 'FE Mesh' A-45
A 2.2.9 The tab sheet 'Calculations' A-46
A 2.2.10 The tab sheet 'Results' A-47
Properties of output zones and downstand beams A-48
Required results for the example A-49
3D Representation A-54
The Layer Switch A-55
A 2.2.11 Analysis Documentation A-56
Report update A-57
Modify report A-58
The Report Generator A-59
Manual print entries A-59
The CubusViewer A-60
Output interfaces to other programs A-61
A 2.3 Introductory Example: Building A-62
A 2.3.1 Problem Definition A-62
A 2.3.2 Prerequisites A-63
A 2.3.3 Program Start A-63
A 2.3.4 Materials A-63
A 2.3.5 Foundation Zone A-63
A 2.3.6 Input of the Basement Slab „UG“ A-64
The tab sheet 'Building Elements' A-65
A 2.3.7 Input of the Slab „EG“ A-66
A 2.3.8 Input of the Master Slab for the OG's A-66
A 2.3.9 Introduction of the Storeys OG1, OG2 and DG A-67
A 2.3.10 Generation of the Building Model A-68
Tabular Output for the Buildings Model A-68
A 2.3.11 Stacked Slab Model A-68
Concerning the model A-68
Analysis A-69
Results in the vertical elements A-69
Results in the storey slabs A-71
A 2.4 Example BIM: Create Buildings from 3D-Data A-71
A 2.4.1 Problem Definition A-71
A 2.4.2 Prerequisites A-71
A 2.4.3 Program start A-71
A 2.4.4 Import 3D BIM file A-72
Coordinate system A-72
BIM elements and element types A-73
A 2.4.5 Baustoffe definieren A-73
A 2.4.6 BIM-Elemente bereinigen A-74
A 2.4.7 BIM-Geschosse generieren A-75

A 2.4.8 BIM-Geschosse bereinigen A-76
A 2.4.9 BIM- und CEDRUS-Geschosse verlinken A-77
Verlinken der Geschosse A-77
Definition von Generierungsumfang und Lasten A-78
A 2.4.10 CEDRUS-Gebäude erzeugen A-79
A 2.4.11 CEDRUS-Platten eingeben A-80
Dialog 'Layer-Eigenschaften': Darstellung der Pläne A-82
A 2.5 Einführungsbeispiel BIM: Modellnachführung aus 3D-Daten A-82
A 2.5.1 Aufgabestellung A-82
A 2.5.2 Voraussetzungen A-82
A 2.5.3 Neue Version der 3D-BIM-Datei importieren A-83
A 2.5.4 Konfiguration der Nachführung A-84
A 2.5.5 CEDRUS-Gebäude nachführen A-84
A 2.5.6 CEDRUS-Platten nachführen A-85
A 2.6 Einführungsbeispiel BIM: Gebäude aus 2D-Plänen erstellen A-86
A 2.6.1 Aufgabenstellung A-86
A 2.6.2 Voraussetzungen A-86
A 2.6.3 Ablauf A-86
A 2.6.4 Programmstart A-86
A 2.6.5 BIM-Geschoss einführen A-86
A 2.6.6 2D-Plan importieren A-87
A 2.6.7 Lage und Orientierung anpassen A-88
A 2.6.8 CEDRUS-Gebäude erzeugen A-89
A 2.6.9 CEDRUS-Platten eingeben A-91
Hintergrundpläne einschalten A-91
Strukturdaten konstruieren A-92

Vol. B Base Module B-1

B 1 Introduction B-1
B 2 Element Model and Methods of Solution B-1
B 3 The Program Window B-2
Control tabsheet B-2
Graphics Editor tools B-2
Layer switches B-3
Menu bar B-3
B 4 Menu Bar B-4
Menu 'File' B-4
Menu 'Settings' B-5
Menu 'Representation' B-8
Menu 'Window' B-9
Menu 'Help' B-9
B 5 All you need to know about Printing B-9
Report Generator B-11
Manual Print Entries B-11
Updating the Report B-12
CubusViewer B-12
B 6 The tab sheet 'Control' B-13
B 7 CAD B-13
B 7.1 Axis Grid B-13
B 7.2 Background Plans, CADview B-16
B 7.2.1 Import of CAD Drawings and CEDRUS Plan Views B-16
B 7.2.2 Working with CEDRUS Background Plans B-16
B 7.3 CADextract B-18

B 8 Structure B-18
B 8.1 What to Pay Attention to in the Input of Geometrical Objects B-19
B 8.2 Outline in Plan View B-20
B 8.3 Openings B-20
B 8.4 Attributes Box for Material Zones B-20
B 8.5 Material separation lines B-22
B 8.6 Material Models B-22
B 8.6.1 Isotropic Material B-23
B 8.6.2 Material without Drilling Moments Stiffness B-23
B 8.6.3 Orthotropic Material B-24
B 8.6.4 Shear Deformations B-25
B 8.7 Downstands B-25
B 8.7.1 Downstand Dialog B-26
B 8.7.2 Genereting of upstanding beams under walls of displayed slabs B-27
B 8.7.3 Downstand Model B-27
B 8.7.4 What to note during Input B-28
B 8.8 Attributes Box for Area Support Zones and Separation Lines B-29
B 8.8.1 Determination of the Modulus of Subgrade Reaction ks B-30
B 8.8.2 Procedure for the bedding process B-31
B 8.9 Columns and Piles B-31
Pile as Special Type of Column B-31
Layer Buttons B-31
B 8.9.1 Tab sheet 'Attributes' of the Column Dialog B-32
Support directions B-33
Definition of column cross sections B-33
Cross sections of Piles B-33
Inclined Columns B-34
B 8.9.2 Tab sheet 'Support' of the Column Dialog B-34
Settlement stiffness Sdz B-35
Spring support for the rotations of the tops of the columns Srx und Sry B-35
B 8.9.3 Tab sheet 'Supported zone' of the Column Dialog B-36
B 8.9.4 Tab sheet 'Options' of the Column Dialog B-37
B 8.9.5 Support Models B-37
Point-supported columns (= point supports) B-38
Area supports (= point support with area compensation) B-38
Element supports (= area supported elements) B-39
B 8.10 Walls / Line Supports B-41
B 8.10.1 Modelling Recommendations B-42
B 8.10.2 Direction of Wall Axis for Output of Reaction Moments B-42
B 8.10.3 Wall openings B-42
Influence of the wall openings on the static models B-43
B 8.11 Lines of Symmetry B-44
B 8.12 Joints B-45
B 8.12.1 Limitations B-46
B 8.12.2 Loads on Joints B-46
B 8.12.3 Results Output for Joints B-46
Comments on the results for joints B-46
B 8.13 Remaining Buttons in the tab sheet 'Structure' B-47
B 8.13.1 Legend of the Structure Data B-47
B 8.13.2 Checking Structure Input B-47
B 8.13.3 Revert to Previous Structure B-47
B 8.13.4 Import of Structure Data B-47

B 9 Punching Shear B-47
B 9.1 Treatment of Punching Shear in CEDRUS B-48
B 9.1.1 Punching Shear Objects (PSO) B-48
B 9.1.2 Input of Punching Shear Objects B-48
Column PSO B-49
Wall PSO B-49
Free PSO B-50
B 9.1.3 Punching Shear Resistance B-50
Influence of a column head plate B-51
Models to determine the resistance B-51
Required bending resistances B-51
Correction values for the punching shear resistances B-52
Model 1: control perimeter B-52
Model 2: control perimeter with punching shear reinforcement B-52
Model 3: resistance is given B-52
B 9.1.4 SIA262 - Specifics B-52
Interpretation of the output resistance values VRd and VRd-e B-55
B 9.1.5 EC2 - Specifics B-55
ENV 1992-1-1 . . . . . B-55
B 9.2 Inputs in the 'Punching' Tab sheet B-57
B 9.2.1 Presentation and Selection of PSO B-57
On the selection B-57
B 9.2.2 PSO Dialogs B-58
Tab Sheet 'Effective perimeter' B-59
Tab Sheet 'Options' B-60
Tab Sheet 'Wall proprieties' B-60
B 9.2.3 The Input of Free Punching Shear Objects B-61
B 9.2.4 Modifying Punching Shear Polygons B-61
B 9.3 Import of Punching Shear Results B-62
B 10 Loads B-63
B 10.1 Important Terms in Connection with Loads B-63
B 10.2 Loadings B-64
B 10.2.1 Loading Type 'Loading' B-65
Area Loads B-66
Line Loads B-67
Point Loads B-68
Prescribed Displacements / Rotations B-68
B 10.2.2 Loading Type „Combination“ B-68
B 10.2.3 Loading Type „Generator (Moving Load)“ B-69
B 10.2.4 Loading Type „Generator (Unfavourable Pattern)“ B-69
B 10.2.5 Loading Type „Influence field“ B-70
B 10.2.6 Loading Type „Export Combination“ B-70
B 10.2.7 Loading Type „Mass Distribution“ B-70
B 10.2.8 Loading List B-71
B 10.3 Actions B-71
Action Dialog B-71
B 10.4 Load Transfer to Several Storeys B-73
B 10.4.1 Description of Problem B-73
B 10.4.2 Implementation in CEDRUS B-73
B 10.4.3 Load Export B-74
Automatic export combinations B-74
Reduction factors in the case of storey groups of the same live load category B-75
Manually created export combinations B-75
Determination of the export loadings B-75
Section lengths B-76
Nonlinearly determined export combinations B-76
Export of export loadings B-76
B 10.4.4 Load import B-77
B 10.4.5 Checklist for the Load Distribution B-78

B 11 FE Mesh B-78

B 12 Analysis B-80

B 12.1 Results Combinations B-80

B 12.1.1 Input B-81

B 12.2 Limit State Specifications B-82

B 12.2.1 Overview of the Limit State Specifications B-82

B 12.2.2 Actions B-83

Action Specification B-83

Automatically generated action specifications B-85

B 12.2.3 Action Combinations B-85

B 12.2.4 Limit State Specifications with Action Sets B-86

B 12.2.5 Limit Values of Nonlinearly Determined Results B-87

B 12.3 Dimensioning specification for orthogonal reinforcement B-88

B 12.3.1 Dimensioning in the Results Points B-89

Basic Reinforcement B-89

B 12.3.2 Dimensioning for Beam Sections B-90

Basic reinforcement B-91

B 12.3.3 Analysis Parameters B-91

B 12.3.4 Dimensioning for the ULS (Ultimate Load State) B-91

B 12.3.5 Dimensioning for the SLS (Serviceability Limit State) B-91

B 12.4 Dimensioning Specification for Skew Reinforcement B-93

B 12.4.1 Input B-93

B 12.4.2 Basis of Dimensioning B-94

B 12.4.3 Section Forces for Dimensioning B-95

B 12.4.4 Analysis Parameters B-95

B 12.4.5 Apportioning Criterion B-95

B 12.4.6 Verification of Concrete Compressive Force B-95

B 12.5 Specifications for Dimensioning with Prestressing B-96

B 12.6 Dynamic Analysis B-96

B 12.7 Specification of Bending Deflection Analyses B-97

B 12.7.1 Input Data B-97

B 12.7.2 Method based on SIA262 . . . . . B-98

B 12.7.3 Method based on Eurocode 2 . . . . . B-99

B 12.7.4 Take note at bending deflection verification B-101

B 12.8 Specification of Crack Analysis B-101

B 12.8.1 Input B-101

Specification dialog B-101

Regarding the reinforcement B-102

B 12.8.2 Method of Analysis B-102

B 13 Results B-104
B 13.1 Organization of the Results Output B-104
B 13.1.1 Starting the Results Analysis and Output B-104
Graphical Outputs B-104
Tabular Output B-105
B 13.1.2 Results Tree B-105
B 13.1.3 Output Quantities, their Type of Presentation and Components B-106
B 13.1.4 Types of Presentation and the Corresponding Settings B-108
Output of Isolines B-108
Numerical-graphical output B-108
Three dimensional output B-108
Output of principal values / Cracks output B-108
Tabular output B-109
Output of lines and point reactions B-109
Output of results for CAD programs B-110
Section results B-110
Beam section results B-110
External section displays B-111
B 13.1.5 Section Input B-111
B 13.2 Standard Results B-113
B 13.2.1 Raw Results B-113
B 13.2.2 Result Quantities for Loadings and Results Combinations B-113
Deformations: B-113
Section forces: B-113
Edge stresses: B-114
Reactions: B-114
Export loadings: B-114
B 13.2.3 Results Quantities for Limit State Specifications B-114
Deformations: B-114
Reactions: B-114
Reinforcement moments: B-114
Maximum shear forces: B-115
B 13.2.4 Required Reinforcement B-115
B 13.2.5 Punching Shear Verification B-115
Punching shear loads B-116
Integration of third-party program for the punching shear verification B-117

Vol. C CADview & CADextract C-1

C 1 Introduction C-1
CADview C-1
CADextract C-1

C 2 CADview C-2
C 2.1 Introduction C-2
C 2.2 Management of CAD drawings C-2
C 2.3 Importing a CAD drawing C-3
Setting the CAD drawing scale C-3
Defining the position of the CAD drawing C-4
Defining a rectangular cut-out C-5
Setting the layer properties C-5

C 3 CADextract C-5
C 3.1 Introduction C-5
C 3.1.1 Outline detection C-6
C 3.1.2 Structural member detection C-6
C 3.1.3 Final check and export of extracted CAD data C-6
C 3.2 Basics of handling C-6
Starting CADextract C-7
Display area C-7
Visibility C-7
Step sequence C-8
Graphics input C-8
Filters C-9
C 3.3 Introductory example C-9
Loading the introductory example C-9
Step 1: Drawing elements for outline detection C-9
Step 2: Outline of the floor slab C-10
Step 3: Drawing elements for structural member detection C-14
Step 4: Wall detection C-15
Step 5: Material assignment for walls C-16
Step 6: Closure of wall openings C-17
Step 7: Column detection C-18
Step 8: Material assignment to columns C-19
Step 9: Final checking and exporting the extracted data C-20
C 3.4 Design of a CAD drawing for an efficient work with CADextract C-20
Recommended preparation C-21
C 3.4.1 Outline detection C-21
Recommended actions C-21
C 3.4.2 Structural member detection C-21
Recommended actions C-21

Vol. D Building Option G D-1

D 1 Introduction D-1

D 2 Starting the Program and Organisation of the Data D-2
D 2.1 Starting the Program and the Building Window D-2
D 2.2 Organisation of the Data D-3
D 2.3 Handling Windows D-3
Program, building and slab windows D-3
Open, active and visible building and slab windows D-3
Change to a slab window D-4

D 3 Geometrical Building Model D-5
D 3.1 Coordinate System D-5
D 3.2 Storys D-5
D 3.2.1 The building Definition dialog D-5
D 3.2.2 Master Slabs / Floor Slabs D-6
D 3.2.3 The Story Properties in the Building Definition Dialog D-6
D 3.3 Creating and Managing Storys and Master Storys D-8
D 3.4 Modifying Storys D-9
D 3.4.1 General D-9
Limitations in the slab input D-9
Extensions for wall and column input D-9
Common characteristics of all slabs D-9
Additional input for building D-10
D 3.4.2 Slab Strips for a Frame Model D-10
D 3.4.3 Wall Pillars D-11
Properties of wall pillars D-12
D 3.4.4 Reinforcement for Wall Pillars D-12
D 3.4.5 Visibility Control for the Wall Pillars D-14
D 3.5 Creation of the Geometric Building Model D-15
D 3.5.1 Assembling the building from storys D-15
Support of wall pillars and supports on lower slabs D-15
Heights, weights and masses of vertical elements D-16
Illustration of vertical elements in the geometry model D-16
Nature of ground surface / basement storys D-17
Permitted slab types in a building analysis D-17
D 3.5.2 Creation and Updating of the Geometric Building Model D-17
Graphical documentation of the building D-17
Tabular documentation of the building D-18
Reset D-19
D 4 Special Ways of Controlling Visibility D-19
Invisible structural parts dimmed visible D-19
D 4.1 Subsystems D-19
D 4.1.1 The Subsystems Dialog D-20
D 4.1.2 Steps in Defining a New Subsystem D-20
D 4.1.3 Automatically-generated Subsystems D-21
D 4.2 Clipping D-22
D 4.3 Name Filter D-23
Special functionalities D-23
D 5 Stacked Slab Model D-23
D 5.1 General Considerations regarding the Stacked Slab Model D-23
D 5.2 Stacked Slab Model with Vertical Loads D-23
D 5.2.1 Overview of Model D-24
D 5.2.2 Treatment of Columns D-26
Skew columns D-26
D 5.2.3 Treatment of Wall Pillars D-26
On the footprint of wall pillars D-27
Load distribution along the footprint D-28
Point support D-28
Long sections of plane walls D-29
D 5.2.4 Limits of the Load Transfer Model D-30
D 5.3 Tab sheet 'Stacked Slab Model' D-30
D 5.3.1 Creation of the Model D-30
D 5.3.2 Starting the Analysis D-31
D 5.3.3 Results in the Individual Floor Slabs D-31
D 5.3.4 Results in the Columns and Walls D-31
Results for D-31
Output of results D-31

D 6 BIM-Schnittstelle D-33
D 6.1 Einleitung D-33
BIM-Modell ≠ Berechnungsmodell D-34
Das Modul BIMlink D-34
Generierung des Berechnungsmodells D-34
D 6.1.1 Beschreibung der BIM-Schnittstelle D-34
D 6.2 Einsatz von CEDRUS in der BIM-basierten Planung D-35
D 6.2.1 Extrahieren des Berechnungsmodelles D-35
Szenario 1: Architektur BIM-Modell direkt in CEDRUS einlesen D-35
Szenario 2: BIM-'Schalungsmodell' in CEDRUS einlesen D-35
Szenario 3: BIM-'Berechnungsmodell' in CEDRUS einlesen D-35
D 6.2.2 Nachführung des Berechnungsmodelles (Update) D-36
D 6.3 BIMlink D-36
D 6.4 Das Register 'Import' D-37
D 6.4.1 Einstellungen IFC-Schnittstelle D-37
D 6.4.2 BIM-Element D-38
D 6.4.3 Elementtyp D-39
D 6.4.4 Elementfilter D-39
D 6.4.5 Zuweisung der Baustoffe D-40
D 6.5 Das Register 'BIM-Geschosse' D-40
D 6.5.1 Attributdialog 'BIM-Geschoss' D-41
Geometrie im Berechnungsmodell D-41
Zugehörige BIM-Elemente D-41
Hintergrundplan D-42
D 6.6 Das Register ‘Tragwerksmodell’ D-42
D 6.6.1 Dialog Generierungsparameter D-43
D 6.6.2 Tabelle Geschosse D-43
D 6.7 Das Register 'Update' D-43
D 6.7.1 Layergruppe Update D-44
D 6.7.2 Dialog BIM-Element: Update D-44
D 6.8 Einsatz von BIMlink bei 2D-Geschossplänen D-45
D 6.8.1 Schritt 1: Geschossstruktur mit 2D-Plänen erstellen D-45
D 6.8.2 Schritt 2: CEDRUS-Geschosse konstruieren D-45
D 6.9 Export in IFC-Datei D-46
D 7 BIM-Interface to Nemetschek Allplan D-46
D 7.1 Introduction D-46
CAD model ≠ analysis model D-46
Creation of the analysis model D-46
D 7.2 Workflow of a 3D CAD import D-47
1. Importing the CAD model from an Allplan XML file D-47
2. Selection and adjustment of the CAD elements to be modeled D-48
3. Definition of building materials and generation parameters D-48
4. Generation of the calculation model D-49
5. Check and refine the generated model D-50

Vol. E Building Option H E-1

E 1 Introduction E-1

E 2 Analysis with the CEDRUS Building Model E-1
E 2.1 Tab sheet 'Horizontal Loads' E-1
E 2.1.1 Management of Loadings E-2
E 2.1.2 Load Input Dialog E-3
E 2.1.3 Options Dialog E-4
E 2.1.4 Updating the Loads E-5
E 2.1.5 Evaluation of the Mass and Stiffness Centres E-5
Centre of mass E-6
Centre of stiffness E-6
E 2.2 Tab sheet 'Analysis' E-8
E 2.2.1 Combination Loadings E-8
Defining combination loadings E-8
Automatically generated combinations E-8
E 2.2.2 Definition of Design Spectra E-10
User-defined spectra E-10
E 2.2.3 Setting-up a Masonry Analysis E-12
E 2.3 Tab sheet 'Stacked Slab Model' E-13
E 2.3.1 Building the Model and Selecting the Options E-13
Analysis model for horizontal loads E-13
E 2.3.2 Results Selection Field and Starting the Analysis E-14
E 2.3.3 Results in Walls and Columns E-15
E 2.3.4 Results in Floor Slabs E-16
E 2.3.5 Design Specifications for Slabs with Seismic Actions E-18
E 2.4 Murus Analysis from A - Z E-19
On creating the building model we have to consider: E-19
Loadings E-19
E 2.5 PYRUS calculation for the dimensioning of columns E-23
E 2.5.1 Overview E-23
E 2.5.2 Steps in a PYRUS-calculation E-24
1. Step: set up a PYRUS-calculation E-24
2. Step: open a PYRUS-calculation E-24
3. Step: Check / modify the column properties E-24
4. Step: Groupings E-26
5. Step: Design E-26
6. Step: Documentation E-28
E 2.5.3 Setting up a PYRUS calculation E-29
The dialog 'Options PYRUS-analysis' E-29
E 2.5.4 The dialog 'PYRUS Calculation' E-29
Open dialog E-29
Overview of the dialog E-30
Set column visibility E-32
Sort the column list E-32
Column list filter E-32
View by changes E-33
Tab sheet 'Columns' E-33
Tab sheet 'Connections' E-34
Tab sheet 'Groups' E-34
Tab sheets 'Preliminary design' and 'Design' E-35
Tab sheet 'Verification' E-37
Tab sheet 'Report' E-38
E 2.5.5 Columns and punching shear E-40
Force flow through slab with continuous columns E-40
E 2.6 Berechnung von Stahlbetonwänden E-40
CEDRUS (S)-Berechnung von Wandscheiben mittels FEM E-41

Alternative Verfahren für Geschosswände E-41
FAGUS (A)-Berechnung als Biegestab oder Spannungsfeld E-41
E 2.7 FAGUS (A)-Berechnung der Wandbewehrung E-42
E 2.7.1 Einleitung E-42
Definition der Wandpfeiler E-42
Spezifikation der Wandbewehrung E-43
Bemessung und Nachweis der Bewehrung E-44
FAGUS (A)-Berechnung mit dem Plattenstapelmodell E-45
FAGUS (A)-Berechnung mit dem Stabmodell E-45
E 2.7.2 Wall reinforcement dialog E-45
Synchronisation von 3D-Grafik und Tabellenansicht E-46
Tabelle Wandpfeiler E-47
Tabelle Knotenbewehrung E-47
Tabelle Wandbewehrung (vertikal und horizontal) E-47
Druckausgabe E-48
E 2.8 CEDRUS (S)-Berechnung von Wandscheiben E-49
E 2.8.1 Einleitung E-49
Wandscheibe aus Teilsystem E-49
Berechnung mit Gleichgewichtskräften E-49
E 2.8.2 Ablauf einer Wandscheiben-Berechnung E-50
1. Teilsysteme bilden E-50
2. CEDRUS(S) Berechnung aufsetzten E-50
3. Horizontallasten aktivieren (falls erwünscht) E-50
4. FE-Berechnung einer Wandscheibe öffnen E-50
5. FE-Scheibenberechnung ausführen E-50

E 3 Model for STATIK E-51
E 3.1 General E-51
E 3.2 Properties and Parameters of the Frame Model E-52
E 3.2.1 Modelling the Slabs E-52
How are downstands made into beams? E-54
What are slab strips and how are they made into beams? E-55
On the 'Slab with Condensed Stiffness' model E-56
E 3.2.2 Modelling the Wall Pillars E-57
With and without resistance to horizontal forces E-58
Embedded in membrane slab E-58
Plane or general wall pillars E-58
E 3.2.3 Modelling the Columns E-58
E 3.2.4 When are vertical elements placed one on top of the other? E-59
Point support of a wall pillar E-60
Segment support of a wall pillar E-60
Point support of a column E-60
E 3.2.5 Node support E-61
Discussion of the node supports E-62
E 3.2.6 Supports of the STATIK model E-63
Storey topology E-63
Base level E-64
Embedment levels E-64
E 3.2.7 Automatically-Generated Partial Systems E-65
E 3.2.8 Generated Loadings and Mass Distributions E-65
Generation of storey weights for dynamic analyzes E-66
Vertical loads E-66
E 3.3 Generation of the Frame Model E-68
Optionendialog für das Stabmodell E-68
Generation of the frame model E-69
Generating the input and call STATIK E-70
E 3.4 Steps to Obtain a Useful Frame Model E-71
E 3.4.1 Preparatory Work E-71
E 3.4.2 The Cantilever Wall Model E-71
Visibilty Control E-72
Tidying up the generated cantilever wall model E-74
E 3.4.3 The Frame Model E-76
E 3.4.4 The Slab Stiffness Model E-76

E 4 Example with Earthquake Analysis E-77
E 4.1 Problem Description E-77
E 4.2 Solution Part 1 with CEDRUS E-77
E 4.2.1 Starting the Program E-77
E 4.2.2 Input of the Ground Floor Slab E-77
E 4.2.3 Input of the Master Story for the Upper Storys E-78
E 4.2.4 Introducing the storys OG1, OG2 and DG E-78
E 4.2.5 Generation of the Building Model E-79
E 4.2.6 Stacked Slab Model E-80
Results in the vertical elements E-80
Results in the floor slabs E-81
E 4.2.7 Frame Model E-81
E 4.3 Solution Part 2 with STATIK E-84
E 4.3.1 Checking the Generated Model E-84
E 4.3.2 Free Vibration Analysis E-85
E 4.3.3 Equivalent Forces E-86
Note on the equivalent force method of analysis: E-87
E 4.3.4 Response Spectrum Method E-88
Torsion E-90
Interstory drift E-91
E 4.3.5 Dimensioning (Reinforced Concrete Analyses) E-92
The reinforcement of load-bearing walls E-92
Dimensioning of the wall pillar reinforcement for equivalent forces E-93
Dimensioning of wall pillar reinforcement for the response spectrum results E-97

Vol. F Dynamic Analysis F-1

F 1 Natural Vibrations F-1
F 1.1 Basic Theory F-1
F 1.2 Mass Distribution F-1
F 1.3 Setup of a Natural Vibration Analysis F-2
F 1.4 Output of Results F-2

Vol. G Walls G-1

G 1 Introduction G-1

G 2 Basic Theory G-1

G 2.1 Element Model and Solution Method G-1

G 2.2 Modelling G-2

G 2.2.1 Geometry G-2

G 2.2.2 Thickness and Material G-2

Isotropic Material G-3

Orthotropic Material G-4

G 2.2.3 Point Supports G-4

G 2.2.4 Line Supports G-5

G 2.2.5 Lines of Symmetry G-6

G 2.2.6 Loads G-6

Area loads (i.e. loads per unit area) G-6

Line loads G-6

Point loads G-6

Displacements of point supports G-7

G 2.3 Results G-7

G 2.3.1 Raw Results G-7

G 2.3.2 The Structuring of the Output of Results G-7

The quantities for load cases and load case combinations G-8

Quantities for limit state specifications (envelope values) G-9

Required reinforcement G-9

Forms of Presentation G-10

G 3 Slab with Normal Forces G-11

G 3.1 Model G-11

G 3.2 Changing the Structural Type G-12

G 3.3 Results G-12

Vol. H Prestressing Module H-1

H 1 Introduction H-1

H 1.1 Overview H-1

H 1.2 Tendons and Supports H-2

H 1.2.1 Tendons H-2

H 1.2.2 Supports H-2

H 1.3 Input Procedure H-2

H 2 Basics H-4
H 2.1 Preliminary Design H-4
H 2.2 Tendon groups H-5
H 2.3 Tendon-Dialog H-6
Attributes H-6
Tensioning process H-6
Options H-8
H 2.4 Supports- and Tendon point-Dialog H-9
Attribute H-9
H 2.5 Tendon Geometry H-10
H 2.5.1 Geometrical description in the plan view H-10
H 2.5.2 Geometrical description in the side view H-10
Segments of tendons H-11
Segments of standard elements H-11
Parabola segments of third order H-14
H 2.6 Dialog 'Output settings' H-15
H 2.7 Standard presentation or Plan view H-15
Standard presentation H-15
Plan view H-15
H 2.8 Deviation Forces H-16
H 2.8.1 Exclusive slab model H-16
H 2.8.2 Slab with normal forces H-16
H 2.8.3 Load generation for slab with constant thickness H-16
Vertical deflection forces H-17
Horizontal deflection forces H-17
Frictional forces H-17
Anchor forces H-18
H 2.8.4 Load generation for slab with different thicknesses H-18
Conditions H-18
Approach solution H-18
H 2.9 Dimensioning of the Non-Prestressed Reinforcement H-20
H 2.9.1 Prestressing as an external action H-20
H 2.9.2 Prestressing as self-equilibrating stress state H-20
The design criteria H-20
The dimensioning conditions H-21
Dimensioning as beams H-21
Cross sectional resistance H-21
Axial force due to prestressing H-23
Constraint moments H-24
Direction of the tendons and of the associated beam sections H-24
Remarks on the Results Tables H-25
H 2.9.3 Use of program H-27
Input in the tabsheet 'Calculation' H-27
Input in the Tabsheet 'Results' H-28

H 3 Examples H-28
H 3.1 Flat Slabs H-28
H 3.1.1 Description of problem H-28
H 3.1.2 Tab sheet 'Structure' H-28
H 3.1.3 Tab sheet 'Loads' H-28
H 3.1.4 Tab sheet 'Prestressing' H-29
Input of tendons and supports H-30
Checks H-32
H 3.1.5 Tab sheet 'Calculation' H-35
H 3.1.6 Tab sheet 'Results' H-36
H 3.1.7 Checklist H-37
H 3.2 Two Span Beams H-38
H 3.2.1 Problem description H-38
H 3.2.2 Results H-39
H 3.3 Tips and Tricks H-42
H 3.3.1 Duplication of groups H-42
H 3.3.2 Generating possibilities H-42
H 3.3.3 Checks H-42
H 3.3.4 Detecting tendons lying on top of each other H-42
H 3.3.5 Reference height of supports with different slab thicknesses H-42

Vol. I Externally Bonded Reinforcement I-1
I 1 Introduction I-1
I 2 Running an EBR Analysis I-1
I 2.1 Analysis procedure I-1
I 2.2 Specification of the reinforcement I-3
I 2.3 Analysis using Beam Sections I-4
I 2.4 Checklist: Verification procedure for EBR I-5
I 2.5 Results I-5

Vol. J Bewehrung + Traglast J-1
J 1 Einleitung J-1
Einschränkung J-1
Kontexthilfe J-1
J 1.1 Bewehrungsmodul J-1
J 1.2 Traglastmodul J-2
J 2 Grundlagen J-2
J 2.1 Materialmodell J-2
J 2.1.1 Momenten-Krümmungsdiagramm J-3
J 2.1.2 Plastizitätstheorie J-3
J 2.1.3 Fliessbedingung und Fliessgesetz J-4
Kinematisches Verfestigungsmodell J-5
J 2.1.4 Plastisches Widerstandsmoment J-6
J 2.2 Bewehrungsmodell J-7
J 2.2.1 Bewehrungsfelder J-7
J 2.2.2 Platten- und Unterzugsbewehrung J-8
Aktive Bewehrungslagen J-8
Unterzugsbewehrung J-8
J 2.2.3 Schiefe Bewehrung J-8
J 2.3 Lösungsverfahren J-10
Implementierung des Materialgesetzes J-10
J 2.3.1 Nichtlineare Analyse J-10
Inkrementelle, iterative Berechnung J-11
Einfluss von Ausgangszustand und Lastgeschichte J-11

J 3 Bewehrungsmodul J-12
J 3.1 Grundkonzepte J-12
J 3.1.1 Bemessung auf Tragsicherheit J-12
J 3.1.2 Elastische Bemessung als Optimierungsaufgabe J-13
J 3.1.3 Plastische Bemessungsoptimierung J-14
J 3.1.4 Bemessung auf Gebrauchstauglichkeit J-15
J 3.2 Bemessungsablauf J-17
J 3.2.1 Vorbereitung der Ausgabezonen J-18
J 3.2.2 Erzeugen eines Layouts J-18
J 3.2.3 Eingabe Bewehrungsfelder J-19
Untere Bewehrung J-19
Obere Bewehrung J-23
J 3.2.4 Elastische Bemessung J-24
J 3.2.5 Bemessungsoptimierung J-24
Bemessungsmonitor J-25
J 3.2.6 Interpretation der Resultate J-26
J 3.2.7 Erstellung Bewehrungsskizze J-26
J 4 Traglastmodul J-29
J 4.1 Grundkonzepte J-29
Nichtlineare Berechnungsverfahren J-29
J 4.1.1 Berechnungsmodell J-29
Nichtlineare Momenten-Krümmungsbeziehung J-30
Verfestigung J-30
Lastgeschichte J-30
Teilsicherheitsbeiwerte J-31
J 4.1.2 Abbruchkriterien J-31
J 4.1.3 Berechnungssteuerung J-34
J 5 Kontexthilfe J-35
J 5.1 Das Registerblatt /Bewehrung/ (Option) J-35
J 5.1.1 Layoutverwaltung J-35
Neuen Layout erzeugen J-36
Layout löschen J-36
Layouteinstellungen ändern J-36
Liste der Bewehrungsfelder J-36
J 5.1.2 Eingabe der Bewehrungsobjekte J-36
Bewehrungsfeld (Schaltfläche im Registerblatt /Bewehrung/) J-36
J 5.1.3 Darstellungs- und Eingabesteuerung J-36
Obere/Untere Bewehrung (Schaltfläche im Registerblatt /Bewehrung/) J-37
Aktuelle Bewehrungsrichtung (Schaltfläche im Registerblatt /Bewehrung/) J-37
Sichtbarkeit der Bewehrungsfelder (Schaltfläche im Registerblatt /Bewehrung/) J-37
J 5.1.4 Das Dialogfenster Darstellung J-37
Toleranzwinkel J-37
Feldbeschriftung J-37
Darstellungsoptionen Bewehrungsgehalt J-38
Plastifizierung J-38
J 5.1.5 Das Dialogfenster Bewehrungsfeld J-38
Zuweisung zu Materialzone oder Unterzug J-39
Feldtyp (in Dialogfenster Bewehrungsfeld) J-39
Verankerung (in Dialogfenster Bewehrungsfeld) J-39
Minimalgehalt (in Dialogfenster Bewehrungsfeld) J-39
Lage (in Dialogfenster Bewehrungsfeld) J-40
Gruppe (in Dialogfenster Bewehrungsfeld) J-40
Darstellung der Feldbeschriftung (in Dialogfenster Bewehrungsfeld) J-40

Stahleigenschaften (in Dialogfenster Bewehrungsfeld) . . . J-40
J 5.1.6 Bemessung . . . J-41
1. Definition der Einwirkung . . . J-41
2. Berechnung Bewehrungsbedarf . . . J-41
3. Eingabe der Bewehrungsfelder (s. auch Grundlagen: Bewehrungsmodell) . . . J-41
4. Elastische Bemessung (im Registerblatt /Bewehrung/) . . . J-42
5. Plastische Bemessungsoptimierung (im Registerblatt /Bewehrung/) . . . J-42
J 5.1.7 Darstellung der Resultate (im Registerblatt /Bewehrung/) . . . J-42
Darstellung der Bewehrungsfelder . . . J-42
Elementweise Darstellung der Resultate . . . J-43
J 5.1.8 Das Bearbeiten-Kontextmenü (Bewehrungfelder) . . . J-44
J 5.2 Das Dialogfenster Monitor . . . J-44
J 5.2.1 Monitorvariablen . . . J-45
Monitorvariablen Bemessungsoptimierung (im Dialogfenster /Monitor/) . . . J-45
Monitorvariablen Traglastanalyse (im Dialogfenster /Monitor/) . . . J-45
J 5.2.2 Das Dialogfenster Monitoreinstellungen . . . J-45
J 5.3 Das Registerblatt /Traglast/ (Option) . . . J-46
J 5.3.1 Verwaltung der Lastgeschichte . . . J-46
Neue Lastgeschichte erzeugen . . . J-46
Lastgeschichte löschen . . . J-47
Lastgeschichte ändern . . . J-47
J 5.3.2 Darstellungssteuerung . . . J-47
Obere/Untere Bewehrung (Schaltfläche im Registerblatt /Traglast/) . . . J-47
J 5.3.3 Traglastanalyse (im Registerblatt /Traglast/) . . . J-47
Ablauf einer Traglastanalyse . . . J-48
Basisdaten einer Traglastanalyse . . . J-48
Traglastoptionen (im Registerblatt /Traglast/) . . . J-48
Berechnung des Widerstandes (im Registerblatt /Traglast/) . . . J-48
Bestimmung des Traglastfaktors (im Registerblatt /Traglast/) . . . J-48
J 5.3.4 Das Dialogfenster Traglastoptionen . . . J-48
Verfestigung . . . J-49
J 5.4 Datenaustausch mit CAD-Systemen . . . J-50
J 5.4.1 Einleitung . . . J-50
J 5.4.2 Bemessung: Schema des Datenflusses . . . J-50
J 5.4.3 Austausch der Struktur/Schalungsgeometrie . . . J-51
J 5.4.4 Austausch Bewehrungsfelder . . . J-51
J 5.4.5 Traglastanalyse: Schema des Datenflusses . . . J-52

Vol. K Foundations . . . K-1

K 1 Introduction . . . K-1
Types of Foundations . . . K-1
Analysis Model . . . K-1

K 2 Modelling the Soil-Structure Interaction . . . K-2
Influence of the Ground . . . K-2
Influence of the Stiffness of the Slab . . . K-3
Influence of the Loading of the Slab . . . K-3
Influence of Piles . . . K-3
Influence of Multiple Slabs . . . K-3
Limits of the implemented Method . . . K-4

K 3 Process of a Foundation Analysis . . . K-4
Step 1: Calculation of the Slab Stiffness in CEDRUS . . . K-4
Step 2: Calculation of the Slab Bedding in LARIX . . . K-4
Step 3: RC design of the Slab in CEDRUS . . . K-4
Design of the Bottom Slab for Limit States . . . K-5
State of the LARIX-Foundation . . . K-5

K 4 Story with LARIX-Foundation K-5
Specification of a LARIX-Foundation K-6
Specification of Piles K-6
Analysis of a LARIX-Foundation K-7
K 5 Building Model with LARIX-Foundation K-7
Dialog LARIX settlement calculation K-7
Opening a LARIX settlement project K-8
Import the LARIX-Foundation K-8
Delete the LARIX settlement project K-8
Visualization and Documentation of the LARIX foundation K-8

Vol. L CAD Interface L-1

L 1 Import L-1
L 1.1 Background drawings L-1
L 1.1.1 DXF-Lines L-1
L 1.1.2 DXF/DWG-Drawings L-2
L 1.2 Structure L-2
L 1.2.1 SIN Interface L-2
Import of structural data L-2
Structure recovery L-2
Menu File > Import L-2
L 1.2.2 Structure import from Allplan L-3
L 1.2.3 IsbCAD Interface L-3
L 1.2.4 Allplan 3D L-3
L 1.3 Loadings L-3
L 1.3.1 LIN Interface L-3
L 1.3.2 IsbCAD Interface L-4
L 1.4 Reinforcement for the reinforcement module L-4
L 1.4.1 ECB Interface L-4
L 2 Export L-4
L 2.1 SIN/LIN Structure- und Loading data L-5
L 2.2 Reinforcement results from Base module L-5
L 2.3 Reinforcement from reinforcement module L-5

Vol. A Introduction

A 1 Overview

A 1.1 Introduction

Basically CEDRUS is a Finite Element Program for the linear-elastic, static and dynamic analysis of plate (slab) and wall structures. It is especially focused on reinforced concrete structures and outputs the corresponding limit state moments for the reinforcement and the required reinforcement contents.

Besides the basic module for plates the following modules are also available as options:

- Buildings module with the stacked slab model for the top-down analysis of vertical loads
- Buildings analysis model: Model to accommodate horizontal loads and space frame models for earthquake analysis with the space frame program STATIK
- Prestressing
- Dynamics (analysis of eigenvalues)
- Optimum design (with possible plastic compensation) of the slab reinforcement
- Determination of ultimate loads for slabs
- Externally bonded reinforcement
- Walls, also combination slabs/walls

CEDRUS can be used to solve difficult structural analysis tasks. However, a number of error sources have to be considered, including modelling of the structure, data input, numerical inaccuracy problems, interpretation of results and of course actual program errors. With such complex software, despite all the care taken in its development, programming errors cannot be completely excluded. Therefore, an important requirement for the successful use of CEDRUS is an adequate understanding of the engineering principles as well as checking the results using simplified calculations and plausibility considerations.

The firm Cubus AG will continue to develop CEDRUS. Critique, suggestions and wishes from users in engineering practice are always welcome. Customers will be informed of major changes or developments.

Cubus reserves the right to make small changes in the software not described in the manual, such as self-explanatory changes in the dialogues.

A 1.2 Documentation

A 1.2.1 CEDRUS: Specific Topics

All topics specific to CEDRUS are treated in the first document of the manual. It is structured as follows:

A1 Overview with the following sub-chapters

A 1.1 **Introduction**

A 1.2 **Documentation**

A 1.3 **Program Installation**

A 1.4 **Slab Analysis Procedure:** The basic range of performance and the user concept for slab analyses are explained.

A 1.5 **Procedure for a Building Analysis:** Since most slabs analysed using CEDRUS are part of a building, here the building analysis procedure is illustrated for vertical loads.

A 1.6 **Program Modules:** Here the performance range of the basic module and all optional program modules is described.

A 1.7 **CubusViewer:** The CubusViewer collects all outputs which together comprise the printable report. It gives a preview of the report, which can still be edited and printed as a whole or in part.

A 1.8 **CubusExplorer:** The CubusExplorer manages the analysis data.

A 2 Introductory examples

To help first-time users of the program, two introductory examples are provided. The autodidact cannot leave them out since the following chapters are of a reference character, i.e. the modules treated are described in detail and are thus not very suitable for gaining an overview of the whole program.

☞ *It is highly recommended that for learning to use CEDRUS these two fairly short chapters are worked through thoroughly before making a serious attempt at analysis.*

At the end of Part A with an overview and the general introduction, there are other parts with detailed descriptions of the base module and all optional program parts. They have a reference character, i.e. they provide all necessary information both to the basic theory and to the use of the program.

A 1.2.2 Shared Program Modules

Separate documents describe in detail the shared parts of the program used by various Cubus programs. Worthy of mention here are the documents for the following modules:

- **Graphics Editor:** It describes the basic program-independent functionality for the input, modification and presentation of objects on the screen.
- **CubusViewer:** This module is for printing and previewing output documents.
- **Construction materials and analysis parameters:** Here the handling of construction materials and the analysis parameters for the cross section program FAGUS described.

A 1.2.3 Context-specific Help

By this one understands information on the current context that the program provides during execution. It is activated using the function key <F1> or using the [?] buttons in the various dialogues.

A 1.2.4 Document Format and Document Access

As mentioned above, the program manual consists of a program-specific and several shared program documents. These documents are in **PDF format** and can be accessed directly in the menu 'Help'. The context-specific help also uses these documents and moves directly to the desired place. As a prerequisite, a version of Adobe ™ 's Acrobat Reader ™ has to be installed. A free version of this program can be downloaded and today it is essential as an addition to any operating system. It is recommended to work with version 9 (or newer) of the program.

Thus all the possibilities of Acrobat Reader are available, especially the full text search and printing the whole or parts of the document.

Supported PDF-Viewer and PdfViewer.ini-file

A PDF viewer must be installed and registered to open PDF files in the operating system. Supported are the official Acrobat Reader „AcroRd32" by Adobe ™ and a few more common readers.
The file ...*Cubus Software**CubusShared**Manuals**PdfViewer**PdfViewer.ini* contains all supported Viewer with the to be used call syntax. More Viewer can be inserted in this file, if they have the necessary call parameters and the call syntax is entered correctly.

If you make changes to the PdfViewer.ini file, copy it to a folder
...*Cubus Software**Config* because it will be overwritten during an installation update to the original location. The program first looks for the file there.

A 1.2.5 Printed manuals

In general, no completely printed manuals are supplied with the program. The corresponding documents are available in PDF-Format (Menu „Help …" and if required can be printed by the user.

A 1.3 Program Installation

Installation

First installation

The Cubus software and the customer-specific licenses are available as installation packages on the Internet for download and subsequent installation.

To perform a first installation on a Windows PC, open the Cubus Installer Homepage in your browser. The URL is as follows:

https:\\install.cubus-software.com

Log in there with your Cubus customer number (BN) and the associated InstallKey.

You will find your required BN and InstallKey on the delivery note of the software, which you have received by e-mail (PDF file) or together with the hardlock by post.

Following the instructions, download the three installation packages CubusServices, CubusSoftware and the CubusLicense (licenses) one after the other and open (or install) them.

Depending on the type of license, the Cubus programs only work with a USB hardlock, which you will receive after purchasing the software licenses together with the delivery note.

USB

Software updates

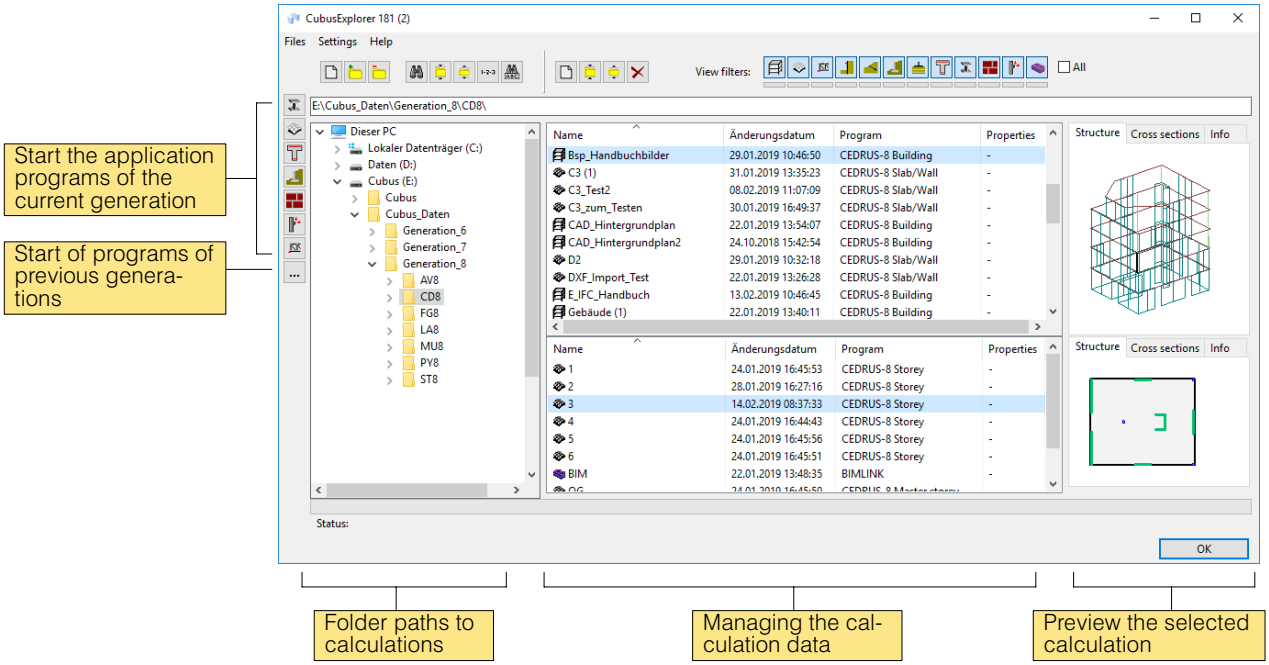
Updates are newer versions of programs within a program generation (e.g. CEDRUS-8). Updates are normally downloaded via the Internet. The CubusExplorer compares the installed program versions with the current versions on the Cubus Server and indicates if newer versions are available for downloading. The prerequisite for downloading program updates is an Update Service Subscription.

Installation Guide for Administrators

Detailed information about the Cubus installation packages can be found on the Cubus Installer Homepage (*https:\\install.cubus-software.com*).

CubusExplorer

Each installation of Cubus-programs containes the CubusExplorer. It is the central module with the use of programs and serves the following purposes:
Start each program
Managing the calculation data
Managing the installation via the CubusManager, which is called from the CubusExplorer.



Program start

CEDRUS can be started explicitly from the CubusExplorer sidebar or implicitly by choosing a CEDRUS project in the CubusExplorer. The CubusExplorer is an independent utility to manage analysis with different Cubus programs.

You may start the CubusExplorer via the Start menu of Windows, for standard installation:

> „Programs“ > „Cubus Software“ > „CubusExplorer“.

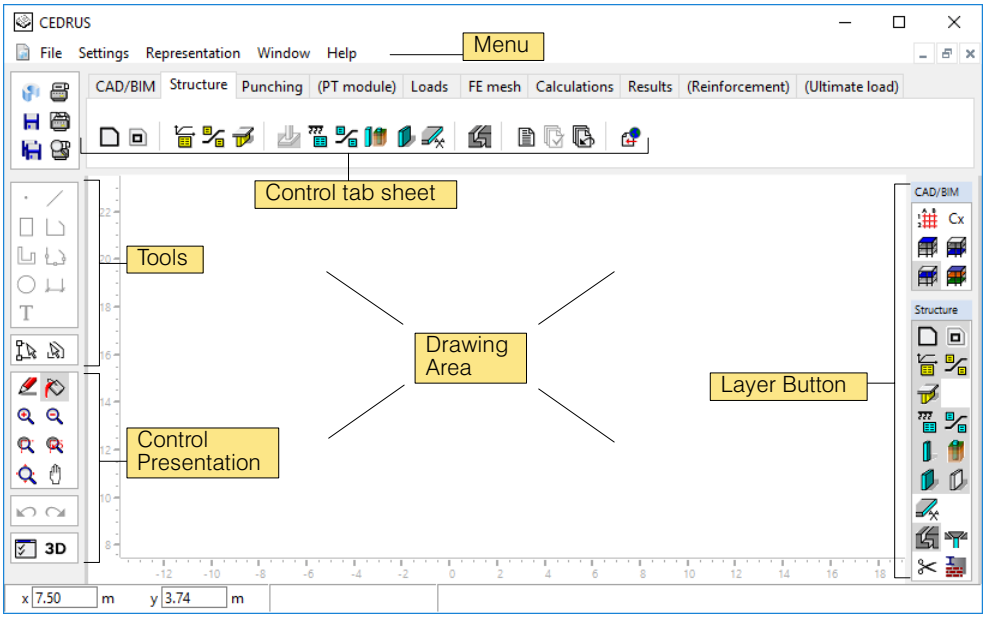
A 1.4 Slab Analysis Procedure

The aim of a standard CEDRUS analysis is the determination of bending moments and deflections, reactions and section forces for a series of given load cases and finally the determination of the design (dimensioning) section forces and from this information the slab reinforcement.

The following explanations should show, in a very summary way, how such an analysis proceeds.

Firstly, with the help of the CubusExplorer (see A 1.7) a new analysis is created. It includes the name of the analysis and the path to the directory in which the analysis data should be stored.

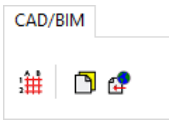
The program CEDRUS then opens with the following window:



The Control tab sheet of CEDRUS gives an overview of the essential steps in a slab analysis. Every tab sheet contains the tools to input and check the corresponding objects.

In the case of slab analysis the following tab sheets are used in a given sequence (from left to right):

Tab sheet 'CAD/BIM'



These are the tools to aid the input of the geometry. With the first button the grid of the axes can be defined. The second button allows the import of DXF/DWG files as well as floor plans from other CEDRUS slabs, with which the input is carried out.

These tools can greatly simplify the input. It is also possible to work without them and begin directly with the input in the 'Structure' tab sheet described below .

Tab sheet 'Structure'

These are the buttons to input and modify structural objects.

If it is a question of purely geometrical objects, such as an outline or openings, these are directly specified as lines, polygons, etc. (= drawing tools of the Graphics Editor).

If the objects possess properties, then a dialogue appears in which these can be specified before they are input. Such a dialogue is shown below in the case of the object Material Box.

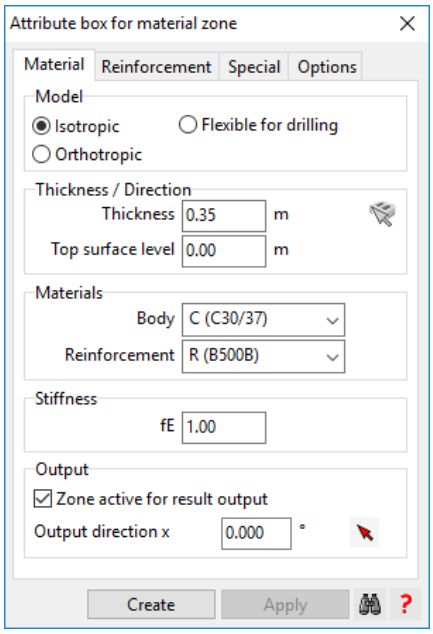
With regard to their geometry and properties, all objects can be very easily modified at any time.

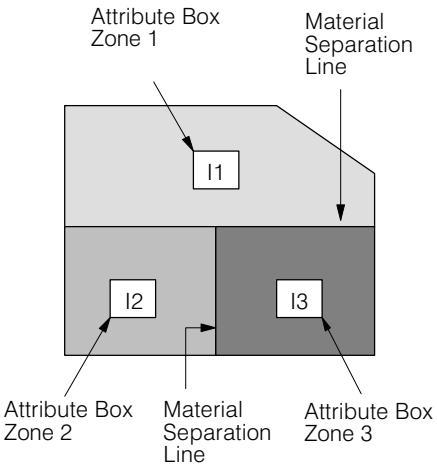
The tab sheet contains the buttons to input the following structural objects;

Outline: arbitrary polygon

Openings: arbitrary polygons

Properties summarized under '***Material***' such as slab thickness, material, etc. If these properties are not constant over the slab, it can be subdivided into individual material zones by means of ***Material Separation Lines***. An Attribute Box for material zones (abbreviated to ***Material Box***) can be placed in each material zone. It contains the values of the material properties. The figure shows the dialogue that belongs to an Attribute Box and a slab subdivided into three zones:





Downstand Beams: implemented as zones with orthotropic material properties.

Area Support Zones: The slab can be elastically-supported (i.e. assuming uniformly distributed springs, i.e. the simplified Winkler model) over a finite area, either wholly or in zones. Zones are bounded by separation lines. Elastically-supported zones contain an ***Area Support Box*** (box with coefficient of subgrade reaction).

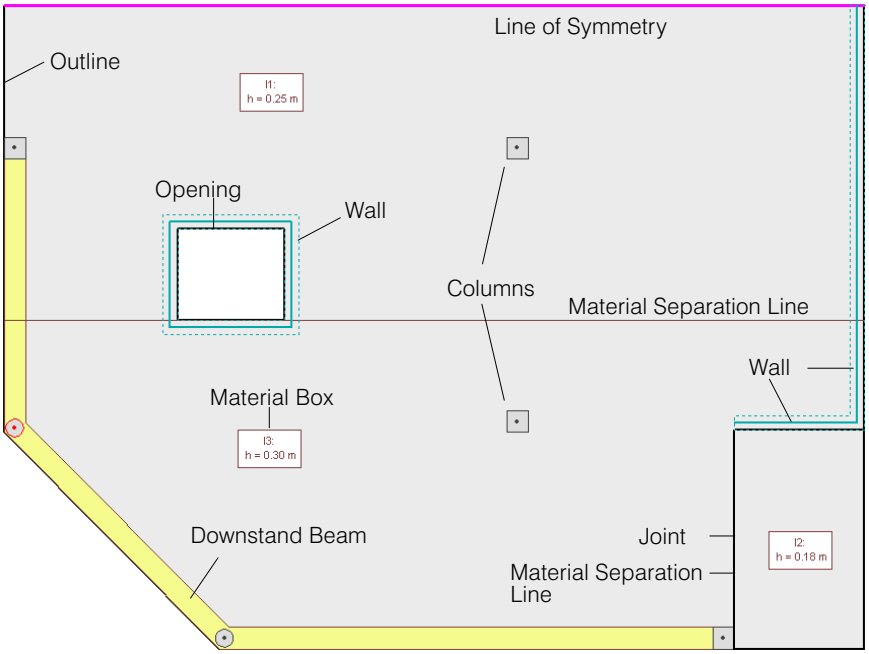
Columns: They may have arbitrary cross sections and define a support and a critical circular cross section to determine the punching shear resistance.

Walls: Walls in plan view are input as double-line polygons. In the analysis walls are modelled as line supports. They can be rotated freely, fixed or provided with spring stiffnesses in regard to settlement and rotation.

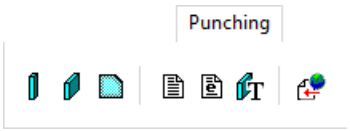
Lines of Symmetry: These are line supports with special support conditions to take into account lines of symmetry, which in principle only occur at plate boundaries.

Joints: There are three types of joint lines: hinge lines with optional spring stiffness, separation lines or closed joints.

As an example the following figure shows the result of a structure input:

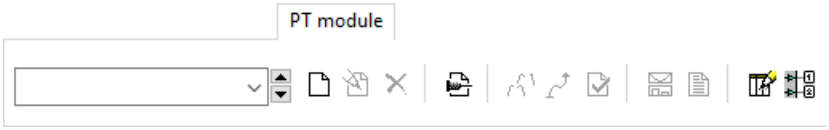


Tab sheet 'Punching'



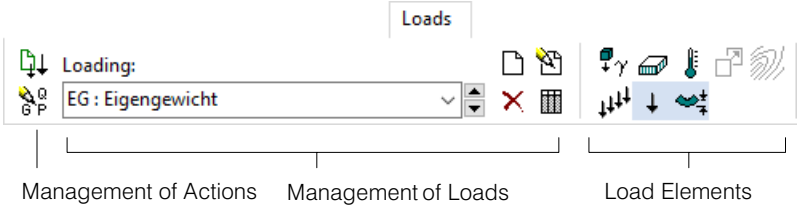
With the buttons above, the punching shear resistance can be determined for columns, ends and corners of walls, as well as for single concentrated loads.

Tab sheet 'PT module'



Prestressing cables are input in this tab sheet. Deviation and anchor forces are determined automatically and allow treatment of prestressing as an external action. In beam sections the prestressing cables can also be considered on the resistance side in the dimensioning of normal (non-pretensioned) reinforcement. Prestressing is an option in CEDRUS.

Tab sheet 'Loads'



Everything concerning load input is done here.

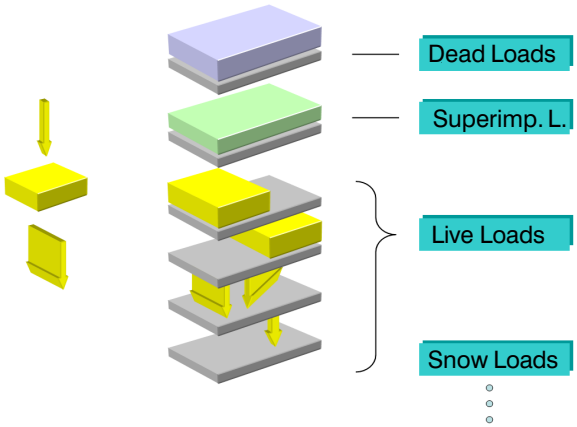
Loads are grouped into **Loadings**. A loading can contain one or more of the following load objects:

- dead load (self-weight)
- area load bounded by an arbitrary polygon
- curvature loads for differential temperature distribution
- constant or trapezoidal line load or line moment
- settlement of point supports

Loadings are the units that can be addressed in the output of results.

Actions are groups of loadings, which are considered as leading or accompanying actions in the automatic determination of limit design values assigned the corresponding factors.

Loads ⇨ Loadings ⇨ Actions



CEDRUS permits the input of **Loading Combinations** (creation of new loadings by combining existing loadings) and provides **Loading Generators** for the evaluation of different load positions.

As a speciality, also **Influence Fields** for sectional forces and reactions of columns with element support can be determined. They have to be specified as loadings.

Tab sheet 'FE mesh'

The FE mesh, especially its fineness, is very important in the modelling when using the FE method. Normally, CEDRUS generates the FE mesh automatically on the basis of the input of a value for the fineness as well as the direction of the mesh sides. These properties are transmitted in the form of an Attribute Box. They may be different for each zone, whereby the zoning is analogous to the material zones with mesh separation lines. In special cases the mesh can also be input manually.

In simple standard cases one comes without the tab sheet 'Calculations', since the limit state specifications to determine the section dimensioning quantities and the dimensioning specifications are created automatically.

The tab sheet provides the following functionality:

1) Definition of result combinations (combinations of loadings, for which results are required)

2) Definition of limit state specifications (superposition rule for dimensioning results)

3) Dimensioning specification with the definition of the criteria for the dimensioning of orthogonal (untensioned) reinforcement

4) Dimensioning specification for diagonal reinforcement

5) Dimensioning specification for prestressed slabs

6) Specification for the analysis of natural vibrations (Option)

7) Specification for displacement analyses in the cracked concrete state

8) Crack analyses

Tab sheet 'Results'

All results are called here using the lightning button in accordance with the other settings in the tab sheet.

1) Everything is listed here for which results can be obtained (Loadings, Result Combinations, Dimensioning Specifications, ...)

2) Depending on selection 1) the various quantities can be selected. If 1 is a loading, then here we get displacements, reactions, section forces, stresses and joint results.

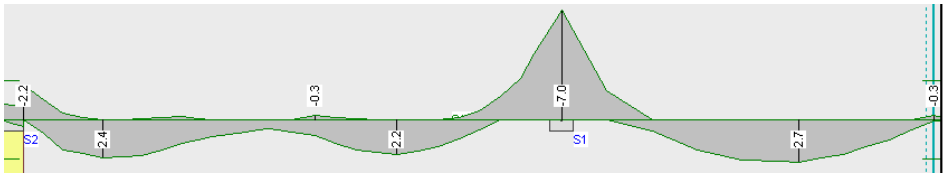
3) The forms of output corresponding to selection 2) are available. In the figure above there are isolines, graphical presentation of principal values, numerical results given on the FE mesh itself, result values, tabular output and section output.

4) If the output values include several components, the desired components can be selected here.

5) The upper button calls up a dialogue with additional output properties (e.g. equidistance of the isolines). Section results are defined with the lower button.

6) The results analysis is started using the lightning button.

Example: Reinforcement requirements for a slab strip of width 1.0 (beam section output)



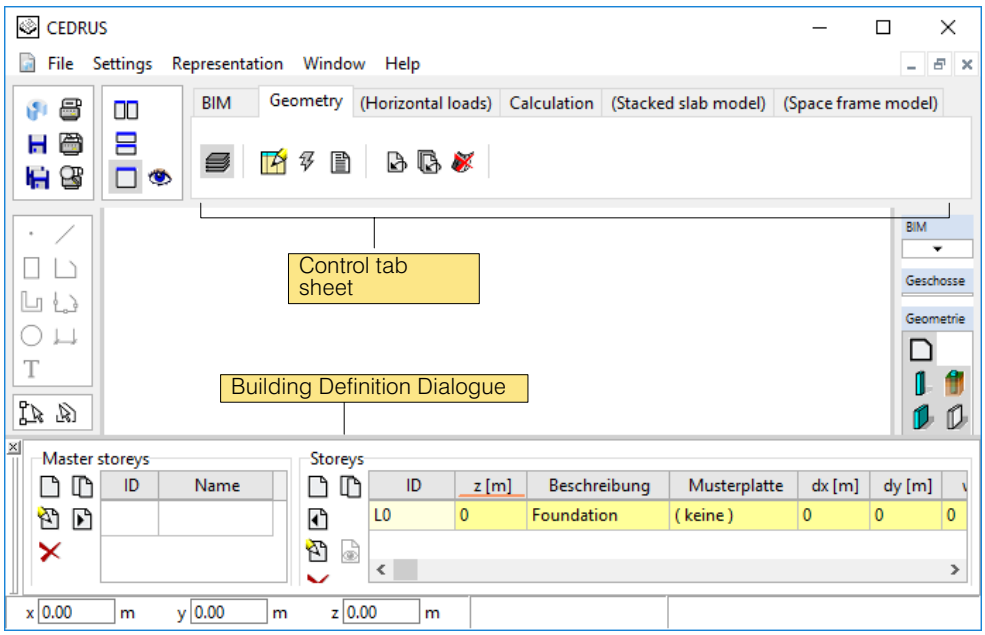
CubusViewer (see A 1.7) is used to print the output.

A 1.5 Procedure for a Building Analysis

The first step in a building analysis is the creation of a building model based on CEDRUS floor slabs. Using this building model various statical models can be derived and analysed. The basic model is the stacked slab model for vertical loads, which serves to analyse the building under vertical loads and especially to dimension the floor slabs. The procedure for such an analysis is described here. But further analysis models can also be developed. The user will find more information in Chapter A 1.6.3.

A CEDRUS analysis can either involve a single slab (or wall) or a building, which can be selected on opening a new analysis.

In the case of a building analysis the program's main window looks as follows:

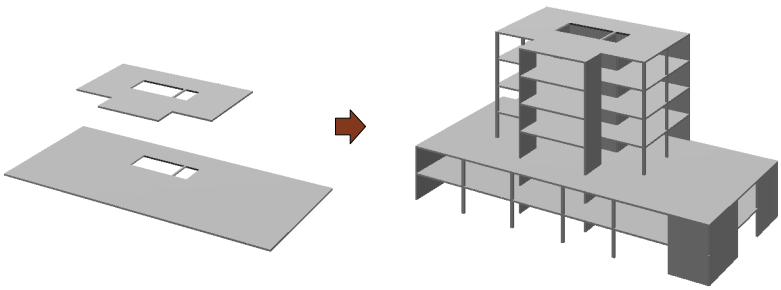


It differs from CEDRUS slab windows mainly by having a different Control tab sheet and by the Building Definition Dialogue.

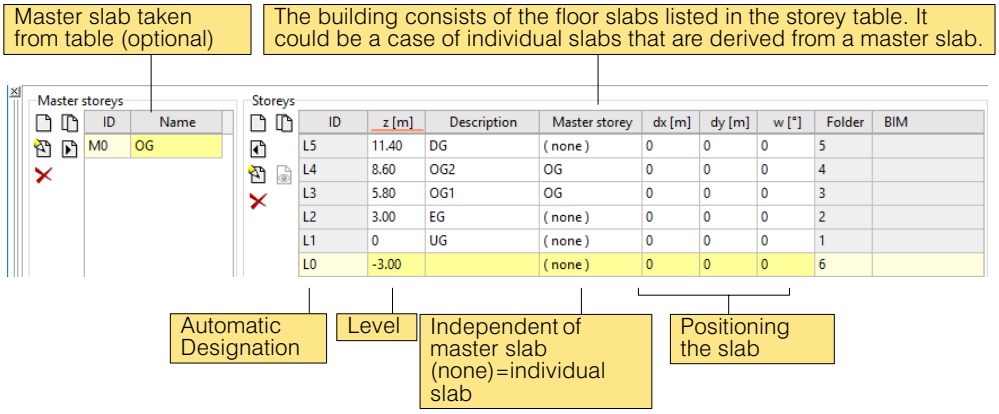
A 1.5.1 Geometrical Building Model

The geometrical model is created in the tab sheet 'Geometry':

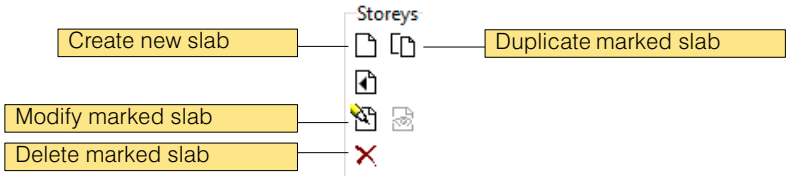
A building model is obtained by stacking the slabs.



For this purpose the visible building definition dialogue in the building window is available:

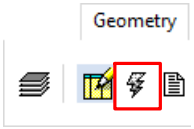


The creation and management of slabs in the storey table is carried out using the buttons on the left side of the table:

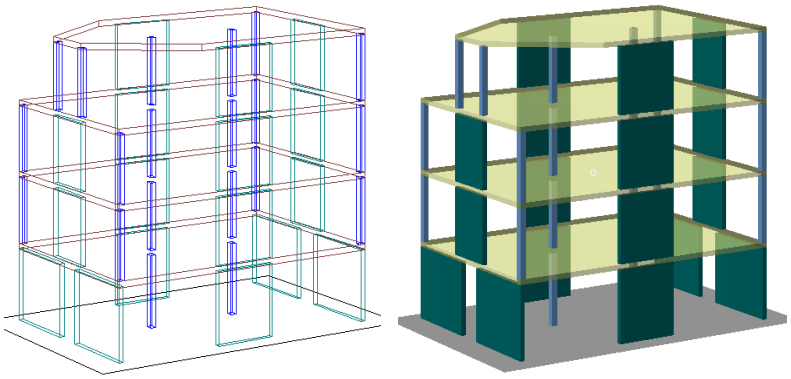


After creating a new slab it can be input directly or an existing slab can be imported. Then it has to be correctly placed by inputting its level (top of slab) and any horizontal positional correction.

If the building has several identical storeys (structure and loads), it is recommended to create a master slab (left table) and then to copy this into the storey table as often as required using the button .



By clicking on the lightning button the input slabs are checked and then the geometric building model is created and displayed:

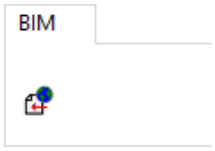


Besides the graphical output, tables with extracted area, volume and masses can be created.

The building model, of course, is only meaningful if an analysis can be performed with it. Thus using the Building module various analysis models can be created. The first such models is the stacked slab model for the top-down analysis of loads.

A 1.5.2 Geometric building model from BIM data

If a BIM file (for example in IFC format) with 3D building data is available, the geometric building model can be created from this. This is done in the tab sheet 'BIM'.



Clicking on this button opens the BIMlink dialog (see below). There, the 3D geometry is read in from the BIM file and from this the slab data for the structural model of CEDRUS are extracted. In this case, the user can influence the extraction of the slab data.

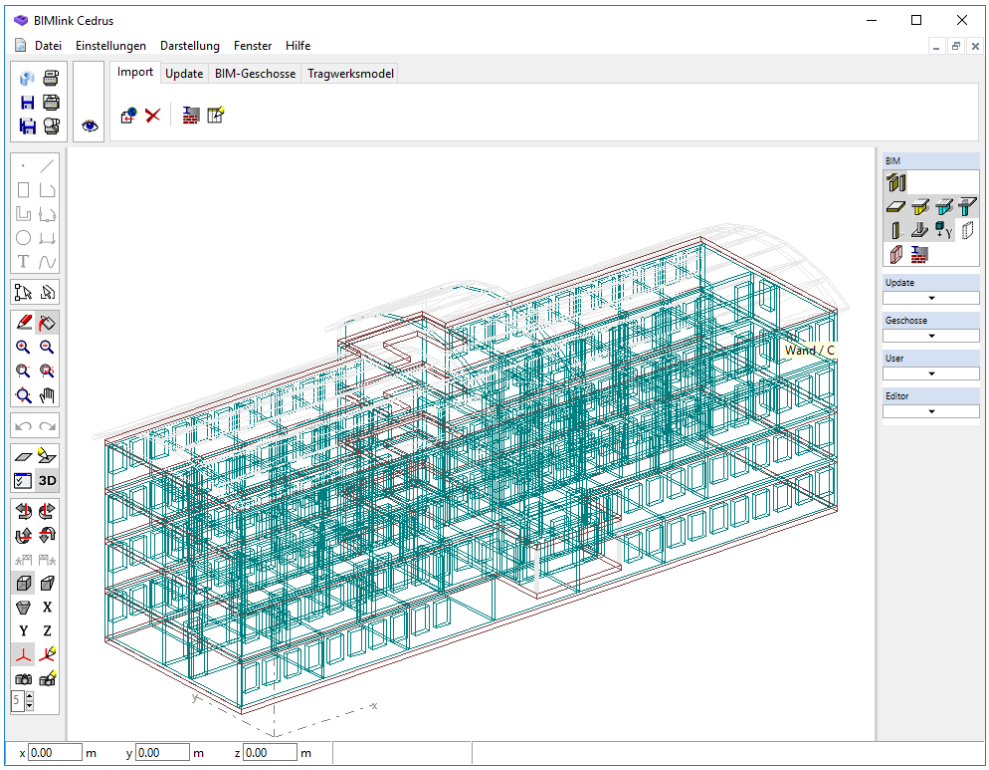
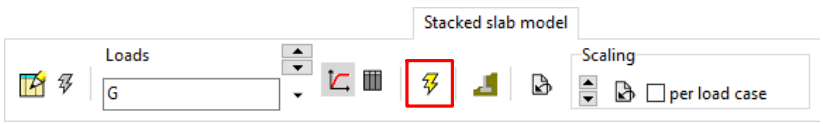


Fig. A–1 *BIMlink–Dialog*

A 1.5.3 Stack of Slabs Model

This is an analysis model for the top-down analysis of loads, which is dealt with in the tab sheet 'Stacked slab model':



After successfully creating the geometrical model the stacked slab model can be analysed using the lightning button. Both vertical loads input for the individual slabs, and horizontal loads defined in the tab sheet 'Horizontal Loads' of the building window can be considered. To dimension the slab the vertical loads are considered, whereas in the case of horizontal loads, above all the effects on the walls are of interest.

Here we consider only the vertical loads and the dimensioning of slabs. The vertical loads are dealt with according to the following principle:

They are exported separately as permanent and variable actions (and possibly with any prestress effect present) from slab to slab downwards in the form of load export above and load import below.

Load export

For each of the three actions (permanent, variable and prestressing loads) all loadings of a slab are combined to to so-called export combinations. The load export is calculated on the basis of the reactions for these export combinations (with the added wall and column weights in the case of permanent actions).

Load import

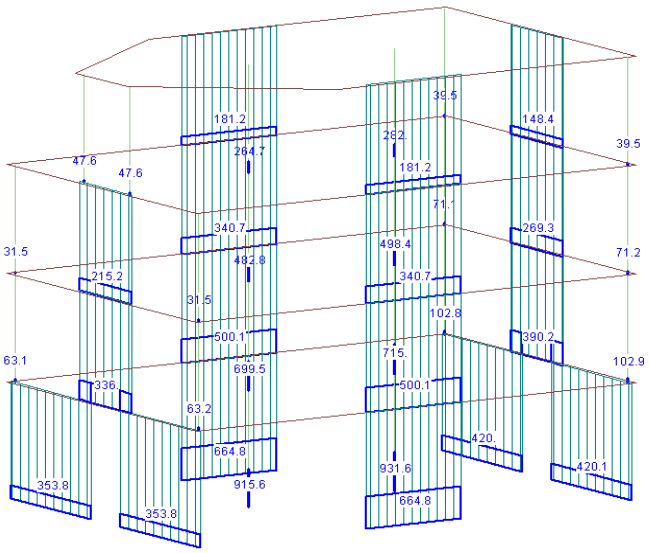
By contrast, the loads exported to the slab to which they are transmitted, appear as imported loadings, again separated according to permanent, variable and, if present, pre-tressing actions.

Shear force compensation

The reactions calculated for the load export contain normal forces and moments. Depending on the support conditions and the dimensions of the vertical elements at their base, an assumption of a moment diagram, which gives the shear force in these elements, can be made. If the sum of the shear forces of all vertical elements is not zero, then a corresponding corrective load has to be introduced in the opposite direction as a horizontal loading for this storey, in order to maintain equilibrium. In Ch. D 5 the effect of such a loading on the vertical elements is described. In this way the final shear forces and thus also the moments in the vertical elements - and especially at their base - is known.

Results

The results obtained, i.e. the forces at the base of the vertical elements, can be output graphically and in tabular form, and they are available in all floor slabs as imported loadings, so that these, taking into account the loads transmitted from above, can be analysed as standard CEDRUS slabs. If changes are made to the floor slabs, the stacked slab model has to be reanalysed, so that the flow of the vertical loads is always up-to-date.



Wall and column forces

A 1.6 Program Modules

The capabilities of the available program options are briefly described in the following sections .

A 1.6.1 Base Module

The Base Module of CEDRUS covers the static analysis of individual slabs including the dimensioning of the reinforcement (see also Chapter A 1.4: Procedure for a Building Analysis). Here are the relevant points:

Structure

The following structural objects are available:

- Outline (arbitrary polygon)
- Openings (arbitrary polygons)
- Arbitrary subdivision into zones for the slab thickness and material properties; various material models: isotropic, no twisting moments, orthotropic
- Downstand beams
- Arbitrary subdivision into zones for elastic support (Winkler model)
- Column and Piles supports, different models; also supports with zero tensile reactions
- Wall supports as line supports with selectable properties; lines of symmetry; supports without tensile reactions
- Joint lines with different coupling properties

Loads

The loads are grouped into individual loadings, which can be combined and superimposed to form limit values. The following load types are possible:

- Dead load (self-weight)
- Arbitrarily-shaped area loads, temperature differences
- Constant and trapezoidally-distributed line loads and line moments
- Concentrated loads and concentrated moments at a point
- Prescribed displacements of point supports

Basic Results

The following basic results are available in various graphical and numerical output forms:

- For individual loadings and combinations of loadings:
  deformations, reactions, section forces, stresses
- The following limit values on the basis of limit state specifications (superposition rules): deformations, reactions, reinforcement moments, maximum shear forces, results for joints, punching shear verifications (see the following chapters)

Punching shear

CEDRUS also performs punching shear verifications. This is carried out in two parts:

1) **Determination of punching shear resistances.** These are determined by the program for columns and end corner points of walls on the basis of automatically generated punching shear objects. Corresponding punching shear objects can be input for concentrated loads posing the risk of punching shear failure. The necessary settings are input in the tab sheet 'Punching Shear'.
2) **Determination of the punching shear loads**. They are determined on the basis of a limit state specification for the ultimate load capacity (ULC). For this purpose the program determines the maximum reactions and derives from them the loads acting within the critical circular sections.

The **Punching Shear Verification** is just a comparison of the punching shear resistances and the punching shear loads. It is called in the 'Results' tab sheet for a limit state specification representing the punching shear loads.

CEDRUS provides an interface, with which the dimensioning programs of product suppliers can be directly integrated.

Dimensioning of the reinforcement

Orthogonal reinforcement

In this standard case the reinforcement bars are arranged in two orthogonal directions x and y, each in two layers at the top and the bottom. The x direction may be specified differently from zone to zone. It is the same for the top and bottom reinforcement layers.

For each of the four layers a distance to the edge (concrete cover) is input and for each layer a basic reinforcement is also input, so that as a dimensioning result only the required additional steel cross sections have to be output.

For dimensioning, the following criteria, which all have to be fulfilled, can be prescribed selectively:

1) **Plastic cross section dimensioning for action effects at the ultimate load state (structural safety).** Here we are concerned with the so called reinforcement moments, which are determined on the basis of a load superposition rule (limit value specification). Regarding reinforcement moments see B 13.2.
2) **Safety with respect to brittle failure** (dimensioning for crack moments)
3) **Safety with respect to cracking** for different requirements. Depending on the requirements they are based on the loading at the ultimate load state for frequent or quasi permanent load cases. Here too the action effect depends on the corresponding limit state specifications.

The above mentioned **limit state specifications** are produced by the program automatically according to the structural codes. For this to be possible, every input loading must be assigned to an action category (dead loads, live loads on office floors, etc.), and for variable loads it has to be known, whether they cancel each other out (vehicle positions) or whether they act simultaneously.

Diagonal reinforcement

diagonalreinforcement can also be dimensioned, not however for limit stress values as with orthogonal reinforcement but for fixed input combinations of loadings, so-called result combinations. For a detailed description see Chapter B 12.4.

Displacements in the cracked state

Displacement analyses that take into account cracking and long-term effects present a very difficult task, for which hardly any exact methods exist, no matter how complex they are. A large number of parameters and boundary conditions play an important role, and in an actual case they can hardly be completely dealt with correctly.

As a result one often depends on estimates of the displacement behaviour. CEDRUS with the methods described in B 12.7 provides a simple solution based on the Swiss Code SIA 262 or on the Eurocode.

Crack analysis

CEDRUS analyses crack widths and spacings for individual loads or result combinations. The analysis is based on a reinforcement content, which is either the result of dimensioning, or is input zone-wise (material zones). The required bar spacings are also specified in the materia l zones.

The crack widths in the x and y directions are output together with the spacings and the maximum crack widths.

A 1.6.2 Buildings Module - Option G

This program option allows the creation of a geometric building model from normal CEDRUS slabs (see Chapter A 1.5.1) and the automatic top-down analysis of the vertical loads with the aid of the stacked slab model (see Chapter A 1.5.3).

Although top-down analysis of the loads is also possible without Option G, working with the Buildings Module has several advantages. These include

- Refined model with shear force compensation (see Chapter A 1.5.3)
- Organisational advantages, since all slabs of a building are included in one building analysis
- Visual checking of the whole building
- Automatic top-down analysis of the loads after any changes made
- Checking the flow and variation of forces in the whole building
- Quantities classified according to structural elements or materials
- Efficient working with identical storeys (master slabs)

The Buildings Module is a prerequisite for the other analyis models of option H.

A 1.6.3 Buildings Module - Option H

Option H represents, on the one hand, an extension of the stacked slab model to the top-down analysis of horizontal loads and provides thereby

- the automatic generation of earthquake loads for the static-equivalent force method
- the combination of the effects of vertical and horizontal loads
- the automatic verification of masonry walls (requires the MURUS program)

On the other hand, option H is able to create space frame models for earthquake analyses with STATIK.

Top-down analysis of horizontal loads

Load input

Loads, as usual, are included in individual loadings. For each storey forces can be specified in x and y direction as well as torsional moments about the z axis. The forces can be referenced to the centre of mass, the centre of stiffness or the origin of coordinates and can have an eccentricity normal to the load direction.

The loadings for the static-equivalent force method of earthquake analysis are automatically created.

Analysis model

Here too the analysis is based on the stacked slab model, i.e. the loads are transmitted from top to bottom for each storey through the wall pillars and columns and from these to the next storey.

The loads are transmitted through the practically rigid floor membrane to the vertical elements corresponding to their stiffnesses in such a way that the resulting shear forces are in equilibrium with the external loads.

The forces obtained at the bottom of the wall pillars and columns are transferred as loads to the next floor slab.

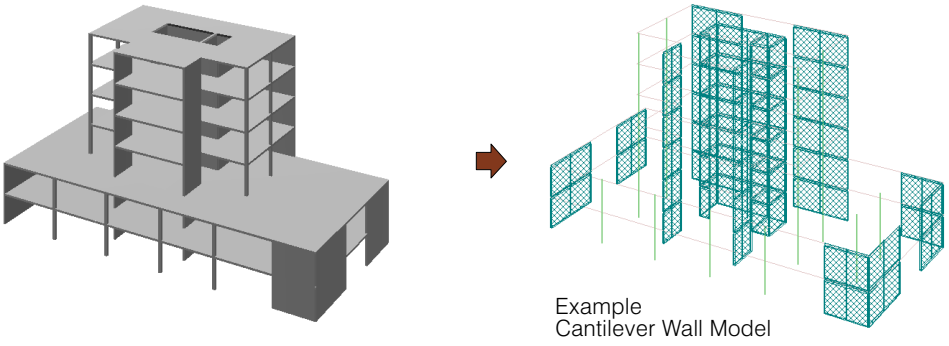
Dimensioning of walls and verification of masonry walls

The effects of vertical and horizontal loads from the stacked slab model can be combined. For these combined section forces an automatic verification of masonry walls can be done using the MURUS program. Automatic dimensioning or verification of reinforcement of reinforced concrete walls is also possible. Depending on the wall type (earthquake wall, simple wall, intercepting wall, etc.), a license of the program FAGUS, the program STATIK or the option "Walls" is required for the calculation of the wall reinforcement.

Dimensioning and verification of reinforced concrete columns

The reinforced concrete columns of a building can be dimensioned and verified using the PYRUS program in a stacked slab model. In this case, the detailed reinforcement arrangement for the stress from the stacked slab model and for impact forces can be verified.

Creation of space frame models for STATIK



Example
Cantilever Wall Model

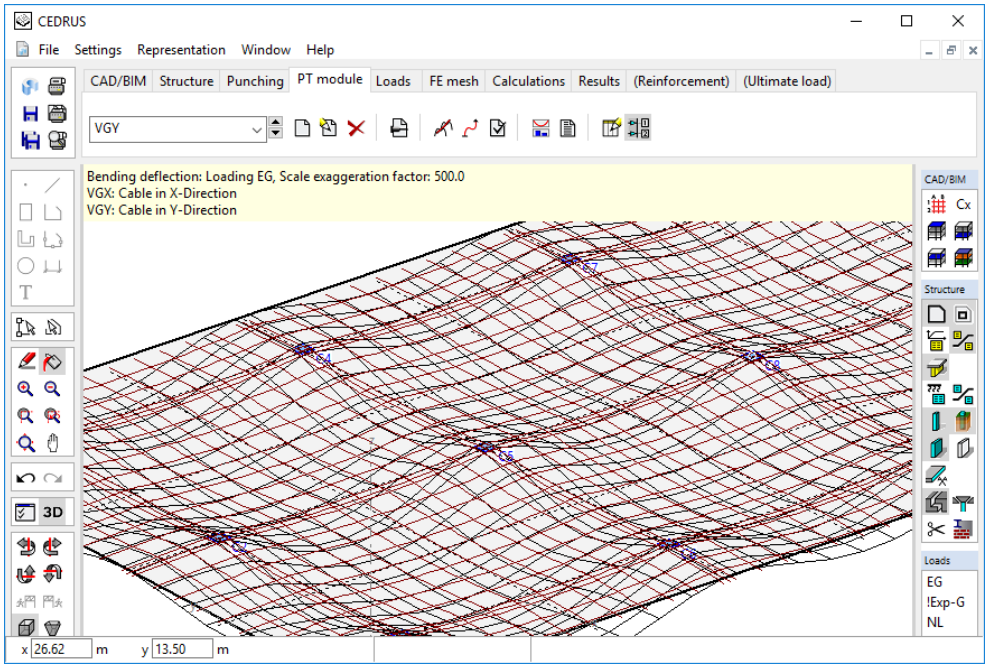
Space frame models of buildings are often a suitable engineering model for earthquake analyses. With the option H such space frame models can be generated from a building model and there is an automatic connection to the program STATIK, which offers various methods of earthquake analysis. Thereby, not only the space frame system for STATIK is generated, but also all loadings and mass distributions are transferred, so that one can begin with the analysis straightaway.

The user can influence the building of the space frame model in various ways. The following points may be mentioned:

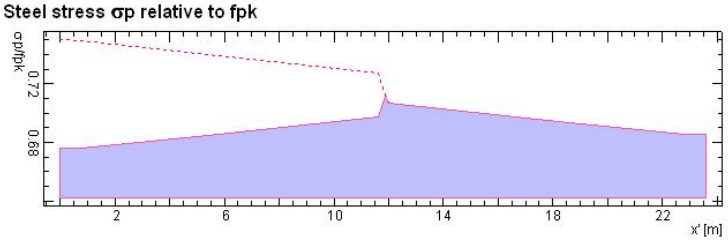
- **Type of model: cantilever wall model or frame model**
  In the case of a space frame model, downstand beams and specially defined plate strips are incorporated in the model as beams and together with the walls and columns form a space frame. The cantilever wall model, on the other hand, only consists of walls and columns, which are connected together by a rigid membrane in each of the storeys.
- **Grouping of walls:**
  Often (e.g. in the case of lift shafts) it makes sense to group together several wall elements, so that only a single frame member results. These decisions have to be made by the user.
- **Structural action of walls:**
  One can specify for each wall whether is should also resist earthquake loads or only vertical loads.
- **Choice of the embedment level:**
  The embedment level can be freely chosen. This is useful in the case of buildings with stiff lower storeys, which are to be neglected in the earthquake analysis.

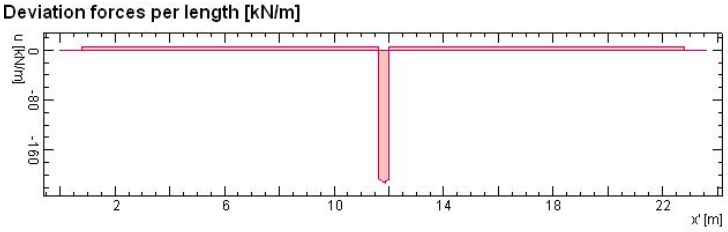
A 1.6.4 Prestressing Module - Option V

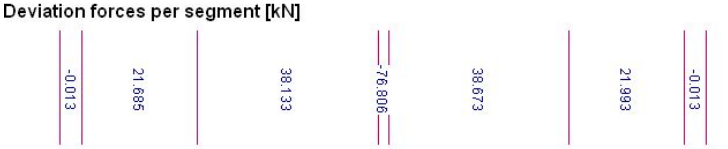
The prestressing module allows an efficient input of arbitrary prestressing tendons in a slab and taking into account their effects in the analysis of a slab.



The force variation in the tendons and the resulting deviation forces are determined by the program on the basis of the specifiable prestressing steps at the ends of the tendons:



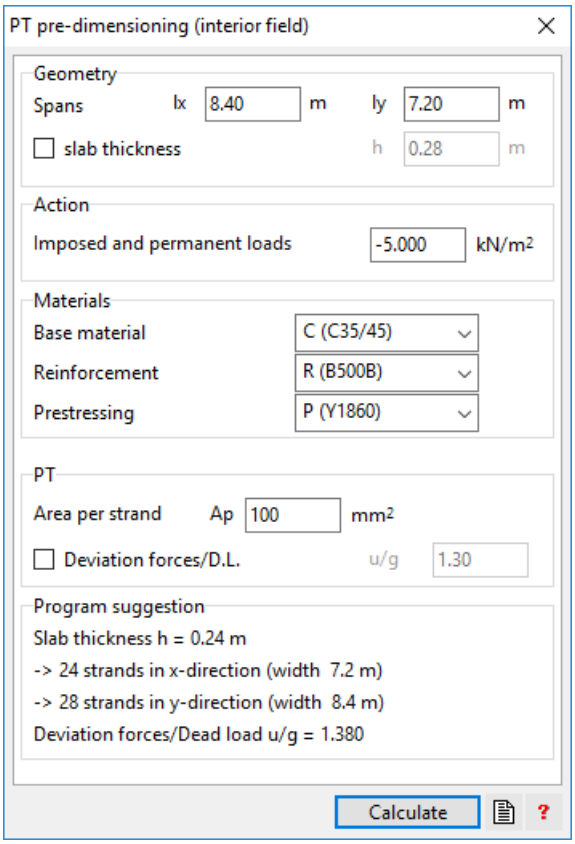




The tendons can be grouped together, whereby for each group a loading is created which includes all deviation and anchorage forces.

To dimension the reinforcement, the prestressing can be considered either as an external action or as an internal stress state with the tendons acting on the resistance side.

A special dialogue permits the pre-dimensioning of an inner span of a uniform flat slab:



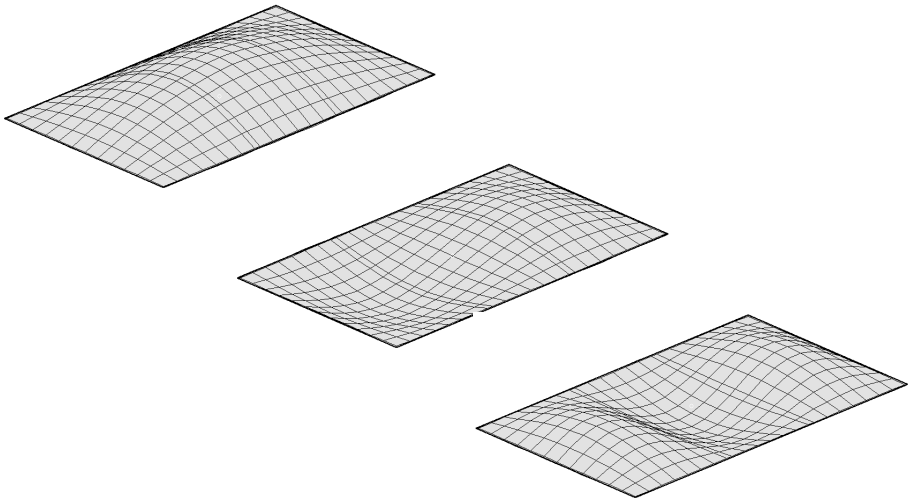
Thus, with just a small amount of input an estimate for an inner span of an (infinitely large) flat slab can be quickly found. The minimum amount of input is: spans, loading (live load), prestressing steel. The program then estimates a slab thickness and the number of required strands.

A 1.6.5 Dynamics Module - Option D

With this module the natural vibrations, consisting of the eigenfrequencies and the corresponding mode shapes, can be determined for a slab with an arbitrary distribution of masses.

NATURAL FREQUENCIES
"for analysis: Dyn; Mass distribution=M"

AC-nr.	ω^2 [(rad/s)^2]	ω [rad/s]	Period [s]	Frequency [s^{-1}]
1	84.85	9.211	0.68211	1.47
2	532.85	23.084	0.27219	3.67
3	1739.44	41.707	0.15065	6.64



A 1.6.6 Reinforcement and Ultimate Load - Options R + T

The base module of CEDRUS determines the required amount of reinforcement at every point of a slab, e.g. in the isoline representation.

The **Reinforcement Module** works with a given reinforcement arrangement in the form of bar and mesh layers. These correspond to a reinforcement plan, in which all reinforcement layers required for the verification of the load-bearing capacity are input by the user. The individual reinforcement layers can have given fixed cross sections or their cross sections can be dimensioned by the program.

The Reinforcement Module then determines the required cross sections on the basis of the stresses calculated by the program CEDRUS for each reinforcement layer selected for dimensioning.

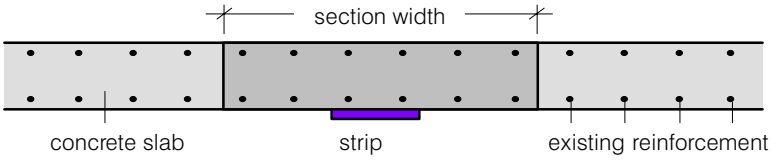
As an option, an optimisation algorithm can be used. This is based on a statically-admissible redistribution of the stresses (analogous to the the displacement of the final deflection line in the case of a continuous beam). The aim is to minimize the required steel content or to find a solution, if one wants to install a given fixed amount of reinforcement in particular zones that cannot resist the purely elastic loading.

For a slab with given reinforcement content the **Ultimate Load Module** can determine how much a particular loading (in the case of a given fixed basic loading) can be increased until the load-bearing capacity is exceeded. This question often arises when changes are made to a structure or in connection with a change in use. The reinforcement is input in the same way as in the Reinforcement Module, whereby however the cross sections of all reinforcing layers have to be known.

A 1.6.7 Externally Bonded Reinforcement - Option K

This module is for the dimensioning and the verification of externally bonded reinforcement strips according to the Swiss Code SIA166.

The analysis is part of the investigation of beam section behaviour and with little extra input it reduces to a standard CEDRUS analysis.



The existing reinforcement is input as the basic reinforcement (property of the material zones).

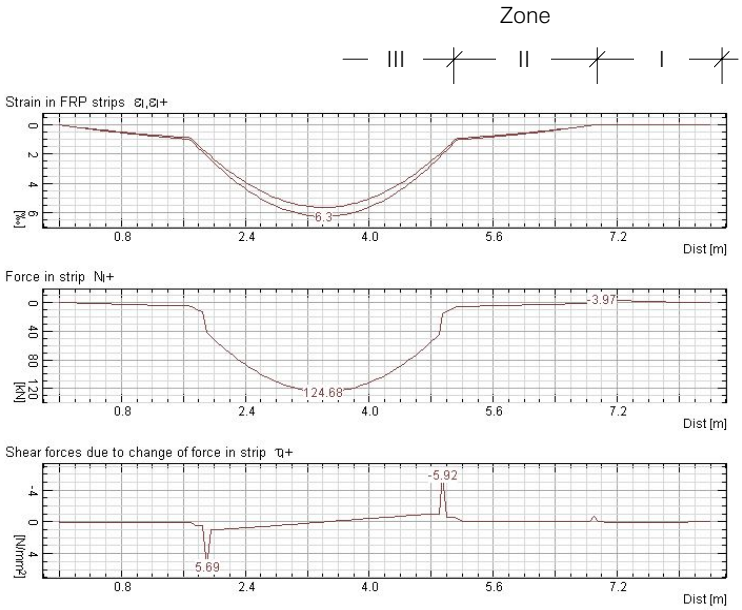
In the beam sections the number, thickness and width of the strips is input as well as the material from a list of given types.

The analysis is carried out in two steps:

1) Application of the permanent loads to the slab without the reinforcing strips

2) Application of the final loading (corresponds to the limit value specification) to the slab with the reinforcing strips

Excerpt from the output of the graphical results:



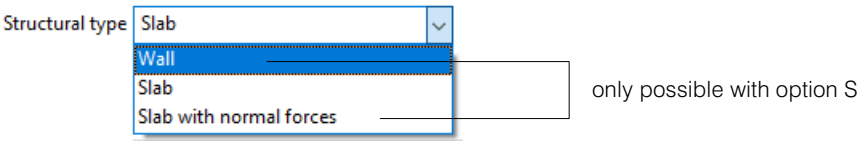
Zone I :	State I	(uncracked concrete), anchorage zone
Zone II :	State II	(cracked concrete), zone of action
Zone III :	State II	(yield of untensioned reinforcement), zone of action

Further output:

- The ratio of the thickness of the concrete compression zone to the effective static height permits a quick assessment of whether the use of adhesive reinforcement has any advantages.
- Effective anchorage length.
- Comparison with admissible values: verification of tensile force, verification of change in tensile force, verification of anchorage

A 1.6.8 Walls - Option S

With this option the FE analysis of walls is also possible. On opening a new CEDRUS analysis, one can choose between the following structure models:



The structure type **Slab with Axial Force** is of interest mainly in connection with the prestressing module. It permits the determination of the axial forces in the slab due to prestressing. Not yet implemented in the current program version is the general dimensioning of the reinforcement for slabs with axial forces (sandwich model). The program, however, supports the dimensioning of beam sections, which is important for prestressed slabs.

A **Wall Analysis** is very similar to a slab analysis, except that one works with the displacement degrees of freedom DX, DY and RZ.

The following material models are available:

- Isotropic material, plane stress state
- Isotropic material, plane strain state
- Orthotropic material

Results of wall analyses

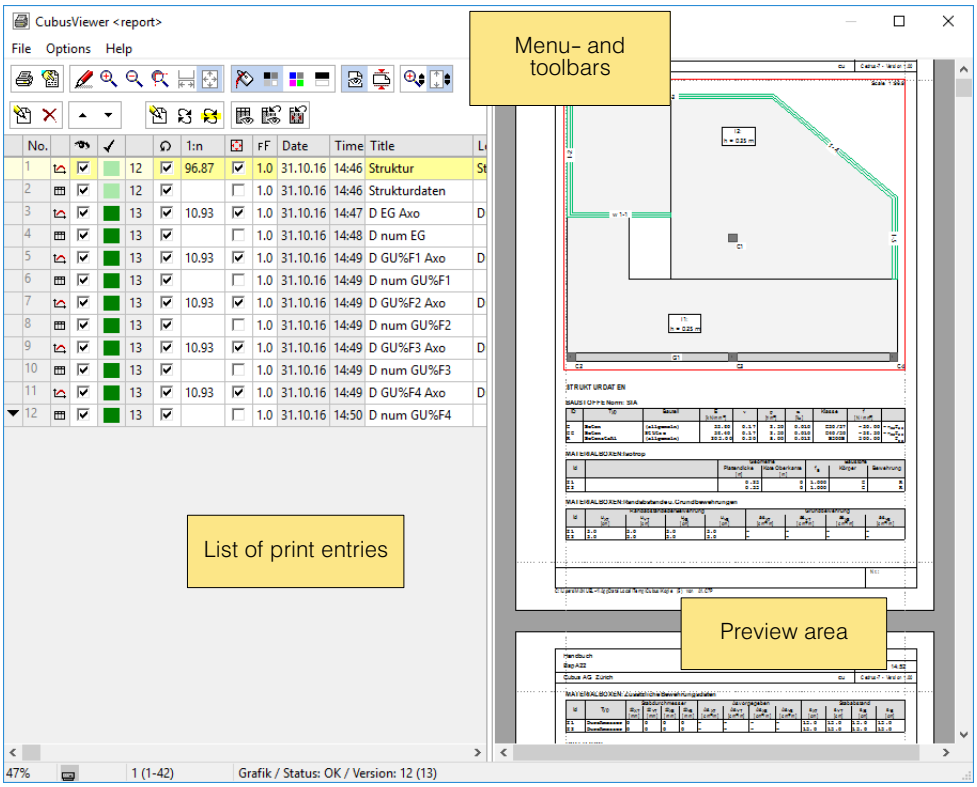
For loadings and results combinations:

- Displacements dx, dy, rz
- Stresses σ_x, σ_y, τ_{xy} in zone-wise definable output directions and the forces n_x, n_y, n_{xy} resulting from the multiplication of the stresses with the wall thickness
- Principal stresses
- Stresses for comparison purposes according to the von Mises and Tresca criteria
- Reactions

For limit state specifications:

- Forces in the reinforcement
- Limit values of reaction components for point and line supports
- Required reinforcement content based on the reinforcement forces described above

A 1.7 CubusViewer



Everything that can be printed in CEDRUS can be printed directly. Normally, however, one would choose the CubusViewer, and one doesn't print directly but creates a **Print Entry.**

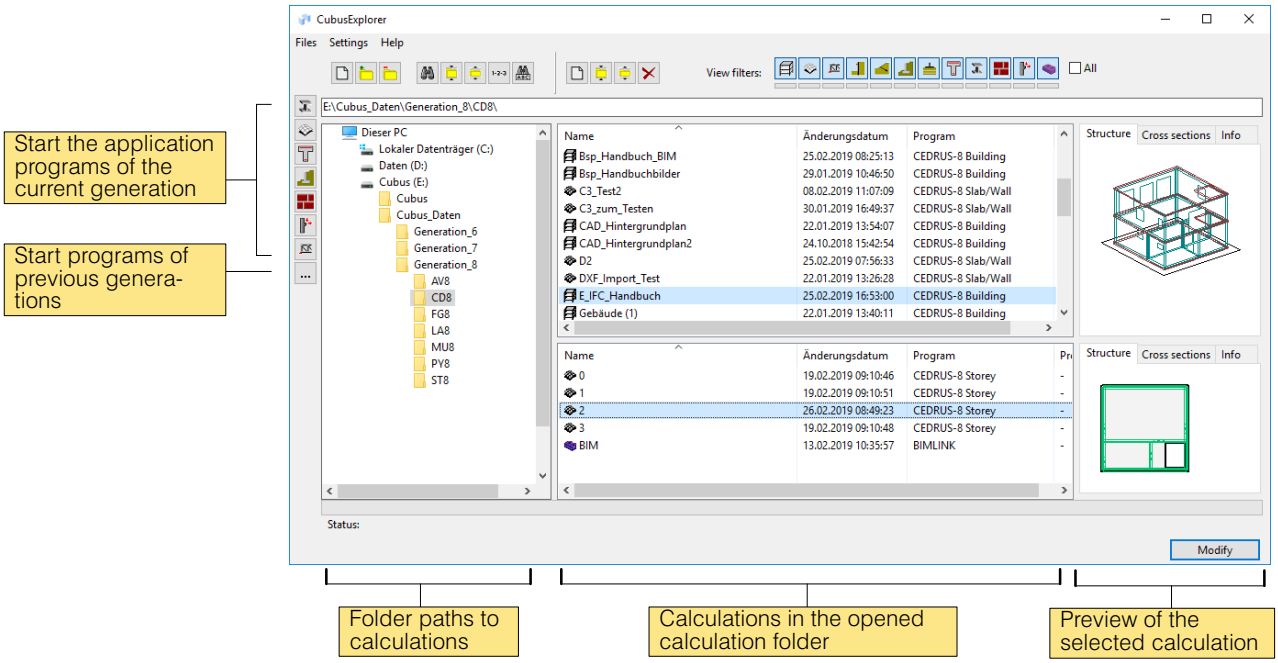
The print entries are managed by the CubusViewer in a list with the aim, at the close of the analysis, of printing a methodically arranged report. For this purpose the following possibilities are available in CubusViewer:

- Print preview
- Moving and deleting of print entries
- Activating or deactivating the print entries for printing
- Selecting the scale for the graphical output
- Inserting a new page command and additional text
- Selection of different output formats (size, orientation: portrait / landscape)
- Free design of the output pages with logo, arbitrary headers / footnotes and frames
- Printing the report with all active entries

The CubusViewer can also be used to store the results of different system states and finally to compare results.

☞ *The complete manual of CubusViewer is available in the Menu* ***Help > CubusViewer****.*

A 1.8 CubusExplorer



All data which belonging to an analysis model is generally identified as an analysis. Thus a CEDRUS analysis, for example, identifies all data of a slab or buildings analysis.

The analysis data can be saved anywhere on a storage device (e.g. hard disk). In the uncompressed state the data is saved in a folder with several subfolders and numerous files

With the CubusExplorer the data from Cubus programs can be found and managed in a user-friendly way. For this purpose it acts as a tailor-made WindowsExplorer window with the following properties:

- Presents the folders created by the user, normally those that contain data from Cubus programs.
- Previews pictures simplify finding the analyses.
- Filter settings make it possible to present the analyses of specific programs in a selected folder
- A search function finds all Cubus analyses stored on a selected drive or in a folder and displays the corresponding folder paths.
- Individual analyses or whole folders containing analyses can be compressed or decompressed (ZIP format).
- New folders for storing analyses can be created.
- The program corresponding to a particular analysis can be started directly from the CubusExplorer.

☞ *From the Help menu in Cubus Explorer window a complete description of CubusExplorers is available.*

A 2 Introductory Examples

A 2.1 Introduction

This chapter includes two introductory examples, the one involving the analysis of a slab the other the Building Module (stacked slab model).

The examples represent an **Introductory Course** into the use of the program and require no previous knowledge in the use of Cubus programs. Starting with the slab example, they are to be worked through from beginning to end exactly according to instructions, since each step builds upon the previous steps. Having carefully worked through the examples, you should be in a position to solve typical problems and if necessary to tackle more complicated problems either with the aid of the manual or the Help System .

The most important source for any information is the **Help System** of **CEDRUS**, whose intensive use we highly recommend. It can be accessed in different ways:

- By clicking on the menu 'Help' in the program's menu bar. Here is a list of all existing help documents that can be displayed directly. One of them is called “Use of Help” and explains the use of the Help System.
- By pressing the key **<F1>** you can obtain context-specific help (e.g. when inputting a point) or for an input element to which the mouse cursor currently points.
- Many of the dialogue windows that appear during input have their own Help button for information on the current dialogue.

This assumes that you have a basic knowledge of the use of the Windows operating system. This includes the manipulation of windows (moving, making larger or smaller), with the Start Menu, the Task Bar, the Clipboard and the Windows Explorer.

☞ *As is usually the case in Windows, one works with the left mouse button. A symbol on the screen **click** or **select** means: move the mouse pointer to the symbol and then press the left mouse button for a short time. The right mouse button is only to display a context menu (see later) in a specific situation.*

Detailed descriptions of the Graphics Editor and the CubusViewer are to be found in the corresponding manuals.

A 2.1.1 Presentation Conventions for the Examples

For all application examples the following presentation conventions apply:

- All actions to be performed are highlighted as follows:

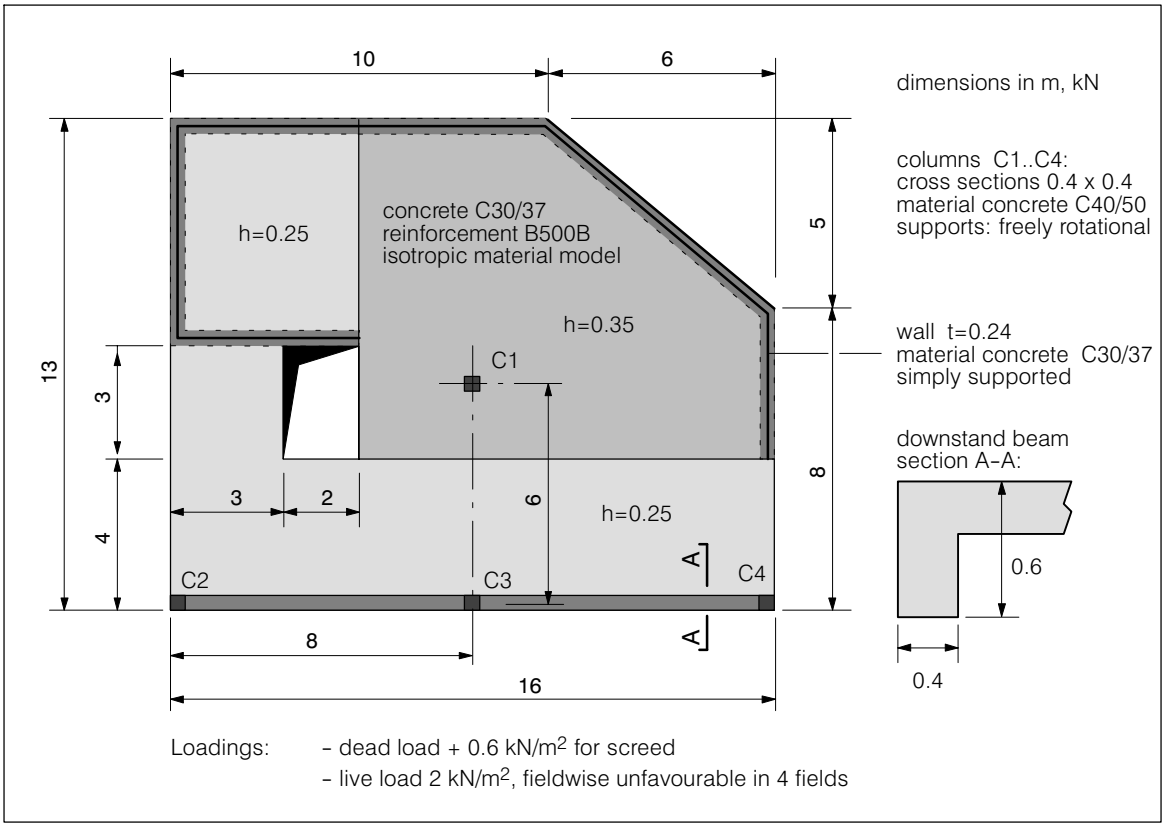
▶ Description of action

- Everything in bold print in the description of an action has to be typed in exactly as it is except the following.
- Special keys are denoted by **<..>** (e.g. **<Enter>**=<↵>, **<Esc>**, **<F1>**, etc.)
- Shortcuts are marked with a '+' and the two keys are to be pressed simultaneously (eg **<Alt> + <F11>**)
- The mouse buttons are abbreviated as **<LMB>** and **<RMB>** (left and right)
- Buttons displayed on the screen are shown in square brackets (e.g. **[Cancel]**)
- A menu entry to be selected is shown with apostrophes, separated for a multistep selection by '>' (e.g. „Options“ > „Language“ > „English“)

A 2.2 Introductory Example SLAB

A 2.2.1 Problem Description

With the aid of the following example a slab analysis is worked through up to the dimensioning of the required reinforcement.



A 2.2.2 Program Start

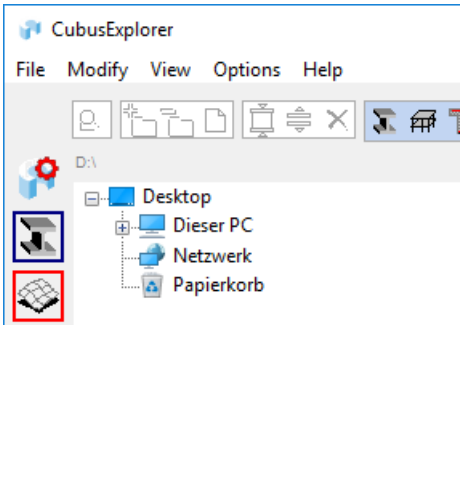
CEDRUS can be started explicitly from the CubusExplorer sidebar or implicitly by choosing a CEDRUS project in the CubusExplorer. The CubusExplorer is an independent utility to manage analysis with different Cubus programs.

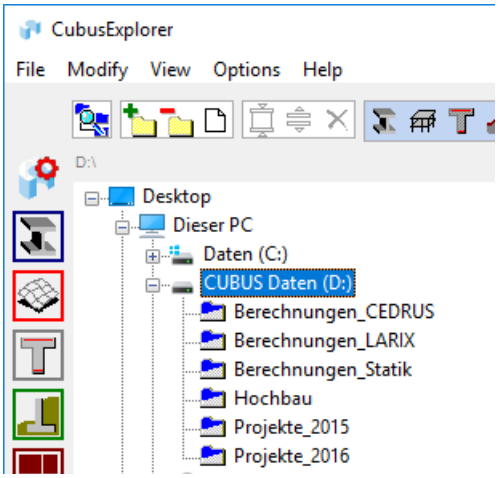
▶ You may start the CubusExplorer via the Start menu of Windows, for standard installation: typing CubusExplorer (usually the first few letters are sufficient and CubusExplorer is suggested).

A 2.2.3 Beginning an Analysis

▶ Start the CubusExplorer via the Windows Start Menu:
[Start] > Programs > Cubus > CubusExplorer >

If you are doing this for the first time, it might look like the following figure on the left. Click on the '+' next to'Working Space' and you will obtain the figure on the right.



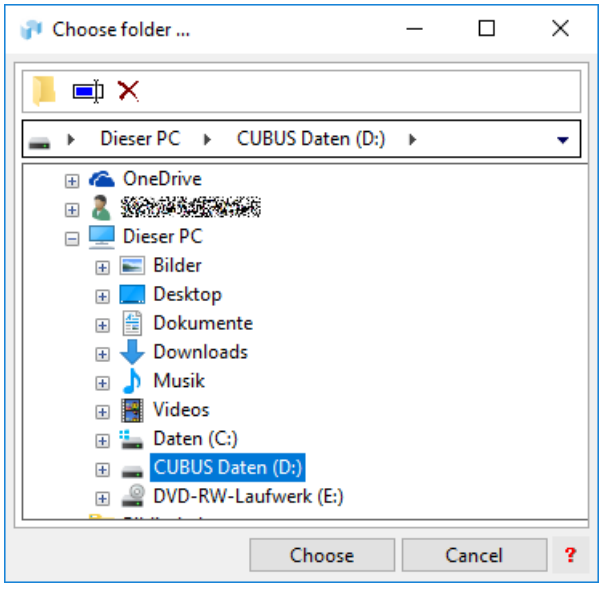


The CubusExplorer is very similar to the Windows Explorer. The main difference is that in the left window only the directories are shown that you require, and these are normally those that only contain analysis data of Cubus programs. For a detailed description of the CubusExplorer see its Help Menu.

For our example we want to create a folder called “Data” on a drive (here G:) , in which we want to save our first analysis. Basically you have complete freedom to create a folder structure to manage your analysis data. At any time you can reorganize the data structure and rename or move folders.

▶ Click first on the symbol of the required drive and then on the symbol shown on the left to create the folders or make them visible.

The figure that appears shows the directory structure of the selected drive:



Now you can choose one of the folders or sub-folders and select it with the button [Choose] in the CubusExplorer.

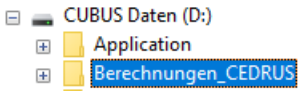
However, we want to create a new folder:

▶ For this purpose select the object (harddisk or folder), where the new folder should be placed - in our case the hard disk symbol G:.

▶ To create a new folder click on the button on the above right in the window.

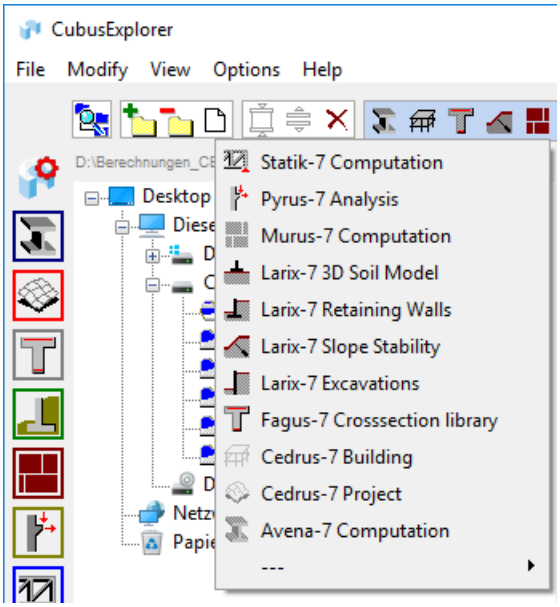
Thereby a new folder appears in the correct directory level.

▶ Overwrite its name with “Data” and using [Select] place it in the CubusExplorer, where it is now in the correct place:



▶ Select the newly created folder and then click on the button shown on the left to create a new analysis.

Directly beneath the button a menu appears showing all installed Cubus programs.

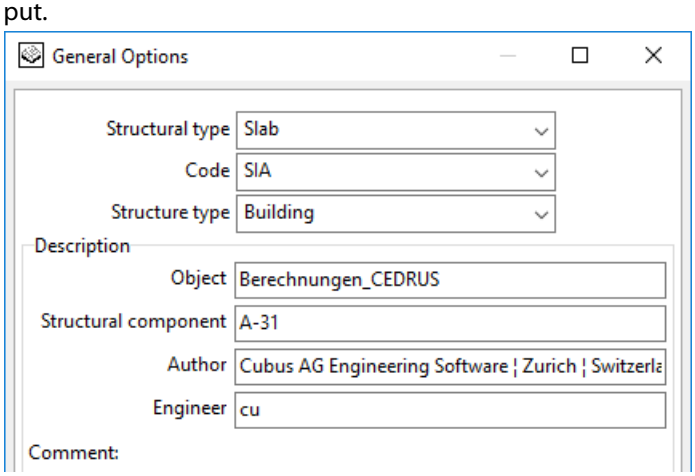


▶ Select 'CEDRUS project', whereupon in the middle window of the CubusExplorers a new analysis with the standard name “CD-Analysis” appears. Since the name of the new analysis is already highlighted, you can rename it directly by typing “Example 1”.

Thus you have already created the desired analysis folder.

▶ Now start CEDRUS with the button [Modify] lower right in the CubusExplorer.

The following dialogue window appears, in which important pre-settings for the new analysis can be carried out. The information under 'Description' appears in the printed output.



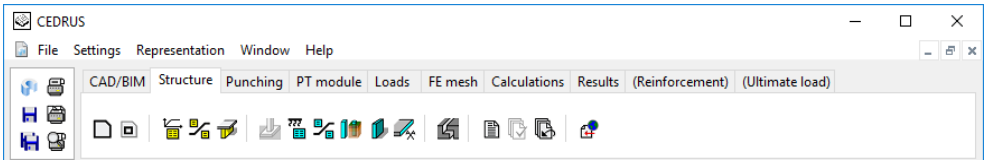
▶ Leave the settings as they are and close the window using [OK].

☞ *The settings (with the exception of the structure type) can be changed at any time using the menu 'Settings' > 'General...' .*

☞ *After starting a new analysis it is possible to import input data from earlier CEDRUS analyses. To do this use the main menu 'File' > 'Import'.*

A 2.2.4 The tab sheets

After starting a new analysis the main program menu appears at the top of the CEDRUS window in the form of a set of tab sheets:

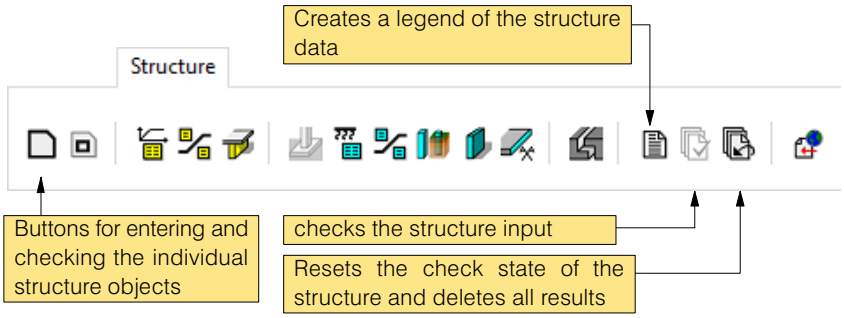


The tab sheets are arranged from left to right corresponding to the procedure for a normal analysis. They are activated by clicking on their flags. Each tab sheet has a number of functions, which can be activated using the corresponding buttons. These buttons are also arranged from left to right, in the sequence they are normally needed in the course of an analysis.

The first thing to do with a new slab is define its structure. Thus the corresponding tab sheet has already been selected.

☞ *The first tab sheet is called CAD. It contains tools to support the input. Grids parallel to the axes can be introduced and DXF/DWG file as well as plan outlines from other CEDRUS slabs can be imported, on which to base the input. These possibilities are not dealt with in the introductory example.*

A 2.2.5 The tab sheet 'Structure'



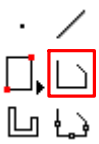
The relevant structure for the FE analysis consists of a number of objects, such as outline, openings, etc. For each of these object types there is a corresponding button, with which new objects of the corresponding type can be input.

Outline, openings

We begin with the input of the outline:

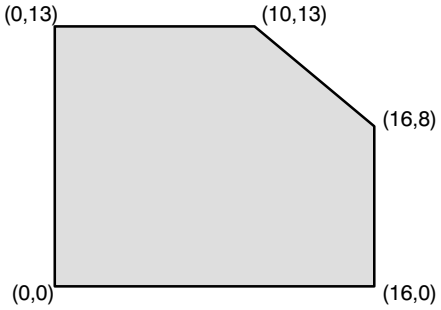
▶ Click on the adjacent button.

Now on the left side of the screen the drawing tools of the Graphics Editor are active, with which the outline or parts of the outline can be created.



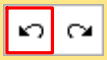
▶ Input the outline depicted below using the polygon tool, by clicking on it and then type in the five coordinate pairs of the corner points sequentially (**<x>** and **<Enter>**, **<y>** and **<Enter>**, etc.). The polygon is closed by clicking on the starting point. It doesn't matter whether one proceeds in clockwise or anti-clockwise fashion.

☞ *With the **<Esc>** key the last input polygon point can be cancelled.*

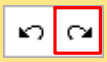


Undo / Redo

Here we want to point out a very useful function of the Graphics Editor. If you have made one wrong input or even several one after another this is not a problem for CEDRUS :



With the Undo function (left edge of window) stepwise as many changes can be undone as one wants (provided the button is active). Key combination: <Ctrl> + <Z>



With the Redo function changes that were made using the Undo function can be redone stepwise. Key combination: <Ctrl> + <Y>

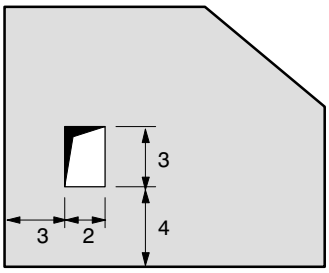
☞ *The outline can be composed of several parts (lines, polygons, circular arcs), so that finally the desired complete outline polygon, which doesn't intersect anywhere, can be viewed on the screen.*

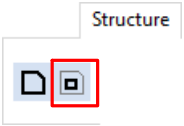
▶ Centre the view; thereby you can also see the neighbouring buttons to control the view (**<F1>** provides help).

☞ *A slab can only have one outline.*

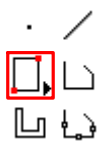
☞ *The Graphics Editor provides various possibilities of inputting new points, which depending on the situation can greatly simplify the work. You will learn more about this below.*

Openings are input in the same way as the outline.





▶ Click on the adjacent button for openings.



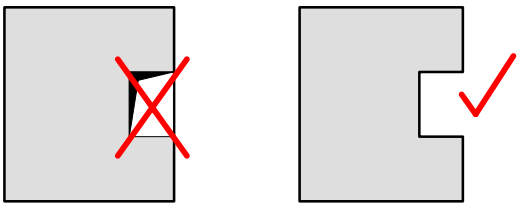
▶ "Draw" the opening shown with a suitable tool. We use the rectangle tool, which is quicker than the polygon tool used above. Here we only have to input diagonally opposite corner points. Use relative input for the second corner point (see below).

Relative input of coordinates

Often it is more convenient to work with relative rather than absolute coordinates. If you type in **<R> before typing in the coordinates** of a point, the values are not with reference to the origin of coordinates but to the so called **Relative Point**.

The relative point is indicated by a small red triangle. Before typing in the coordinate values it can be moved to any existing point by clicking on it. As default value the program always chooses the last point to be input.

☞ *An opening may not intersect or touch the outline or another opening. Thus a notch (recess), i.e. an opening at the edge of a slab, has to be input as part of the outline.*



Inputting geometric objects

This is an important note, which not only concerns the input of an outline and openings, but all geometric structural objects.

The geometric definition of a slab structure consists essentially of points and lines. Higher level objects like rectangles, polygons, wall-like objects, etc. consist of several lines. Since the FE mesh has to take all these points and lines into account, these are subject to the following conditions:

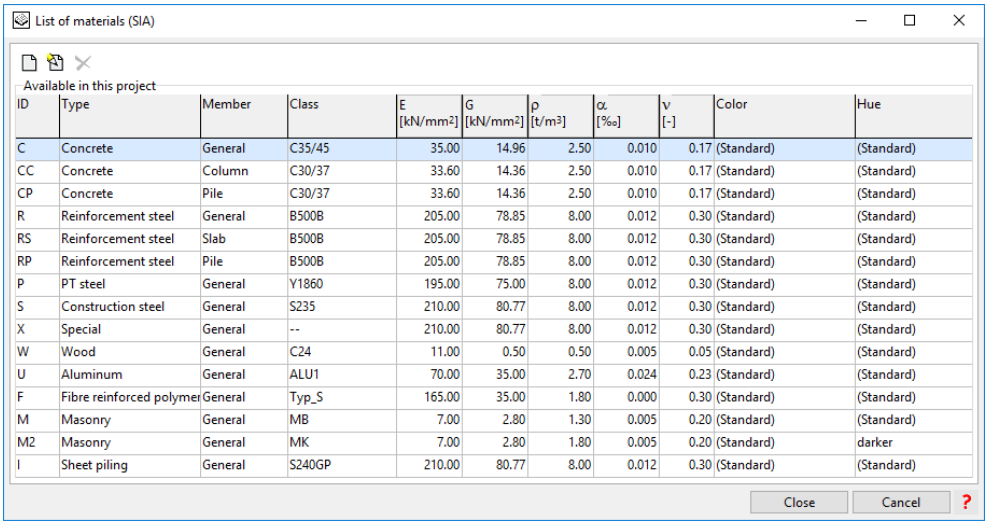
- All points have to be spaced at a minimum distance of d_{min}. The value of d_{min} can be viewed in the menu *Settings > Input* under 'minimum side length' and in special cases also changed.
- The distance of a point from a line must also be at least d_{min}.
- If two points are in the same position (e.g. two lines that meet at a point), then depending on the type of input of these points, then due to rounding errors they may not have exactly the same coordinates. For this purpose CEDRUS has a tolerance value d_{tol}. If two points are closer than d_{tol}, then the program 'merges' them. The value of d_{tol} an be viewed in the menu *Settings > Input* under 'Visibility Limit' and can in special cases also be changed.

In summary: If the distance between two points or that of a point from a line is greater than d_{tol} but smaller than d_{min}, the program will not accept this when checking and report an error.

Materials

The concrete class C30/37 will be used for the slab and C40/50 for the columns. The reinforcement steel should be of class B500B.

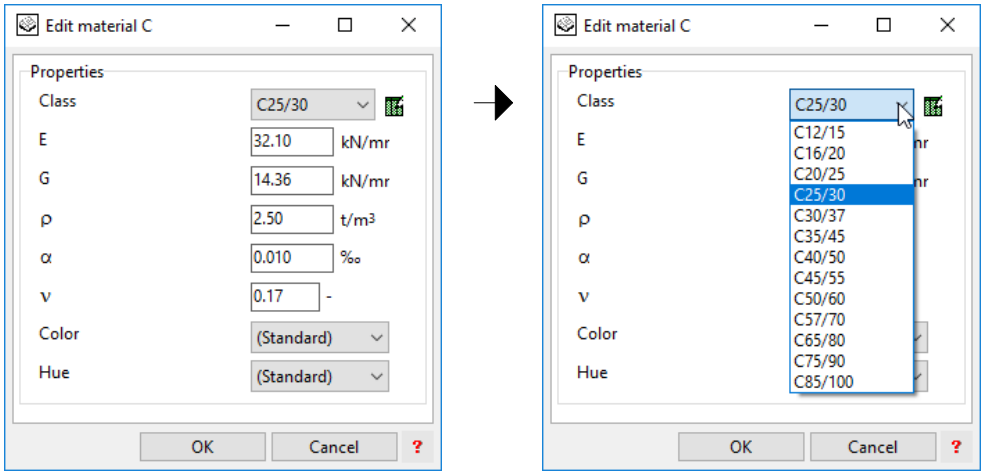
Before entering objects that use materials (materials box, downstanding beams, columns and walls), the needed materials should be made available. This is done in the 'List of materials' table, which is accesses by the menu 'Settings > Materials' and looks as follows:



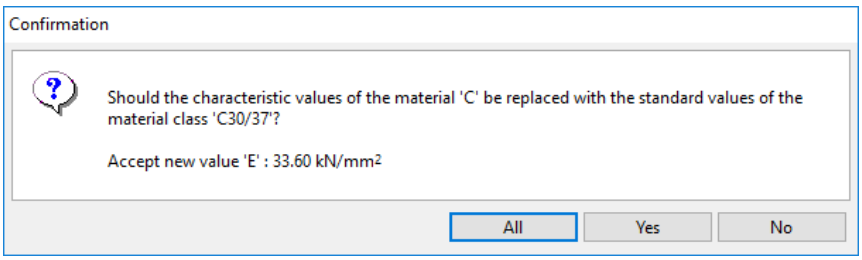
ID	Type	Member	Class	E [kN/mm²]	G [kN/mm²]	ρ [t/m³]	α [‰]	ν [-]	Color	Hue
C	Concrete	General	C35/45	35.00	14.96	2.50	0.010	0.17	(Standard)	(Standard)
CC	Concrete	Column	C30/37	33.60	14.36	2.50	0.010	0.17	(Standard)	(Standard)
CP	Concrete	Pile	C30/37	33.60	14.36	2.50	0.010	0.17	(Standard)	(Standard)
R	Reinforcement steel	General	B500B	205.00	78.85	8.00	0.012	0.30	(Standard)	(Standard)
RS	Reinforcement steel	Slab	B500B	205.00	78.85	8.00	0.012	0.30	(Standard)	(Standard)
RP	Reinforcement steel	Pile	B500B	205.00	78.85	8.00	0.012	0.30	(Standard)	(Standard)
P	PT steel	General	Y1860	195.00	75.00	8.00	0.012	0.30	(Standard)	(Standard)
S	Construction steel	General	S235	210.00	80.77	8.00	0.012	0.30	(Standard)	(Standard)
X	Special	General	--	210.00	80.77	8.00	0.012	0.30	(Standard)	(Standard)
W	Wood	General	C24	11.00	0.50	0.50	0.005	0.05	(Standard)	(Standard)
U	Aluminum	General	ALU1	70.00	35.00	2.70	0.024	0.23	(Standard)	(Standard)
F	Fibre reinforced polymer	General	Typ_S	165.00	35.00	1.80	0.000	0.30	(Standard)	(Standard)
M	Masonry	General	MB	7.00	2.80	1.30	0.005	0.20	(Standard)	(Standard)
M2	Masonry	General	MK	7.00	2.80	1.80	0.005	0.20	(Standard)	darker
I	Sheet piling	General	S240GP	210.00	80.77	8.00	0.012	0.30	(Standard)	(Standard)

The List of materials table can be extended or modified as desired. We need here to adapt the two concrete materials.

▶ Doubleclick on the row containing the concrete with the Id C and change the class to C30/37:



▶ For the correct adoption of the new material properties, select [All]

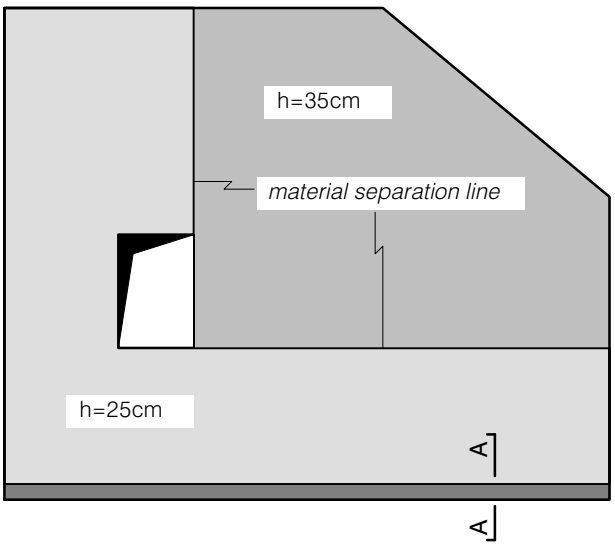


▶ Adapt also the column concrete 'CC' to 'C40/50'.

The material for the reinforcement should be set correctly.

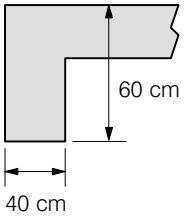
Material zones

The material and thickness properties in our slab example are defined as follows:



concrete C30/37
reinforcing steel B500B
isotropic material model

downstand beam
section A-A:



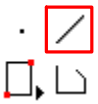
Material separation lines

The slab consists of two zone of different thickness. Thus we will have to divide up the slab using separation lines into the corresponding two **material zones**. They are called material zones because within these zones all the properties that are constant like material model, thickness and material type are included under the term 'Material'. Thus the dividing lines are called **material separation lines.**

Downstand beams represent special material zones, which we will discuss later. For now we only have to know that the outline lines of downstand beams act automatically as material separation lines.

Now let us input the material separation lines.

▶ Click on the button for material separation lines and draw the two lines that limit the zone with the slab thickness 35 cm.

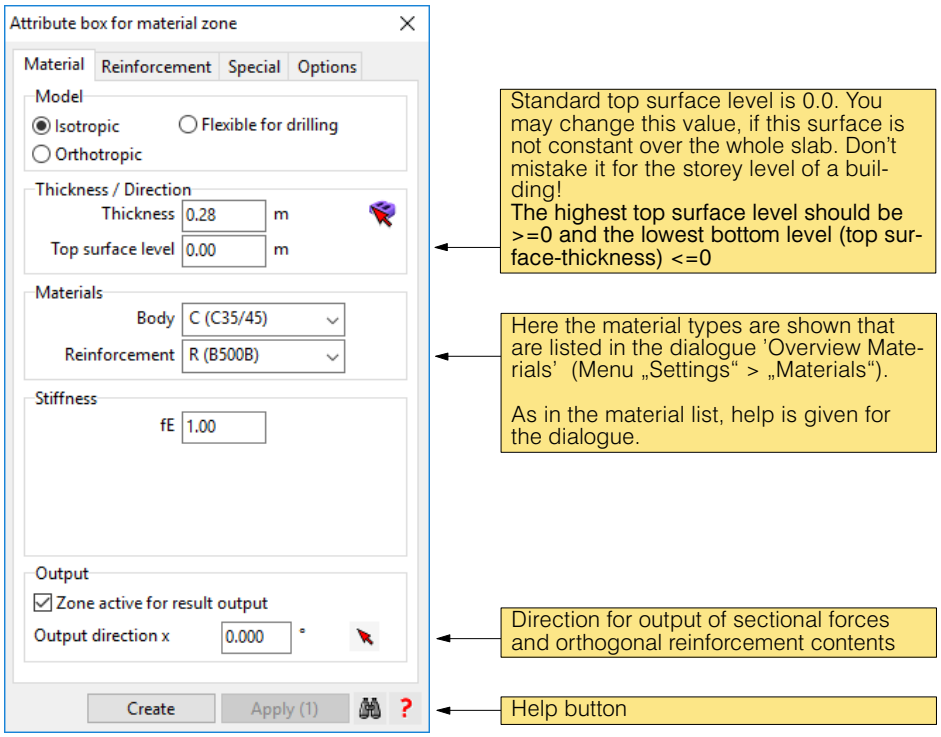


▶ The simplest way is to input the lines as follows:
Click on the line tool of the Graphics Editor. For the starting point of the horizontal separation line click on the lower right point of the opening. Then press the key **X** (= line in X direction) and click on the far right on the outline line. Proceed analogously with the vertical separation line by drawing a line from the top right point of the opening in the Y direction upwards. No line of separation is needed along the side of the opening.

Attribute Boxes

Thus the slab has now been divided up into the two required material zones. In each material zone an Attribute Box is required with the corresponding material properties.

▶ Click on the button 'Attribute Boxes for Material Zones'.
The dialogue window for material properties appears. Set the first tab sheet as follows:



The two middle tab sheets of the dialogue contain information on the reinforcement, which we will not discuss here. However, look in the tab sheet 'Options', in which you can select everything that should be shown in the subsequently introduced Attribute Box.

▶ Click on the button [Introduce] and place a Material Box on the top right zone of the slab by pressing the left mouse button.

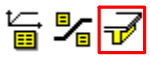
These material properties now apply at the position of the Material Box up to the edge of the zone by the boundary of the slab or by material separation lines.

▶ In the dialogue window that is still on the screen change the thickness to 0.25 and in the same way with [Introduce] also place the Attribute Box for the second material zone in the corresponding area.

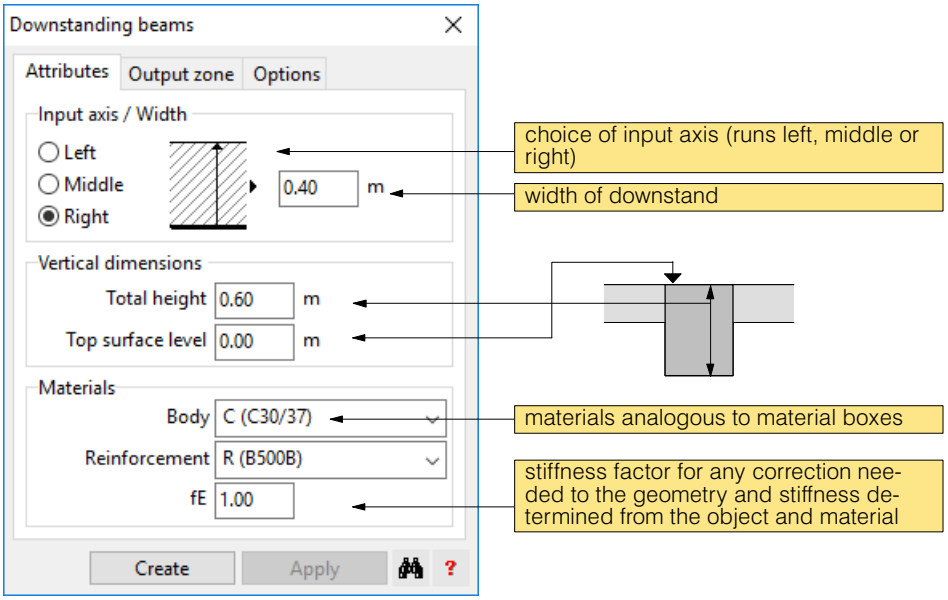
For further explanations regarding the material properties click on the Help button of the dialogue window.

Downstand beam

Downstand beams (or simply downstands) are modelled in CEDRUS with orthotropic slab elements. With the introduction of a downstand beam therefore a corresponding material zone is automatically generated, but it doesn't contain a Material Box .



▶ To input a downstand beam at the lower edge of the slab click on the downstand button. The dialogue window of the downstand object, in which the settings in the desired attributes of the downstand beam are carried out:



▶ Input the settings according to the problem description and the figure. Then with [Introduce] start the double line polygon tool of the Graphics Editor and define the downstand beam by clicking on the lower left and the lower right corners of the outline.

The double line polygon, which here only consists of one segment, has to be closed. This is done by clicking again on the end point (or use key **<Q>**).

☞ *If by mistake you have input the wrong point, then with the polygon input you can go back a point at a time using the <Esc> key.*

For further information on downstand beams click on the Help button in the dialogue window.

▶ After inputting all material objects you can start a provisional structure check (using the adjacent button) to see whether the material zones have been defined correctly. This check is always carried out automatically before the solution stage and at the latest it will then be reported that something is not in order.

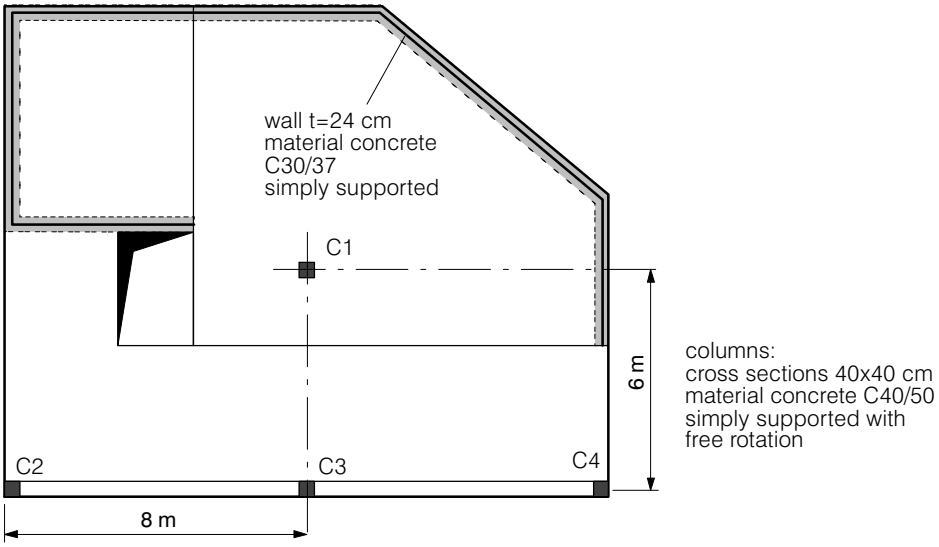
Start the structure check here to obtain the warning that the slab is not supported, which is true for the present state of the input.

Supports

The following buttons are available for defining the type of support for the slab:

The first two buttons serve to define zones of area support and concerning input they function with attribute boxes and separation lines as with the material zones described above. They are mainly used for foundation slabs analysed with the method of subgrade reaction and like lines of symmetry are not treated in this example.

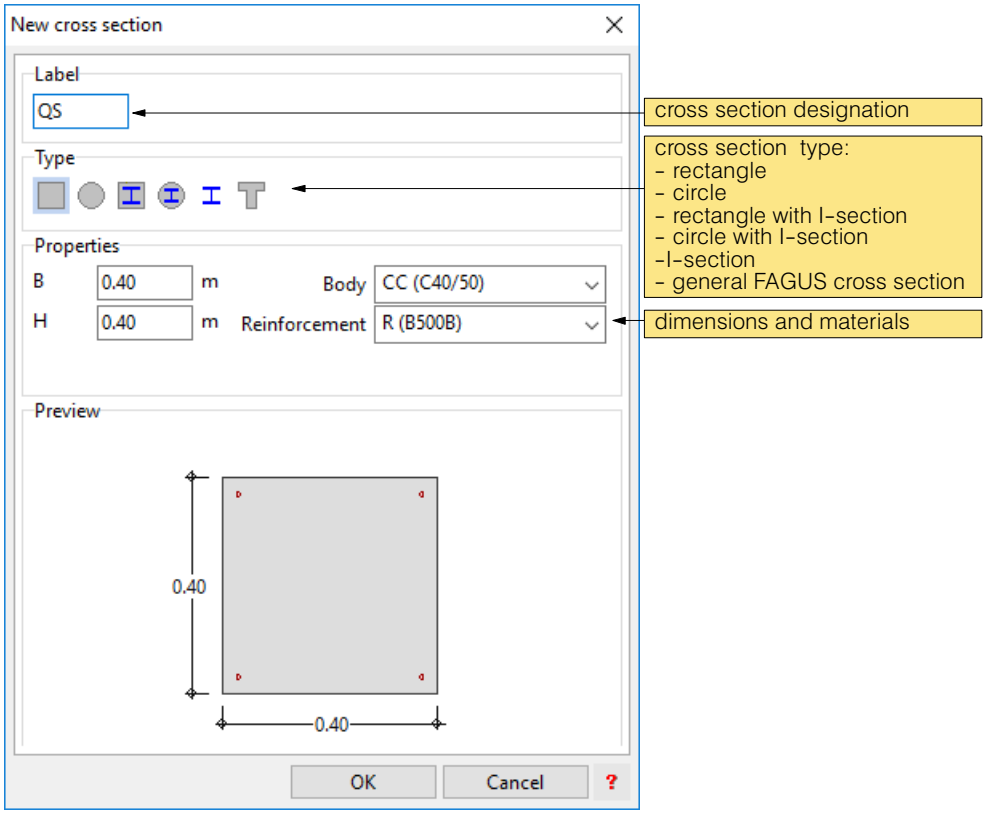
The problem description envisages four columns and the walls shown in the figure



Columns

First of all we want to input the four columns that act as supports for the slab.

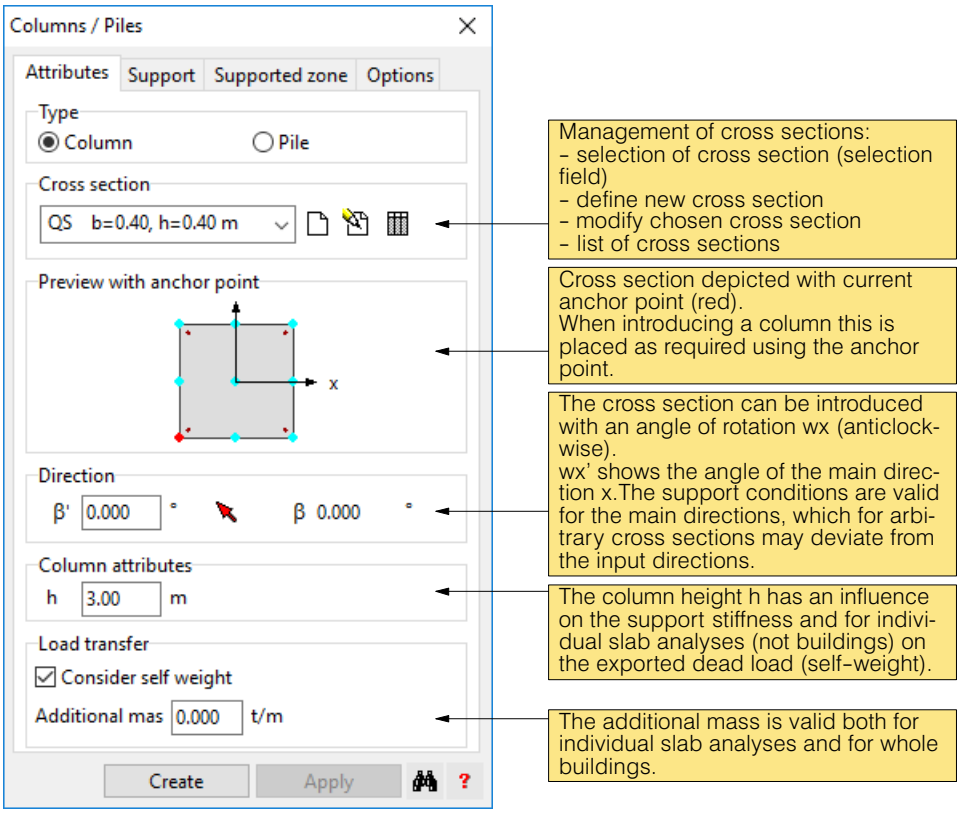
▶ Click on the Columns button. Normally, the dialogue window for columns appears. Since in this analysis column cross sections have not yet been defined, here at first the dialogue to define column sections appears:



▶ Input a rectangle of dimension 0.40 x 0.40 m and check the correct setting of the body material (CC with C40/50), then click on [OK].

☞ *In reality the material used for columns is much stiffer than the one used for slabs and walls. This also due to the usually strong reinforcement of columns. You should respect this fact in the calculation model. Otherwise you may get too big settlements of the column supports which means too small column reactions because a part of them are transferred to neighbouring walls. This is the reason why the program suggests a weaker concrete C for slabs and walls and a stiffer one (CC) for columns. Note also the possibilities to work with stiffness factors for slabs, columns and walls.*

After defining the column cross section, CEDRUS shows the main column dialogue:



Here we do not consider in detail the different support models for the columns. For this purpose activate the Help dialogue. We do not change the default settings, which assume a freely rotatable area support with automatically calculated vertical bearing stiffness.

To introduce the different columns first select a suitable anchor point in the figure (within the Column dialogue window), then select the button [Introduce]. Input a new column by inputting a new point or by clicking on an existing point. The input point corresponds to the selected anchor point.

▶ For the column C1 select the anchor point in the middle, click on [Introduce] and input a new point in the position (8.0,6.0).

▶ For the column C2 select the anchor point bottom left, click on [Introduce] and click on the outline point bottom left.

▶ For the column C3 select the anchor point bottom middle, click on [Introduce] and input a new point at (8.0,0.0).

▶ For the column C4 proceed as with C2.

☞ *If you input an incorrect point by mistake, use the Undo function!*

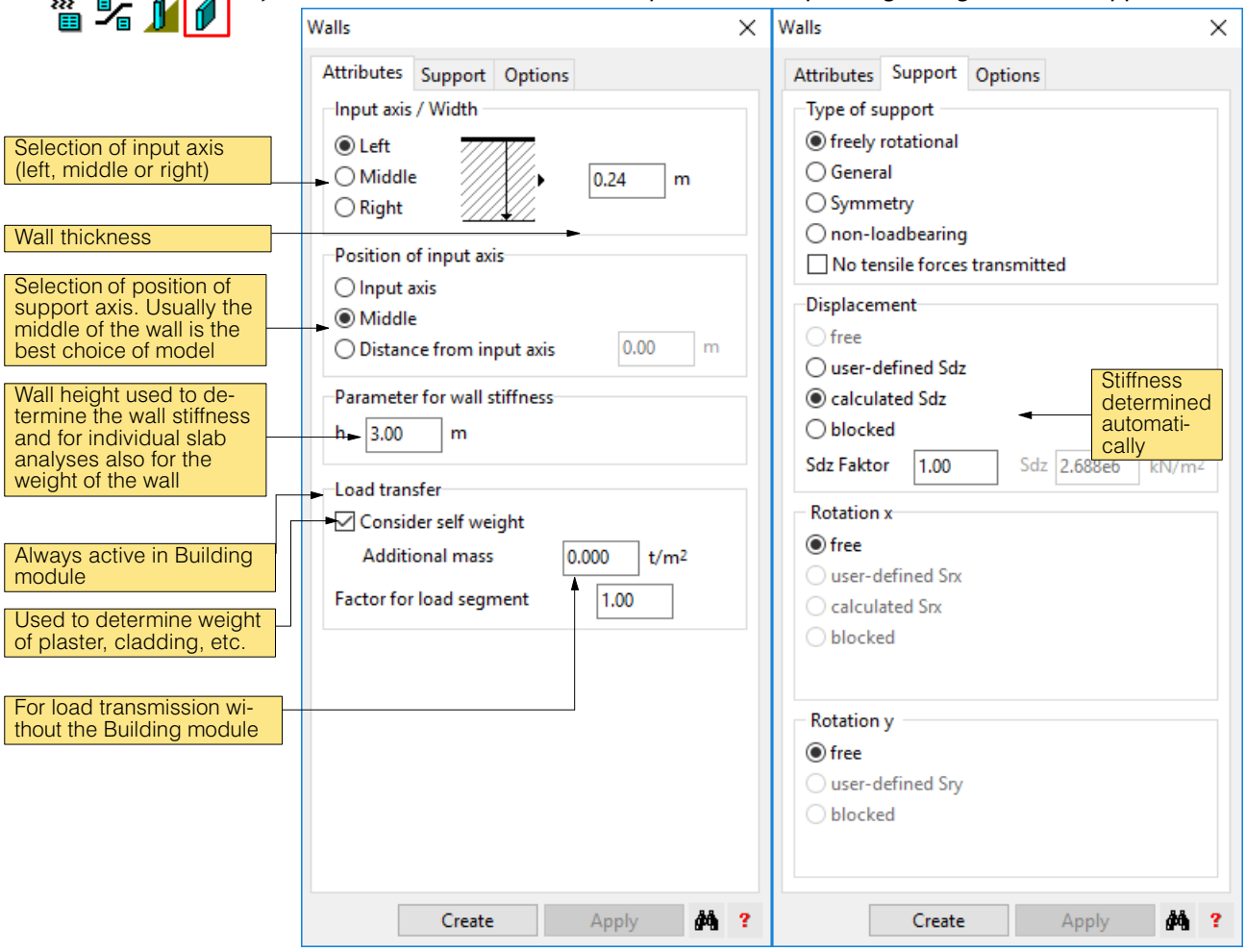
Walls

In our example, besides the columns we have a wall consisting of several segments, on which the slab is supported.

A wall is input as a double line polygon with the actual wall thickness and the associated material. From this information, together with the wall height, we can determine the dead load for the load transmission and also corresponding values for the support stiffnesses. In the FE model the walls are considered as line loads.

In our example the wall consists of several interconnected segments of uniform thickness but otherwise the same properties. Thus it can be input as a double line polygon.

▶ Click on the Wall button, whereupon the corresponding dialogue window appears:



▶ For our example set all input fields of the Wall dialogue box according to the above default values and then click on [Create]. The simplest way to input the polygon points is as follows, with the input axis set to 'left':

1) For the first point of the wall polygons click on the upper right point of the opening.

2) The second point lies on a horizontal line from the first one. So just press the key <X> and and click on the left edge of the slab.

3) The remaining four points can be clicked on directly.

4) Quit the wall polygon input using the key 'Q' or by clicking on the last point.

For detailed information on the Wall dialogue window click on its Help button.

Selecting objects and the right mouse button

This insertion familiarizes you with some very important information regarding CEDRUS. It is decisive to understand it. Up till now, for inputting structural objects you have worked with menus (e.g. column button to input columns), but this method is much less suited for changing existing objects, as you will see in the next example:

We want to move the opening and the adjoining separation line bottom right by 50 cm to the left. You should do this using the following steps:

1) If the structural objects button is activated, click on it to switch it off. In this way all structural objects can be selected (if a button is pressed, then only objects of that type can be selected).

2) Now you have to **select those objects**, which have to be moved. Firstly, the most important points regarding the selection of objects is explained:

   - Objects are normally selected by clicking on the left mouse button or by creating a window. If a window is created by stretching it from left to right, then only the objects lying within the window are selected, and by stretching it from right to left also the intersecting objects are selected.
   - Select the objects displayed in red colour (can be set).
   - In the case of selection by means of clicking or using a window, objects already selected or still selected are automatically deselected by the program beforehand. If one wants to avoid this, then during selection the <Shift> key has to be pressed. If the <Shift> key is pressed then already selected objects can also be deselected.
   - Geometrical objects normally consist of lines, rectangles or polygons. If one wants to select the end or corner points of such objects, the object has to be selected first. In the selected state it shows its points, which can then be selected in the usual way.

   For our example we have to select the end point of the horizontal material separation line together with the opening. Taking into account the above points we proceed as follows:

   a) Click on the material separation line and then on the left end point of the line. When the point is selected the selection of the line is lost, which is what we want here.

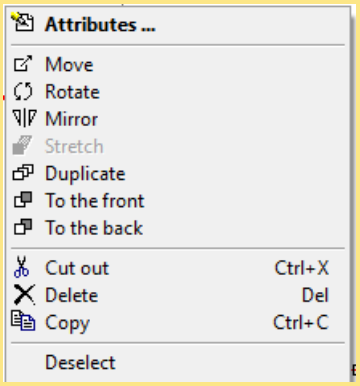
   b) In addition to the selected point the opening now has to be selected. For this purpose with the <Shift> key pressed click on the opening. Note the preselect text on the mouse pointer, which always shows which kind of object will be selected at this point.

3) Now press the **right mouse button**, and the menu shown below appears on the screen, which lists all possible modifications that can be made with the selected objects. This menu is called a context menu. If nothing is selected, the context menu comes to the graphics area.

4) Click on “Move” in the context menu.

5) The text at the mouse pointer tells you that you should select the point to be moved. Click on a corner point of the opening. By moving or with the new position of this chosen point you specify the new position of the selected objects.

6) The point now to be input defines the new position of the point to be moved. In our case there is no target point to grab so we have to input it. This is best done with the *relative input*.



Press the key <R>. The relative point, marked by a small red triangle, must be placed on the point to be moved. If it is not there click on this point. As a confirmation the marking triangle is moved to this position. Now type in the relative coordinates (-0.5 ↵,0↵).

Were you successful? In any case undo the change using the Undo function.

☞ *The procedure described above (selecting objects, then calling the context menu with the right mouse button) is called an object-oriented procedure. The more different objects you select, the smaller the number of active functions of the context menu, since it comprises the complete set of methods that can be applied to all the objects. CEDRUS is firmly based on this principle.*

☞ *On the various techniques for selecting objects the user should read the corresponding chapter in the Help document "The Graphics Editor".*

☞ *Changing the attributes of objects is a very important function, which can also be carried out using the right mouse button. The line "Properties" in the context menu, however, is only active if a single object or several similar objects are selected. Important information on this topic can also be found in the Help document of the Graphics Editor under "Working with Attribute Dialogues" .*

A 2.2.6 The tab sheet 'Punching'

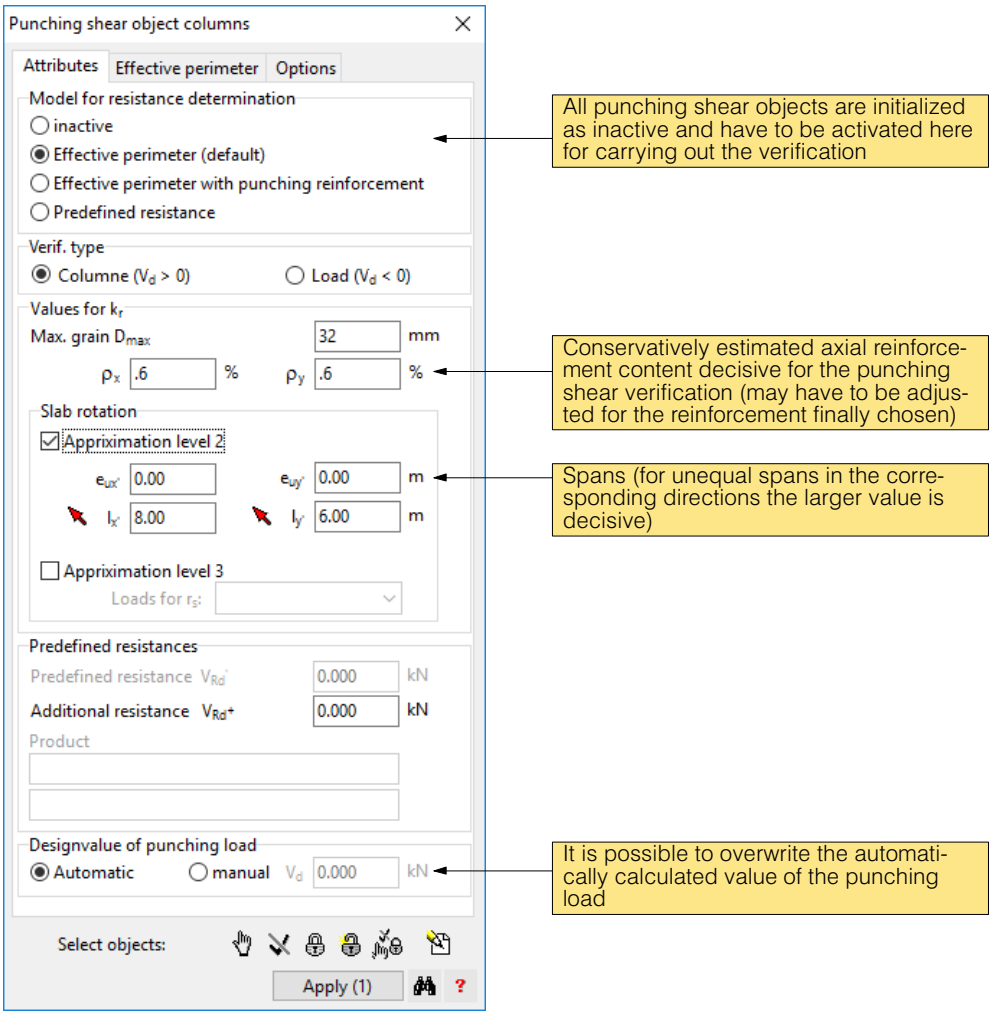
For a short overview on how to deal with punching shear in CEDRUS see Page A-15.

We want to carry out a verification of punching shear for the middle column.

▶ Change to the tab sheet 'Punching'

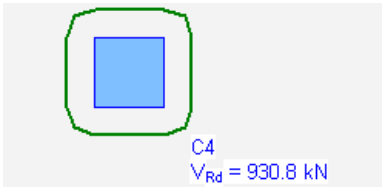
☞ *When changing to the tab sheet 'Punching Shear' the structure input is automatically checked for errors. If the check finds errors, then one is returned automatically to the tab sheet 'Structure', in order to be able to carry out the necessary corrections or additions.*

▶ Double click on the middle column, in order to show the Properties dialogue for the object to be investigated for punching shear. Set the properties as follows:



▶ In the tab sheet 'Options' in the Labelling dialogue, activate the identification and the punching shear resistance and then select [Apply] for the dialogue.

The punching shear object is now shown as follows and possesses the punching shear resistance shown.

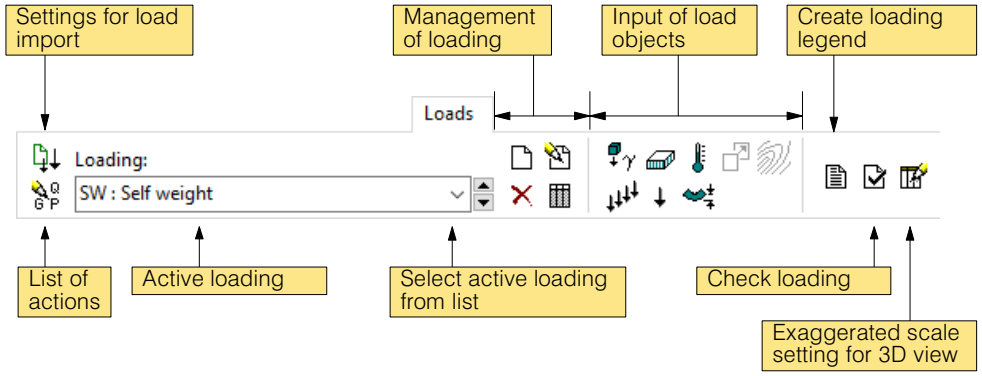


Whether this resistance is sufficient or not, will be shown later in the output results of the punching shear verification.

A 2.2.7 The tab sheet 'Loads'

In this tab sheet the loadings with their loads are input and modified.

▶ Click on the slider and you will obtain the following tab sheet:

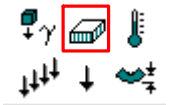


According to the problem description the following loadings have to be introduced:

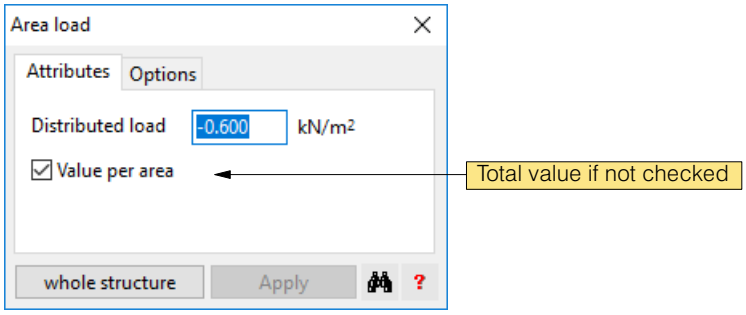
- Dead load of slab comprising the weight of the input slab and a uniformly distributed load of 0.6 kN/m^2 for the surfacing.
- Live load of 2 kN/m^2 in the most unfavourable positions on four fields (category: residential building).

A loading with the dead weight of the slab is introduced automatically by the program. It is considered as a gravity (acceleration) load, which leads automatically to the desired dead load on the basis of the selected materials and slab thicknesses in the material boxes.

We still have to add to this (pre-selected) loading the weight of the surfacing. One could introduce a new loading, but this is not necessary here since it is also a case of a permanent load.



▶ Select the button for distributed loading and in the dialogue set the desired value of -0.6.



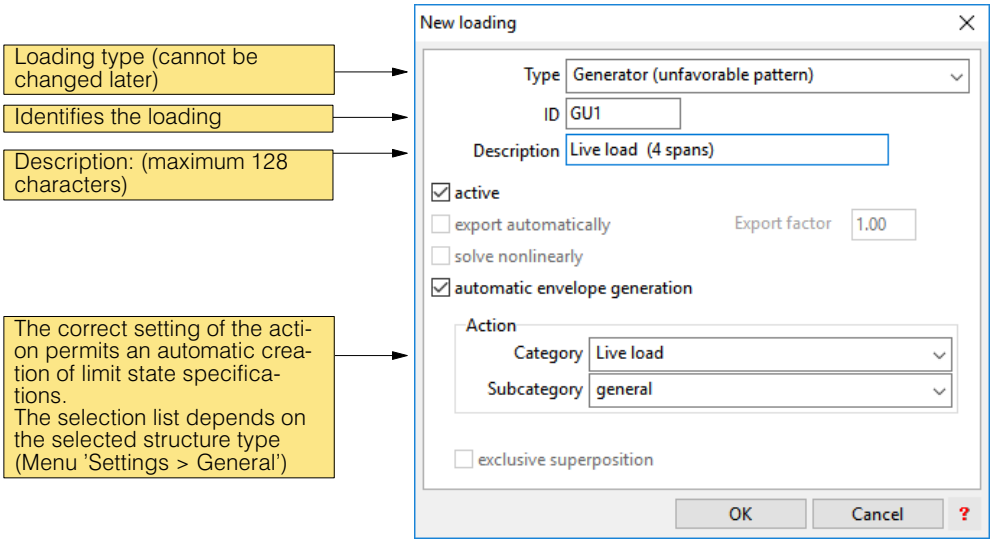
☞ *Since the Z axis points upwards, loads are normally input with a negative sign.*

▶ Click on [Whole Structure] and place the Load box with the <LMB> anywhere within the slab

We come now to the live load, which has to be considered fieldwise unfavourably. For this we need the four corresponding loads, which we have to input individually. It is easier, however, with the corresponding 'Generator for an Unfavourable Load Pattern', for which all load fields have to be input explicitly, but in the same loading, the *Generator Loading*. The program then performs the input of the required loadings and their correct introduction into the automatic limit state specification.

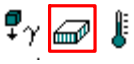
▶ Click on this button to create a new loading.

The following dialog window appears:



▶ Make the settings according to the figure and click on [OK].

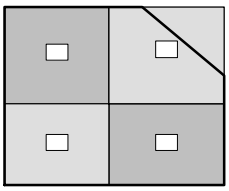
The loading is now active for the input. The buttons to input load objects are also active in the tab sheet. Now the distributed loadings for the four fields have to be input.



▶ Click on this button to introduce a distributed loading and in the Load dialogue set the load value to the required value of -2 kN/m^2.

▶ With the rectangle tool of the Graphics Editor draw the four load rectangles, which together cover the whole slab.

4 load rectangles



☞ *Click twice on a tool for it to remain active, and you can then input several objects one after another. It is deactivated again using <Esc>.*

From the generator loading just defined, for which no direct results are obtained, the program creates as many individual loadings as loads are defined. In our case just 4.

☞ *If later you want to change something in the generated loadings, then you always have to make the corrections in the loading generator. CEDRUS then generates afresh the corresponding loadings automatically.*

☞ *To simplify input the load areas of the distributed loads can be drawn over openings and also over slab boundaries. The program only considers the load on the effective slab area.*

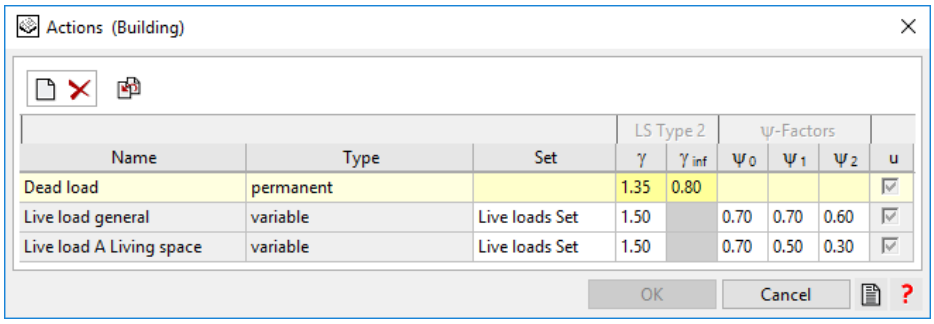
▶ Click on the button 'List of Loadings' and you will see a list of the previously defined and generated loadings with the corresponding properties. Close the dialogue again.

☞ *There is also a generator of moving loads, with which you can place at any position a vehicle previously defined by arbitrary loads using a sequence of positioning lines. The program then introduces the corresponding loadings and takes these into account in the limit state specification as 'either-or' loadings (more information to be found in the context help).*

Before we continue the input, we want to look at two other important dialogues in the Load tab sheet.

The dialogue with the List of Actions

▶ Click on this button on the far left of the Load tab sheet to open the following dialogue:



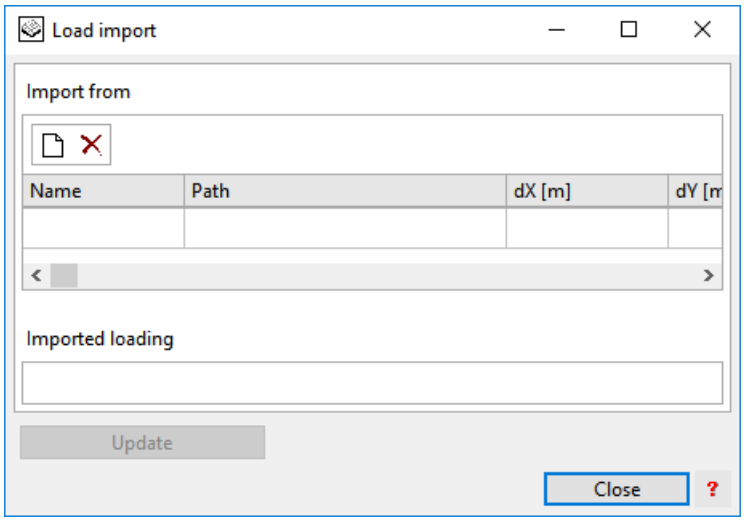
It contains all actions that we have assigned to the input loadings. If we want to work with actions that are not available in the corresponding list field of the Loading dialogue, then in this dialogue we can define beforehand our own actions.

The actions and their load and combination factors are required for the limit state specification. More information on actions and limit state specifications is to be found in section B 12.2 as well as under context help for the dialogue.

If you look more closely you will see that an action 'General Live Load' is entered in the list, that we have never explicitly assigned to a loading. What we have tacitly assumed is the setting 'Automatic Export' in the Loading dialogue, which leads to an export combination of the action in question. Here it is a matter of a load transmission to an underlying storey. All relevant information is to be found in section B 10.4.

The dialogue for load import

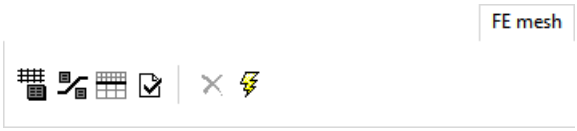
▶ We exit the dialogue with the list of actions and click on the adjacent button on the far left in the Load tab sheet to open the load import dialogue:



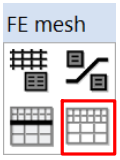
Here we can specify from which overlying slab (possibly more than one) loads have to be imported, and with the button [Update] the import of the loadings can be activated. More information on load transmission is to be found in section B 10.4.

☞ *In the building module of CEDRUS the load transfer works differently. The dialog shown above is only of interest if you work with individual slab calculations outside the building module.*

A 2.2.8 The tab sheet 'FE Mesh'



The FE mesh required for the numerical analysis is automatically generated by the program. You only have to provide some parameters, usually just the maximum element side length. This is done more or less analogously to the material properties, by placing an attribute box for FE mesh zones. The mesh parameters can vary from zone to zone. In this case it requires the necessary zone separation lines and per zone one attribute box.



☞ *Normally the program automatically places an attribute box with some pre-settings. Make sure in these cases - by making the mesh visible (using ,the Layer button on the right, that after solving or after its explicit creation the mesh is available with the lightning button shown below), - that you are able to accept the mesh. As mentioned above, the FE method is an approximate numerical method, whose accuracy increases with the fineness of the mesh. It is the user's responsibility to ensure the suitability of the mesh.*

▶ In our example we want to work with a mesh with a maximum side length of 0.8m. If a mesh attribute box exists select it and set the maximum side length to 0.8. Then click on [Apply]

Otherwise, click on the button for attribute boxes and in the dialogue window that appears set the maximum side length to 0.8. Click on [Introduce] and place the box anywhere on the slab.

You can generate the mesh with the adjacent button. This, however, is not really necessary. The mesh generation is carried out automatically by the program, if required.

In special cases you can also define zones with a manually produced mesh. More on this and also on the other mesh parameters can be found using the Help function in the dialogue window for the Attribute Boxes.

☞ *General remarks on FE meshes can also be found in section B 11.*

☞ *A subsequent modification of the automatically generated mesh is not possible in CEDRUS. If you feel that the generated mesh is not suitable try a finer mesh, and adjust the other mesh parameters or work with several mesh zones.*

A 2.2.9 The tab sheet 'Calculations'

This tab sheet includes the following briefly described buttons. If you do not need to input anything here, then you can proceed directly to the 'Results' tab sheet and the standard solution procedure is executed automatically by the program, if results are requested.

Results for all input and active loadings are always available in the 'Results' tab sheet. However, if you want combinations of results (results of loadings combined with arbitrary factors) then you have to define these combinations using the adjacent button. The details can be obtained with the help of the dialogue window that appears.

☞ *The results of derived nonlinear loadings should strictly speaking not be superimposed. For this purpose we have combinations of loadings, in which the loadings are multiplied by selected factors and comprehended in a new loading. This loading can then be analysed nonlinearly.*

This is the button for the input of limit state specifications. In standard cases, as in our example, the following limit state specifications are created automatically on the basis of an assignment of the loadings to specific actions:

for the limit state of ultimate load:

- !ULS

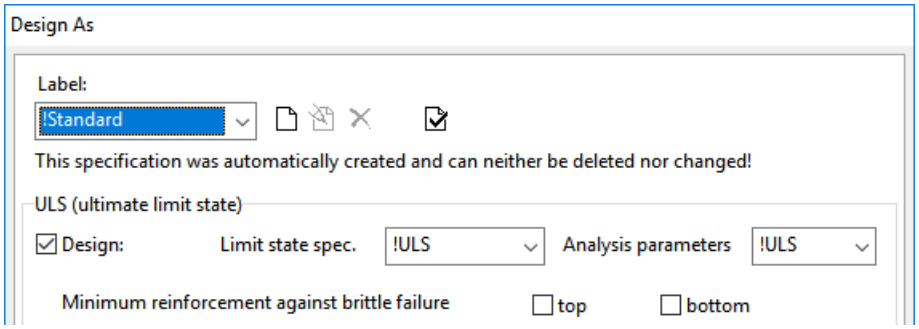
and for the limit state of serviceability:

- !SLS (quasi-permanent)
- !SLS (frequent)
- !SLS (accidental)

These non-modifiable specifications can be viewed by clicking on the button for limit state specifications and also by entering them in the print output list. More information can be obtained by using the Help function in the dialogue window.

▶ Click on the button to check the automatically generated limit state specifications.

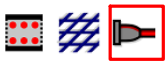
This button calls the Specifications dialogue for the dimensioning of orthogonal reinforcement. Here you specify which criteria the dimensioned reinforcement have to fulfil. Without special requirements it normally suffices to dimension for the ultimate load state, which is why there are two given and unmodifiable specifications called '!Standard' and '!Standard+':



With '!Standard+' the switch for the 'Minimum Reinforcement to avoid Brittle Failure' is active.

We will use this specification in our example. For a detailed description of general dimensioning specifications please refer to chapter B 12.3 .

For diagonal reinforcement we have this Specification dialogue. Details on this are to be found in chapter B 12.4.



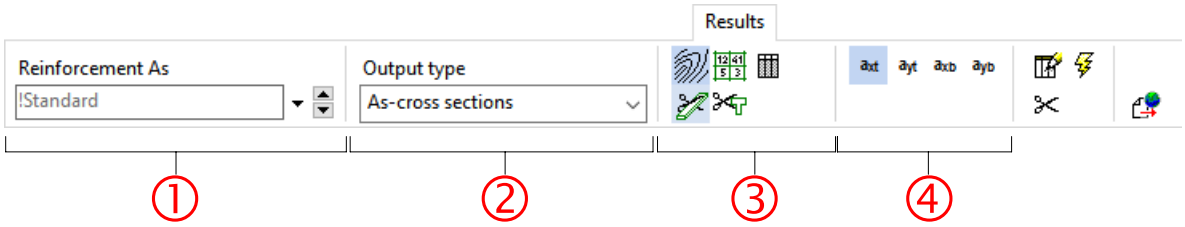
For prestressed slabs there are special dimensioning specifications, which can be modified using this button. More on this in chapter B 12.5.

This button is for the specification of dynamic analyses (modal analyses), see chapter B 12.6

This button serves to specify the analysis of bending in the cracked state (see chapter B 12.7).

This button serves to verify cracking (see chapter B 12.8)

A 2.2.10 The tab sheet 'Results'



With the „lightning" button on the far right the analysis and presentation of a result corresponding to the settings in the tab sheet are initiated.

In the individual parts of the tab sheet you can make the following settings (the content for each zone depends on the settings on the left of it):

① In the results tree that can be called, everything is listed for which there are results, i.e. normally it contains a list with all input loadings, results combinations, limit state and dimensioning specifications. Click on the small button with the arrow in the text field and select the desired entry in the list that appears.

② Depending on the setting in the field ① all the possible output quantities are offered for selection. Select the desired quantity from the list, if there is more than one to choose from.

③ Here you select the type of presentation. The choice depends on the selected output quantity. If you point the mouse at a button, the explanatory Help text appears. The six buttons shown in the above tab sheet (for section forces results) represent from left to right: isolines, graphical representation of principal values, numerically labelled mesh, tabular output, normal sections and beam sections (sections, for which the results values are integrated over a specified width).

④ If an ouput quantity has different components, then these are available for selection.

There are two remaining buttons:

With this one the results sections are defined. These must exist before the section results discussed under ③ are available. A dialogue window appears in which you can set certain parameters, before the sections are introduced [Introduce]. Details can be had using the Help function for the dialogue window.

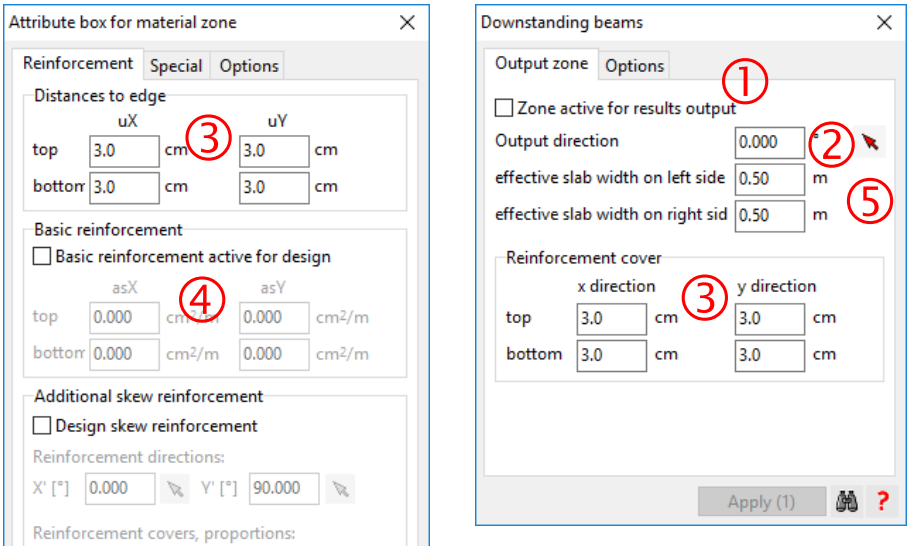
☞ *Downstands are implicitly beam sections and as other sections can be selected for beam section output. The section width has to be specified in the case of downstands. It is the sum of the effective width left and right in the tab sheet 'Output Zone' of the downstand properties dialogue. These widths are measured from the axis of the downstand.*

For some types of presentation additional parameters can be specified, such as the equidistance of isolines or the exaggerated scale factor for sections. See the dialogue window of the button on the left.

For exportation of reinforcement results.

Properties of output zones and downstand beams

The material zones defined in the 'Structure' tab sheet as also downstands also contain some properties needed for the output. If a corresponding Attribute Box or a downstand (with an active 'Results' tab sheet) is selected and in the context menu 'Properties' is requested, the following dialogue windows appear (if the boxes are not visible, these can be activated using the corresponding layer switch on the right side of the window):



① Zone active for output of results

Since the downstand results are likely to be output as beam sections, for surface types of output like isolines, they are switched off by default. Since the section forces in downstands are also often much greater than in slab zones, poorly readable isolinine plots would result (much too concentrated in downstand zones or too few lines in slab zones). This setting has no effect on cross section outputs.

② Output direction

The output direction specifies the x direction for the section force output and the output of orthogonal reinforcement. Beam sections are an exception, since this direction is always transformed into the direction of the beam axis. This also applies to downstand sections that are output as beam sections.

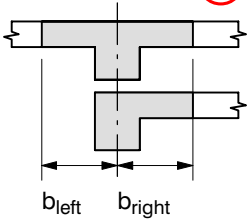
③ Distance to the edge of the reinforcement layers

The distances to the edge refer to the reinforcement layers in the abovementioned output directions. This also applies for downstands. If the downstand's axis does not lie in an output direction, then the edge distances in the beam section output are interpolated between the two values of the output directions.

④ Basic reinforcement

Basic reinforcement can be specified for slab zones. Then in the area outputs CEDRUS only shows the required additional reinforcement. The values of basic reinforcement have no influence on the section outputs.

⑤ Effective width for downstands



With this setting the width of the slab strip containing the downstand is specified taking into account the effective width to the left and right of the downstand's axis (looking in the output direction), over which the section forces in the beam section output are to be integrated. These widths have no effect on the analysis model for the slab.

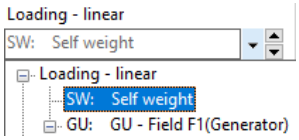
Required results for the example

As an example the following results will be created:

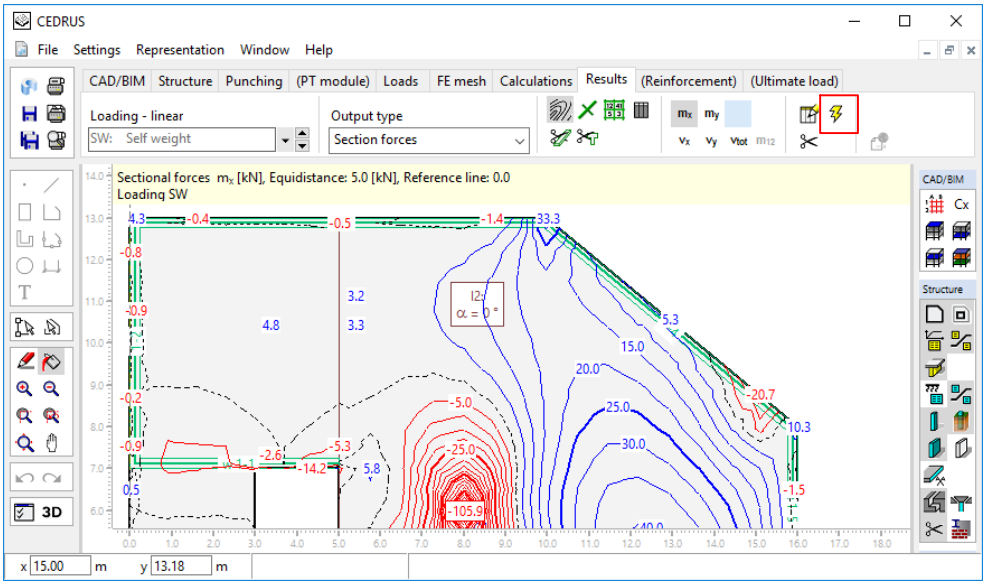
- Section forces for the loading dead load
- Displacements for the full load, i.e. for the combination of dead load + live load everywhere
- Reinforcement moments for the ultimate load state
- Isoline plot or the required bottom reinforcement in the x direction
- Reinforcement in the beam section of width 2 m in the x direction through the middle column
- Verification of punching shear for the middle column

Section forces for the loading Dead Load

▶ Change to the 'Results' tab sheet and in the Results Tree select the loading 'EG' :



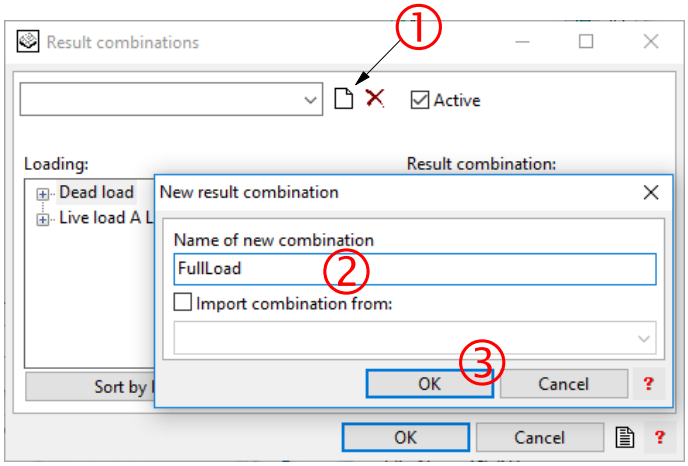
▶ Set the 'Results' tab sheet as follows; by clicking on you obtain the following result:



Displacements for full loading

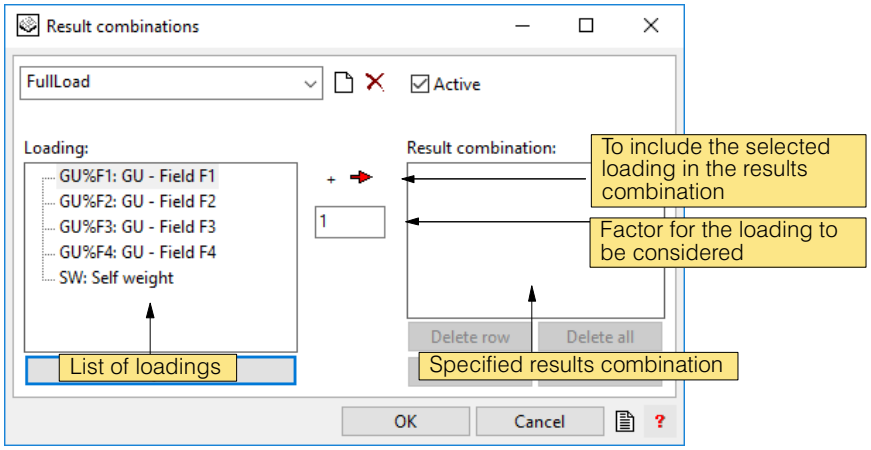
▶ Change to the tab sheet 'Calculations', in order to define the required results combination.

▶ Select the button for result combinations and in the dialogue the button for a new results combination ① :



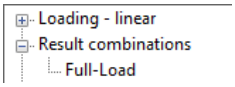
▶ Give the new results combination the name 'FullLoad' ② and then press [OK] ③ .

▶ Click on the button [Sorting according to Loading Identifier' at the bottom of the left dialogue window. You will obtain the following presentation:

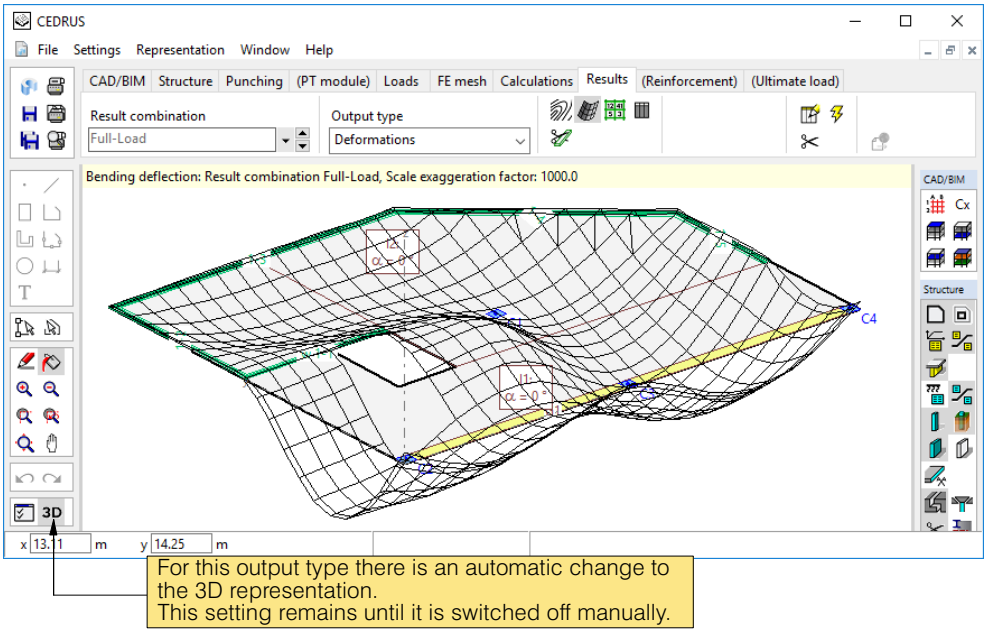


▶ Include all 5 loadings with the factor 1.0 in the results combination and then close the dialogue with [OK]

▶ Change once more to the 'Results' tab sheet and in the Results Tree select the result combination 'Full-Load':

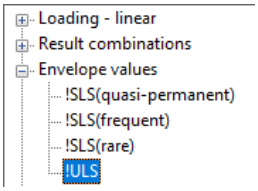


▶ Set the 'Results' tab sheet as follows. Click on and you will obtain the corresponding result:



Reinforcement moments for the ultimate load state (ULS)

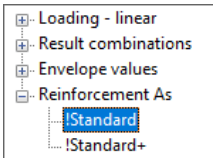
▶ In the Results Tree select the automatically created limit state specification '!ULS':



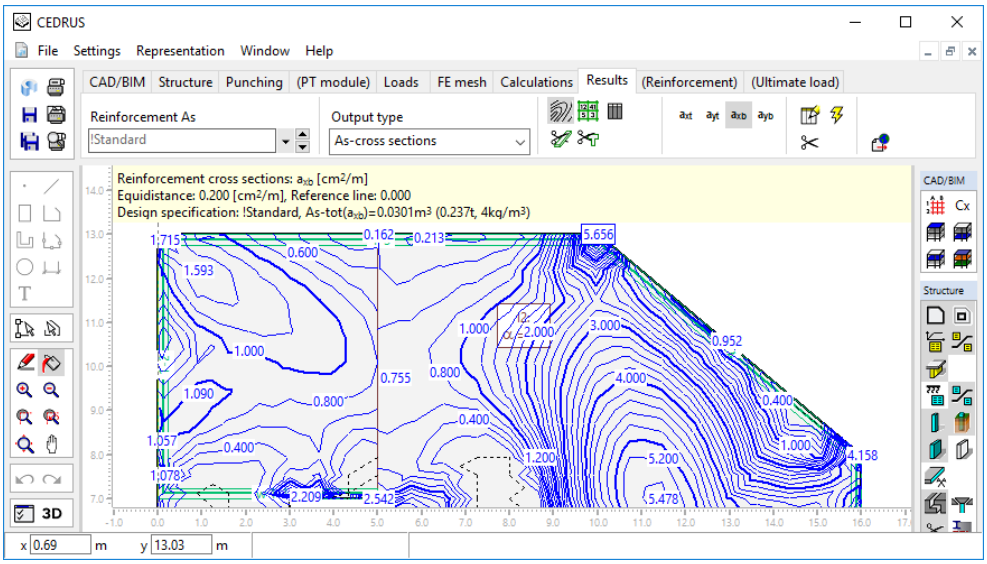
▶ Set the 'Results' tab sheet as follows. Click on and you will obtain the reinforcement moment for the top layer of reinforcement in the x direction:

Isolinine plot for the bottom reinforcement layer in the x direction

▶ In the Results Tree select the automatically created dimensioning specification '!Standard':



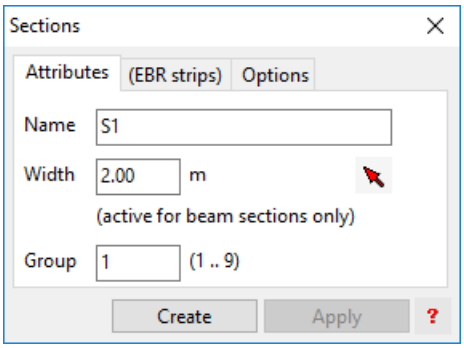
▶ Set the 'Results' tab sheet as follows. Click on and you will obtain the required reinforcement for the bottom layer in the x direction:



Reinforcement in the beam section

Required is the reinforcement in a beam of width 2 m in the x direction through the middle column. Thus this section must first be defined:

▶ Click on the button for the section objects. Set the width to 2 m and then select [Create].



We want to take the section though the column axis, and it should run from the opening to the right slab boundary. To avoid having to type in coordinates proceed as follows:

▶ Type the letter **A** for absolute coordinate input. Instead of typing in the X coordinate , grab it by clicking on the bottom right corner of the opening. For the Y coordinate click on the middle point of the column.

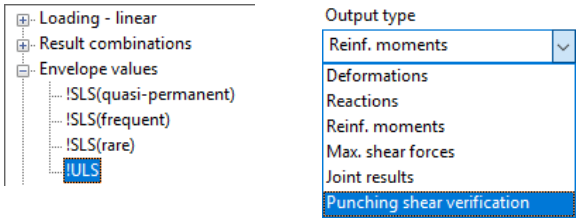
▶ To input the end point type the letter **X**. By clicking on the right edge of the slab the end point of the section is specified.

▶ Set the 'Results' tab sheet as follows and select the section just defined. Click on and you will obtain the required reinforcement in the beam section:

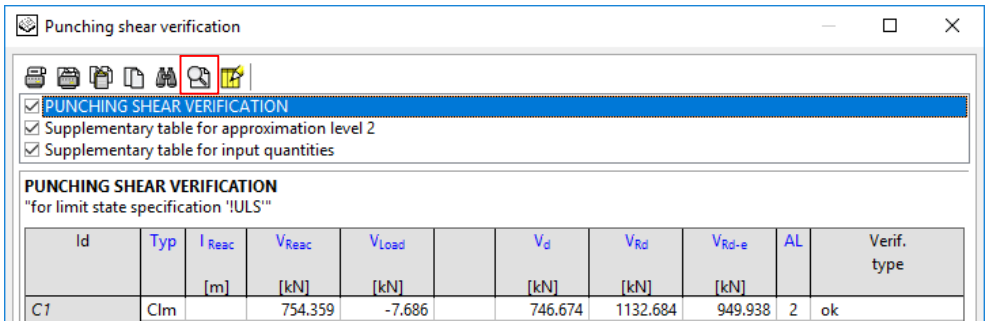
Verification of punching shear

The punching shear resistances for the object to be checked for punching shear were determined in the tab sheet 'Punching Shear'. For the verification of punching shear we have to compare this with the punching shear loads. The dimensioning values of the punching shear loads are given by the maximum reactions at the ultimate limit state.

▶ In the results tree select the limit state specification !ULS and as output quantity 'Punching Shear Verification'



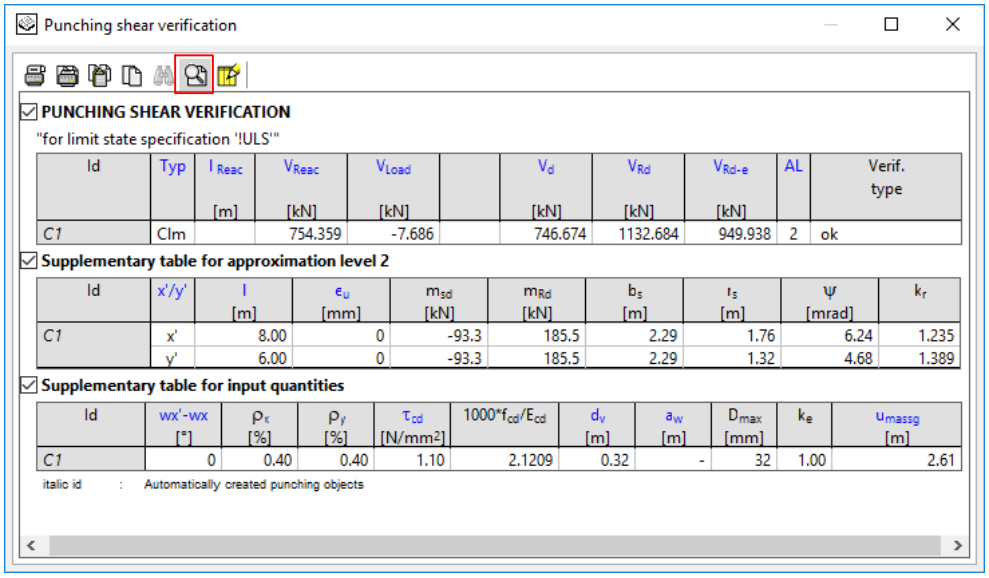
▶ By clicking on the punching shear verification is output in tabular form:

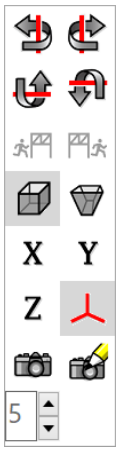


Id	Typ	l Reac [m]	VReac [kN]	VLoad [kN]		Vd [kN]	VRd [kN]	VRd-e [kN]	AL	Verif. type
C1	Clm		754.359	-7.686		746.674	1132.684	949.938	2	ok

In the tab sheet 'Punching Shear' we only activated as punching shear object the middle column, which is why the verification table only consists of one line. In this line the dimensioning value of the punching shear resistance V_{Rd} is compared to the punching shear load V_d. Since the resistance is greater the verification is fulfilled. V_{Last} is that part of the load that in the case of the maximum reaction acts within the critical core section. For V_d it may be subtracted from the maximum reaction.

If necessary, switch to the page view with the adjacent button.



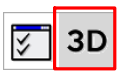


3D Representation

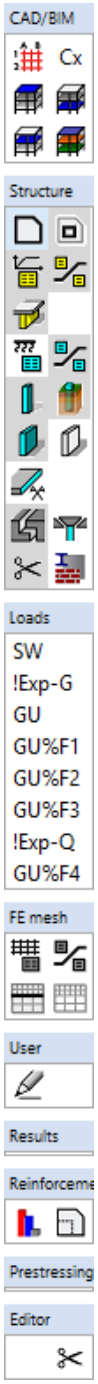
Certain types of results presentations are intended for a 3D view. In this case the program switches automatically to the 3D mode (parallel projection), in which you can select the projection direction arbitrarily. This is accomplished using the self-explanatory buttons palette, which is always shown on the screen in the 3D view. It is quicker with the mouse as shown below:

Keep the keys <Ctrl> and <Alt> pressed and move the mouse with the left button pressed. A horizontal mouse movement rotates the projection direction about a vertical axis and a vertical movement tilts the projection direction about a horizontal axis (one that is normal to the current projection direction).

Also try out once the rendering representation (camera button) and with the key <F1> call the Help functions for the individual buttons.



With this button on the left edge of the window you can switch between 2D and 3D representations.

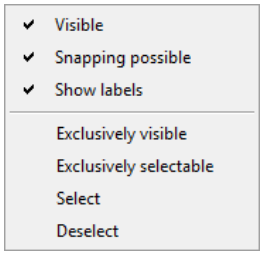


The Layer Switch

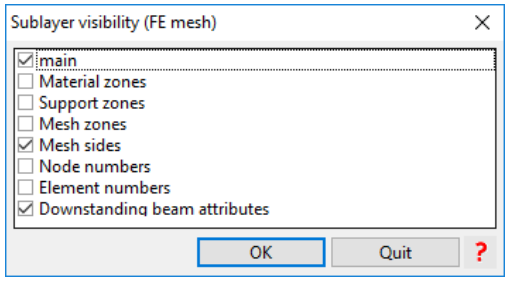
The whole of the graphics information is distributed among individual layers, whose visibility can be turned on or off individually.

The band displayed here contains the layer switches shown on the right side of the screen, which are concentrated in individual groups. By clicking on the button of one of the layers its visibility is switched on or off. This is important above all in the 'Results' tab sheet. During input (e.g. in the 'Structure' tab sheet) the layers wil be automatically set by the program.

Click with the right mouse button on a layer switch, then the following context menu is opened, which depending on the layer type may vary slightly from the shown one. The listed menu points should be self-explanatory (if necessary get Help using <F1>). The menu points refer to the selected (clicked on) layer.



Certain layers have a series of sublayers, whose visibility one may sometimes also want to turn on or off individually. In the context menu click on the menu line “Sublayer Visibility”, and the following dialogue appears, in which the buttons (check boxes) of the listed sublayers can be set as desired.



This is of interest, e.g., for the following layer:

- „FE Mesh”: you can show the nodes and/or element numbers
- In the case of graphically displayed numerical values, for each selection the node and element middle values are labelled. Since these each have their own sublayer, they can also be turned on or off individually.

The header with a blue background of each layer group also has a small self-explanatory context menu (click on it with the right mouse button). One of its important functions is changing the presentation between lists (as in the group 'Loadings'), to save space and the listing of all layer switches, as in the group Structure. Often, with the graphical results the labelling gets in the way of the structural object. Switch this off by selecting in the context menu of the layer group 'Structure' the line 'Labelling Invisible'.

The layer groups 'User' and 'Results' need some further explanation:

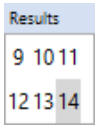
The Layer group 'User'

If you want to print the graphics with additional information like dimensioning lines or additional labelling, for this we have the layer group “User”, in which the button for a first User Layer is already given. If you want to distribute additional graphics information in different layers using the context menu of the group header “User” you can create as many User Layers as you want.

Also use the User Layer for preparatory work (auxiliary construction lines, help points, etc.) for the graphical input of the structure.

To be able to draw in a User Layer this must first be made the active Layer (don't confuse this with making it visible). Click with the right mouse button on the button of the desired User Layer and then in the context menu on the line “active” or use the corresponding menu item in the context menu of the graphics area (this appears if you haven't selected anything and with the right mouse button click anywhere on the graphics area). Now you can use all active Graphics Editor drawing tools.

The Layer group 'Results'



The last six 6 created graphics for the results remain in the clipboard provided you do not end the analysis and are made available here with the consecutively numbered buttons. Here above all the following possibilities were envisaged:

1) You can “superimpose” several graphics for the results by switching on the corresponding Results Layer. For this you must keep the <Shift> key pressed, otherwise the already activated results layers will be switched off.
2) You can switch quickly back and forth between the last pair of results.
3) Saving graphics results can also be useful for the structure or load input. With the input of critical vehicle load positions, e.g., one can show previously calculated influence fields.

Regarding this point we want to make the following remarks:

- Limiting it to the last six graphics was a deliberate choice, because the graphics can use very large amounts of storage. This limit cannot be changed by the user.
- During storage this results buffer is always deleted. A factor is also the storage economy, and that loading an analysis could also take a long time.
- By making an entry in the output list as many graphics as desired can be saved and inspected again at any time (see below).

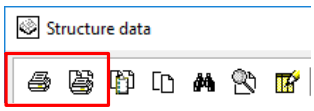
A 2.2.11 Analysis Documentation

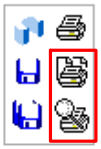
Print immediately

Create print entry

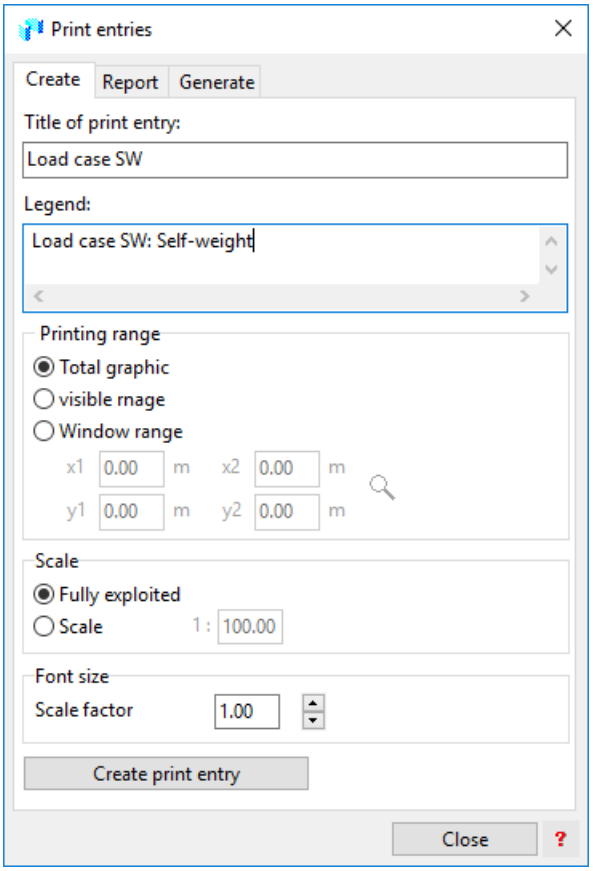
All printable data, whether in text or graphics form, can be printed directly in the corresponding parts of the program using the buttons shown on the left or entered as a print entry in an output list for subsequent printing.

Text is always output in separate table windows, which provide buttons in their header for printing or making a print entry:





In the case of graphics it is a question of the content of the graphics area or a part of it. Printing and print entries for graphics is carried out with the button at the top left of the program window. Before printing or making a print entry the following dialogue window always appears, which is largely self-explanatory:



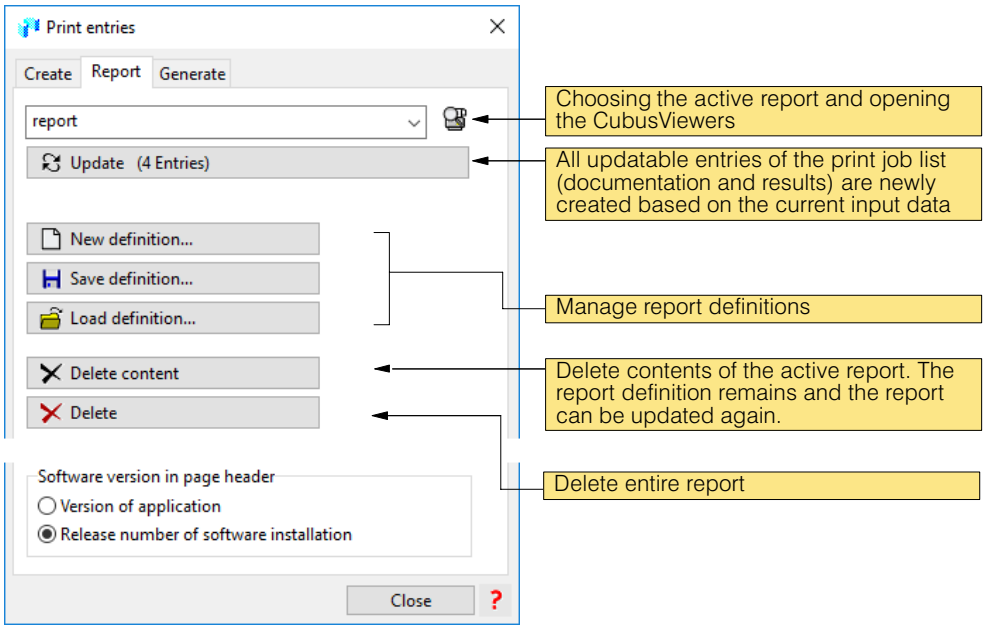
For the complete documentation of an analysis one works with print entries, so that before printing one can have a preview of the whole "report" in the CubusViewer and corrections can still be made with regard to sequence, scales, page breaks, page numbering, etc.

As a **report**, therefore, the totality of the created print entries, which are managed in the CubusViewer, is meant.

Report update

The first version of a report is basically created by means of individual print entries, whereby for the documentation of the input data a report generator is available. After making changes in the input an existing report can be automatically updated.

In the second tab sheet of the dialog 'print entry' are the features for managing and updating reports:

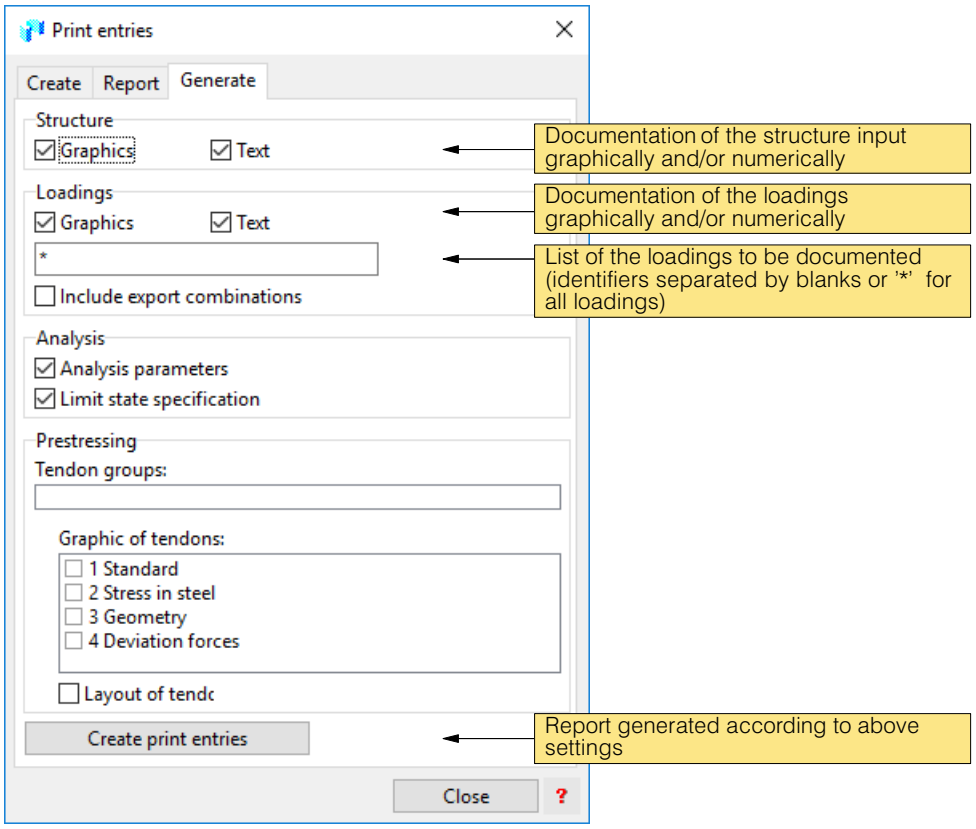


Modify report

In CubusViewer print entries can be moved or deleted and it may be the insertion position for future print entries are defined. In addition, print entries can be edited, that is whose content can be adapted: for example, the segment can be varied, layers can be turned on or off, other results can be selected.

The Report Generator

Click on the button for a print entry of the graphics area, and the following dialogue appears whose second tab sheet contains a configurable report generator, whose possibilities are explained in the figure.



The print entries for the results have to be created manually. When updating the report, however, they are also updated.

Manual print entries

The following list shows all points in the program, where print entries are possible and are in fact in the sequence in which they would usually be needed for a report:

Structure

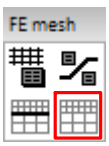
- For documentation of the structure input it is best to enter in the 'Structure' tab sheet at least one graphic of the slab as a graphical description of the slab. Make sure that all required layers are active. If you have a large slab with many structural objects, it is probably better to enter several graphics with different layer switches or additional excerpts from the graphic.

  In the case of objects with Labelling Boxes don't forget that the size of the labelling for most objects can be modified in their attributes dialogue in the tab sheet 'Options'.

  Furthermore, we would point out the possibility of dimensioning the graphics with a view to an optimum documentation and to add to the graphics as required. This is done in layers of the group "User".

- As legend to the structure graphics the tabular listing of the structure objects is useful, which can be created with the button shown (far right in the tab sheet 'Structure').

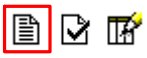
FE mesh



- The documentation of an FE analysis may also include the FE mesh. For this purpose, open the layer for the FE mesh using the layer switch and make the corresponding print entry.

If you use tabular results, it may also be necessary that you add a graphic with the node and element numbers. These can be added by clicking with the right mouse button on the layer switch for the FE mesh and in the menu “Sublayer Visibility” activate the desired sublayer.

Loads



- In the tab sheet 'Loads' with the button 'Legend of the Loading Data' tabular documentations of the loadings can be created. The loadings list (see below) also provides generation possibilities for this. The tabular documentations can also be extended to include graphics. You can move from loading to loading and for each “interesting” loading you can make a graphics entry. It is also possible to include several loadings by adding the corresponding loading Layers to one graphic.

  Further, it is also possible to provide the graphics with dimensions and other graphics features. Don't forget the possibility of creating additional layers in the layer group “User”, whereby any structural dimensions or other dimensions are not deleted first, but only hidden visually.

- The dialogue window 'Loading List', which you can display on the screen using the button shown left, has at the bottom a print entry button for entering the loading list.

  With these buttons at the top of the dialogue window 'Loading List' tabular documentationof the selected or active loadings can be generated. The loadings are selected in the usual Windows way - <Ctrl><A> to select all loadings is also supported.

Results combinations and limit state specifications

- The documentation of any results combinations and limit state specifications is made in the tab sheet 'Calculations' in the corrresponding input dialogue windows.

Results

- All graphical and text results documents can be entered in the usual way in the output list. In the case of graphics any unwanted layers can be made invisible and any relevant structure, load or FE mesh layers of interest to the displayed results can be included. The user is reminded of the possibility described earlier of superimposing several layers of graphical results.

The CubusViewer

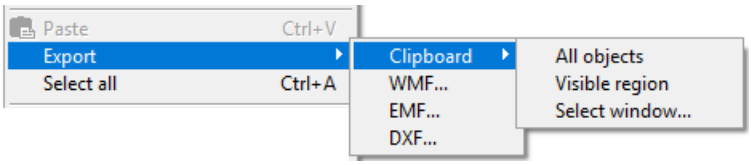
The CubusViewer is used to provide a preview before printing the report, i.e. of all previous print entries. On the left side of the CubusViewer there is a list of all entry titles and on the right the print preview. The CubusViewer provides the following possibilities (for the details consult its Help menu and the CubusViewer Handbook accessible via the Help menu):

- Activation and deactivation of print entries
- Switch on or off the possibility of updating entries
- Moving entries (print sequence)
- Changing the scale of an entry
- Creating page breaks
- Definition of ones own page layout (with logo, etc.)
- Selection of a page layout / paper format
- Selection of printer
- Zooming in the print preview
- Switching off the print preview for quicker modification of the print entry list
- Selection of print output as coloured, black and white or with grey shading
- Printing the active print entries
- . . .

Output interfaces to other programs

Graphics

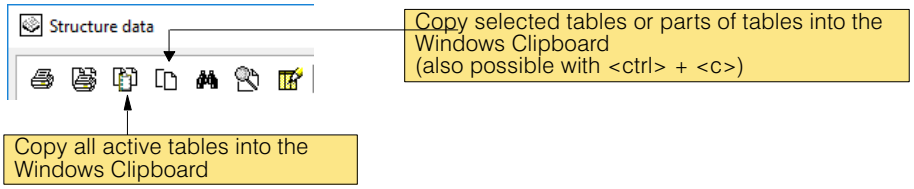
If nothing has been selected then in the context menu of the graphics area (<RMB>- click on an empty place in the graphics area) offers the following possibilities of exporting graphics:



In the case of 'WMF' (Windows Metafile), 'EMF' (Enhanced Metafile) and DXF corresponding files are created at a location that is selectable. Graphics copied to the Windows Clipboard can be exported using the command 'Paste' directly to Word or other documents.

Texts

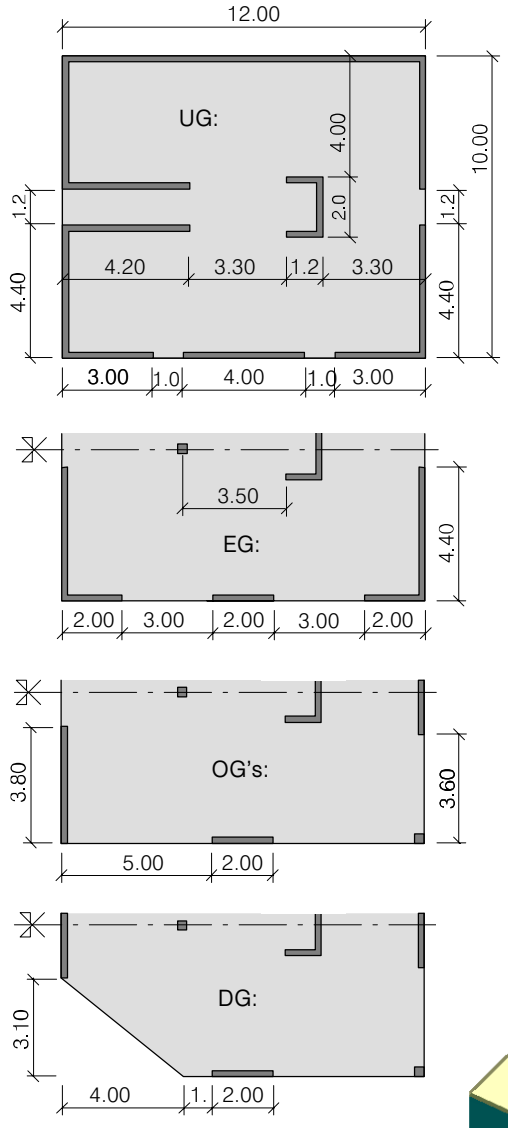
Texts from tables can also be imported into Windows documents like Excel and Word.



A 2.3 Introductory Example: Building

A 2.3.1 Problem Definition

The model of a five-storey building with a basement UG, ground floor EG, two identical upper floors OG1 and OG2 and an attic floor DG has to be generated and using the stacked slab model the loads should be analysed top-down.



Dimensions in m;
Objects without dimensions are same as those of the underlying storey

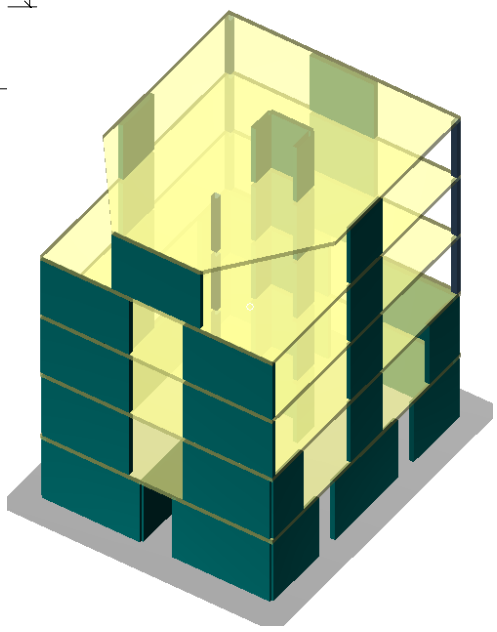
Slab thickness:	UG	0.26
	else	0.24
Wall thickness:		0.20
Columns:		0.25x0.25
Materials:	slabs, walls	C30/37
	columns	C40/50
Elevations	Found.	-3.0
	UG:	0.0
	EG:	3.0
	OG1:	5.8
	OG2:	8.6
	DG:	11.4

Permanent loads:

all storeys	dead load
UG . . . OG2	+ 1 kN/m^2
DG	+ 5 kN/m^2

Variable loads (living areas):

UG	5 kN/m^2
EG . . . OG2	3 kN/m^2
DG	2 kN/m^2



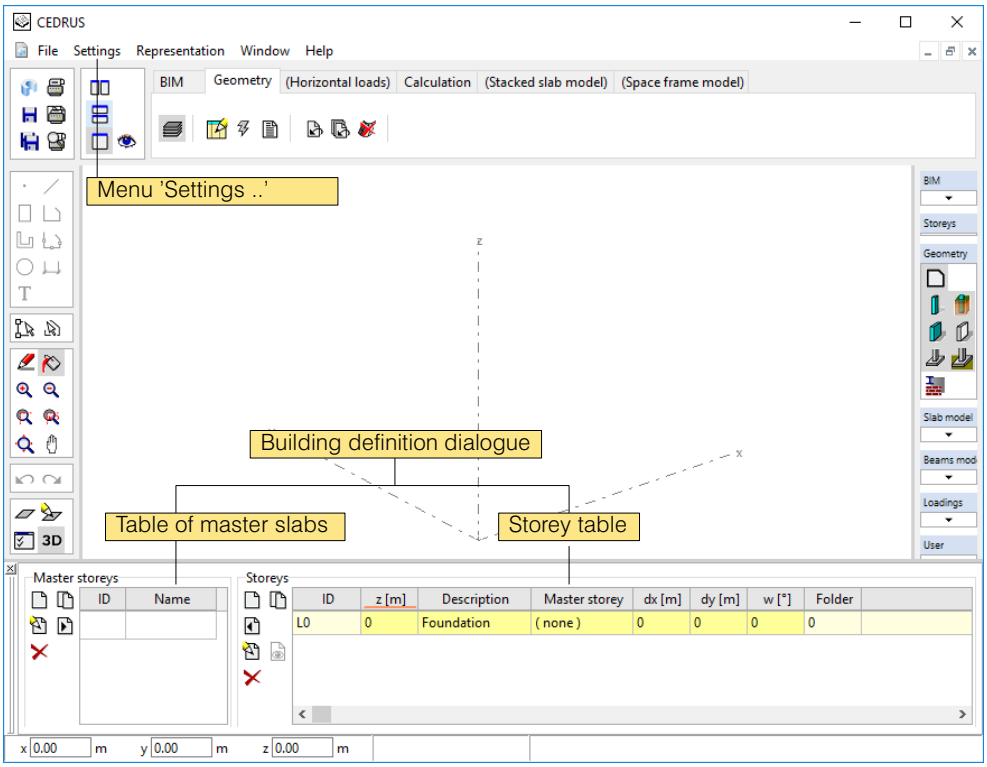
A 2.3.2 Prerequisites

- For the following description it is assumed that you have some experience in the use of the basic version of CEDRUS.
- You should first read Chapter A 1.5. It will provide a brief overview of the building analysis procedure.

A 2.3.3 Program Start

▶ Start CEDRUS and open a new building analysis using the menu „File“ > „New...“ > „Building Model..“.

You will now see the Building window with the building definition dialogue:



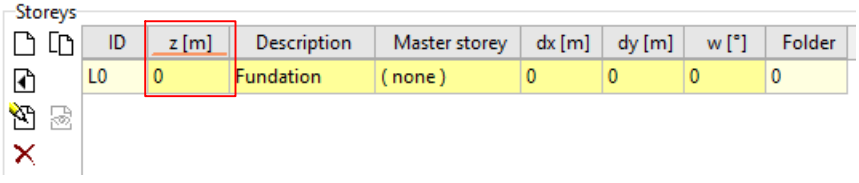
A 2.3.4 Materials

▶ If necessary, in the menu 'Settings > Materials', select the required building material class 'C30/37' for the given material 'C'. For 'CC' set 'C40/50'.

☞ *The materials can only be changed in the Building window and apply to all master and storey slabs.*

A 2.3.5 Foundation Zone

On opening a new building analysis the program automatically creates a storey slab called 'Foundation' in the storey table of the Building Definition dialogue and assigns it the elevation 0.0.



Here we have a case of a „degenerated“ slab that only consists of an area support box. Such a slab forms an unlimited foundation zone, that acts such that walls and columns resting on it are supported at this level.

These 'degenerated' slabs can, like every other slab, be modified and if necessary be extended to become a foundation slab. We want to leave it as it is, but just give it the prescribed elevation:

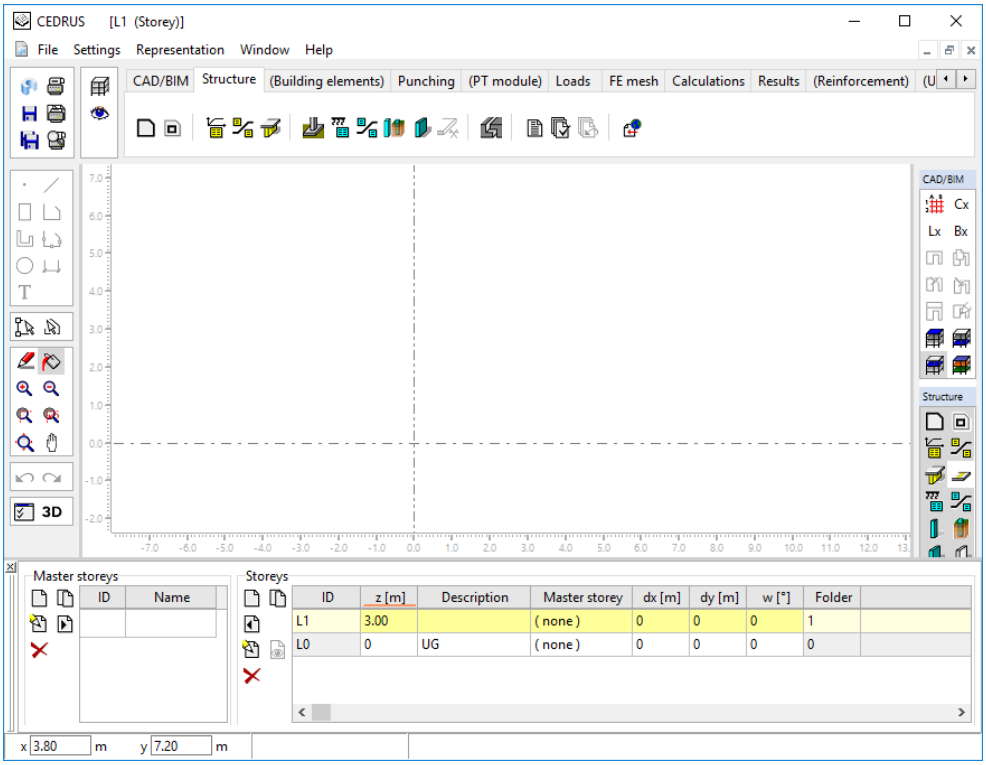
▶ Click in the storey line L0 on the field in the column 'z' and overwrite the default value by -3.0 [m].

A 2.3.6 Input of the Basement Slab „UG”

Since there is only one UG slab in the building, we want to input it directly as a storey slab (and not as a master slab).

▶ Click on the button for a new empty slab (in the storey part of the Building Definition dialogue). Write 'UG' in the field of the column 'Description'.

In this way a new floor is added, the corresponding (still empty) storey slab was opened and its document window was placed over the building window.



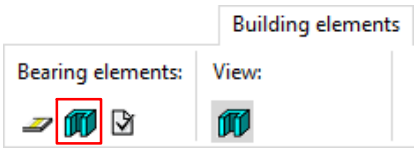
The shown elevation of 0.0 corresponds to the value specified in the problem description.

If the UG slab already exists from an independent CEDRUS slab analysis, we could now import it using the menu „File“ > „Import“ > „CEDRUS-?“. Here this is not the case, which is why we now input the slab, as we would in the base module of CEDRUS.

▶ Input the UG slab including the loads in the usual way corresponding to the problem description.

Special:
- wall support allows free rotation, settlement stiffness with calculated Sdz

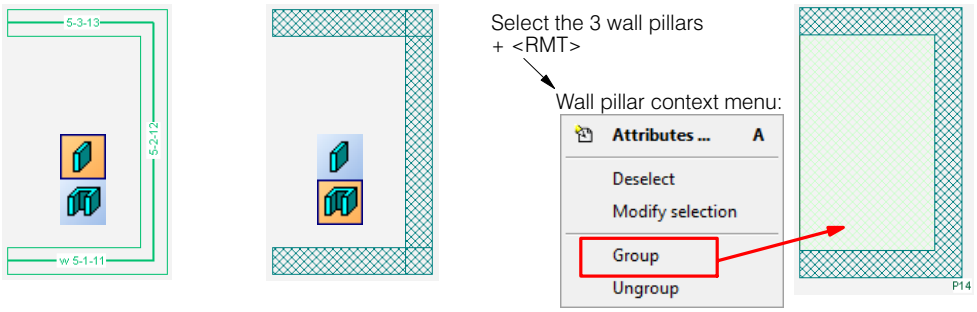
The tab sheet 'Building Elements'



The tab sheet only appears for building slabs. Except for the marked button for wall pillars all the functions available here serve to create the frame model, but we will not discuss this now. The properties of the wall pillars, however, also play a part in the stacked slab model.

Wall pillars

In the building model, walls, as we know from the CEDRUS slab program, have a different meaning and extended properties, which is why they are called wall pillars. The following figure shows walls in a building slab, as they are represented in a structure or building elements tab sheet:



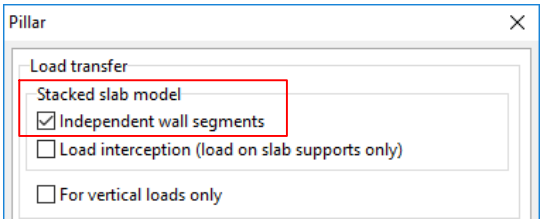
tab sheet 'Structure': wall consisting of 3 segments

tab sheet 'Building Elements': the 3 wall segments appear by default as 3 independent wall pillars

If required, several wall pillars can be grouped using this procedure, i.e. merged to a single wall pillar.
'Undo' is the Undo function.

Each segment of the walls input in the 'Structure' tab sheet is by default its own independent wall pillar. By grouping, several wall pillars (see above figure) can be merged to a single wall pillar.

This grouping of wall pillars is of importance above all in regard to a frame model. Depending on the setting in the Properties dialogue of a wall pillar it may also be important in the stacked slab model:

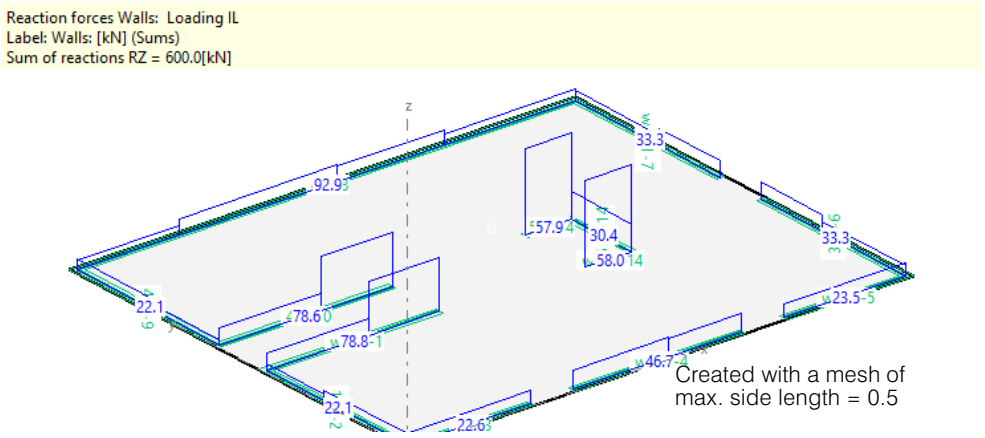


With the setting 'Independent Wall Segments' for the stacked slab model the grouping of wall pillars is of no importance. If one deactivates this property, the grouped wall pillar acts as a single cross section in the determination of the forces at its bottom. You can also find more on this with the setting 'Support (load only on slab support)' using the Help function of this dialogue.

▶ Group the three wall pillars in the inner part of the slab and for the properties of the newly created wall pillar switch off 'Independent Wall Segments'.

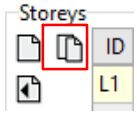
▶ Save the slab data.

▶ As a plausibility check analyse, e.g., the deflection due to bending and the reactions for the live load.



A 2.3.7 Input of the Slab „EG“

There is also only one EG (ground floor) slab, so that, as already with the UG (basement) slab, we input it directly as a storey slab. Since it is partly the same as the UG slab, the quickest procedure is probably to work with a copy of it.



▶ Select storey 'L1' and then with the button shown create a new storey L2 as a copy of L1.

▶ Select the new storey 'L2' and then the button shown to modify the slab.

▶ Modify the slab according to the problem description (including loads).

▶ Save the slab data.

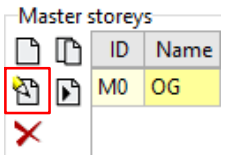
Special:
- columns: with area supports, stiffness with calculated Sdz , rotations free
- walls: supports with free rotation, stiffness with calculated Sdz

A 2.3.8 Input of the Master Slab for the OG's

Since the two upper storeys are identical, it makes good sense to work with a master slab. Since this master slab is only slightly different from the EG slab, we can make use of the possibility of creating a master slab as a copy of an independent storey slab.

▶ Select the storey 'L2' and then the button shown.

This creates a master slab as a copy of the slab 'L2'.



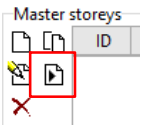
▶ Rename the master slab 'OG' and then click on the button to modify the slab.

▶ Modify the slab according to the problem description.

▶ Save the slab data.

A 2.3.9 Introduction of the Storeys OG1, OG2 and DG

The two upper storeys OG1 and OG2 as well as the attic storey DG can now be generated from the master slab „OG“.



▶ Select the master slab „OG“ in the Building Definition dialogue and then click once on the button shown.

In this way a new independent slab „L3“ is created from the master slab „OG“ .

▶ Modify the elevation of „L3“ to 5.8.

▶ Repeat the last point twice to generate the storeys „L4“ and „L5“. For the new elevations the storey height of the underlying storey is given automatically, which in the present example leads directly to the desired elevation:

Storeys

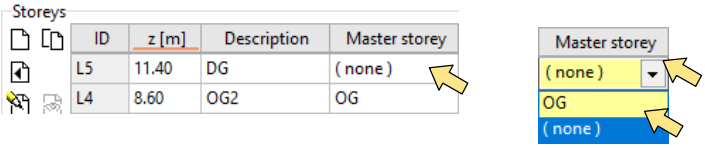
ID	z [m]	Description	Master storey
L5	11.40	DG	(none)
L4	8.60	OG2	OG
L3	5.80	OG1	OG
L2	3.00	EG	(none)
L1	0	UG	(none)
L0	-3.00		(none)

Storey slabs dependent on the master slab „OG“

Independent storey slabs

In the attic storey „L5“ according to the problem description we now have to adjust the floor outline and the loads, for it is in this way that the storey slab differs from the master slab and before we are able to make the changes to „L5“, we have to separate this storey slab from the master slab. In the column „Master Storey“ of the Building Definition dialogue for each storey slab there is either the name of the master storey or „(none)“, if we have an independent slab.

▶ Click on the field „L5“ / „Master Storey“. Then press the arrow button to open the selection list and select '(none)'.



In this way the storey „L5“ is uncoupled from the master storey and is now independent. Thus changes to the master slab only apply now to „L3“ and „L4“.

▶ Now modify the outline according to the problem description.

▶ In the live load loading, change the load value to -2 [kN/m^2] and the additional permanent loads to -5 [kN/m^2].

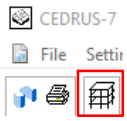
▶ Save the slab data.

▶ Change the descriptions of the storeys as follows:

Storeys

ID	z [m]	Description
L5	11.40	DG
L4	8.60	OG2
L3	5.80	OG1
L2	3.00	EG
L1	0	UG
L0	-3.00	

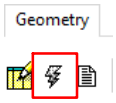
A 2.3.10 Generation of the Building Model



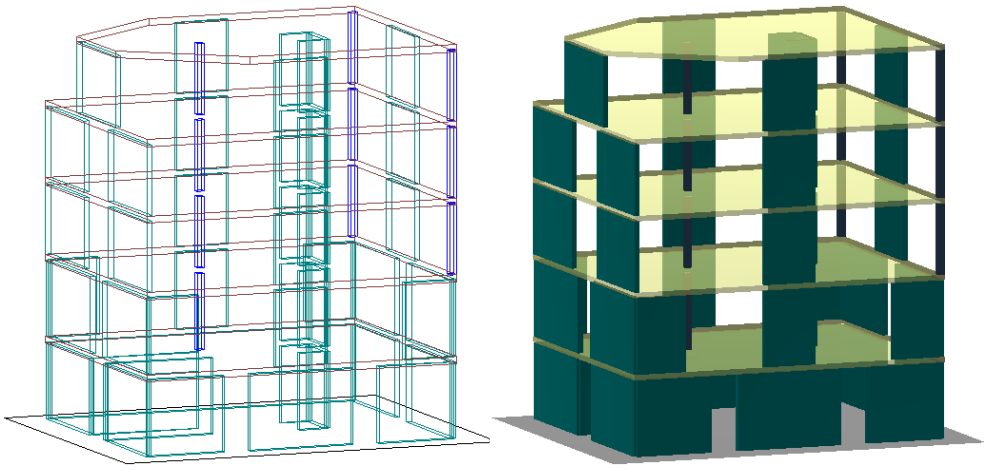
We now have to change to the Building window. This is best done as follows:

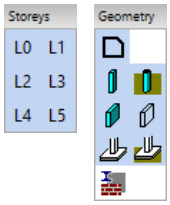
▶ Press the button shown here located top left in the program window.

This causes the Building window to appear.



▶ Press on the lightning button to generate the Geometrical Building Model. Answer any questions from the program in the affirmative.

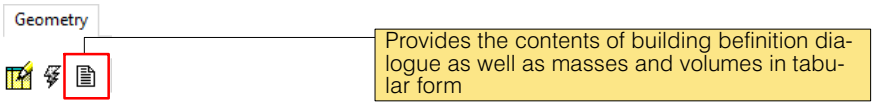




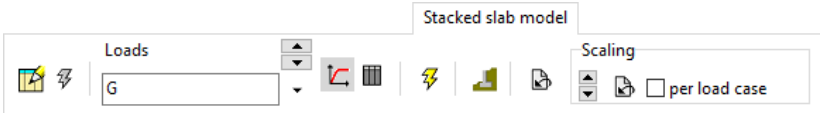
Note the layer switch for controlling the visibility. With the switches in the group 'Storeys' the visibility of the various storeys can be set individually and with the switches in the group 'Geometry' the visibility of structural elements can be selected.

☞ *In the rendered presentation (figure above on the right) many functions, especially also the change of tab sheet, are blocked.*

Tabular Output for the Buildings Model



A 2.3.11 Stacked Slab Model



Concerning the model

The stacked slab model is used for the top-down analysis of vertical and horizontal loads in the building. For horizontal loads Option H is required.

Storey-by-storey the loads acting on the slabs are transmitted to the supporting vertical elements (walls and columns), and the resulting forces at the bottom of the elements are transmitted to the storeys on which they rest. Details of the model may be found in Part D and especially in Chapter D 5.

The vertical loads, as in the CEDRUS slab program, are input in the building's slabs. The input of horizontal loads is carried out in the tab sheet 'Horizontal loads', which is not discussed further here.

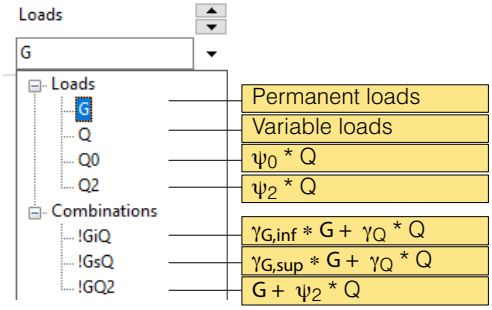
Analysis

▶ Change to the tab sheet 'Stack of slabs model' and analyse the stacked slab model with the lightning button.

After the analysis of the stacked slab model the imported loadings are generated in each storey slab together with the loads coming from above it, i.e. the slabs can be analysed and dimensioned.

Results in the vertical elements

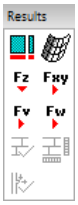
▶ In the Results selection field select the desired loading or combination, of which the following are given by default:



☞ *In the tab sheet 'Calculation' there is the possibility of defining further loading combinations (e.g. G * 1.0 + Q * 0.6). These also appear in the results selection field.*

Graphical

To present the forces at the bottoms of walls and columns in the layer group 'Results' the following layers are available:



Presentation of the footprints of the vertical element, i.e. where they transmit their forces.

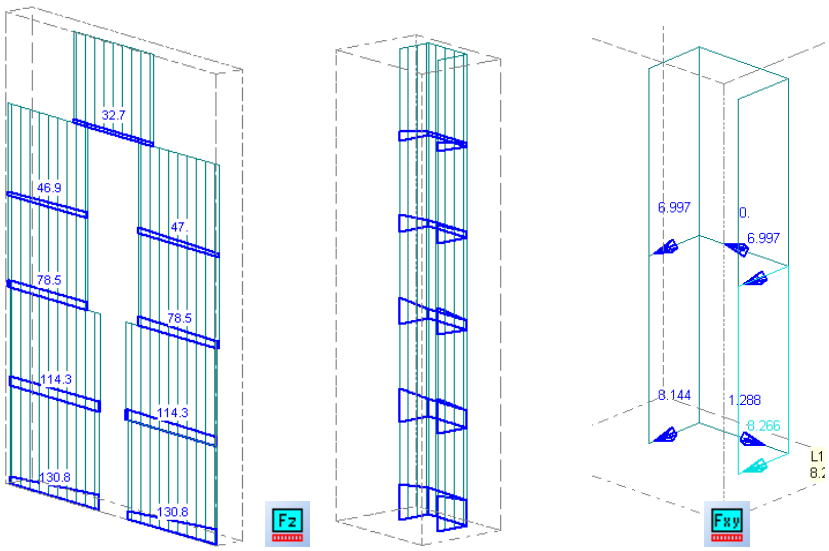
This Results Layer shows the bending deflections of the slab in the Building view. Normally, however, one would view the deflections in the individual slabs.

Vertical forces at the bottom of wall segments;
Labelling: Nu (= total vertical force)
Pointer on cursor: Nu, Mw (= moment about the main principal axis of the segment)

Horizontal force at the bottom of the vertical element in vectorial form.

Wall segments: Shear force at bottom along the wall segments
Columns: Shear force at bottom in the direction X of the slab coordinate system

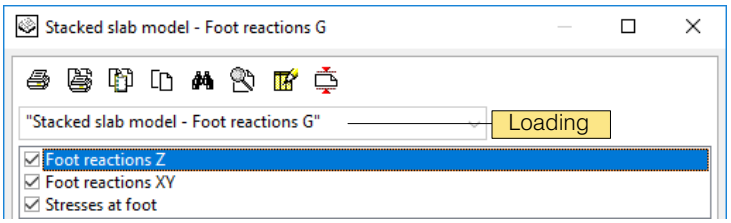
Wall segments: Shear force at bottom normal to the wall segment
Columns: Shear force at bottom in the direction Y of the slab coordinate system



☞ *Don't forget the possibilities offered by the clipping box (figure on the right) or of the partial systems for restricting the visibility*

Tabular

The output for the selected load case (in the figure = G) creates and consists of the three listed tables:



The table headings are as follows:

Foot reactions Z

Storey	Element	Wall segments				Pillars			Columns	Foundation	Sum Fz
		Foot [%]	fz_{min} [kN/m]	fz_{max} [kN/m]	Fz [kN]	fz_{min} [kN/m]	fz_{max} [kN/m]	Fz [kN]	Fz [kN]	Fz [kN]	[kN]
L5	C1								168.707		
L5	C4								30.518		
L5	C5								30.518		

Fz corresponds to the quantities described above in the graphics output. Besides the total forces for each wall segment in the tabular output the forces per unit length fz_{min} and fz_{max} are output (different values in the case of trapezoidal distribution). The values for the wall pillars are also output, which in the case of grouped wall pillars differ from the segment values.

Foot reactions XY

Storey	Element	Wall segments		Pillars		Columns		Foundation		Sum	
		Fx [kN]	Fy [kN]	Fx [kN]	Fy [kN]	Fx [kN]	Fy [kN]	Fx [kN]	Fy [kN]	Fx [kN]	Fy [kN]
L5	C1					0	0				
L5	C4					0	0				
L5	C5					0	0				

Here the components of the horizontal forces at the bottom of the elements are output in the global coordinate system of the building.

Stresses at foot

Storey	Element	Stresses					
		Nu [kN]	Vv [kN]	Vw [kN]	Mu [kNm]	Mv [kNm]	Mw [kNm]
L5	C1	168.71	0	0	0	0	0
L5	C4	30.52	0	0	0	0	0
L5	C5	30.52	0	0	0	0	0

The directions u,v,w are defined here as follows:

u: vertical

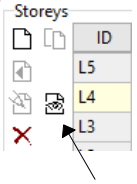
v: **wall segment**: in the direction of the wall segment
wall pillar (grouped): direction X of the slab coordinate system
columns: direction X of the slab coordinate system

w: **wall segment**: normal to wall segment direction
wall pillar (grouped): direction Y of the slab coordinate system
columns: direction Y of the slab coordinate system

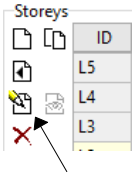
Results in the storey slabs

The results in the individual storey slabs are also available. Open the storey slab with the button shown below and request the results, just as in the base module of CEDRUS.

Dependent on master slab:



Independent of master slab:



A 2.4 Example BIM: Create Buildings from 3D-Data

A 2.4.1 Problem Definition

From a building, a 3D BIM model is available in the form of an IFC or Allplan XML file. Based on this 3D data, a CEDRUS building model is to be created.

A 2.4.2 Prerequisites

- For the following description it is assumed that you have some experience in the use of the basic version of CEDRUS.
- Please read first the chapter A 1.6. It provides a brief overview of the procedure of a building calculation.
- You have a 3D BIM file (IFC or XML format) that describes the geometry of the building.
- You own the program option „Building model, vertical loads“ (G) of CEDRUS.

A 2.4.3 Program start

▶ Start CEDRUS and open a new building calculation via the menu „File“> „New ...“> „Building model..“.

▶ Change to the 'BIM' tab and open the BIMlink window by clicking the 'Start BIMlink' button.

A 2.4.4 Import 3D BIM file

▶ In the 'Import' tab of BIMlink, press the adjacent button and select the file to be imported.

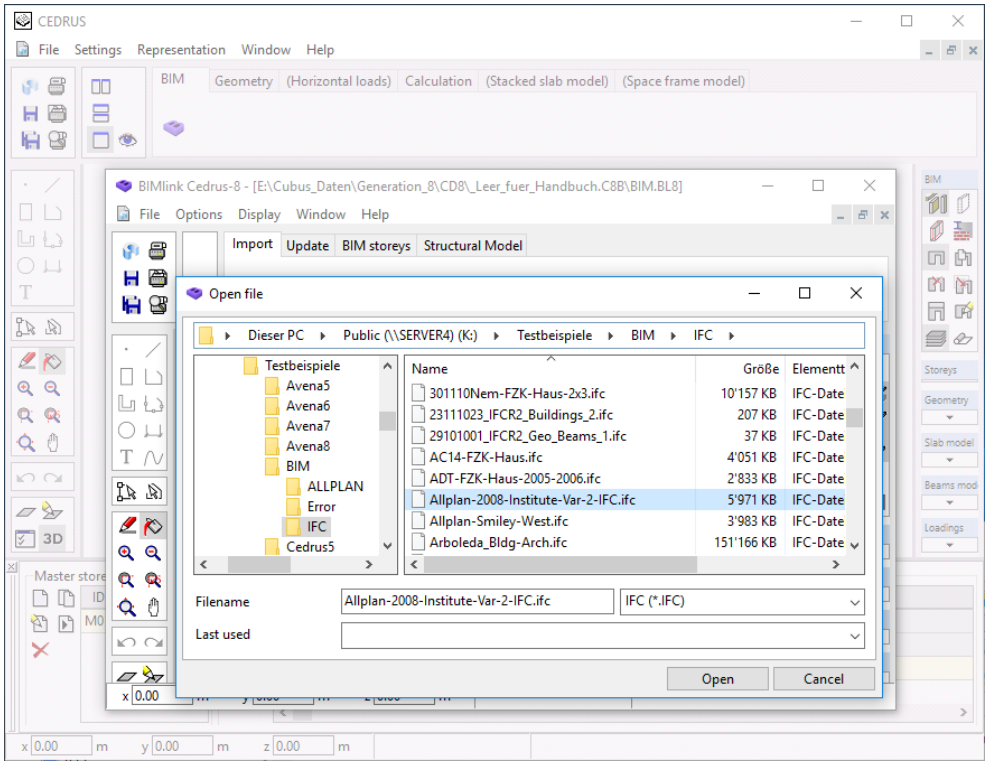


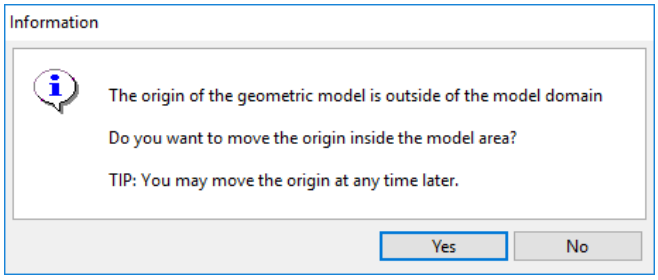
Fig. A–2 *BIMlink window*

☞ *If the BIM file is an IFC file, you can configure the size of the imported data in the main menu „Settings“ > „IFC Interface“ before importing the file.*

The 3D building data is now read from the BIM file. The imported model is stored in the first layer within the layer group 'BIM' (see adjacent image).

Coordinate system

In order to facilitate the work in the building model of CEDRUS, it makes sense to define a coordinate system, which has coordinate values with few digits. Therefore, the following message is displayed when the origin of the coordinate system of the data is outside the area of the model:



▶ Answer [Yes], which automatically converts the coordinates in the BIM file into a new system whose origin is at the edge of the model range. If you answer [No], the system will be adopted from the BIM file.

☞ *The coordinate system selected here will later also be used for the entire CEDRUS building model. By means of the layer menu (ie right mouse button) of the first layer of the group 'BIM', the origin of the coordinate system can still be defined manually after the import.*

BIM elements and element types

The 3D building data from the BIM file are now displayed in the graphic area. The imported data consists of a large number of individual objects (here called BIM elements) whose geometry is described in 3D. As explained in more detail in the D 6.3 section, the BIM elements relevant for the structural calculation are filtered out of the mass of the read-in and assigned to a type of component (eg slab, column, wall).

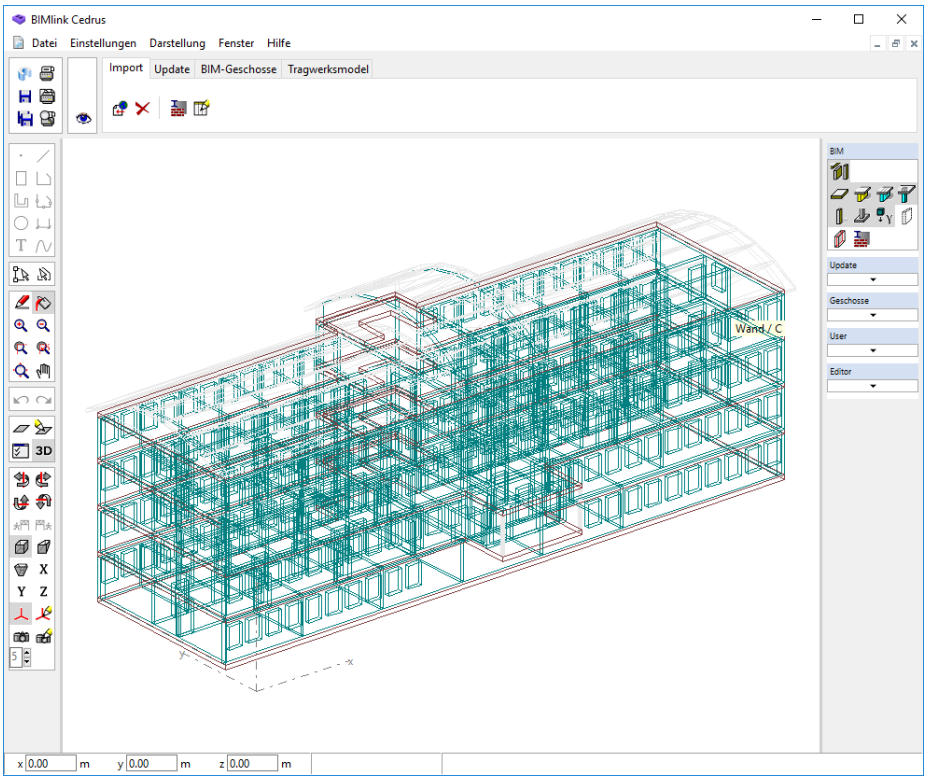
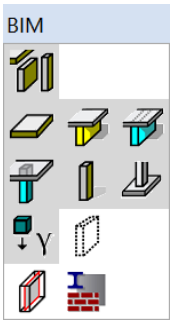


Fig. A–3 *BIM elements*



Jedes BIM-Element wird in der dem Elementtyp entsprechenden Farbe gezeichnet. In der BIM-Layergruppe gibt es einen Sichtbarkeitslayer für jeden unterstützen Elementtyp (s. nebenstehendes Bild). Neben den Elementtypen für Bauteile (z.B. Stütze, Wand etc.) gibt es auch einen Elementtyp 'undefiniert'. Dieser wird all jenen BIM-Elementen zugewiesen, welche für die Tragwerksberechnung nicht notwendig sind (z.B. Treppen).

☞ *Die BIM-Elemente und ihr Elementtyp werden aufgrund der in der BIM-Datei enthaltenen Information und dem sogenannten Elementfilter ermittelt. Die Einstellungen des Element-filters können bei Bedarf über die Schaltfläche* *im Import-Register angepasst werden.*

A 2.4.5 Baustoffe definieren

In einer BIM-Datei sind die Materialdaten als Namen gespeichert. Um eine Tragwerksberechnung zu ermöglichen, muss deshalb jedem Materialnamen ein CEDRUS-Baustoff zugewiesen werden:

▶ Öffnen Sie dazu den Dialog „Zuweisung der Baustoffe“ durch betätigen der nebenstehenden Schaltfläche im Import-Register. Weisen Sie nun jedem Namen einen Baustoff zu und schliessen Sie den Dialog mit [OK].

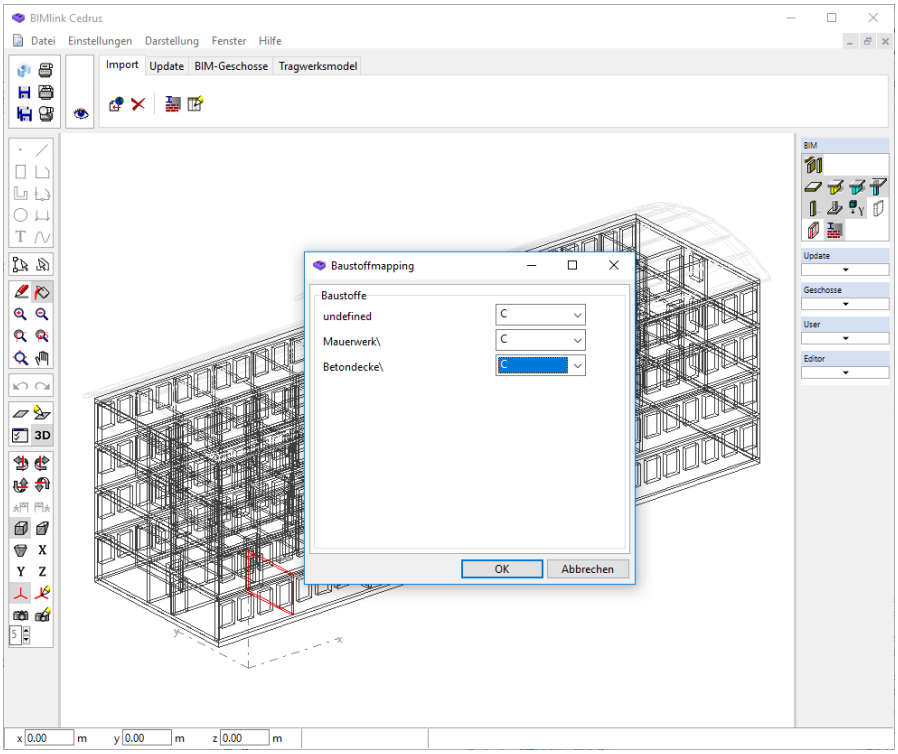


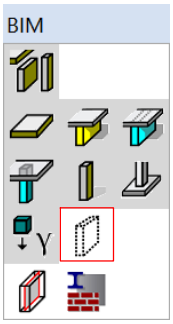
Fig. A–4 *Dialog „Zuweisung von Baustoffen“*

Nach dem Schliessen des Dialogs wird jedem BIM-Element automatisch der entsprechende Baustoff zugewiesen.

A 2.4.6 BIM-Elemente bereinigen

Wie in Abschnitt D 6 detailliert erläutert, versucht BIMlink die für die Tragwerksberechnung erforderlichen BIM-Elemente aus der grossen Datenmenge der BIM-Datei automatisch zu extrahieren. Um Fehler in der BIM-Datei oder im Extrahierungsprozess zu erkennen, sollten die importierten BIM-Elemente deshalb kontrolliert werden.

Zur Kontrolle des Elementtyps können die entsprechenden Sichtbarkeitslayer der Layergruppe ‘BIM’ eingesetzt werden:



- Schalten Sie jeden Sichtbarkeitslayer der Elementtypen jeweils exklusiv ein und kontrollieren Sie die entsprechenden BIM-Elemente visuell. Überprüfen Sie insbesondere den Elementtyp ‘undefiniert’ (s. Layerschalter rechts): Dort sollten nur Elemente sichtbar sein, welche für die Tragwerksberechnung nicht relevant sind (z.B. Treppen, Dachkonstruktion, Fassade).

  Haben Sie einen Fehler entdeckt (z.B. ein angehängtes Fassadenteil wurde als Wand erkannt), so können Sie den Elementtyp des entsprechenden BIM-Elementes wie folgt ändern:

- Öffnen Sie den Eigenschaftsdialog durch Doppelklick auf das fehlerhafte BIM-Element. Ändern Sie z.B. den Elementtyp durch Deaktivieren der Schaltfläche ‘automatisch’ und durch Wahl des richtigen Typs. Drücken Sie danach die Schaltfläche [Anwenden].

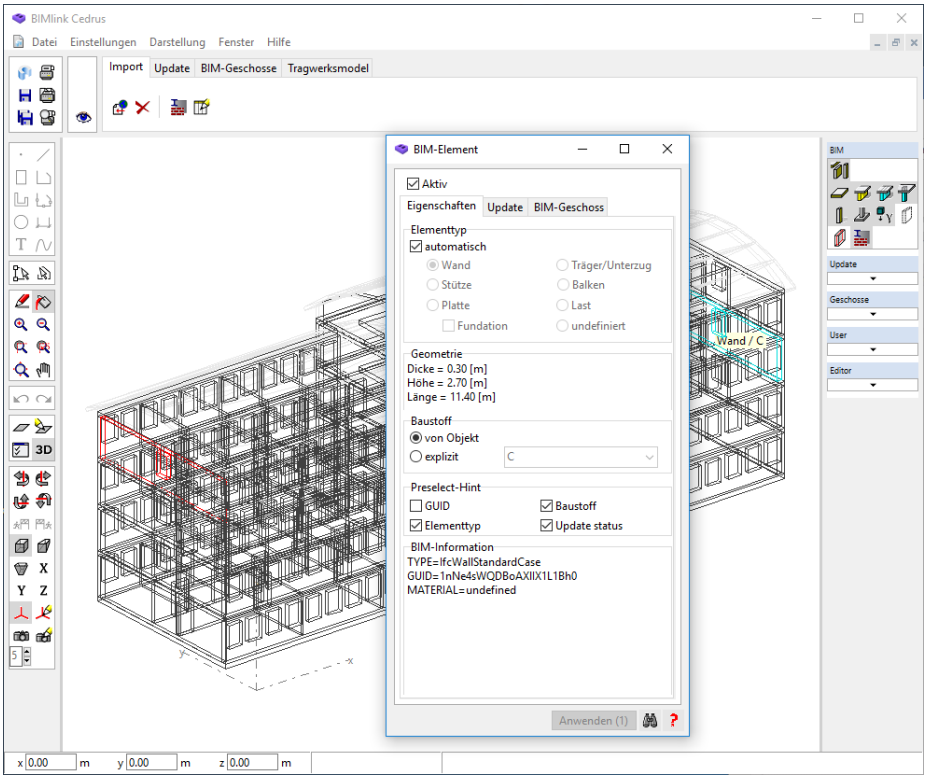


Fig. A–5 *Eigenschaftsdialog „BIM–Elemente“*

Die Kontrolle der Baustoffe erfolgt analog mit Unterstützung des Sichtbarkeitslayers ‘Baustoffe’ aus der Layergruppe ‘BIM’. Bei Bedarf kann der Baustoff für jedes BIM-Element im Eigenschaftsdialog verändert werden.

A 2.4.7 BIM-Geschosse generieren

Aus den bereinigten BIM-Elementen sollen nun die Geschosse für das Gebäudemodell von CEDRUS extrahiert werden.

▶ Wechseln Sie ins Register ‘BIM-Geschosse’ und drücken Sie auf die nebenstehende Schaltfläche.

BIMlink versucht nun die Geschosse automatisch anhand der BIM-Elemente zu finden und erzeugt für jedes entdeckte Geschoss ein Grafikobjekt im Geschoss-Layer der Layergruppe ‘Geschosse’ (s. Bild links).

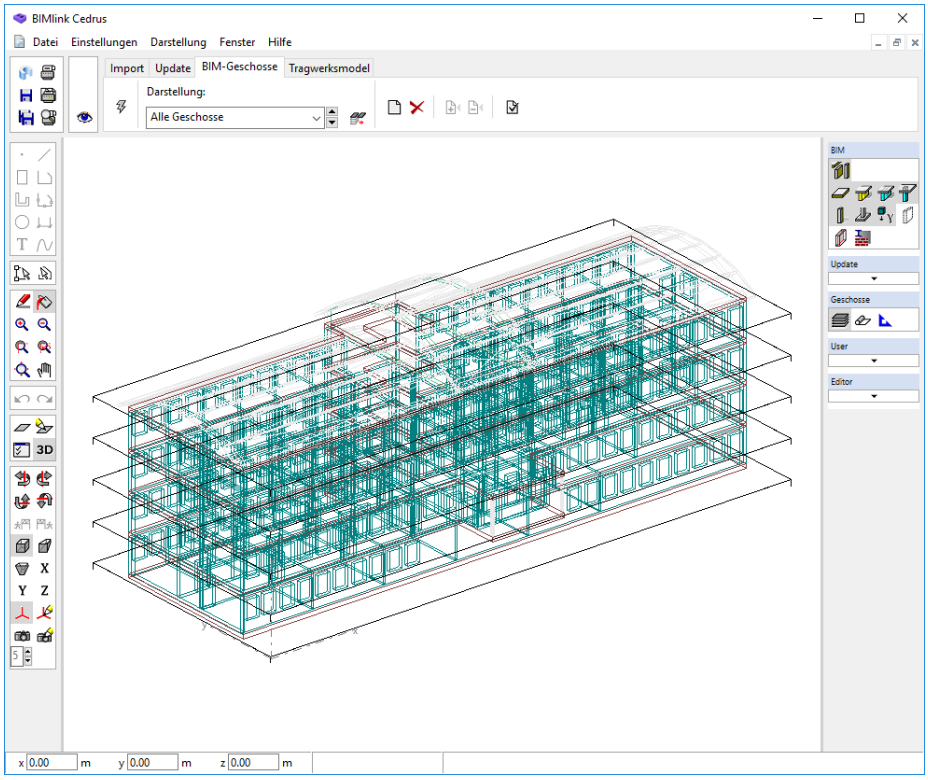


Fig. A–6 *Generierte BIM–Geschosse*

Jedes BIM-Geschoss weist eine Z-Kote auf und erhält eine automatisch erzeugte Bezeichnung ‘Bx’, wobei ‘x’ der Rangordnung aller Geschosse nach Z entspricht (höchste Zahl = zuoberst). Zu einem Geschoss gehört eine Platte mit Brüstungen/Unterzügen und alle untenstehenden Wände und Stützen. In der Grafikfläche wird ein Geschoss als ebenes Polygon dargestellt, welches den Bereich der zugehörigen Elemente umschliesst (s. Bild).

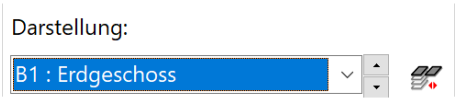
A 2.4.8 BIM-Geschosse bereinigen

Da die Generierung der Geschosse aus den ungeordneten BIM-Elementen nicht in jedem Fall fehlerfrei ist (s. Abschnitt D 6), empfiehlt es sich, die Geschosse zu kontrollieren und allenfalls zu bereinigen.

Verwenden Sie zur Kontrolle der Geschosse die Darstellungseinstellungen im Register ‘BIM-Geschosse‘ und die Layerschalter in den Gruppen ‘BIM’ und ‘Geschosse’. Gehen Sie wie folgt vor.

Wiederholen Sie den folgenden Schritt für alle Geschosse:

▶ Wählen Sie im Register ‘BIM-Geschosse‘ bei ‘Darstellung’ (s. unten) ein Geschoss, womit die zugehörigen BIM-Elemente dargestellt werden. Kontrollieren Sie diese mit Hilfe der nebenstehenden Schaltfläche ‘Darstellung von inaktiven Geschossen / Elementen’.



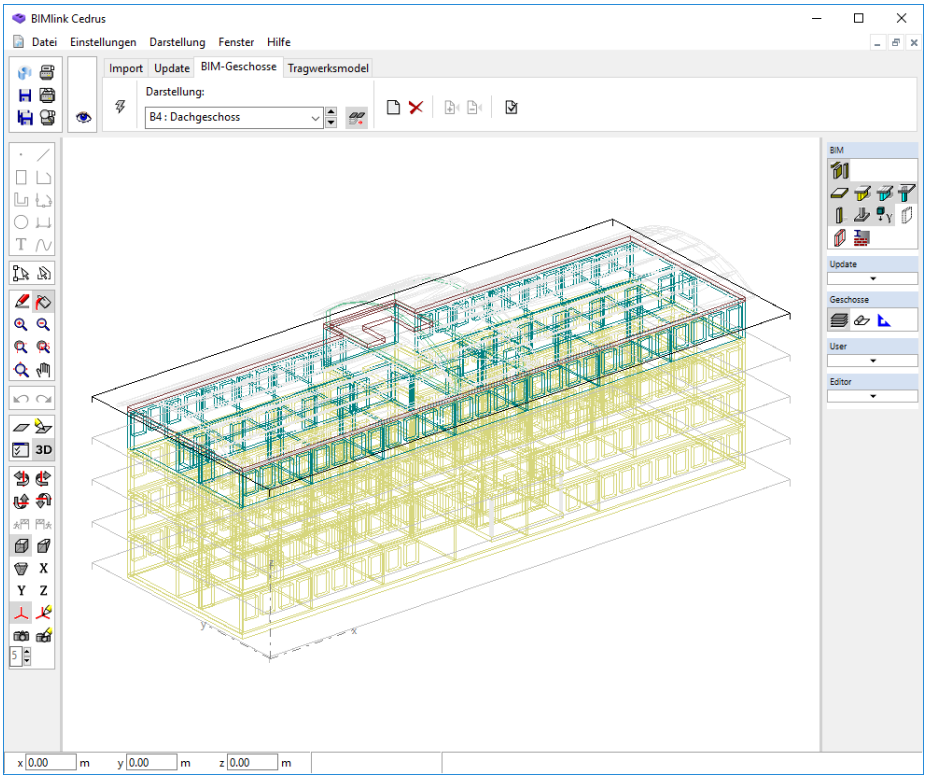


Fig. A–7 *Ansicht bei gewähltem Geschoss*

Das Geschoss und die zugehörigen BIM-Elemente können verändert werden. Folgende Schaltflächen stehen im Register 'BIM-Geschosse' dafür zur Verfügung:

Das aktuelle Geschoss wird gelöscht

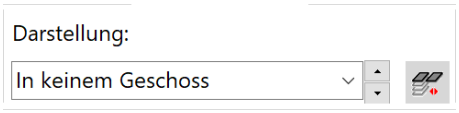
Die selektierten BIM-Elemente werden zum aktuellen Geschoss addiert

Die selektierten BIM-Elemente werden aus dem aktuellen Geschoss entfernt

Ein neues Geschoss manuell einführen: Dabei öffnet sich der Dialog 'BIM-Geschosse' wo Sie die entsprechende Z-Kote angeben können.

Damit kein BIM-Element vergessen geht, lohnt sich am Ende ein Blick auf alle unberücksichtigten BIM-Elemente:

▶ Wählen Sie im Register 'BIM-Geschosse' bei 'Darstellung' die Einstellung 'in keinem Geschoss', womit die Elemente ohne Zuweisung dargestellt werden. Vergewissern Sie sich, dass nur unwichtige Elemente dort dargestellt werden.



A 2.4.9 BIM- und CEDRUS-Geschosse verlinken

Im letzten Schritt innerhalb des BIMlink-Fensters muss noch spezifiziert werden, welche BIM-Geschosse ins Tragwerksmodell von CEDRUS einfliessen und welche Daten pro Geschoss für CEDRUS generiert werden sollen.

Verlinken der Geschosse

In der Regel soll für jedes BIM-Geschoss ein entsprechendes Geschoss im Tragwerksmodell von CEDRUS erzeugt werden. Manchmal trifft dies jedoch nicht zu, weshalb diese sogenannte Verlinkung zwischen BIM- und CEDRUS-Geschoss hier individuell gesteuert werden kann.

▶ Wechseln Sie ins Register 'Tragwerksmodell', wo unterhalb der Grafikfläche eine Tabelle aller BIM-Geschosse dargestellt wird.

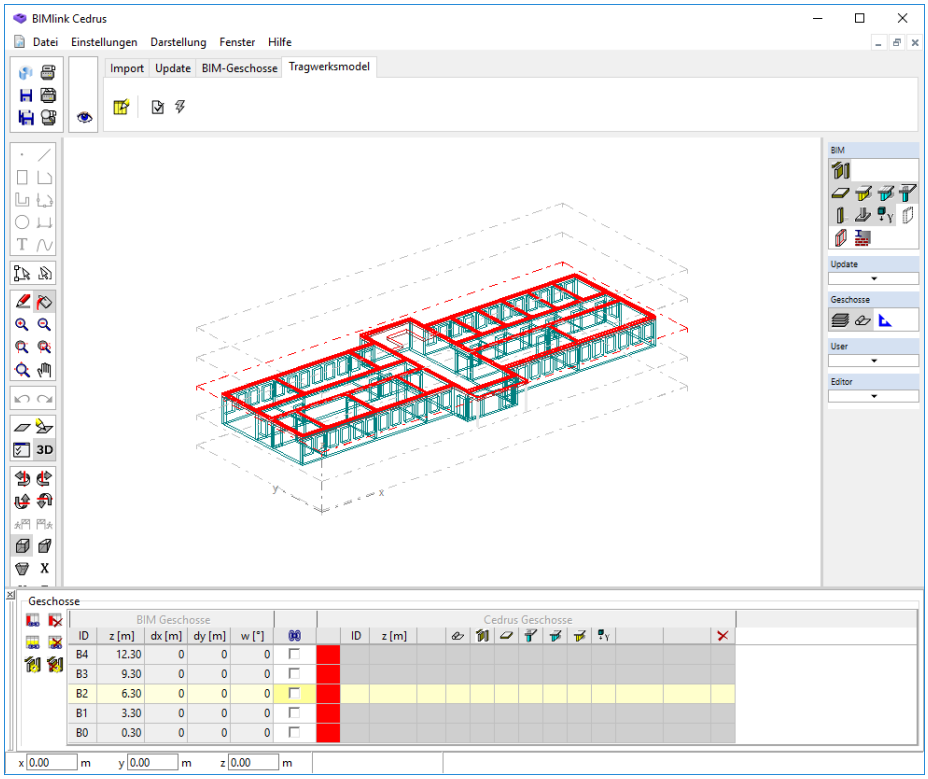
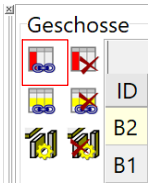


Fig. A–8 *Register 'Tragwerksmodell' mit Geschosstabelle*

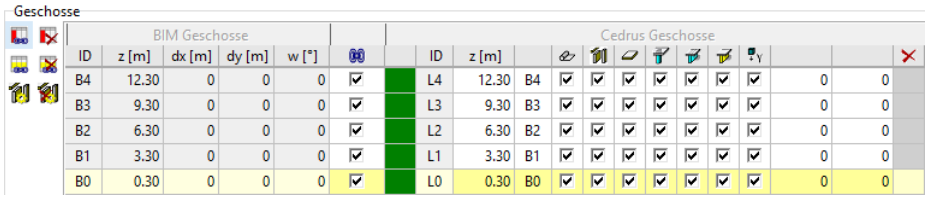
Alle noch nicht verlinkten BIM-Geschosse werden in der Tabelle mit rot markiert.



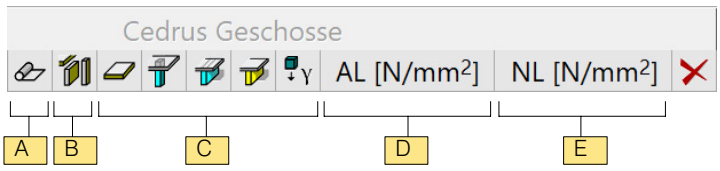
▶ Drücken Sie nun die nebenstehend markierte Schaltfläche in der Geschosstabelle, womit alle BIM-Geschosse automatisch verlinkt werden.

Definition von Generierungsumfang und Lasten

Für jedes verlinkte BIM-Geschoss wird ein Geschoss im CEDRUS-Gebäudemodell generiert. Der Umfang der generierten Daten kann in der Geschosstabelle für jedes Geschoss individuell gesteuert werden.



Durch Einschalten der entsprechenden Option in der Tabelle können folgende Daten in das CEDRUS-Geschoss übertragen werden:



A: 2D-Hintergrundplan der BIM-Daten (horizontaler Schnitt durch 3D-BIM-Modell)

B: Alle Elemente als CEDRUS-Struktureingaben

C: Element individuell als CEDRUS-Struktureingaben

D: Auflast für ganze Geschossplatte

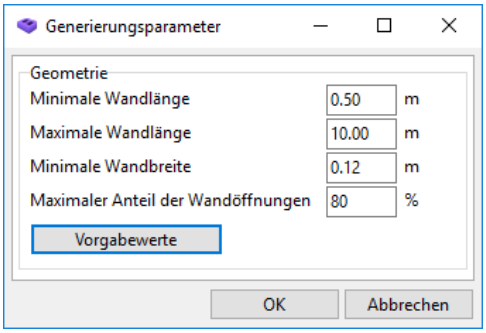
E: Nutzlast für ganze Geschossplatte

☞ *Ob es sich lohnt, die Strukturelemente (d.h. Platte, Wände etc.) für das CEDRUS-Geschoss zu generieren, hängt sehr stark von der Qualität der 3D-BIM-Daten ab. Wie in Abschnitt D 6 detailliert erläutert kann es effizienter sein, nur den Hintergrundplan plus evtl. den Plattenumriss und die Stützen zu übernehmen und die Wände später mit den Mitteln von CEDRUS zu konstruieren.*

A 2.4.10 CEDRUS-Gebäude erzeugen

Damit sind nun die Geschosse für das CEDRUS-Berechnungsmodell definiert. Wie in Abschnitt D 6 erläutert, ist das geometrische BIM-Modell jedoch nicht direkt als Berechnungsmodell verwendbar, wenn man sinnvolle Resultate erhalten möchte. BIMlink führt deshalb im letzten Schritt geometrische Vereinfachungen und Modellanpassungen durch, um ein brauchbares CEDRUS-Berechnungsmodell zu erzeugen.

☞ *Die Modellvereinfachung kann im Dialog 'Generierungsparameter' konfiguriert werden. Um den Dialog zu öffnen, drücken Sie die Schaltfläche* *im Register 'Tragwerksmodell'.*



Die Erzeugung des CEDRUS-Gebäudemodelles aus den BIM-Daten ist nun vorbereitet und kann wie folgt ausgeführt werden:

▶ Drücken Sie die nebenstehende Schaltfläche, womit ein Bestätigungsdialog erscheint. Drücken Sie [OK]. Das BIMlink-Fenster wird geschlossen und Sie befinden sich wieder im

BIM-Register des CEDRUS-Fensters, in dem jetzt die importierten BIM-Daten dargestellt sind und die Geschosstabelle alle neuen Geschosse zeigt.

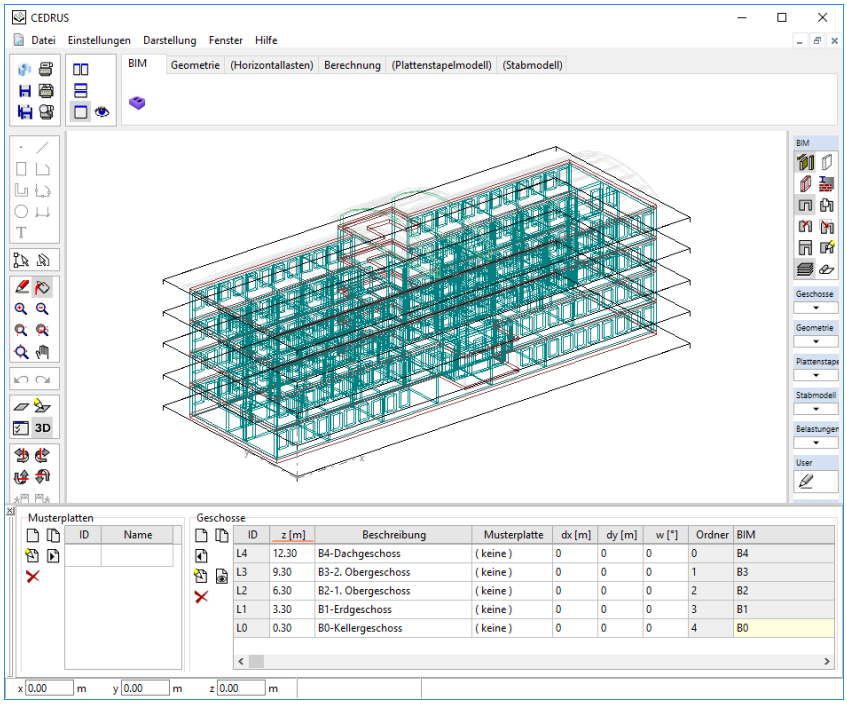


Fig. A–9 *Register 'BIM' von CEDRUS*

☞ *Manchmal kann es vorkommen, dass bei der Überprüfung einer Geschossplatte Fehlermeldungen auftreten, da die generierten geometrischen Daten die in CEDRUS eingestellten Toleranzregeln verletzen können. In diesem Fall müssen Sie die betroffenen Geometrien mit den bekannten Mitteln der Grafikeingabe anpassen.*

A 2.4.11 CEDRUS-Platten eingeben

Die Generierung der CEDRUS-Daten aus BIM ist nun abgeschlossen. Was noch fehlt, ist die Vervollständigung der Eingabedaten der einzelnen Platten.

▶ Öffnen Sie dazu jede Platte und kontrollieren Sie die erzeugten Daten. Vervollständigen Sie wo nötig die Eingaben.

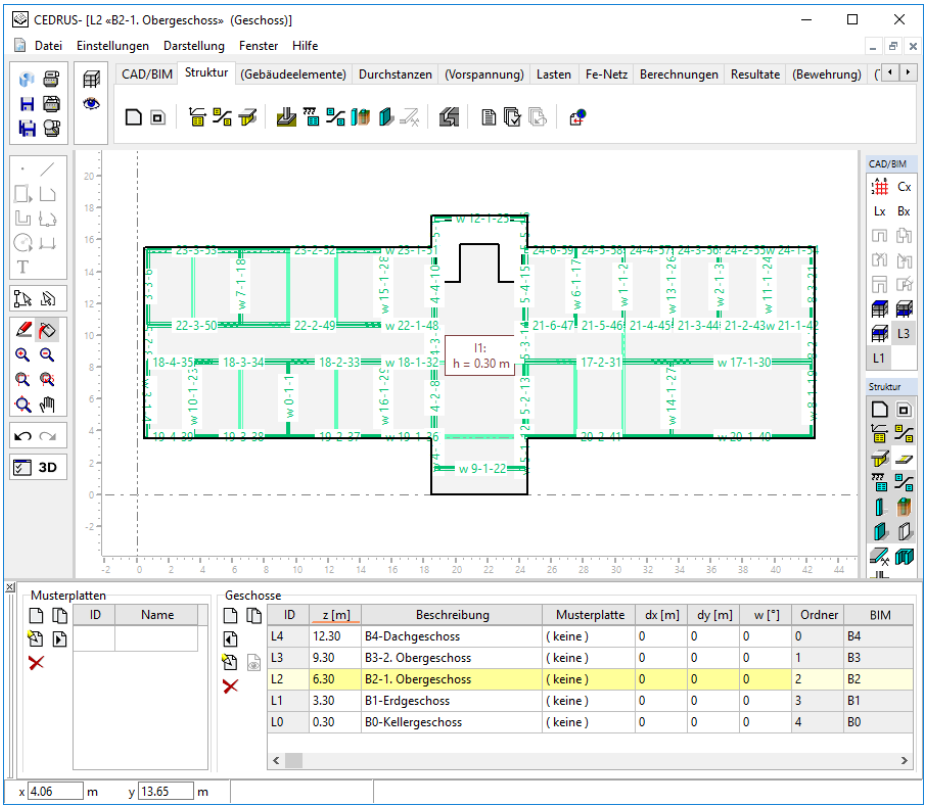
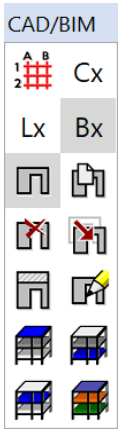


Fig. A–10 *Geschoss mit generierten Eingabedaten*



Die von BIMlink erzeugten Hintergrundpläne können hier sehr hilfreich sein.

Die Pläne des aktuellen, oben- und untenstehenden BIM-Geschosse können über die folgenden Layerschalter in der Layergruppe ‘CAD/BIM’ dargestellt werden:

- Bx alle BIM-Hintergrundpläne ein/aus
- BIM-Hintergrundplan des Geschosses darüber ein/aus
- BIM-Hintergrundplan des aktuellen Geschosses ein/aus
- BIM-Hintergrundplan des Geschosses darunter ein/aus
- Farbe der Hintergrundpläne aufgrund der Geschosslage:

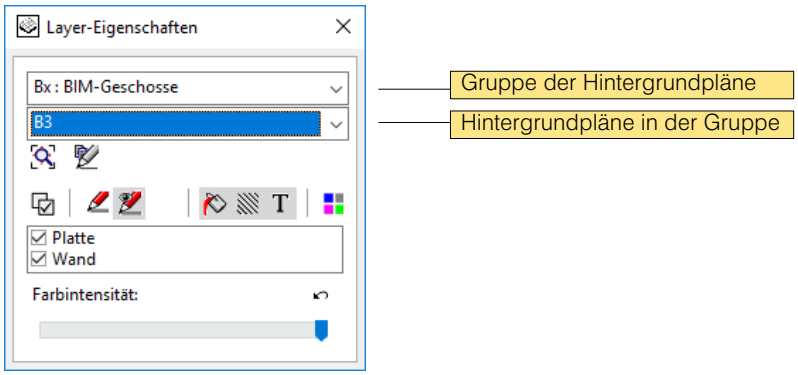
  Blau: Geschosse darüber

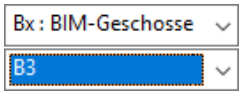
  Orange: aktuelles Geschoss

  Grün: Geschosse darunter

Dialog 'Layer-Eigenschaften': Darstellung der Pläne

Zudem können Sie bei BIM-Hintergrundplänen die Darstellung der Elementtypen einzeln festlegen, indem Sie den Dialog 'Layer-Eigenschaften' öffnen (d.h. rechte Maustaste auf nebenstehender Schaltfläche 'Bx').





Durch Wahl der entsprechenden Gruppe der Hintergrundpläne (Cx, Bx oder Lx) und des entsprechenden Namens können die Darstellungseigenschaften eines Planes angezeigt und verändert werden. In der Liste der Planlayer können einzelne Layer ein- und ausgeschaltet werden. Zudem stehen folgende Schaltflächen zur Verfügung:

Durch Drücken der Taste <F3> oder mit der Schaltfläche im Layereigenschaften-Fenster lässt sich die aktive Zone für das Zentrieren der CAD-Zeichnung mit einem Rechteck festlegen.

Die Zeichnungselemente eines Hintergrundplans lassen sich in einen Layer der User-Gruppe kopieren. Der neu erstellte Layer trägt denselben Namen. Dort sind die Zeichnungselemente des Hintergrundplans als einzelne Grafikeditor-Objekte editierbar. Kopiert werden nur die in der Layer-Liste (s. Bild C-2) ausgewählten Layer des Hintergrundplans. Bei vielen Zeichnungselementen kann das Kopieren einige Zeit in Anspruch nehmen. Ferner ist zu beachten, dass eine grosse Anzahl von kopierten Objekten sowohl das Arbeiten im Grafikeditor als auch Laden und Speichern der Projektdaten verlangsamt.

A 2.5 Einführungsbeispiel BIM: Modellnachführung aus 3D-Daten

A 2.5.1 Aufgabestellung

Ein CEDRUS-Gebäudemodell wurde basierend auf einer 3D-BIM-Datei erstellt (z. B. wie in A 2.4). Nun ist eine neuere Version der BIM-Daten mit potentiell relevanten Änderungen vorhanden. Das CEDRUS-Gebäudemodell soll deshalb nachgeführt werden.

A 2.5.2 Voraussetzungen

- Für die folgende Beschreibung wird angenommen, dass Sie das Programm CEDRUS gestartet und ein CEDRUS-Gebäudemodell geöffnet haben, welches Sie mittels BIMlink aus einer ersten Version einer 3D-BIM-Datei erstellt hatten.
- Von der oben erwähnten BIM-Datei ist eine neue Version vorhanden, welche Sie jetzt zur Nachführung des CEDRUS-Gebäudemodelles verwenden wollen.

A 2.5.3 Neue Version der 3D-BIM-Datei importieren

Als erster Schritt wird die neue 3D-BIM-Datei importiert.

▶ Wechseln Sie dazu in der CEDRUS-Gebäudeberechnung in das BIM-Register und öffnen Sie das BIMlink-Fenster. Dort gehen Sie ins Register ‘Update‘ und lesen die neue BIM-Datei ein.

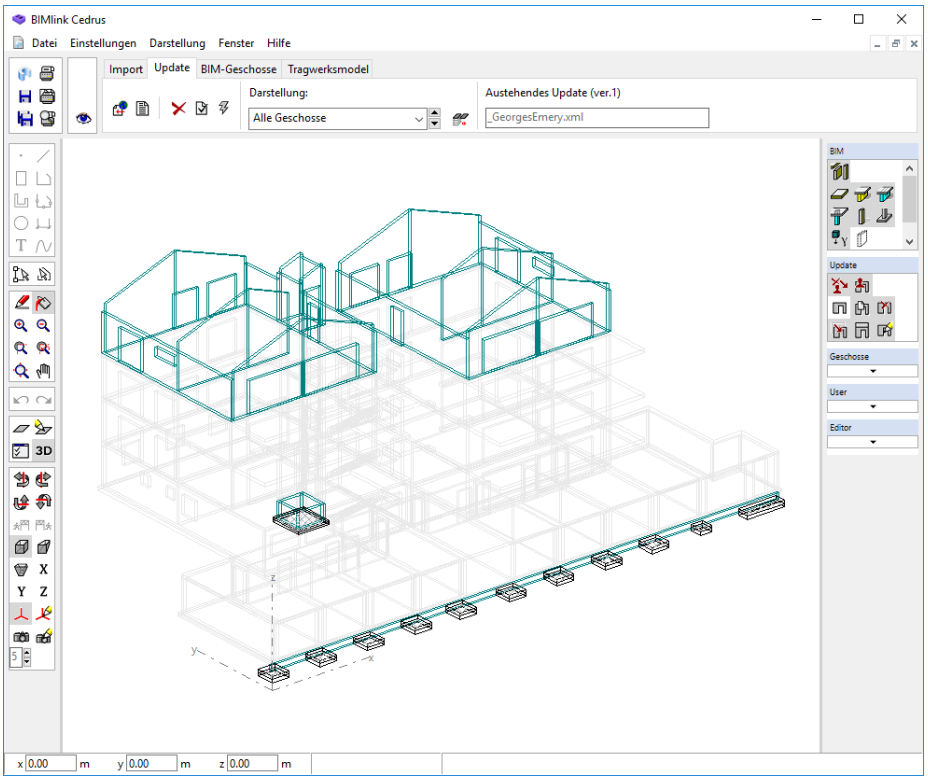
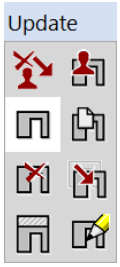


Fig. A–11 *BIMlink nach dem Einlesen der neuen BIM–Daten*

Nach erfolgreichem Datei-Import werden die neu eingelesenen BIM-Elemente mit den bereits Bestehenden verglichen und die gefundenen Änderungen umgehend grafisch dargestellt (s. Fig A-11).

☞ *Hinweis: Behalten Sie die dieselben IFC/XML-Einstellungen wie beim ersten BIM-Daten-Import (s. A 2.4.4), sonst werden zu viele Änderungen angezeigt.*



Die grafische Struktur-Darstellung zeigt einen guten Überblick über alle Neuerungen im Modell. Unveränderte BIM-Elemente werden dabei hellgrau dargestellt. Neue, betreffend Geometrie oder Baustoff veränderte Elemente werden farbig dargestellt. Die Änderungen werden in folgende 5 Gruppen unterteilt, deren Sichtbarkeit durch je einen Schalter in der Layergruppe ‘Update‘ gesteuert werden kann:

Neu

Gelöscht

Verschoben (bei unveränderten Dimensionen)

Veränderte Geometrie (z.B. Dimensionen, Öffnungen)

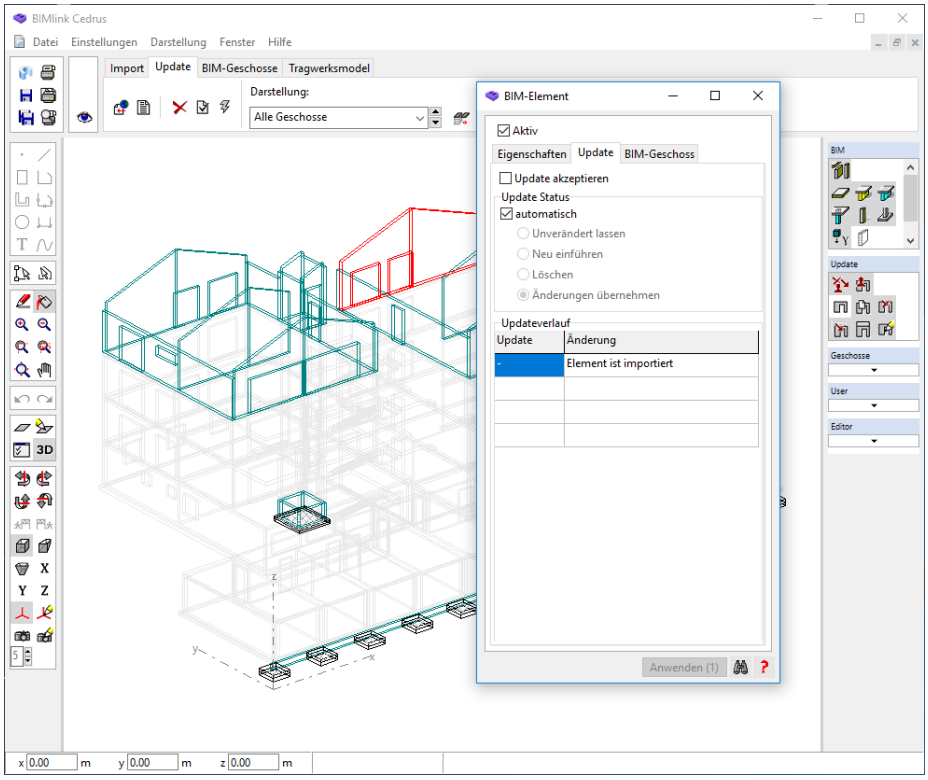
Übrige Eigenschaften (z.B. Baustoffname)

A 2.5.4 Konfiguration der Nachführung

Im nächsten Schritt können die Modelländerungen einzeln eingesehen und die Nachführung konfiguriert werden (Wie in Abschnitt D 6 erläutert macht es keinen Sinn, alle Änderungen unkontrolliert zu übernehmen). Es empfiehlt sich, die Modelländerungen stockwerksweise zu studieren:

▶ Wählen Sie in der Combobox 'Darstellung' ein Geschoss aus und studieren Sie die Modelländerungen unter Verwendung der Layerschalter in der Gruppe 'Update'.

▶ Wollen Sie eine Änderung (z.B. Baustoffname) an einem BIM-Element nicht in Ihr Modell übernehmen, so öffnen Sie dessen Eigenschaftsdialog (Doppelklick auf BIM-Element), deaktivieren im Register 'Update' den „Update-Status" > „automatisch" und wählen „Unverändert lassen" (s. Bild unten).



Sind alle Änderungen in allen Geschossen kontrolliert und gemäss Ihren Vorstellungen konfiguriert, so können Sie die Nachführung starten.

▶ Drücken Sie die Blitz-Schaltfläche, womit das BIM-Modell von BIMlink nachgeführt wird.

⚠ ACHTUNG: Diese Aktion kann nicht rückgängig gemacht werden!

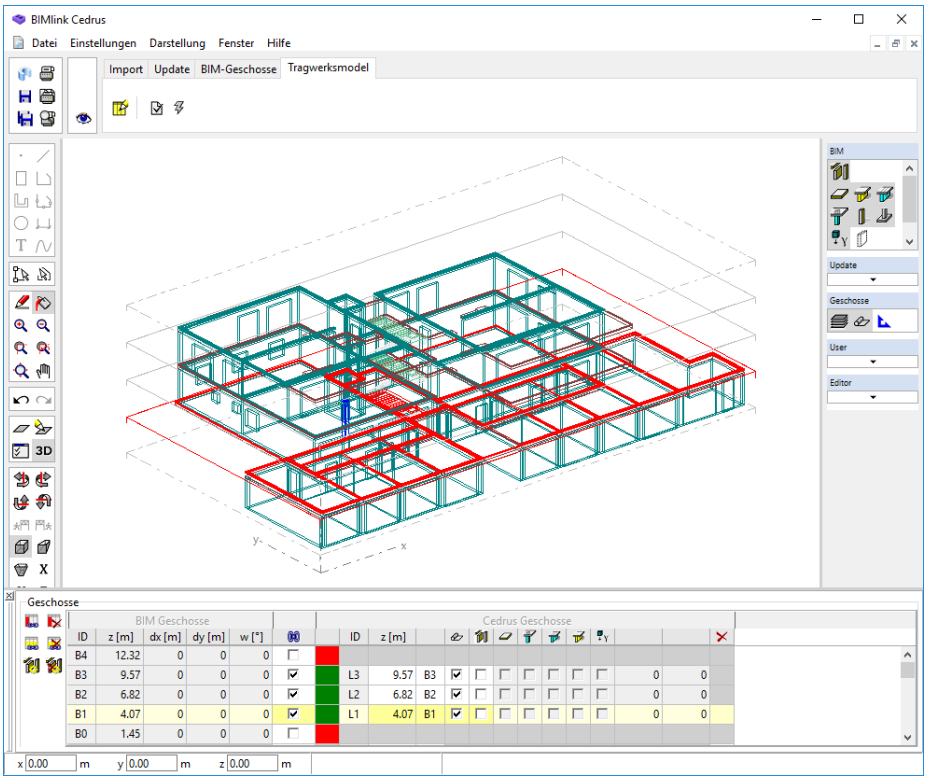
Die 3D-BIM-Elemente von BIMlink aber auch die automatisch generierten Hintergrundpläne (s. D 6) sind nun nachgeführt.

☞ *Die Nachführungsaktionen der 5 Update-Gruppen (neu, gelöscht, etc.) gehen dabei nicht verloren, sondern werden als separate Layer im Hintergrundplan dargestellt.*

A 2.5.5 CEDRUS-Gebäude nachführen

In diesem Schritt soll das Tragwerksmodell von CEDRUS aufgrund des nachgeführten BIM-Modelles aktualisiert werden.

▶ Wechseln Sie in das Register 'Tragwerksmodell'



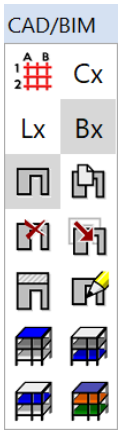
In der Tabelle sind nun alle Geschosse mit grünem Marker versehen, welche von der Nachführung betroffen sind (s. Bild). Das Generieren des CEDRUS-Modells aufgrund der BIM-Daten kann nun wiederum in der Tabelle gesteuert werden (s. A 2.4.9).

☞ *Bei den betroffenen (grünen) Geschossen ist die Generierung der CEDRUS-Strukturelemente standardmässig nicht aktiviert, sondern nur die Aktualisierung des Hintergrundplanes.*

▶ Um das CEDRUS-Gebäudemodell nachzuführen, drücken Sie nun die Blitz-Schaltfläche, womit das BIMlink-Fenster geschlossen wird.

A 2.5.6 CEDRUS-Platten nachführen

Die nachgeführten Daten in 3D und 2D stehen nun in CEDRUS zur Verfügung, um das Berechnungsmodell anzupassen. Im letzten Schritt geht es nun darum, diese CEDRUS-Geschosse nachzuführen.



▶ Öffnen Sie jedes der betroffenen Geschoss und aktualisieren Sie die CEDRUS-Eingabedaten. Dazu stehen Ihnen die BIM- Hintergrundpläne in der Layergruppe 'CAD/BIM' zur Verfügung (s. Bild links).

Mit den folgenden Layerschaltern aus dieser Gruppe können Sie die Änderungen des BIM-Modells anzeigen lassen:

- Bx alle BIM-Hintergrundpläne ein / aus
- neue BIM-Elemente ein / aus
- gelöschte BIM-Elemente ein / aus
- verschobene BIM-Elemente ein / aus
- BIM-Elemente mit veränderter Geometrie ein / aus
- BIM-Elemente mit veränderten Attributen (z.B. Baustoff) ein / aus

A 2.6 Einführungsbeispiel BIM: Gebäude aus 2D-Plänen erstellen

A 2.6.1 Aufgabenstellung

Zur Erstellung eines CEDRUS-Gebäudemodelles steht eine Reihe von 2D-Geschossplänen als DXF- bzw. DWG-Dateien zur Verfügung. Das CEDRUS-Gebäudemodell soll mit Hilfe dieser Pläne eingegeben werden.

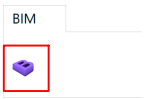
A 2.6.2 Voraussetzungen

- Für die folgende Beschreibung wird angenommen, dass Sie Erfahrung mit der Verwendung des Basismoduls von CEDRUS haben.
- Lesen Sie bitte zuerst das Kap. LEERER MERKER. Es liefert einen kurzen Überblick über den Ablauf einer Gebäudeberechnung.
- Sie haben einen oder mehrere digitale 2D-Pläne im DXF- bzw. DWG-Format, in denen die Grundrisse der Geschosse gezeichnet sind.
- Sie verfügen über Lizenzen für die Programmoption „Gebäude“ (G) und die Service-Option „CADview“ (CV) von CEDRUS (s. B 7.2).

A 2.6.3 Ablauf

In einem ersten Schritt wird für jeden 2D-Plan ein BIM-Geschoss mit der entsprechenden Z-Kote definiert, die DXF/DWG-Datei eingelesen und der Plan in der Geschossebene platziert. In einem zweiten Schritt können dann für jedes Geschoss die Eingabedaten des CEDRUS-Berechnungsmodelles konstruiert werden. Dabei stehen die eingelesenen 2D-Pläne als Hintergrundpläne zur Verfügung.

A 2.6.4 Programmstart



▶ Starten Sie CEDRUS und eröffnen Sie eine neue Gebäudeberechnung über das Menu „Datei“ > „Neu...“ > „Gebäudeberechnung“.

▶ Wechseln Sie ins Register ‘BIM’ und öffnen Sie das BIMlink-Fenster mittels der nebenstehenden Schaltfläche.

A 2.6.5 BIM-Geschoss einführen

Um ein neues Geschoss für einen vorhandenen DWG/DXF-Plan einzugeben gehen Sie wie folgt vor:

▶ Wechseln Sie ins Register ‘BIM-Geschosse’ und öffnen Sie dort den Dialog ‘BIM-Geschoss‘ mit der nebenstehenden Schaltfläche.

▶ Im Dialog 'BIM-Geschoss' geben Sie nun die Höhenkote des Geschosses bei „Geometrie" > „Oberkanten Z" ein und drücken die Schaltfläche [Einführen]. Das neue Geschoss wird nun als ebenes Rechteck in der Grafikfläche dargestellt.

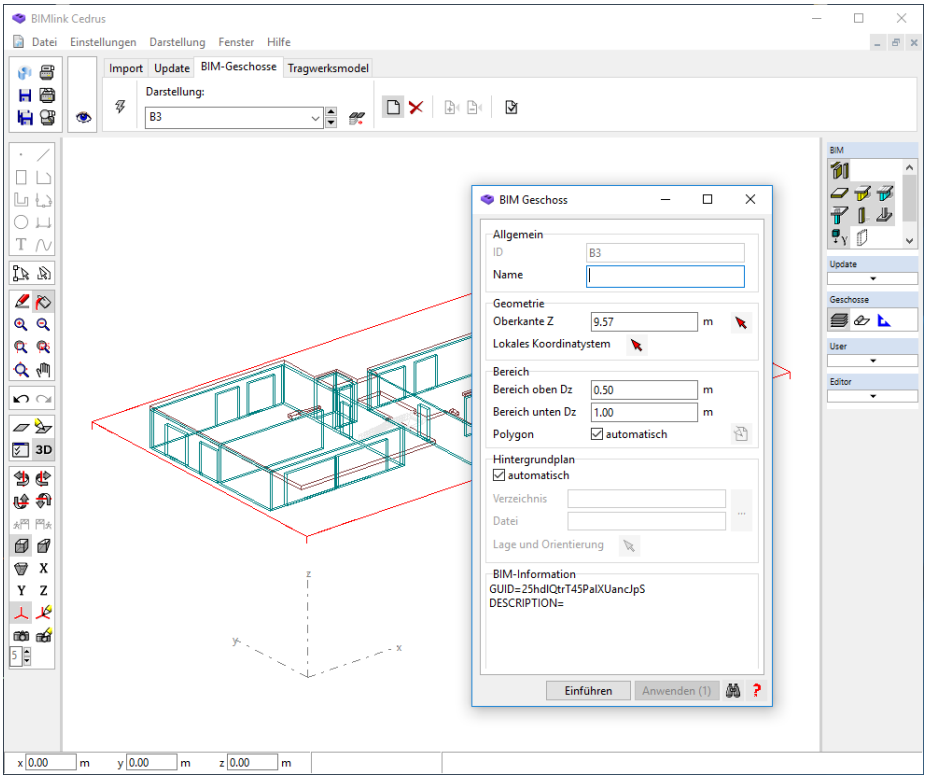


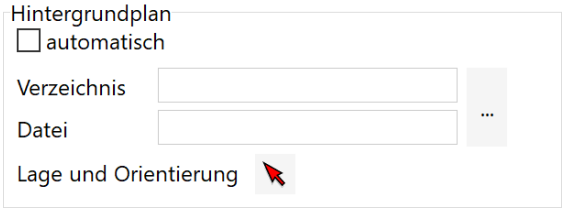
***Fig. A–12** Neues Geschoss und Dialog „BIM–Geschoss"*

A 2.6.6 2D-Plan importieren

Um den DWG/DXF-Plan einzulesen gehen Sie wie folgt vor:

▶ Selektieren Sie das soeben eingeführte BIM-Geschoss in der Grafikfläche.

▶ Im Dialog 'BIM-Geschoss' deaktivieren Sie unter „Hintergrundplan" die Schaltfläche „automatisch" und drücken Sie die […]-Schaltfläche , womit sich der CADView-Dialog öffnet.



▶ Laden Sie im CADView-Dialog die gewünschte DXF/DWG-Datei. Der 2D-Plan wird nun dargestellt. Um die Datenmenge zu reduzieren, markieren Sie in der Liste der Layer nur diejenigen Layer als sichtbar, welche Sie für die Eingabe des CEDRUS-Geschosses benötigen.

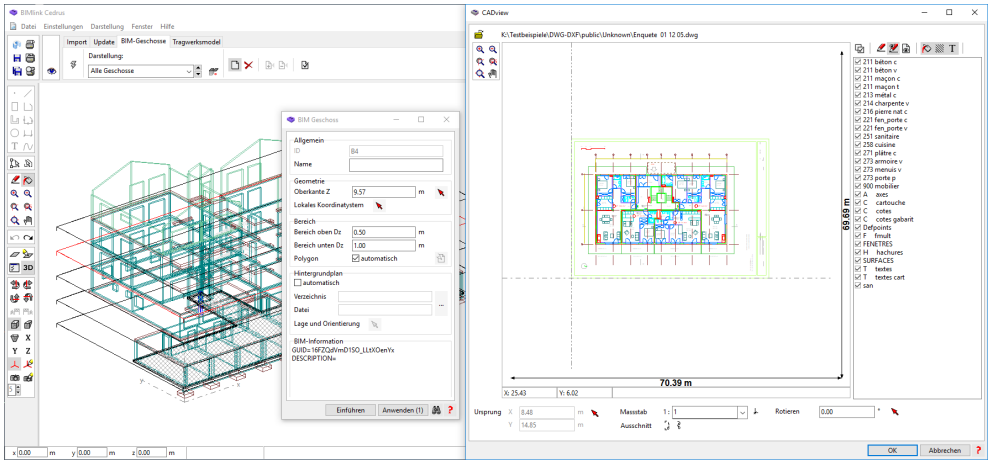
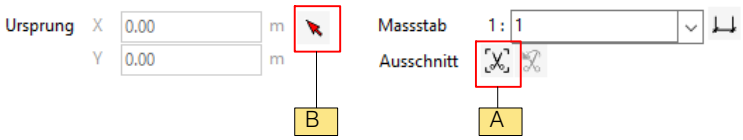


Fig. A–13 *Dialog „CADview“*

▶ Überprüfen Sie den angezeigten Massstab und passen Sie diesen bei Bedarf an.



▶ Wählen Sie mit der Schaltfläche [A] (s. oben) den benötigten Ausschnitt aus.

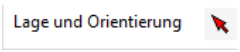
▶ Setzten Sie bei [B] den Ursprung der Zeichnung an einen bekannten Referenzpunkt. Dies hilft später bei der Platzierung des Planes innerhalb des Geschosses.

▶ Um den 2D-Plan zu importieren schliessen Sie nun den CADView-Dialog mit [OK].

A 2.6.7 Lage und Orientierung anpassen

Zur definitiven Platzierung des 2D-Planes und zur Angabe des lokalen Koordinatensystems des Geschosses sind noch folgende Eingaben notwendig:

▶ Selektieren Sie ein BIM-Geschoss. Drücken Sie im Dialog ‘BIM-Geschoss‘ die untenstehende Schaltfläche unter ‘Hintergrundplan‘, um in die 2D-Ansicht der Geschossebene zu wechseln.



Das nun angezeigte Koordinatensystem entspricht den X-Y-Koordinaten des Gebäudes.

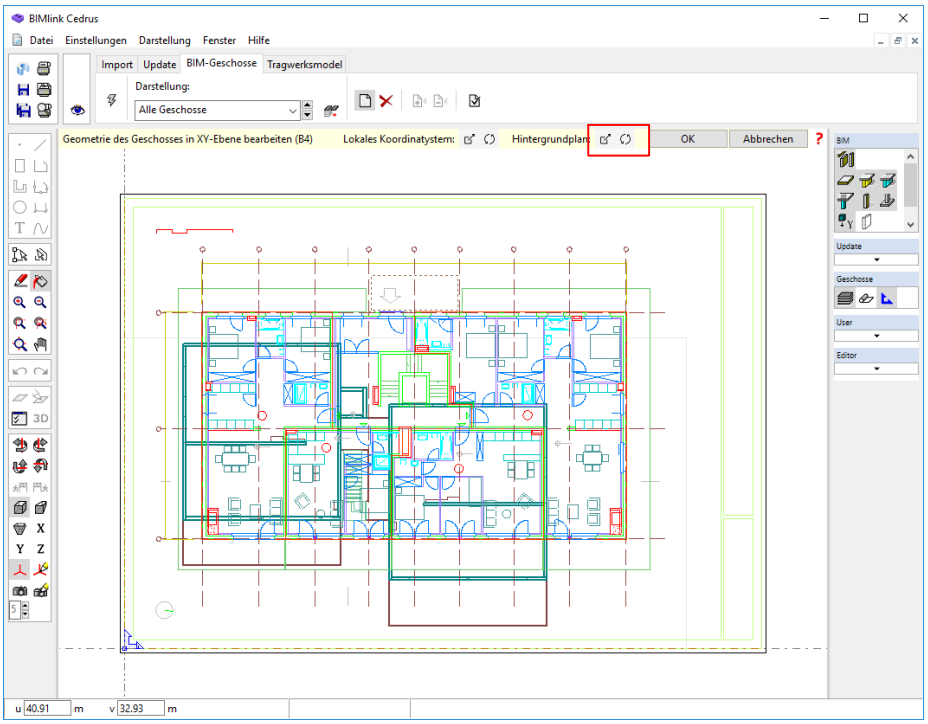


***Fig. A–14** 2D–Ansicht des BIM–Geschosses mit DXF/DWG–Plan*

Um den 2D-Plan innerhalb dieses Systems zu platzieren, stehen zwei Schaltflächen am oberen Rand der Grafikfläche bei 'Hintergrundplan:' zur Verfügung:

- Mit kann der Plan verschoben werden.
- Mit kann der Plan um den Ursprung herum rotiert werden.

▶ Schliessen Sie nun die 2D-Ansicht durch drücken auf [OK], womit Sie sich wieder im Dialog 'BIM-Geschoss' befinden.

▶ Zurück im Geschossdialog drücken Sie nun [Anwenden], um die Änderungen zu übernehmen.

Wiederholen Sie die beiden letzten Schritte „2D-Plan importieren" und „Lage und Orientierung anpassen" für alle benötigten 2D-Pläne.

A 2.6.8 CEDRUS-Gebäude erzeugen

Alle 2D-Pläne sind nun am richtigen Ort in 3D platziert. Um aus diesen BIM-Geschossen das CEDRUS-Tragwerksmodell zu erzeugen gehen Sie wie folgt vor:

▶ Wechseln Sie ins Register 'Tragwerksmodell', wo eine Tabelle aller BIM-Geschosse dargestellt wird.

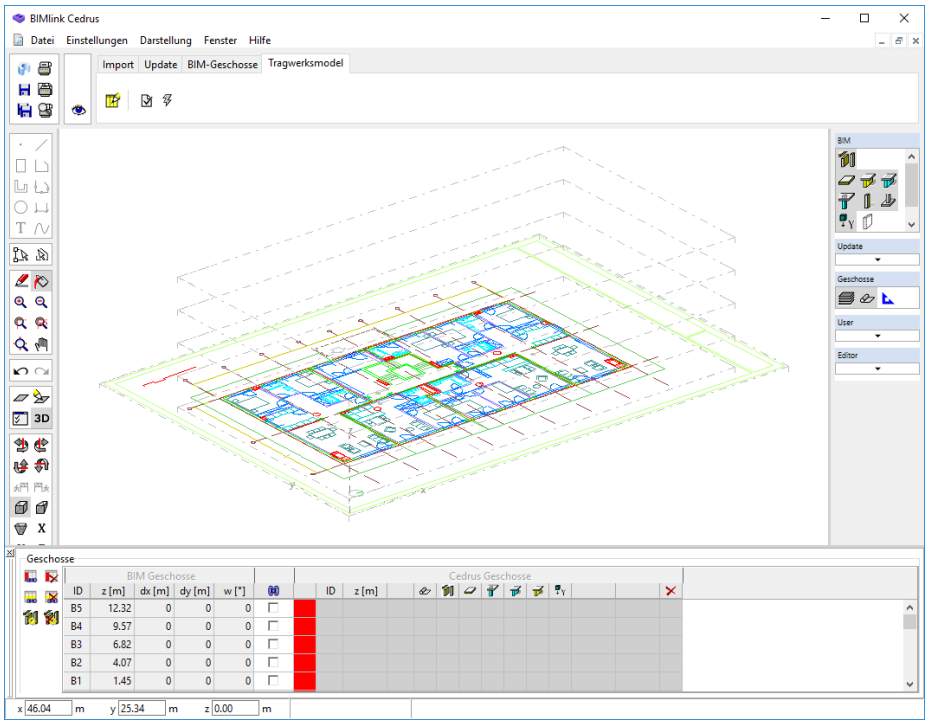
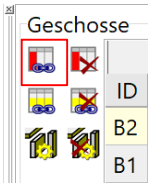
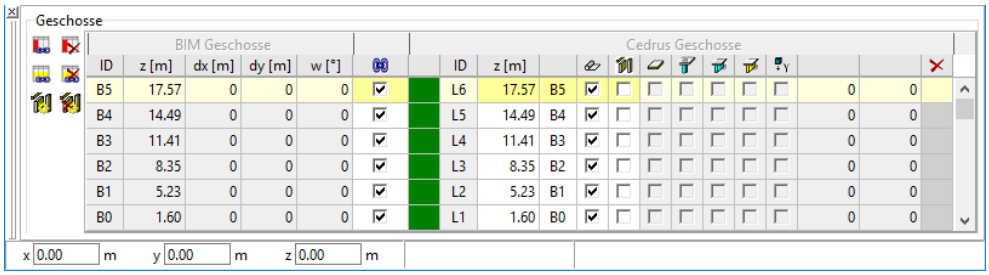


Fig. A–15 *Register 'Tragwerksmodell' mit Geschosstabelle*

Um für ein BIM-Geschoss ein entsprechendes CEDRUS-Geschoss im Gebäudemodell zu erzeugen, müssen die beiden noch verlinkt werden. Alle noch nicht verlinkten BIM-Geschosse werden in der Tabelle mit rot markiert.



▶ Drücken Sie nun die nebenstehende Schaltfläche in der Geschosstabelle, womit alle BIM-Geschosse automatisch verlinkt werden.



Die Erzeugung des CEDRUS-Gebäudemodelles aus den BIM-Daten ist nun vorbereitet und kann wie folgt ausgeführt werden:

▶ Drücken Sie die nebenstehende Schaltfläche, womit ein Bestätigungsdialog erscheint. Drücken Sie [OK]. Das BIMlink-Fenster wird geschlossen und Sie befinden sich wieder im BIM-Register des CEDRUS-Fensters, wo die importierten 2D-Pläne dargestellt sind und die Geschosstabelle alle neuen Geschosse zeigt.

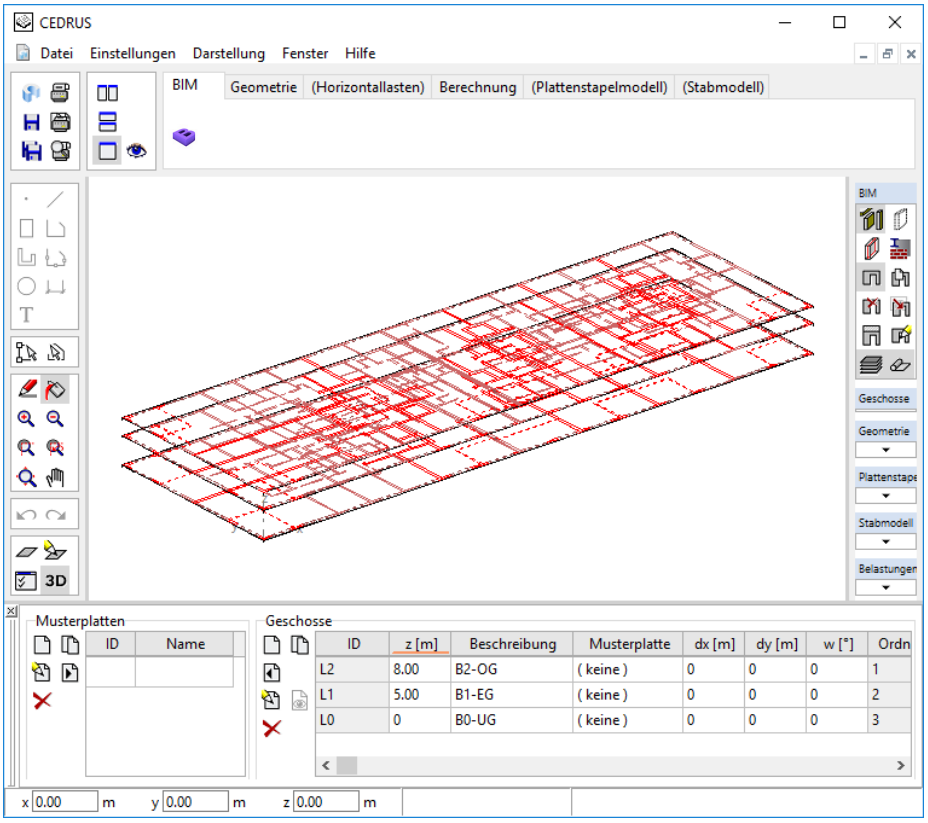


Fig. A–16 *BIM–Geschosse und Geschosstabelle*

Die Sichtbarkeit der BIM-Geschosse und der Hintergrundpläne kann über die nebenstehenden Schaltflächen in der Layergruppe 'BIM' gesteuert werden.

A 2.6.9 CEDRUS-Platten eingeben

Die Geschossstruktur des Gebäudes ist nun festgelegt und es geht darum, die Struktureingaben (Platten, Stützen, Wände) der einzelnen Geschosse zu erstellen. Dies geschieht geschossweise:

▶ Wechseln Sie ins Register 'Geometrie' des Gebäudemoduls.

▶ Öffnen Sie das Geschoss (z.B. Doppelklick auf Geschoss-ID in der Geschosstabelle).

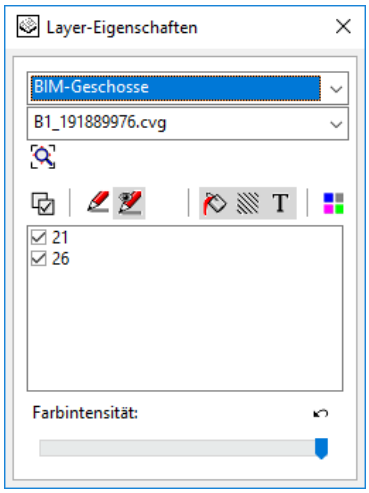
Hintergrundpläne einschalten

Zur Eingabe der Strukturdaten können Sie nun die Hintergrundpläne des aktuellen, oben- und untenstehenden BIM-Geschosses einblenden. Dies geschieht über die Layerschalter in der Layergruppe 'CAD/BIM':

- Bx alle BIM-Hintergrundpläne ein/aus
- BIM-Hintergrundplan des Geschosses darüber ein/aus
- BIM-Hintergrundplan des aktuellen Geschosses ein/aus
- BIM-Hintergrundplan des Geschosses darunter ein/aus
- Farbe der Hintergrundpläne aufgrund der Geschosslage:

  Blau: Geschosse darüber
  Orange: aktuelles Geschoss
  Grün: Geschosse darunter

Bx ☞ *Zudem können Sie für jeden Hintergrundplan die Sichtbarkeit der Layer einzeln festlegen. Dies erfolgt über den Dialog 'Layer-Eigenschaften' des Layers 'Bx' (d.h. via rechte Maustaste). Dort wählen Sie das entsprechende BIM-Geschoss und können dann die Layersichtbarkeit steuern.*



Strukturdaten konstruieren

Die für eine CEDRUS-Plattenberechnung notwendigen Eingabedaten (Umriss, Materialzonen, Wände, Stützen) können nun mit den Mitteln des Grafikeditors konstruiert werden. Durch Abgreifen der Koordinaten aus der Darstellung der Hintergrundpläne kann dies nun speditiv erfolgen.

Vol. B Base Module

B 1 Introduction

The Base Module of CEDRUS is a Finite Element program for the linear-elastic, static analysis of plate structures. It is specially designed for reinforced concrete structures and provides the corresponding reinforccement moment limits and also the reinforcement content.

For an overview of the complete CEDRUS program with all its options the reader should refer to Chapter 'A 1 Overview'.

The present part of the manual describes all relevant aspects of the Base Module with respect to the theoretical basis and also with respect to use of the program. Provided it is a matter of specific questions relating to CEDRUS , this document also uses context-related Help.

The Graphics Editor is a central feature of most Cubus programs. It essentially comprises the program-independent ways of inputting and modifying graphics objects, the control of the display and the use of dialog windows. Since it is not only used in CEDRUS, it is described in a separate chapter. A basic knowledge of the use of the Graphics Editor is a prerequisite for the use of CEDRUS. It may be gained, together with other basic know-how, by completing the Introductory Example on slabs (see Chapter A 2.2).

☞ *All first-time users of CEDRUS are strongly recommended to work through the Introductory Example of see Chapter A 2.2 completely. Before this, one should not attempt serious analyses. It is assumed that the user is already familiar with the terms used in the Introductory Example and the given basic paradigm for using the program.*

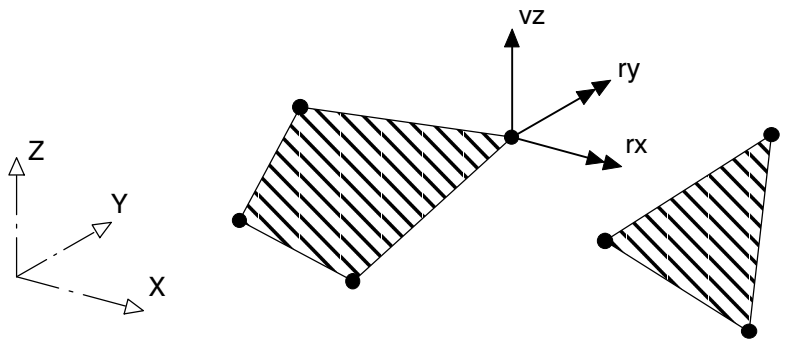
After working through the Introductory Example and gaining some experience with simple calculations it is recommended that you study the manual of the Graphics Editor. The know-how gained thereby will help you to work much more efficiently with the program. If you use other Cubus Software this will pay off, since all programs in this generation uses the same Graphics Editor.

B 2 Element Model and Methods of Solution

The Base Module of CEDRUS ist a Finite-Element program to analyse and dimension reinforced concrete slabs. The linear-elastic FE analysis also permits supports that cannot resist tensile forces. The element models used involve hybrid triangular and quadratic elements of arbitrary shape with the three displacement degrees of freedom vz (bending), rx, ry (rotations about the axes x,y) in the corner nodes.

The state of stress in bending is investigated but not that of the membrane stresses. The elements are based on Kirchhoff's plate theory (lines normal to the neutral axis remain plane in the deformed state). Shear deformations are not considered in this theory, i.e. it is only of limited usefulness for very thick plates and slabs for which the structural behaviour depends greatly of shear deformations. The most important element properties are:

- Quadratic assumptions for the distribution of moments within the elements.
- Cubic assumptions of the displacements along the boundaries of the elements.
- Section force results at the corner nodes and at the centre of the elements.
- Possibility of area (i.e. distributed) support (if necessary without tensile forces).



The elements are still considered to be among the best in this area of application. They were first proposed by Pian and are used in many FE programs. For a thorough study of the element model you are referred to the following literature:

- J.P. Wolf: "Generalized Stress Models for Finite Element Analysis", Institut für Baustatik und Konstruktion, ETH Zürich, Bericht Nr. 52, 1974 Birkhäuserverlag Basel.
- U. Walder: "Beitrag zur Berechnung von Flächentragwerken nach der Methode der Finiten Elemente", Institut für Baustatik und Konstruktion, ETH Zürich, Bericht Nr. 77, 1977 Birkhäuserverlag Basel.

The Finite Element (FE) Method as used in CEDRUS involves essentially the following steps:

1. Subdivision of the slab into finite elements (to provide a suitable FE mesh)
2. Determination of the element matrices according to the hybrid method.
3. Summation of the element stiffness matrices to build the global stiffness matrix.
4. Determination of the load vectors (right hand side of the system of equations).
5. Solution of the resulting system of linear equations for the unknown nodal displacement parameters (iterative in the case of no-tension supports).
6. Determination of the section forces in the elements using the already determined nodal displacement parameters.

You are reminded that the Finite Element Method is an **approximate method**. However, within the limits of numerical accuracy, the method converges to the theoretically exact solution (for Kirchhoff's plate theory) if one takes an increasingly finer FE mesh.

B 3 The Program Window

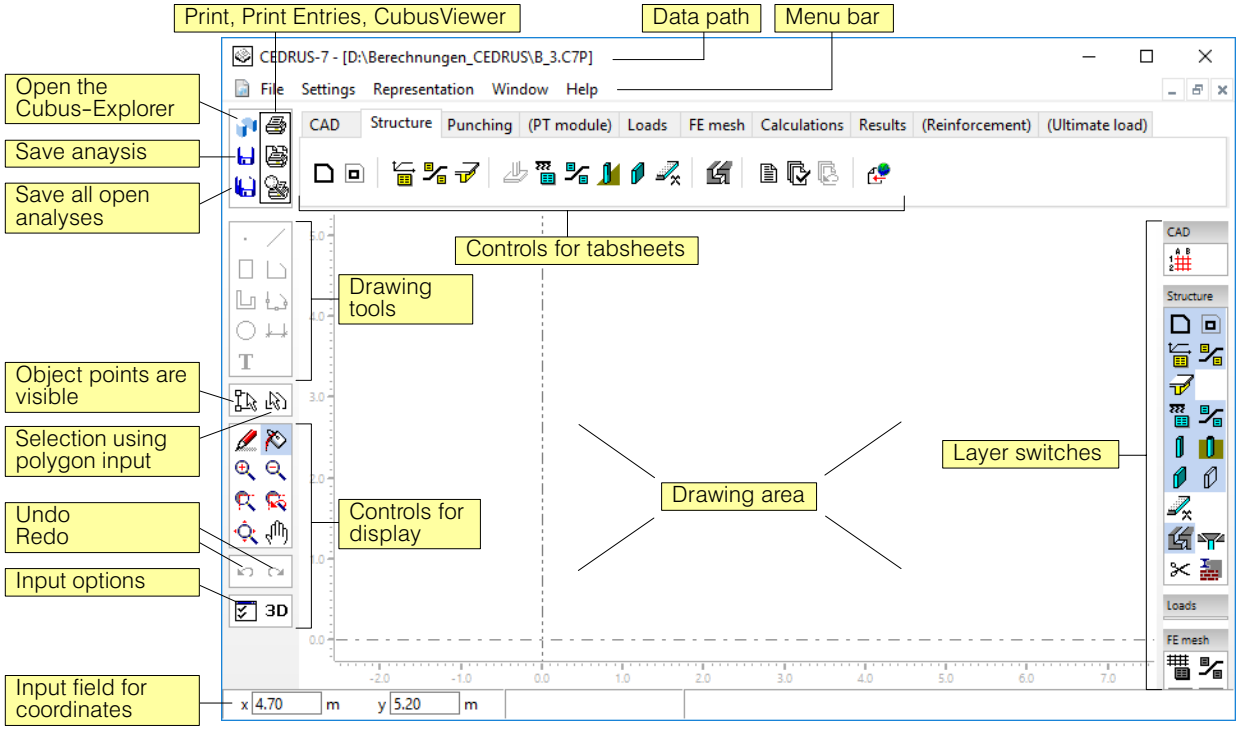
The Program Window of CEDRUS as a slab program looks as follows:

Control tabsheet

The Control tabsheet is in effect the main menu of a Cubus program and here of CEDRUS. It is described in Ch. B 6.

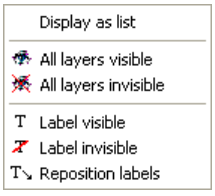
Graphics Editor tools

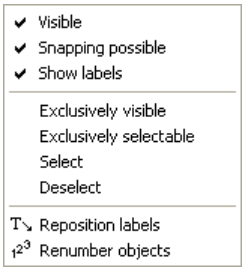
The tools on the left side of the figure influencing the display, etc., are part of the Graphics Editor. They are dealt with in the Introductory Example for a slab (chapt. A 2.2) and described in detail in the chapter on the Graphics Editor. Direct information is also given by the context-related Help, which is activated using the key <F1>.



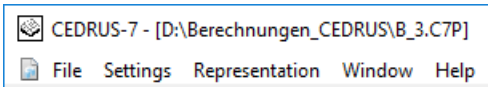
Layer switches

The layer switches arranged in groups on the right side of the figure serve to switch on and off the visibility of the corresponding objects. Both group titles and layer switches have context menus, which are opened using the <RMB> and for which one can get Help using the key <F1>. Here are examples of the context menus of the group Structure and of the layer switches for attribute boxes:





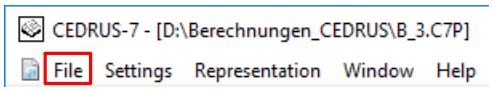
Menu bar

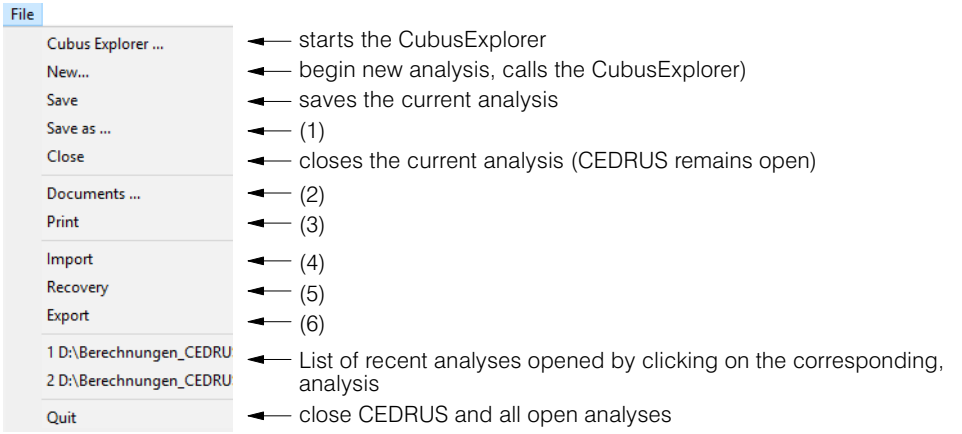


The menu bar, typical of Windows programs, directly below the title bar in the program window, contains some elements specific to CEDRUS and some elements that are found in all Cubus programs. It is described in detail in Ch. B 4.

B 4 Menu Bar

Menu 'File'

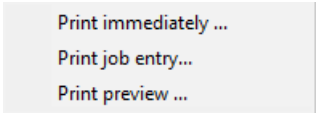




(1) File> Save as ...
The current state of the active analysis is saved under a new name. To select the desired folder and give the new name, the CubusExplorer is called with the corresponding limited functionality (for details see there). The active analysis, without saving, is closed, and work is resumed on the new analysis.

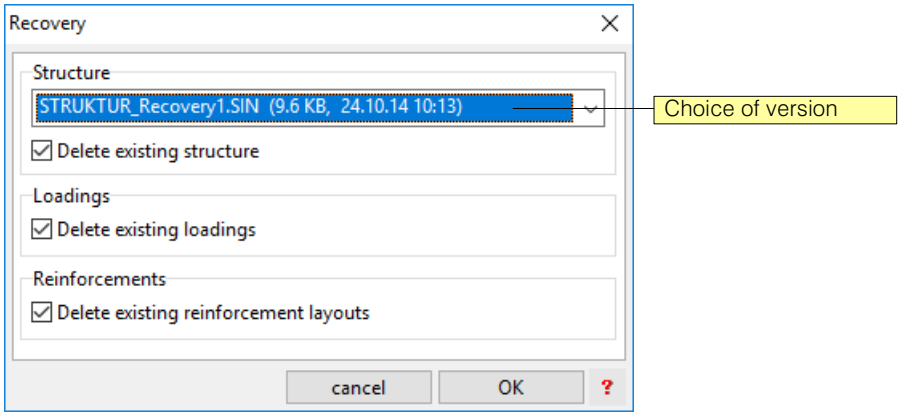
(2) File > Documents ...
In the sub-folder called 'User' of a CEDRUS analysis, as much data can be stored as one wishes, which then belongs to the analysis and is compressed with it, copied and archived. With the menu 'Documents...' an Explorer window is opened, which shows all the files in the user folder. A file can be dragged into this window with the left mouse key, in order to store it in the folder User belonging to the analysis.

(3) File > Print: The following self-explanatory sub-menu appears:



(4) File > Import: The self-explanatory sub-menu appears, with which the data from earlier CEDRUS-analyses can be imported.

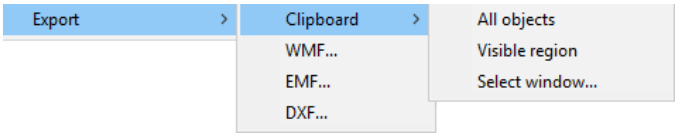
(5) File > Recovery
Each time an analysis is saved a copy of the structure and the load data is saved in text form, whereby several generations of the structure data can be managed. If some data is inadvertently deleted or project data can no longer be read, as a rule this Recovery function can help. The Recovery function has the following dialog:



The first entry in the selection list (STRUKTUR_Recovery1.SIN) corresponds to the last saved state. If this doesn't give you everything you want, you can try with the last-but-one state (Recovery2), etc..

(6) File > Export

Here the contents of the graphics area can be exported in various data formats. The following sub-menu appears:



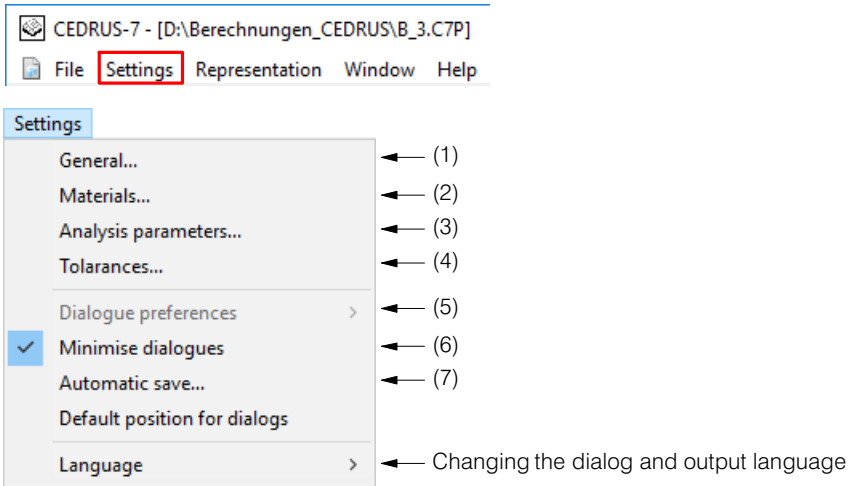
Intermediate storage (clipboard): Creates a copy of the graphics in the Windows clipboard, whereby one can still choose whether all the graphics, or just the visible part or an excerpt to be defined is copied. In this way figures can easily be copied to other applications.

WMF..., EMF...: A file with all the graphics is created in the Windows-metafile or in the Extended-metafile format, for which purpose a dialog to select the folder and the file name appears.

DXF...: For data export in DXF format a specific dialog window appears, whose Help function gives you information about all the details..

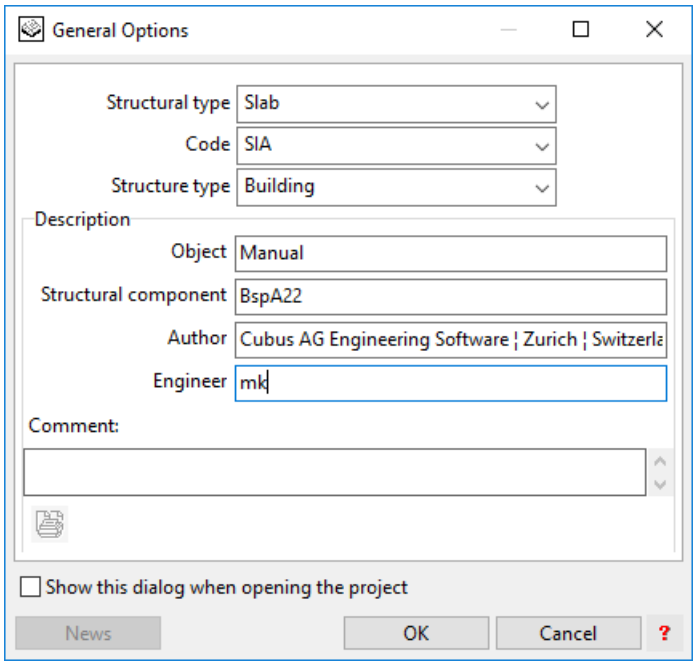
The Export menu can also be shown in the graphics area using the <RMB> function.

Menu 'Settings'



(1) Settings > General ...

The dialog 'General Settings' appears on opening a new analysis. The settings made there can be modified here.



Structure type: the following can be selected 'Slab', 'Shear Wall', 'Slab with Normal Force'. For the last two the program option 'Shear Wall' is needed.

Code: For the current analysis one can change here to one of the other supported codes. After any change in the choice of code the Material List must be changed, since the Material Classes assigned there are code-dependent.

Building type: By correctly setting the building type in the loading input the corresponding category of action can be selected. If one does not need any limit state results this setting is of no consequence.

Description: The text in the four input fields 'Object', 'Part of Structure', 'Author' and 'Design Engineer' appear, when using the standard page layout, in the page headers of the printed output.
The text under 'Comments' is meant for desired comments. It appears in the CubusExplorer in the preview of the selected analysis and can, of course, be inspected here in the described dialog at any time. At the bottom of the dialog there is a print entry button, with which the comment can also be included in the report.

(2) Settings > Materials

Using this menu the materials are managed for the current analysis.

Detailed information is provided by the separate manual 'Materials and Analysis Parameters', which can be opened in the menu 'Help'. Context-related Help (Click on the [?] buttons of the dialogs) also appears in this document.

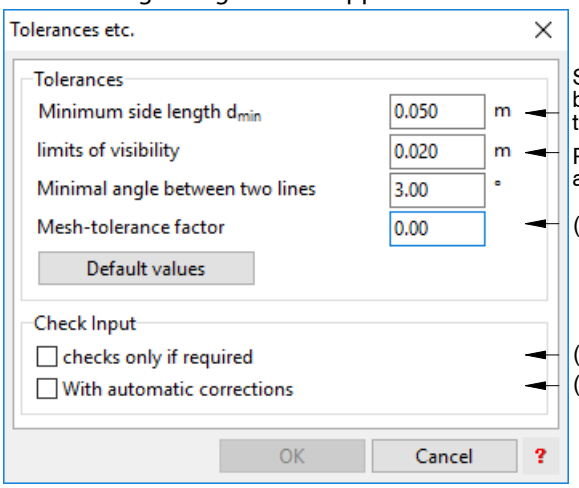
(3) Settings > Analysis Parameters

Analysis parameters are needed for verifications and dimensioning of the cross section using FAGUS procedures. For the current analysis they are managed using this menu.

Detailed information is provided by the separate manual 'Materials and Analysis Parameters', which can be called in the menu 'Help'. Context-related Help (Click on the [?] buttons of the dialogs) also appears in this document.

(4) Settings > Tolerances

The following dialog window appears:

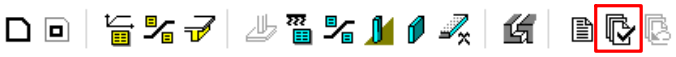


☞ *The tolerances are set for typical applications in structural engineering. If errors are displayed during FE mesh generation, the input geometry should primarily be simplified. The reason is sometimes that the input is taken directly from architect plans without further editing. A change in the default values is often successful only in exceptional cases (and the Cubus hotline only helps with projects with default settings)! They should therefore only be changed in special cases, since the correct functioning of the geometric algorithms is only possible with suitable tolerance values.*

☞ *The minimum side length must be at least twice as large as the visibility limit.*

(a) In the generation of the FE mesh a tolerance value is used, which can be modified here. The initial setting is 0.00. If problems are encountered in the mesh generation one can try a value in the range 0.1 ... 1.0.

(b) After completing the input of structure objects (on leaving the input dialog or when changing to another button in the tabsheet) the structure is checked automatically for input errors. Using this option this checking process can be deactivated (makes sense in the case of large projects). There is always the possibility of checking the structure (or parts of it) manually using the button.



(c) When checking the structure incorrect objects (points, lines) are marked and highlighted. When this option is activated the program tries to automatically correct errors in structure objects found during the checking process. The following errors can be dealt with:
- sides shorter than the visibility limits are eliminated.
- sides lying on top of one other that belong to the same object type are deleted.

The automatic correction, however, causes the input objects to be changed during checking as follows:
- outline, openings: changes lines and rectangles into polygons.
- material and area support separation lines, fixed mesh lines: changes the rectangles and polygons into lines.
- outline, opening, material and area support separation lines, fixed mesh lines: connecting nodes of two lines, which form a straight line (angle = 180 degrees), are eliminated.

(5) Settings > Dialog Settings

Many dialog windows (e.g. attributes box for material zone) function according to the pre-settings as follows:

If a dialog window is used in an analysis, it saves its values as the default values for the next task within the same analysis.

The default values for the first use of a dialog window in an analysis are set as follows: without any special provisions the default values specified by the program are given.

These default values set by the program can be overwritten by a user (identified by Login Name) on his computer as follows:

First one has to set the dialog as one wants it for the default values. Then the menu command **'Settings' > 'Dialog Settings' > 'Save as Default Values'** is called. The settings saved in this way are called User Settings.

As already mentioned, within an analysis the last values used are always taken as the default values. These can be overwritten using the menu command **'Settings' > 'Dialog Settings' > 'Reading in Default Values'** with the user settings described above.

An alternative to these menu commands is use of the key combinations
<Ctrl><F9> for saving as default values and
<Ctrl><F10> for reading in default values.

This is the only possibility if the menu bar is not active. The abovementioned two commands are confirmed in a confirmation dialog.

Whether or not a dialog window functions using to this procedure is detected if the menu **'Settings' > 'Dialog Settings'** is active, if the corresponding dialog window is the only one that is open, or if after activating the corresponding key combination the confirmation dialog appears.

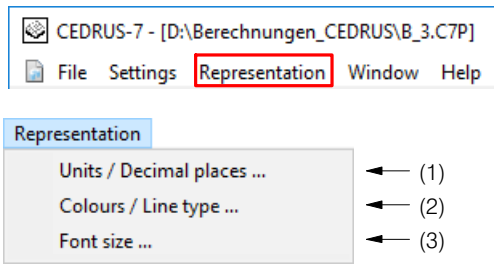
(6) Settings > Reduce Dialog

When introducing objects with a Properties dialog, one often needs to move it, since the position where one wants to place an object is taken by the dialog. By means of '**Settings' > 'Reduce Dialog**' the dialog is reduced to the size of its header. When the input is finished or if one moves the mouse pointer over this header, the complete dialog is shown.

(7) Settings > Automatic Save

The menu causes a dialog window to appear, in which you can specify whether a Save should be automatic and in at what time intervals.

Menu 'Representation'



(1) Representation > Units / Decimal Places

A dialog window appears, in which the units and the number of decimal places can be set for most of the quantities used by the program. The settings made here are saved as user-specific and not program-specific.

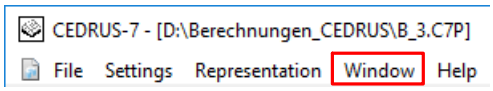
(2) Representation > Colours / Line Types

In a dialog window the type of display of most objects of CEDRUS can be specified with regard to line thickness, line type, colour and filling, separately for the screen, and black and white and colour printers. The settings made here are saved as user-specific and not program-specific.

(3) Representation > Font Size

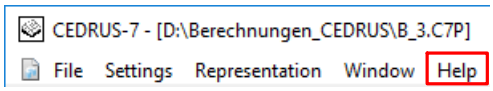
Here the font size applying to text objects in the graphics area can be influenced by specifying a scaling factor once for all objects. This setting has no influence on the font size in the printed output. One can specify the latter using the Print Entry dialog or the CubusViewer.

Menu 'Window'



This menu is only relevant if several analyses (= windows) are open simultaneously. Then it allows the windows to be arranged according to various criteria and to select one of the windows as the active one.

Menu 'Help'



This menu item provides access to the different help documents of CEDRUS.

The short Help document 'Use of Help' gives important tips on the use of the Help system.

The sub-menu 'About CEDRUS' opens a dialog window, in which the current program version, the versions of the module being used and the Read-Me file on CEDRUS can be seen.

Using the sub-menu 'Load Examples' application examples that are part of the purchased package can be loaded.

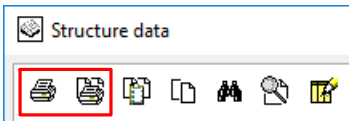
B 5 All you need to know about Printing

Print directly

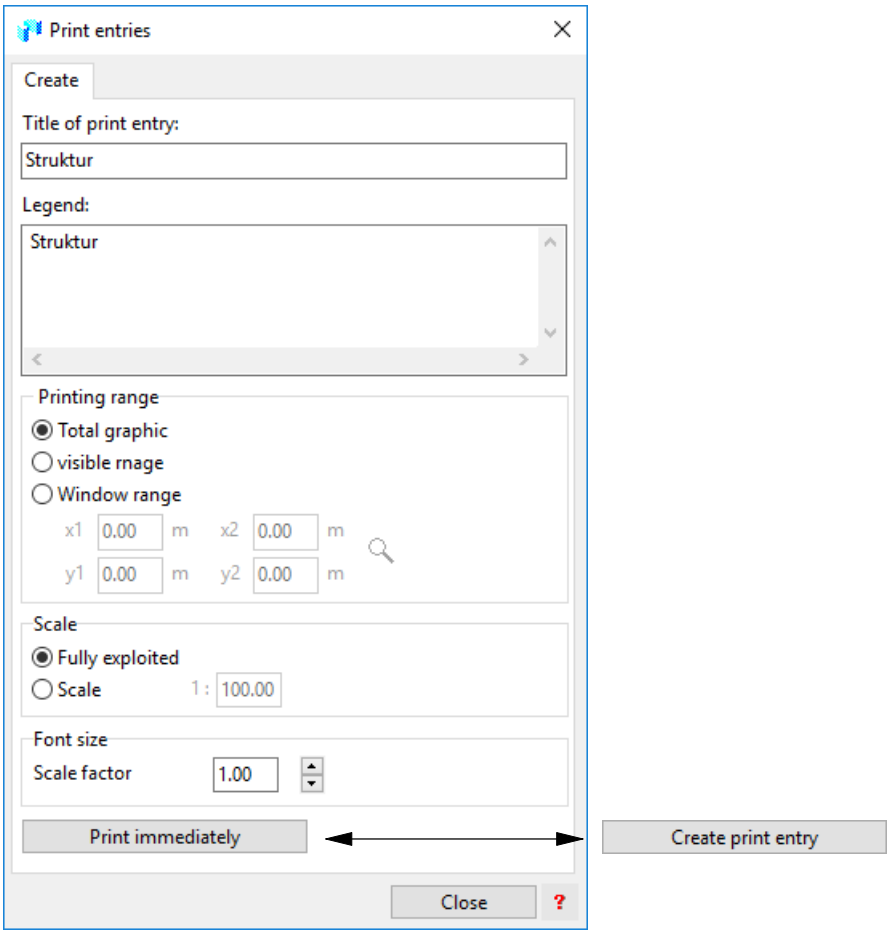
Print Entry

All printable data, whether text or figures, can be printed directly at the corresponding places in the program using the button on the left or a print entry can be made in an output list for printing later.

Text output is always executed in separate table windows, which have buttons for printing or making a print entry in their headers:



Figures involve the contents of the graphics area or a part of it. Printing or making a print entry for figures is executed using the buttons shown on the left at the top of the program window. Before printing or making a print entry, each time the following dialog window appears, which is largely self-explanatory:



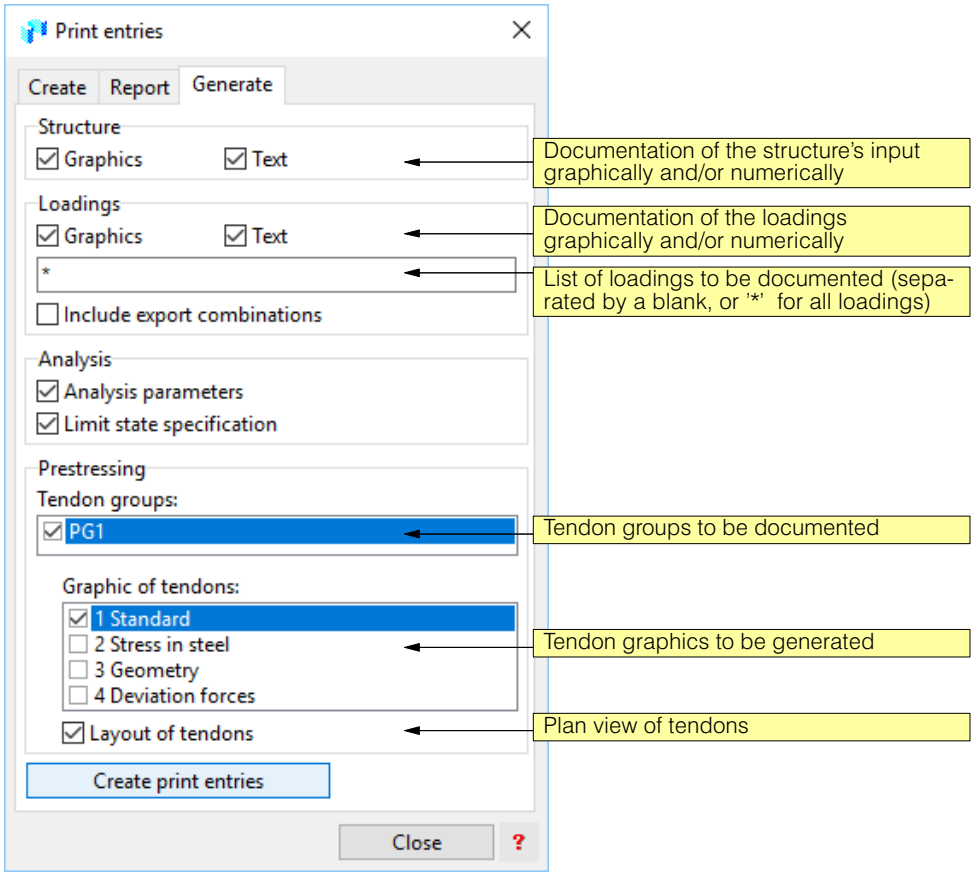
For the complete documentation of an analysis one works with print entries, so that before printing one can have a preview of the whole 'Report' in the CubusViewer and corrections can still be made regarding sequence, dimensions, page breaks, page numbering, etc.

Thus **Report** denotes the sum of all the generated print entries that are managed in the CubusViewer.

The first version of a report is created basically from the individual print entries, whereby for the documentation of the input there is a Report Generator. After any changes to the input an existing report can be automatically updated.

Report Generator

By clicking on a button for a print entry of the graphics area the following dialog appears, whose second tabsheet contains a configurable report generator, whose possibilities are explained in the figure.



Manual Print Entries

The following list shows all points in the program where print entries are possible and in roughly the sequence, in which they would be required for a report:

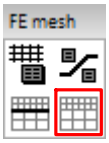
Structure

- Regarding the documentation of the structure input one would include at least one figure of the slab, best of all in the tabsheet 'Structure'. One should ensure that all required layers are activated. If it concerns a large slab with many structure objects, it is better to have print entries for several figures with different layer switches or additional excerpts from figures.

  In the case of objects with labelling boxes one should remember that the amount of labelling for most objects can be influenced in their Attribute dialogs in the tabsheet 'Option'.

  Further, we would like to point out the possibility of dimensioning the figures with a view to their optimum documentation and to elaborate the figures in regard to the graphics. This is done in layers of the group "User".

- As legend for the figures of the structure, we could use a tabular listing of the structure's objects, which can be created using the button shown on the left (located on the right side in the tabsheet 'Structure').



FE mesh

- To document an FE analysis we may also need to include the FE mesh. The FE mesh is made visible using the adjacent layer switch and can be included in the report as a print entry.

For tabular results one might need a figure with the node and element numbers. These can also be output by making the corresponding sublayer (by clicking the right mouse button when placed over the FE mesh layer switch and activating the desired sublayer in the menu 'Sublayer Visibility').

Loads

- In the tab sheet 'Loads' a tabular documentation of the loadings can be created using the button „Legend of the loading data“. In the List of loadings (see below) there are generating possibilities for doing this. The tabular documentations can be extended by including figure entries. One can go from loading to loading and for each “interesting” loading a figure entry can be made. There is also the possibility of including several loadings in one figure by activating the corresponding Loading layer.

  Further, it is also possible here to provide the figures with dimensioning lines and added graphics. One should not forget the possibility here of creating additional layers in the layer group „User“, so that any structure dimensions or dimensions of other loadings are not first deleted, but simply made invisible.

- The dialog window „List of loadings“ can be shown using the button on the left. At the bottom it has a Print Entry button to create an entry for the loading list.

  With these buttons at the top of the dialog window „List of loadings“, tabular documentations of the selected or the active loadings can be generated. The selection of the loadings is done in the usual Windows fashion - <Ctrl> + <A> to select all loadings is also supported.

Result combinations and limit state specifications

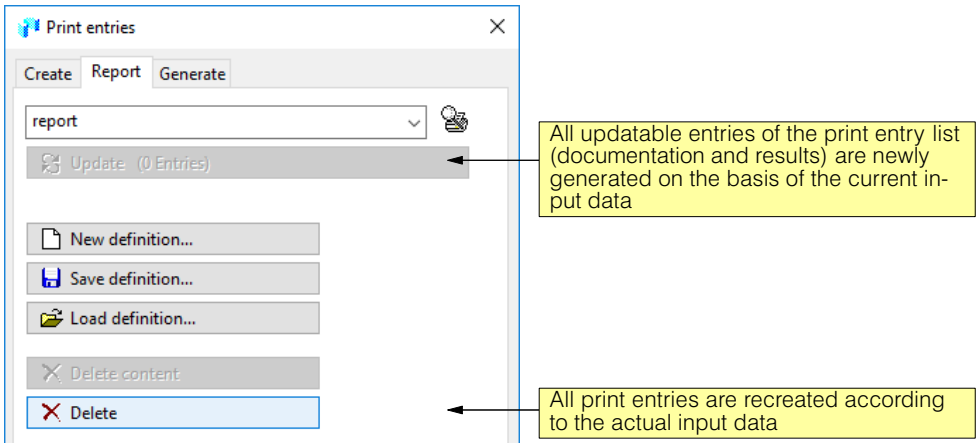
- The documentation of any results combination and limit state specifications is done in the tabsheet 'Analyses' in the corresponding Input dialog windows.

Results

- All figures and text documents showing results can in the usual way be entered in the Output List. In the case of figures, layers that get in the way should be made invisible and in connection with the presented results, any interesting structure, load or FE mesh layers should be made visible. We remind users of the possibility described earlier of superimposing several figures showing the results.

Updating the Report

The print entry dialog offers on its third tabsheet a button to update the entire report according to the acual input:



CubusViewer

The CubusViewer serves the purpose of showing a preview and printing the report, i.e. of all previous print entries. The details are found using their Help menu as well as by referring to the CubusViewer manual, also reachable using the Help menu.

B 6 The tab sheet 'Control'

The Control tabsheet is really the main menu for using CEDRUS. In the following, each of its sheets is described in detail in its own chapter .

Before beginning with this detailed description, one should get a broad overview of a slab analysis, which is given in Chapter A 1.4.

'CAD' see page B-13

'Structure' see page B-18

'Punching' see page B-47

'Loads' see page B-63

'FE-mesh' see page B-78

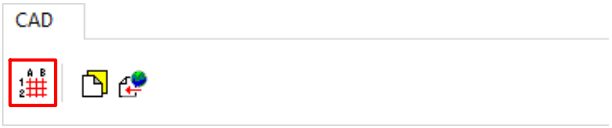
'Calculations' see page B-80

'Results' see page B-104

B 7 CAD

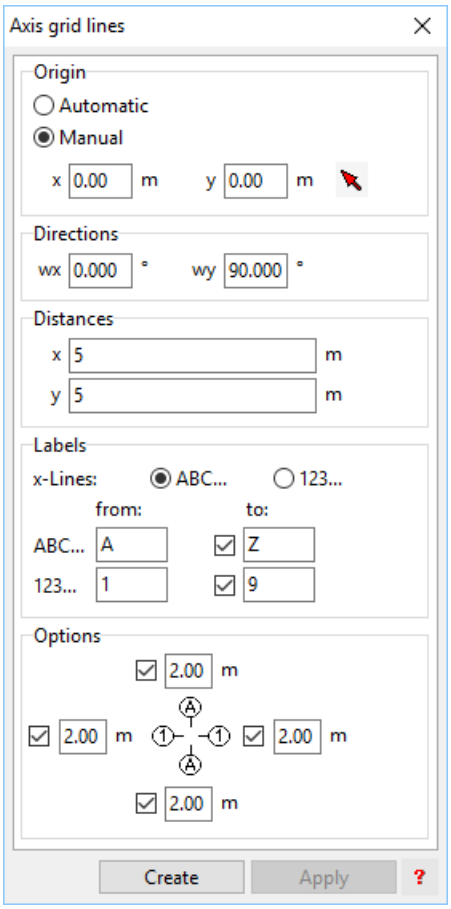
The tab sheet 'CAD' provides various ways of increasing efficiency in the input of the structure.

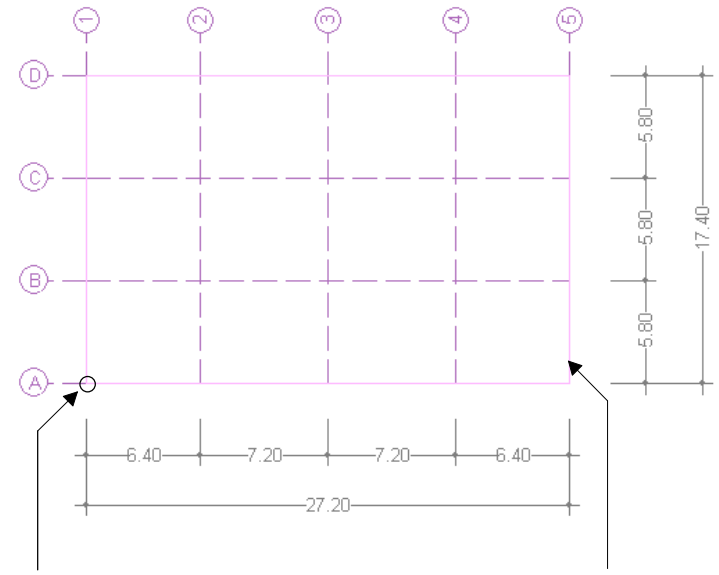
B 7.1 Axis Grid



An axis grid serves two purposes: The intersection points of the axes can be snapped when inputting geometrical objects and the identifiers for the columns can be adjusted automatically to the axis grid.

The button shown in the CAD tabsheet causes the dialog for axis grid objects to appear, which, with the given settings, creates the axis grid object shown, if with [Introduce] a rectangle of dimensions 27.2 x 17.4 is input:





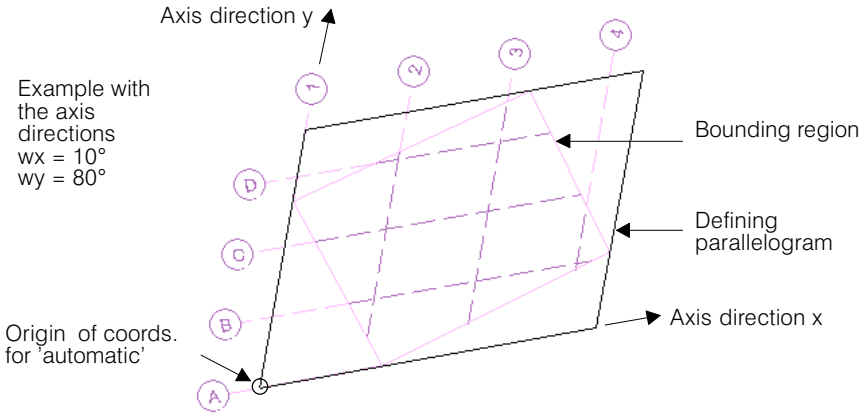
Origin of coordinates:
This is the first grid point, i.e. the intersection of the first two grid lines (A) and (1)

Bounding region of axis grid, here a rectangle.

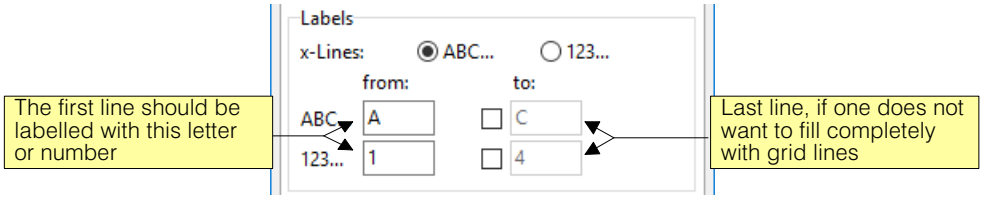
The following points should be observed in the case of axis grid objects:

- When introducing an axis grid object its **bounding region** is drawn. If one uses the button [Introduce] in the dialog, the rectangular drawing tool is activated automatically. Alternatively, one can click on the polygon tool of the Graphics Editor to draw a polygon.
- The **axis directions x and y** are set using the angles **wx** and **wy** in the dialog. The angles are measured with respect to the global X direction in the anticlockwise direction.
- The first two grid lines intersect at the **origin of coordinates** of the axis grid. Corresponding to the setting in the dialog it can be input manually at any desired position or determined by the program automatically.

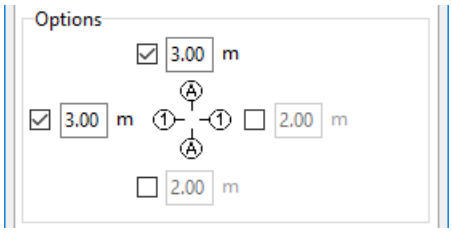
In the 'automatic' case the origin is located at the lower left corner of a parallelogram in the directions of the axes, which defines the bounding region:



- The **spacings** along the x grid lines beginning at the origin, are measured in the axis direction y and that of the y grid lines in the axis direction x. The spacings are input in the corresponding input fields, separated by blanks. Several equal spacings can be specified by n*spacing (e.g. 3*5.4). With the last input spacing the limiting region is filled with grid lines. Negative values are not permitted.
- **Labellings**: The x lines can, if desired, be labelled with letters or numbers. For the y lines the other type of labelling is automatically chosen.



- The labelling is placed along the circumscribing parallelogram (see above) at a selected spacing above and/or below and left and/or right:



- Several axis grid objects can be input, which may be helpful in the case of non-orthogonal structures.

CAD

- The visibility of the defined axis grid can be controlled using a layer switch in the layer group 'CAD'.
- The intersection points of two axis lines (= axis points) have an invisible identifier, e.g. „B3“. If one introduces columns in the vicinity of axis points, the column identifier provides the axis point identifier automatically, if for the identifier one specifies in the Options tab sheet of the Columns dialog „according to axis grid“.

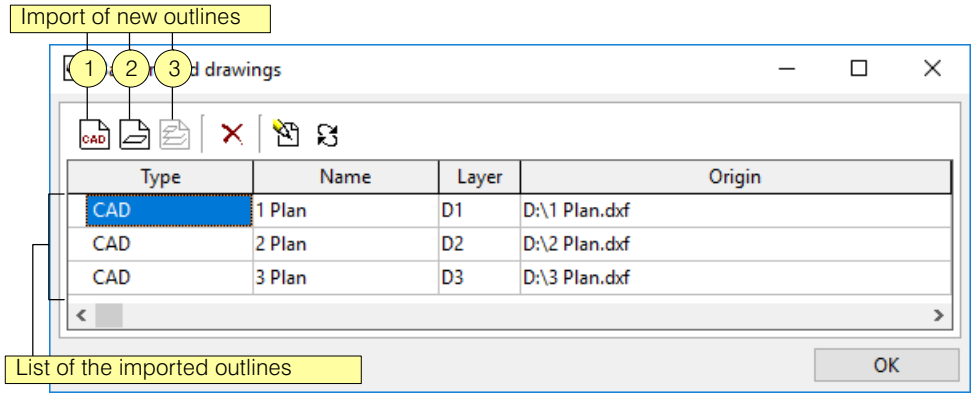
B 7.2 Background Plans, CADview

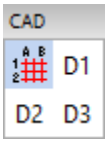
It is often efficient, to input a slab using as a background a construction plan, from which, e.g., points can be grabbed using a mouse. CEDRUS can work with three types of background plan:

1) CAD outline in DWG or DXF format.
   For this one needs the program option 'CADview'. Working with CAD outlines is described in detail in Part C.
2) Outlines from any CEDRUS analysis. One can access the analysis data of earlier program generations.
3) Floor slabs of building analyses.

B 7.2.1 Import of CAD Drawings and CEDRUS Plan Views

The import of plans is carried out in the Management dialog for background plans, which can be opened with the button shown on the left. It has three buttons for the import of the three outline types mentioned above and shows the imported outline plans in a list.



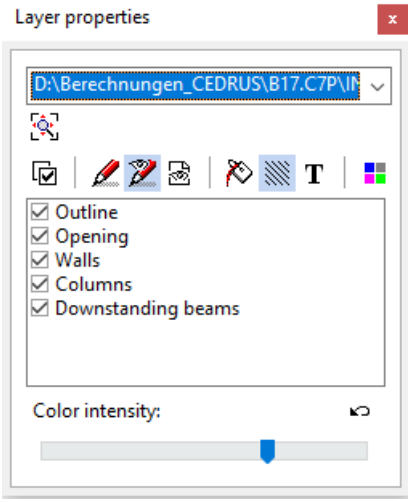


For each background plan shown in the above Management dialog there is a layer switch in der layer group 'CAD'. Thus the plans can be made visible in any arbitrary combination.

B 7.2.2 Working with CEDRUS Background Plans

Placing CEDRUS outlines in the background, especially outlines of other building storeys, is an invaluable help for placing the floor slabs one on top of the other with geometric accuracy. Several outlines can be imported as backgrounds for a slab, e.g. those of the neighbouring storeys below and above.

The CEDRUS outlines imported for backgrounds have the sublayers shown in the Layer Properties dialog, whose visibility can be controlled as desired in this dialog. The dialog is displayed on the screen using the context menu of the layer switch and looks as follows:



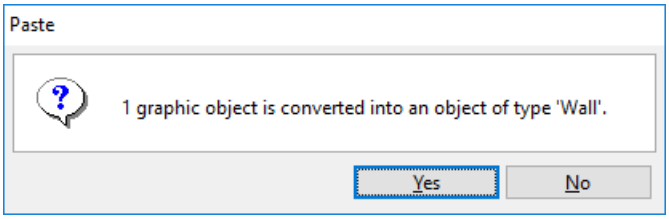
The background outlines can be used as follows:

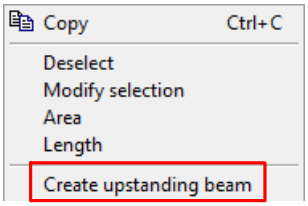
- **snapping:** The graphics objects of the background outlines can be grabbed, i.e. new objects can be drawn exactly over them.
- **Importing objects:** Objects of the background outlines can be imported into the slab directly. This is done as follows:

  1) The desired object (e.g. a wall or an opening) in the background outline is selected and copied to the clipboard <Ctrl+C>

  2) The button for the corresponding input object (e.g. Wall dialog or button for openings)

  3) The object from the clipboard is pasted using <Ctrl+V>. If this object matches the input object, then a Confirmation dialog appears:



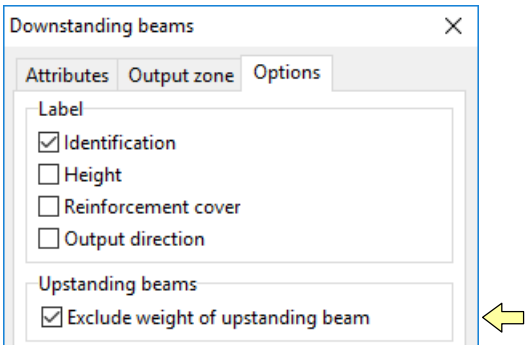


- **Create upstands automatically:** Using the context menu background walls can be imported directly as downstands in the slab. For example if a concrete wall in storey L2 lies directly over two columns in storey L1, then this can be satisfactorily modelled as an upstand in storey L1, which lies exactly under the wall. When automatically creating upstands the following points should be observed:

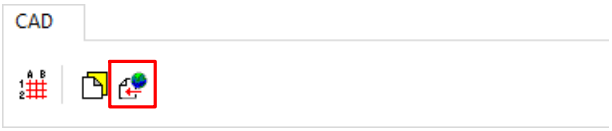
  a) For the height of the upstand, half the storey height is assumed.

  b) In the outline geometry the created upstand corresponds exactly to the wall from which it is derived. This can lead to geometrical conflicts with other structure objects that have to be solved by the user.

  c) To avoid taking into account the weight of the wall in the building twice, the generated upstand is given the following property automatically (Downstand Properties dialog):



B 7.3 CADextract



The CADextract function is based on imported CAD outlines (see B 7.2) and it is able to detect objects like walls and columns in CAD outlines and provides more efficient methods for generating CEDRUS slabs from CAD outlines than simply placing a background and tracing it.

To use CADextract the program options 'CADextract' and 'CADview' are required. Working with CADextract is described in Part C.

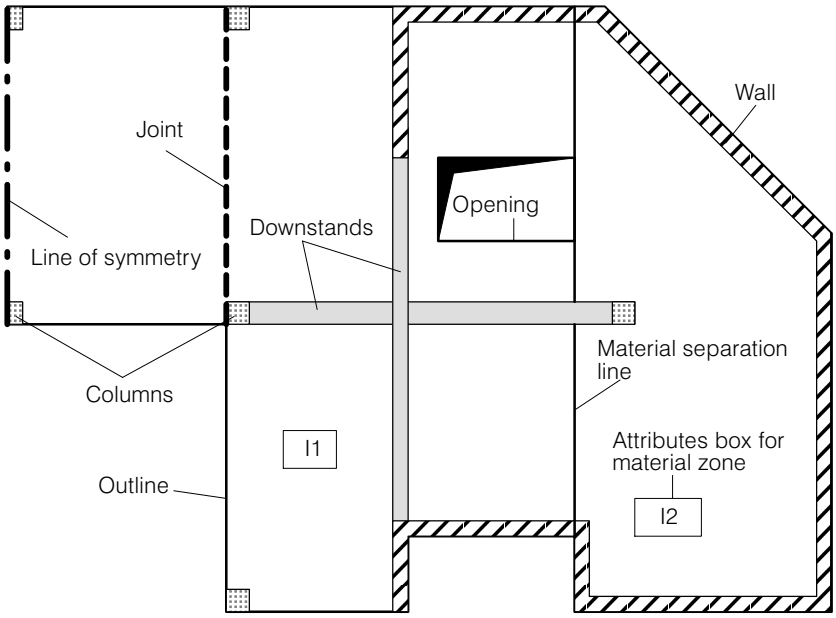
B 8 Structure

In the tab sheet 'Structure' basically one models a slab in terms of geometry, materials and supports.

The tabsheet contains a number of buttons, by means of which individual objects can be input. They are, as in the sheets of tabsheet themselves, arranged from left to right, as they are normally used and then described.

The following figure shows an outline with the most important structure elements and how these are denoted in CEDRUS:

Certain conditions have to be observed in the input of geometrical objects, which are discussed here before describing the individual structure objects.

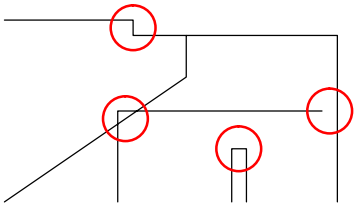


B 8.1 What to Pay Attention to in the Input of Geometrical Objects

The geometrical definition of a slab structure consists essentially of points, lines and objects that consist of several lines. Since the FE mesh has to take all these points and lines into account, they are subject to the following conditions (in the case of walls only the support axis is relevant):

- In relation to one another, all points have to be at a distance of at least d_{min}. The value of d_{min} can be seen in the menu „Settings“ > „Tolerances“ and in special cases can also be changed.
- The distance of a point from a line must likewise be at least d_{min}.
- If there are two points in the same position (e.g. two lines that meet at a point), then depending on the method of inputting these points it may happen that due to rounding errors they do not have exactly the same coordinates. For this situation CEDRUS also has a tolerance value d_{tol}. If two points are nearer than d_{tol}, then they are merged by the program. The value d_{tol} can be seen in the menu „Settings“ > „Tolerances“ and in special cases can also be changed.

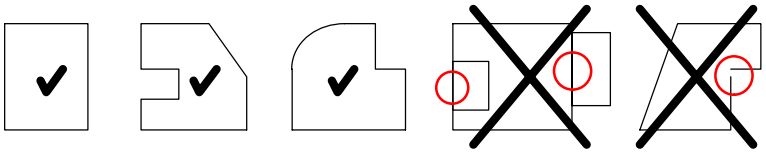
Summarizing: If the distance between two points or between a point and a line is greater than d_{tol} but smaller than d_{min}, then during checking the program will not accept the input and will give an error message.



Some critical places for the minimum distance

B 8.2 Outline in Plan View

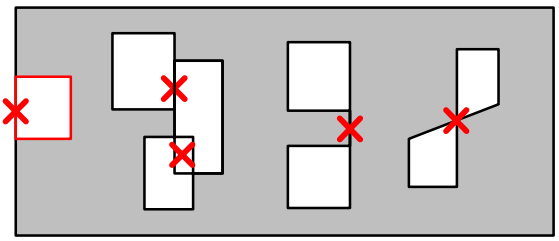
The outline of a slab is its boundary line, which must be represented visually by a closed polygon. By clicking on the button the required drawing tools of the Graphics Editor are activated, with which the boundary line is input. This can comprise several pieces (lines, polygons, circular arcs), independent of order and direction. Finally, one must be able to see a closed polygon on the screen.



A slab may only have one outline, and may not touch or intersect openings.

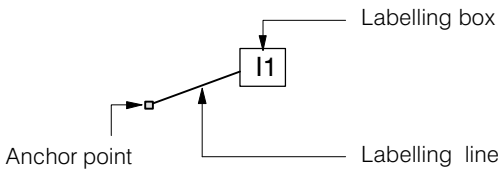
B 8.3 Openings

Openings are closed polygons. They may be of arbitrary shape. Openings may not touch or intersect one another or the outline.



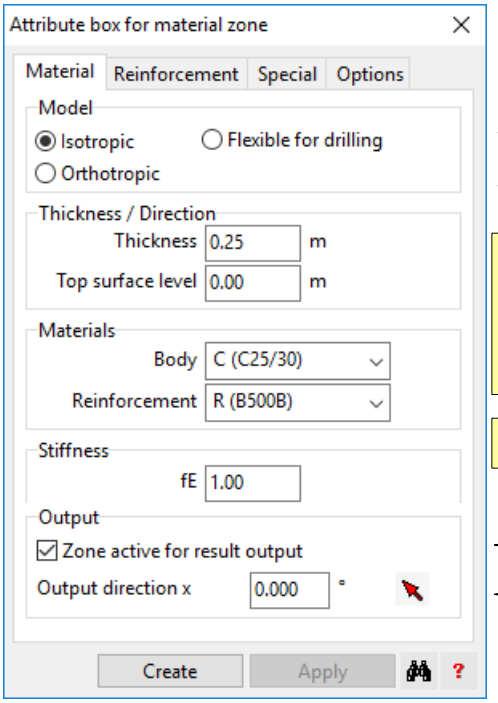
B 8.4 Attributes Box for Material Zones

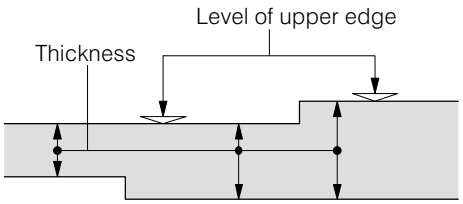
The slab properties summarized under 'Material' are described in a box called '**Attribute box for material zone**'. The corresponding dialog is shown below. The attribute box is an extended point object with a labelling box, which may be placed anywhere on the slab.



The anchor point defines the position, i.e.the zone in which the properties apply. These are presented in the labelling box as a short description. The amount of descriptive text can be specified in the tabsheet 'Options' of the corresponding dialog. With the introduction of an attributes box for material zones the anchor point and the labelling box are at the same place, i.e. only the labelling box is visible. To move just the labelling box in the context menu of the attributes box there is the command 'Move Labelling'.

Tabsheet 'Material' in the dialog for the attribute box for material zones:





The level of the upper edge is only of importance for the static model for slab zones that border on a downstand (see B 8.7).
It is of no consequence in the dimensioning of beam cross sections (see page B–90) and in the reinforcement module.

On the material model, materials and stiffnesses see B 8.6

← (1)

← (2)

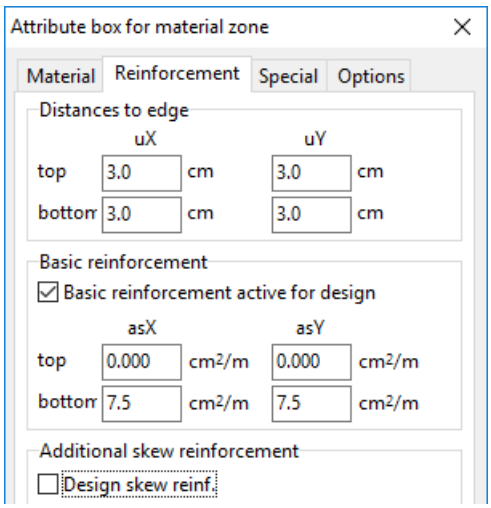
(1) Active zone for results output

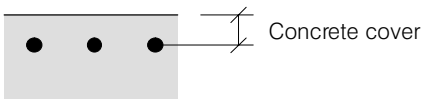
In the case of plots (isolines, principal values of stresses etc., numeric-graphics) this option must be active for all output zones for which results are required.

(2) Output direction

The output directions are those in the orthogonal directions (x, y), in which plots of section forces and reinforcement are output (not important for section outputs). These directions can be specified to be different for each material zone. The default output directions correspond to the global coordinate directions (X, Y). The angle of the output direction x (with respect to X in the anticlockwise direction) may be input either directly or using the arrow button by selecting two points.

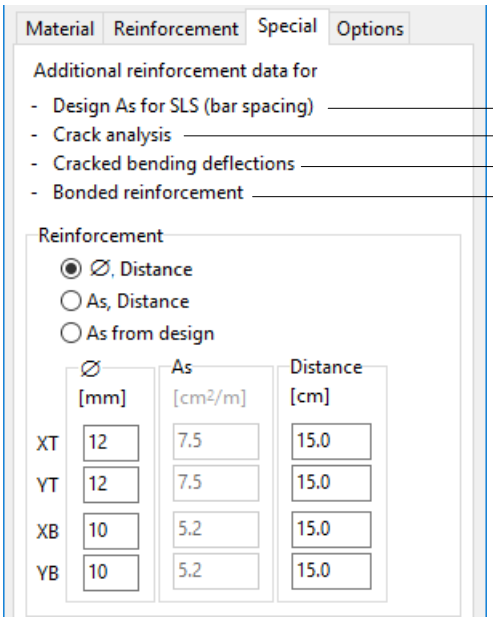
In the tab sheet 'Reinforcement' of the corresponding dialog, the concrete cover at the edges and the basic concrete reinforcement can be specified for the four reinforcement layers (below and above in the default output directions)





For plot outputs of the required concrete reinforcement any basic reinforcement is deducted, i.e. only the additional requirements are output

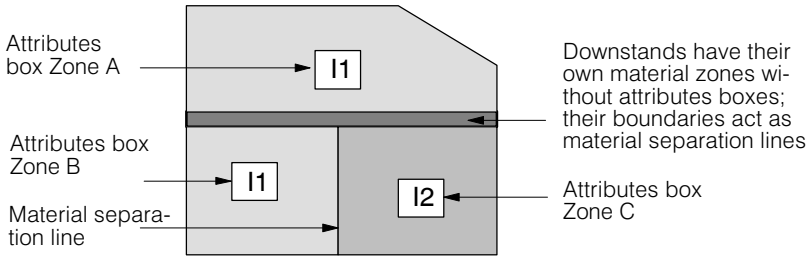
To input skew reinforcement see chapt. B 12.4



Finally, in the tab sheet 'Options' one can specify which properties should be output in the labeling box.

B 8.5 Material separation lines

A slab can be subdivided into several zones using **Material separation lines**. To ensure that unique zones are obtained, material separation lines must begin and end at the outline, at an opening, at another material separation line or at the edge of a downstand. In each material zone there must be one (and only one) attributes box (see following figure).



The following chapter describes the material models supported by CEDRUS with the corresponding material properties.

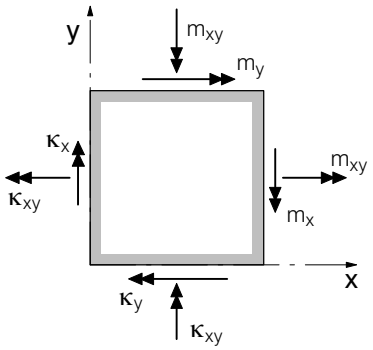
B 8.6 Material Models

A material model is a property of the material zones. It is specified in the 'Attribute box for material zones'.

CEDRUS allows the material models **isotropic**, **torsionally weak (i.e. with regard to drilling moments)** and **orthotropic.** They are all based on Kirchhoff's plate theory and do not take into account shear deformations. Thus slabs that are very weak in shear are not satisfactorily modelled.

Downstands are special orthotropic material zones; see B 8.7.

The following notation is used in the description of the material models:



mx, my, mxy:	slab moments
κ_x, κ_y, κ_{xy}:	curvatures und drilling
E,ν:	modulus of elasticity, Poisson's ratio
h:	slab thickness
f_E:	correction factor for the stiffness

B 8.6.1 Isotropic Material

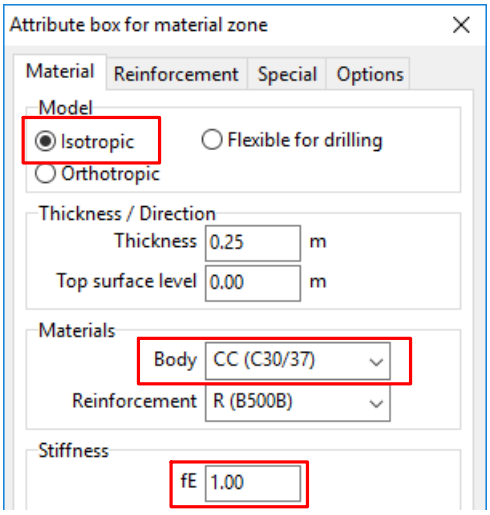
An isotropic material is directionally independent and is described completely by the modulus of elasticity E and Poisson's ratio ν according to Kirchhoff's plate theory. The relationship between plate curvatures and plate moments is determined by the elastic matrix :

$$\begin{Bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix} = \frac{12}{f_E E h^3} \begin{bmatrix} 1 & -\nu & 0 \\ -\nu & 1 & 0 \\ 0 & 0 & 2(1+\nu) \end{bmatrix} \begin{Bmatrix} m_x \\ m_y \\ m_{xy} \end{Bmatrix}$$

The first line, written out, is:

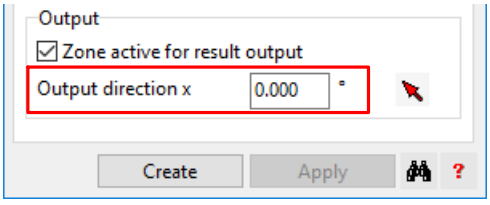
$$\kappa_x = \frac{12}{f_E E h^3}(m_x - \nu m_y)$$

E and ν are material properties of the material zone. The default value of 1.0 of the stiffness factor f_E was introduced, so that the slab stiffness can be simplly multiplied by a factor, without having to change the modulus of elasticity given as a material property or introduce a new material.



B 8.6.2 Material without Drilling Moments Stiffness

This model corresponds to the isotropic material with the difference that the third diagonal coefficient of the above elastic matrix is assigned a very large number. Thus the slab is weak in torsion and cannot resist drilling moments m_{xy}.



Or, expressed in a different way: the principal moments are forced to act in the output direction x,y.

Material very flexible in torsion can be useful in special cases, if one does not want to include drilling reinforcement and instead accept higher m_x and m_y values. However, it should be noted that in some cases equilibrium is not possible without drilling moments.

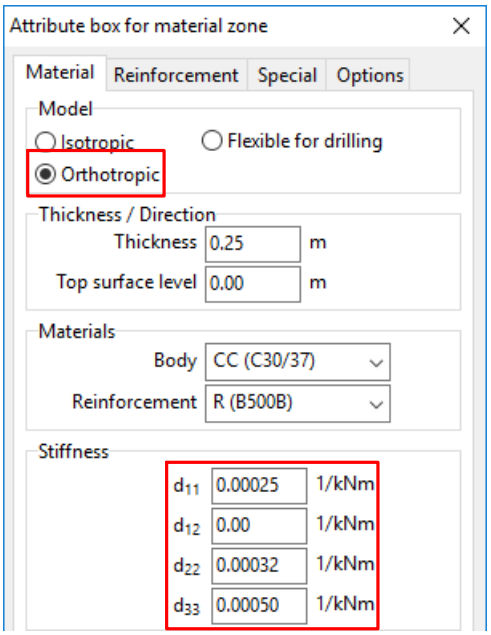
B 8.6.3 Orthotropic Material

Orthotropic material exhibits different material properties in the two orthogonal directions x and y (output directions). It is described by the following elastic matrix, whose coefficients (4 values) have to be input:

$$\begin{Bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix} = \begin{bmatrix} d_{11} & d_{12} & 0 \\ d_{21} & d_{22} & 0 \\ 0 & 0 & d_{33} \end{bmatrix} \begin{Bmatrix} m_x \\ m_y \\ m_{xy} \end{Bmatrix}$$

$$(d_{12} = d_{21} \leqq 0)$$

Here the condition $d_{11}*d_{22} > d_{12}*d_{21}$ has to be fulfilled.
The x (material) direction can be chosen.



Application Example 1:

Slab with different elastic moduli E_x, E_y in the x and y directions

$$\kappa_x = \frac{12}{E_x h^3} m_x - \nu \frac{12}{E h^3} m_y \quad ; \quad \kappa_y = \frac{12}{E_y h^3} m_y - \nu \frac{12}{E h^3} m_x \quad ; \quad \kappa_{xy} = \frac{24}{E h^3}(1+\nu) m_{xy}$$

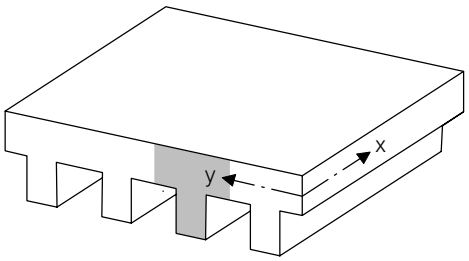
Thus the coefficients to be input are

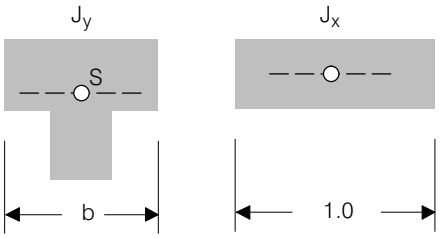
$$d_{11} = \frac{12}{E_x h^3} \quad ; \quad d_{22} = \frac{12}{E_y h^3} \quad ; \quad d_{33} = \frac{24}{E h^3}(1+\nu) \quad ; \quad d_{12} = d_{21} = -\nu \frac{12}{E h^3}$$

If in an actual case one introduces the highest value of E_x,E_y for the usually unknown value of E, then the above condition $d_{11}*d_{22} > d_{12}*d_{21}$ is always fulfilled.

Application Example 2: Ribbed slab with ribs in the X direction

To determine J_y one cuts out a piece repetitively between two axes of symmetry and then determines the moment of inertia with respect to the centre of gravity:





Assumptions: $\nu = 0$, $E_x = E_y = E$

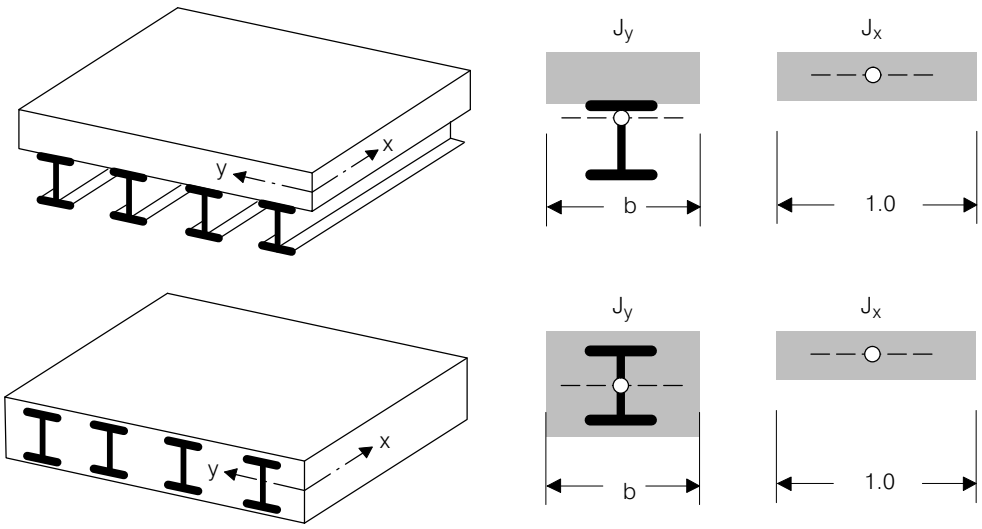
$$\kappa_x = \frac{b}{E J_y} m_x \quad \text{--> } \quad d_{11} = \frac{b}{E J_y}$$

$$\kappa_y = \frac{1}{E J_x} m_y \quad \text{--> } \quad d_{22} = \frac{1}{E J_x}$$

If one only considers the upper slab for the drilling stiffness, we obtain

$$\kappa_{xy} = \frac{2}{E J_x} m_{xy} \quad \text{--> } \quad d_{33} = \frac{2}{E J_x}$$

Ribs made of steel profiles could be treated analogously:

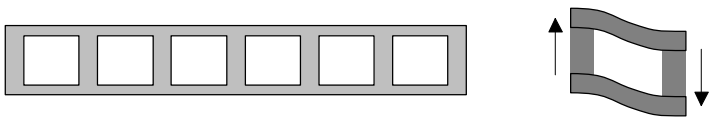


To determine the ideal moment of inertia J_y the steel parts are weighted by the factor $E_{Steel}/E_{Concrete}$.

B 8.6.4 Shear Deformations

In Kirchhoff's plate theory shear deformations are neglected. Thus plates exhibiting substantial shear deformations cannot be modelled accurately using CEDRUS.

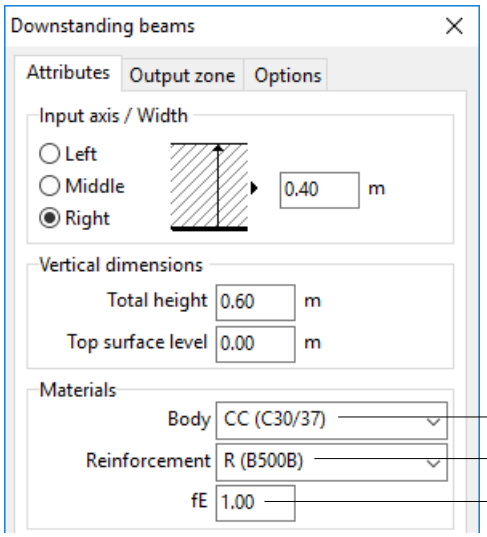
Example: slab with hollow boxes



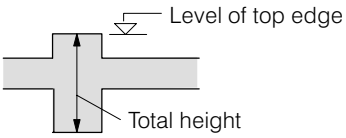
B 8.7 Downstands

Downstands - i.e. the downstanding beams in ribbed slabs - are modelled in CEDRUS as zones with special orthotropic material. They are specific input objects with the Attributes dialog shown below, input as double-lined polygons. Downstand outlines have the function of material separation lines.

B 8.7.1 Downstand Dialog



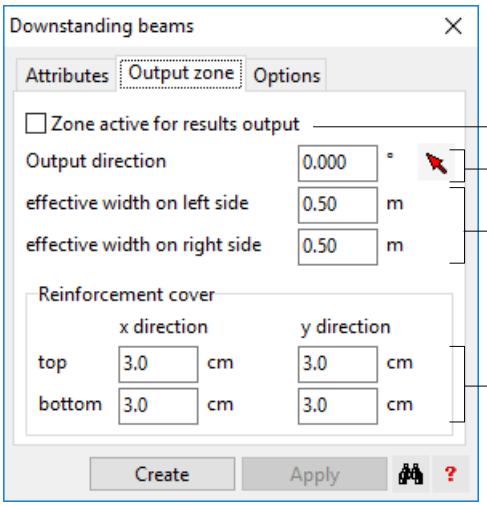
The position of the input axis has no influence on the model



For stiffness and dimensioning (see downstand model)

For dimensioning

For stiffness (see downstand model)



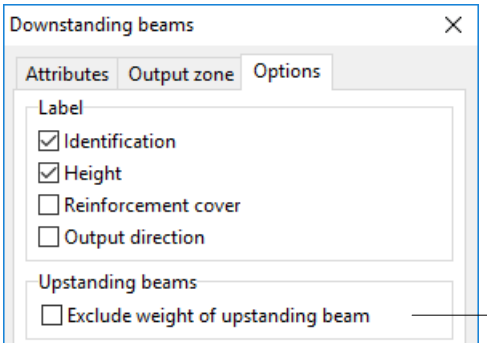
With reference to element-related output such as isolines, etc.; normally switched off, as otherwise poor quality isolinine plots are obtained

For the element-specific output (isolines, etc.)

Defines the section width for the beam section output; no influence on the analytical model

The edge distances (i.e. reinforcement cover) refer to the output directions defined above; in the case of beam sections output interpolation is carried out if the lower tensile axis does not lie in the output direction or normal to it

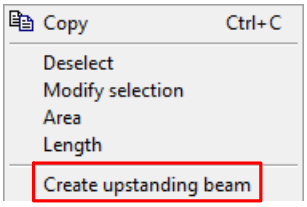
For the output of results it is normally recommended to use beam sections. Downstands can directly be used as beam sections, i.e. they do not need a results section. The section width over which the results are integrated corresponds to the sum of the input values for the effective width left and right, measured from the axis of the downstand.



To prevent the weight of the upstanding part being included in the self weight of the slab. This is important in the building module if upstanding beams are used for modeling walls whoose self weight is already taken account of in the building.

B 8.7.2 Genereting of upstanding beams under walls of displayed slabs

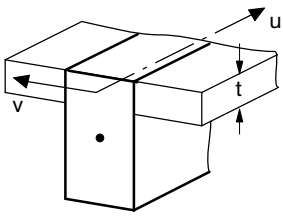
If the reinforcing effect of a wall is to be modeled with a upstand beam on the ceiling slab which supports it, then there is an input aid if the slab with the relevant wall is shown as a background plan (chapter B 7.2). Select the relevant wall of the embedded slab and select the 'Create upstanding beam' line from the context menu (right mouse button):



The following remarks:

- The generated upstanding beam has precisely the outline of the wall used. If this leads to geometric problems with the slab input, the conflict has to be remedied by hand.
- The upstanding beam is inserted with half the height of the wall. This is a relatively arbitrary assumption, but generally leads to a reasonable stiffness.
- The upstanding beam is introduced weightless (according to settings in the options register of the downstand beams dialog)

B 8.7.3 Downstand Model



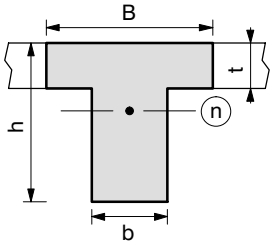
For downstand zones the following orthotropic material model is used:

$$\begin{Bmatrix} \kappa_u \\ \kappa_v \\ \kappa_{uv} \end{Bmatrix} = \frac{1}{f_E} \begin{bmatrix} \frac{12}{E\bar{h}^3} & 0 & 0 \\ 0 & \frac{12}{Et^3} & 0 \\ 0 & 0 & \frac{24}{Et^3} \end{bmatrix} \begin{Bmatrix} m_u \\ m_v \\ m_{uv} \end{Bmatrix}$$

where $\bar{h}$ = effective depth of downstand (according to Model 4 below)
t = thickness of the neighbouring elements determined by program)
E= modulus of elasticity (for body of material in the Downstand dialog)
f_E = factor for downstand stiffness, default value 1.0 (Downstand dialog)

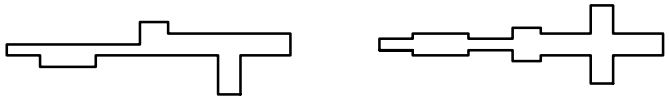
Thus the downstand elements exhibit in the longitudinal direction u the full stiffness (for $\bar{h}$ see below), whereas for the lateral and drilling stiffness the thickness of the neighbouring slab is used. This thickness is determined by the program. If it is not the same on both sides of the downstand, the larger value is taken.

To determine the effective depth of the downstand the $\bar{h}$ Model 4 below is used, which lies between the extreme cases of Models 2 and 3.

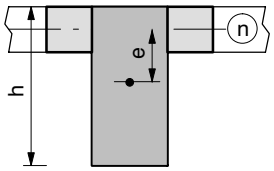


Model 1: T section with effective width B

This model is the most realistic. However, it cannot be applied because an eccentric connection of the T beam with respect to the middle plane of the slab would be necessary. This is not possible since CEDRUS takes no account of membrane strains. All elements of a slab included in the downstands are arranged in the FE model centrally about the common middle plane.



Actual shape of cross section — Cross section assumed in FE model

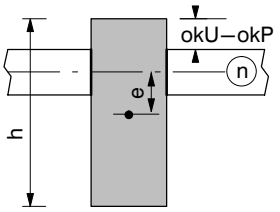


Model 2: Eccentric downstand, neutral axis n in the middle plane of the slab

$$I_n = \frac{bh^3}{12} + bhe^2 + I_{nPlatte} \qquad (I_{nPlatte} = I \text{ of the portion of slab of width } B - b)$$

e is the eccentricity of the centre of gravity of the downstand rectangle with respect to the middle plane of the adjacent slab.

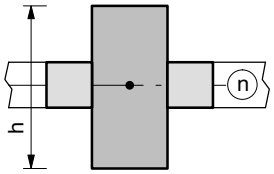
with okU = level of top edge of downstand and
okP = level of top edge of adjacent slab



we obtain for e:

$$e = \frac{h}{2} - \frac{t}{2} - (okU - okP)$$

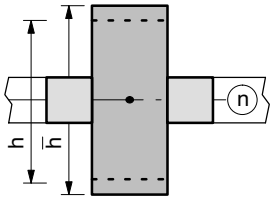
The stiffness of this model compared to Model 1 is large since the neutral axis lies in the middle plane of the slab,



Model 3: Centric downstand

$$I_n = \frac{bh^3}{12} + I_{nPlatte}$$

The stiffness of this model compared to Model 1 is small since the eccentricity is neglected.



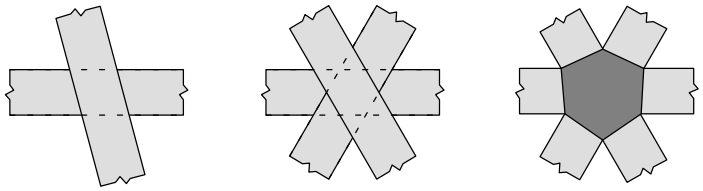
Model 4: Mean of Model 2 and Model 3 = CEDRUS Model

$$I_n = \frac{b\bar{h}^3}{12} + I_{nPlatte} = \frac{1}{2}\left(\frac{bh^3}{12} + bhe^2 + \frac{bh^3}{12}\right) + I_{nPlatte} \quad ; \quad \bar{h} = \sqrt[3]{h^3 + 6e^2h}$$

If downstands intersect, in the zone of intersection an orthotropic material is used, which, if one of the downstands is deeper, the material is assigned the stiffness of the deeper of the two in its direction and normal to it that of the other downstand and likewise with the drilling resistance.

B 8.7.4 What to note during Input

- Only two downstands may intersect in one point. For two identical downstands, in the zone of intersection an isotropic material zone results, for two different downstands a corresponding orthotropic zone.



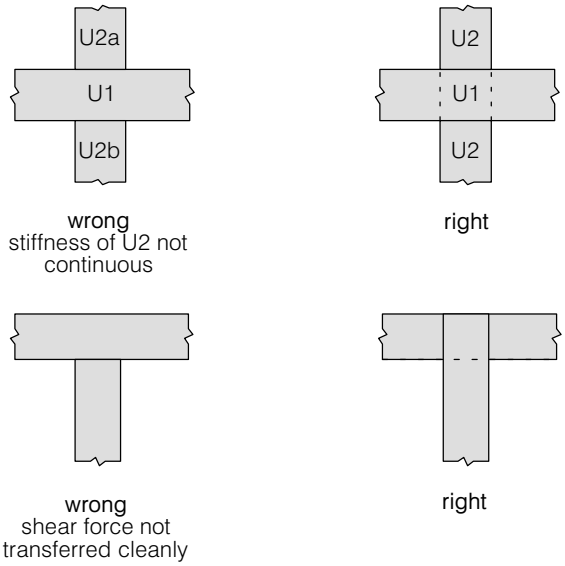
Ok

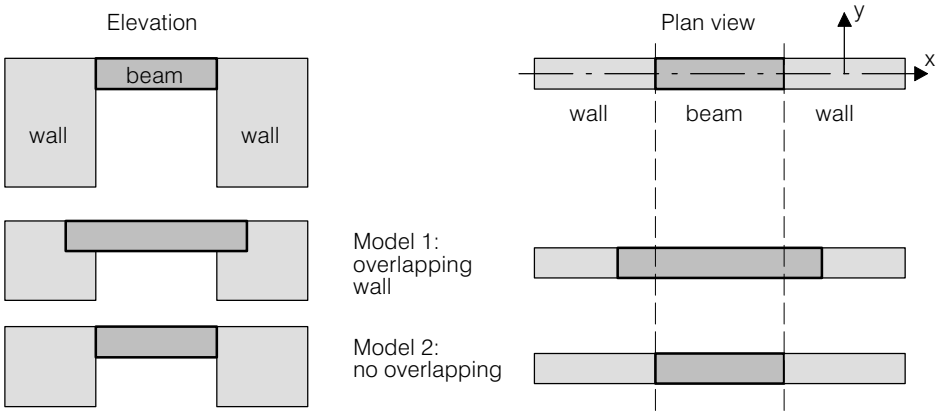
wrong
crossing of more than 2 downstands

right
crossing region with own isotropic material zone

- If two downstands "penetrate" each other, they must be input as follows.
- If one downstand is connected to another one it must penetrate it, as otherwise the shear force cannot be transferred properly.
- Downstand between two walls:

  Which of the two models is better depends on the support conditions of the walls with respect to the rotation about y (direction normal to the bearing axis):



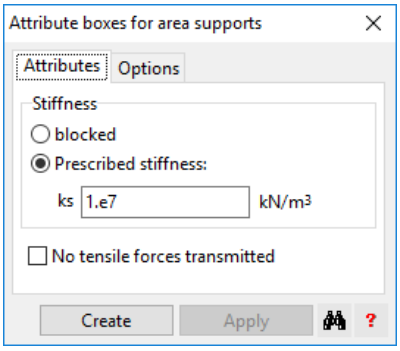


Wall rotation y free (standard model):
Model 1, since otherwise the beam is not properly fixed

Wall rotation y fixed (in general a poor model):
Model 2, since the fixed-end moment at the end of the wall is otherwise too small (just one half, because of taking the mean of the nodal results of the adjacent elements).

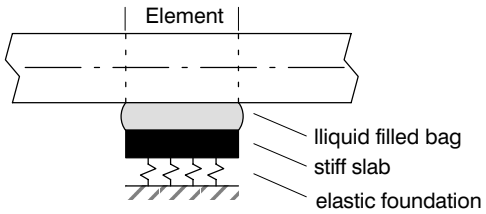
B 8.8 Attributes Box for Area Support Zones and Separation Lines

By introducing an attributes box for area support zones a slab zone can be supported either over a finite area or elastically. Area support zones can be defined with the aid of separation lines. The attributes boxes have the following dialog:



Slab zones supported over an area exhibit for each finite element a uniformly distributed reaction pressure and the average settlement is calculated using Winkler's theory. If the corresponding modulus of subgrade reaction k_s has a very high value, an element is able to bend and rotate freely, but the average settlement remains zero. With a single area support element or several placed along a line, a stable support of the slab is not possible.

A good model for the area-supported element is a liquid-filled bag:



modulus of subgrade reaction k_s:

$$k_s = \frac{\text{soil pressure}}{\text{settlement}} \quad [kN/m^3]$$

An application of area-supported elements is the **method of subgrade reaction (Winkler's model)**. Area support elements with or without tensile stresses can be used. In the latter case the program finds the correct solution by iteration.

Another application is the modelling of column supports, which is treated in the chapter on columns.

B 8.8.1 Determination of the Modulus of Subgrade Reaction k_s

The Modulus of Subgrade Reaction k_s as it is used for foundation slabs is not a soil constant but based on settlement calculations. It is influenced by:

- Bedding of the soil and compressibility of its layers (M_E values)
- Outline of the foundation slab
- Loading

There is no formula for general situations. For simple cases however, like e.g. rectangular slabs and constant M_E value, you will find approximate formulas in the literature.

The following table out of „Bodenmechanik und Grundbau“ from Lang, Huder, Amann und Puzrin (Springer 2007) shows some common values:

Tabelle 11.9. Einige gebräuchliche Werte für den Bettungsmodul k_s

Bodenart	Bettungsmodul k_s in MN/m³ ca.
Torf, Humus	5– 20
Ton, weich	20– 40
Ton, plastisch	30– 60
Ton, steif	50– 90
Ton, sehr steif	100–120
Ton, sandig	80–100
Sand, dicht gelagert	80–100
Kies, fein mit Sand	100–120
Kies, mittel mit Sand	120–150
Kies, grob mit Sand	180–240
Kies, sehr dicht gelagert	200–300

B 8.8.2 Procedure for the bedding process

1) identify subgrade reaction moduli and elastic foundation zones define ('no tensile forces')
2) Determine the reinforcing effect of walls with upstanding beams (see Chap. B 8.7.2)
3) Define combined loading of all loadings (loads from columns and walls, local loads, possibly water pressure) at design level and provide for non-linear calculation (evtl. mehrere Alternativen)
4) Enter the limit state specification for the design ("either or" for all non-linear combination loads introduced in the only active action)
5) Define the design specification with the limit state specification

B 8.9 Columns and Piles

Column objects are important in three respects:

- for support conditions
- to investigate punching shear
- as self-weight in the top-down analysis of loads

Pile as Special Type of Column

Piles and columns share a common attribute dialog. In normal slab analysis (outside a building model) there is no difference between the two types of objects: both are vertical supports and need special punching analysis.

In a building model however, there are significant differences between a pile and a column. These are explained in D 3.5.1.

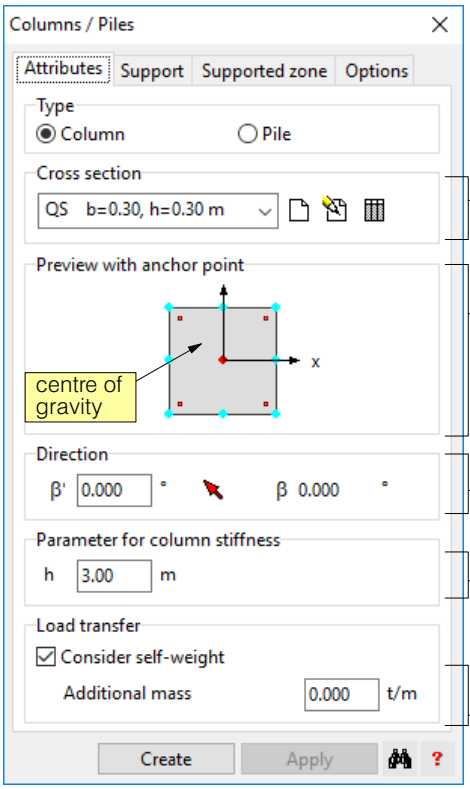
In the software and the manual both object types are generally called ‘columns’.

Layer Buttons

With the left layer button you can control the visibility of all columns and piles, while the right button controls the piles only.

They have the following dialogs, which are spread over 4 tabsheets:

B 8.9.1 Tab sheet 'Attributes' of the Column Dialog



Abbreviation: QS=cross section

QS choice and management (all QS defined in the current analysis appear in the selection field; if none have yet been defined, then the QS input dialog appears automatically, see below)

shows
- QS shape
- QS orientation x
- anchor point for the input (usually the centre of gravity of the QS)
- support area (if not a point support)

β': positioning angle
β: angle of 1st QS principal axis with respect to x'

h is used for the analysis of
- the column stiffnesses (slabs and building)
- for the weight of the column (for slabs, in the case of buildings through the difference in levels)

For non-vertical columns (only visible in building models)

'Self-weight taken into account' (not active in the case of buildings)

mass acting in addition to the mass (e.g. for cladding)

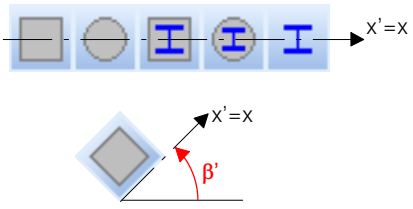
With [Create] a point is input (where the anchor point should be)

for selection of 'Pile':

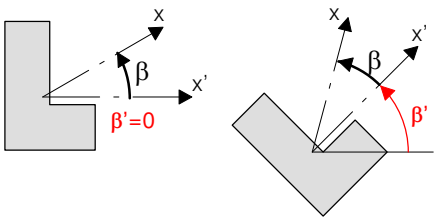
pile diameter (only circle cross section possible) and cross section name

Positioning angle β'

Standard cross sections



General FAGUS cross sections



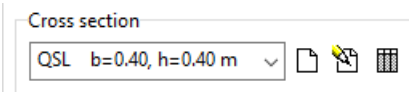
If β' = 0 the columns are positioned in such a way that their cross section axis x' is in the global X direction of the slab. If β' > 0 they are input in a direction with respect to this angle in the anticlockwise direction.

The above figure shows the x' direction for standard cross sections. In the case of general FAGUS cross sections the x' direction corresponds to the input direction y in the FAGUS program. The **principal direction x** coincides in the case of symmetric standard cross sections with x'. In general FAGUS cross sections it may be rotated about x' by an angle β.

Support directions

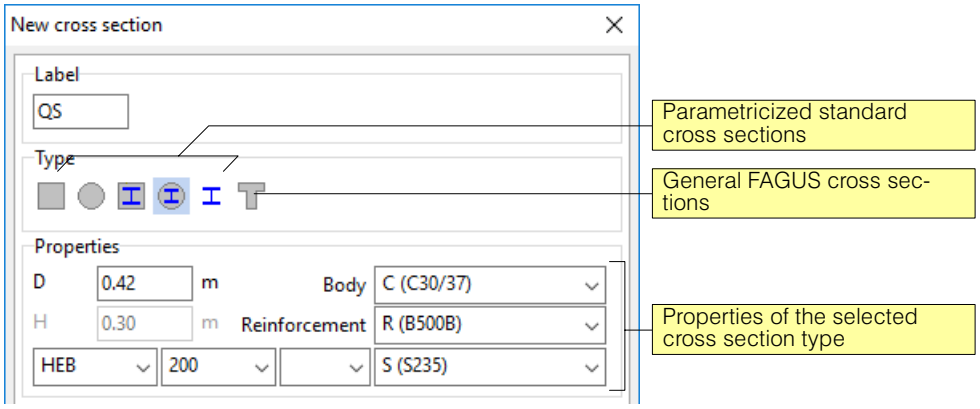
The direction of the support rotation RX and of the (for support not free to rotate) corresponding fixed-end moment MX of a column support points in the direction x, which, as shown above, is defined by the positioning angle β' and any principal axis rotated with respect to x' by the angle β. The direction of RY is normal to it.

Definition of column cross sections



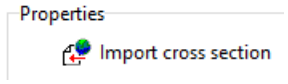
For a desired cross section to appear in the selection field of the dialog, it has to be defined by the user beforehand. With the three buttons for the management of the cross section next to the selection field new cross sections can be input, existing ones can be modified and the list of all existing cross sections can be displayed and managed.

The dialog to input/modify cross sections looks as follows:



Both the type and the corresponding properties can be changed.

A special case is given by the general FAGUS cross sections, in which the properties area of the dialog contains only a single button for the import of a cross section:

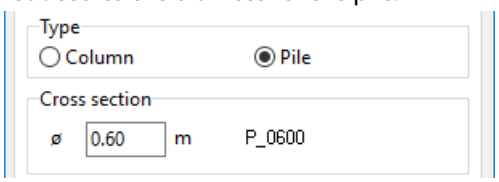


Clicking on the button causes the CubusExplorer to appear, in which one can select an analysis that contains FAGUS cross sections. From the cross sections that appear in a list, one can be selected for import.

An overview of the imported cross sections is provided by this dialog, in which cross sections can be modified, deleted, duplicated and imported. Only those cross sections can be deleted that are no longer referenced in the analysis.

Cross sections of Piles

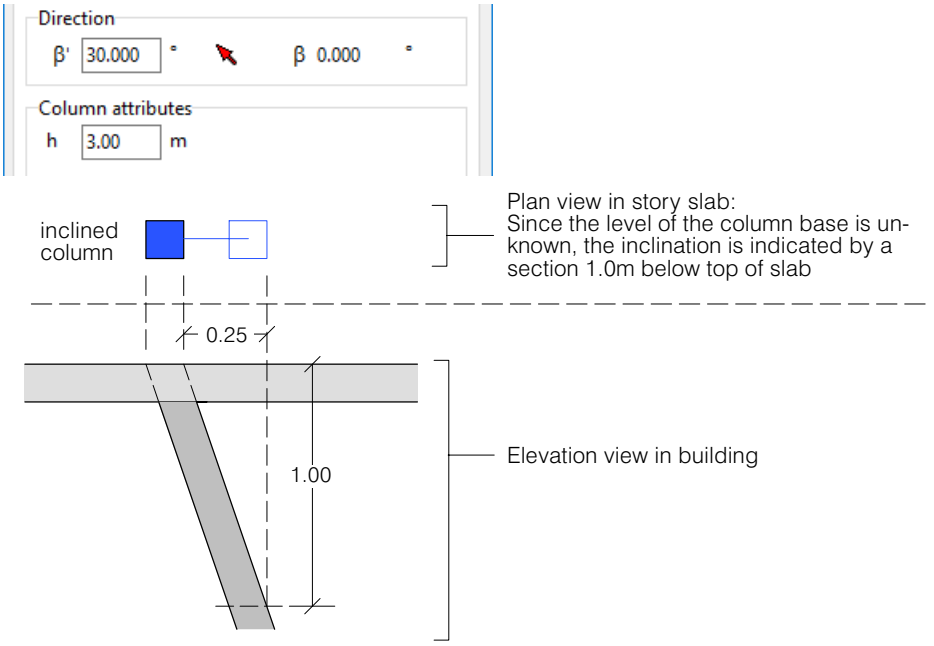
In contrast to columns, piles can have circular cross section only. The user input is therefore reduced to the diameter of the pile.



From the value of the diameter a specially named cross section is automatically generated and added to the list of sections: its name starts with 'P_' and ends with the diameter in [mm] (e.g. 'P_0800' for diameter = 0.8 [m]). All cross section of piles are mode from the concrete 'CP', which can be edited in the list of materials.

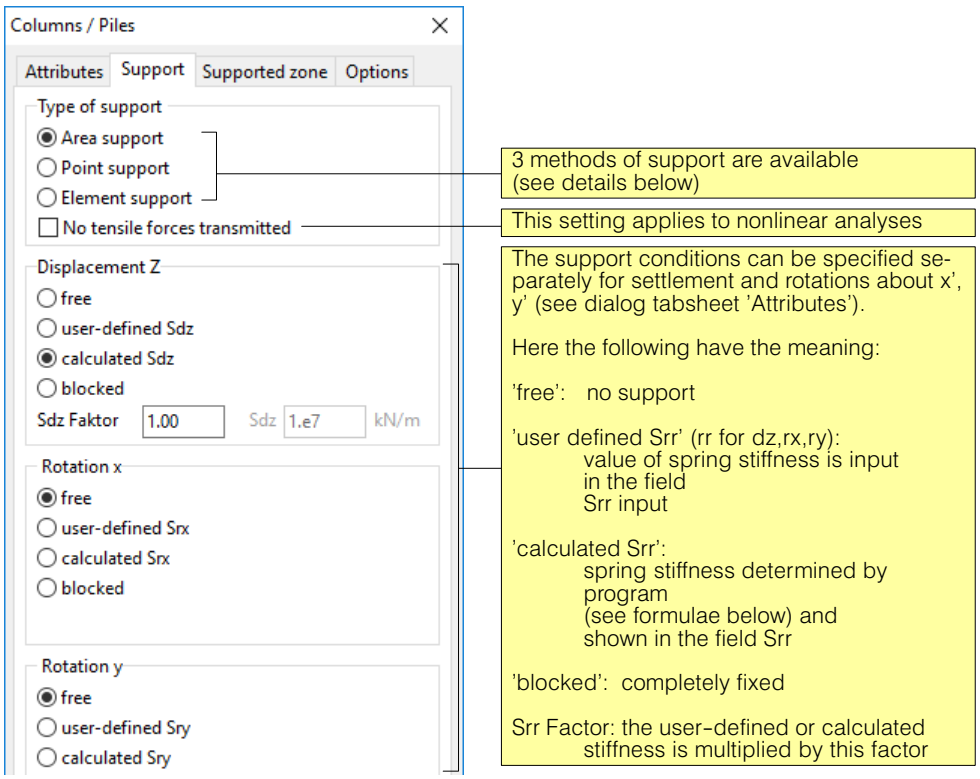
Inclined Columns

In a building model it is possible to define inclined (i.e. non-vertical) columns. The inclination can be defined in the local x- or y-direction of the column and corresponds to the horizontal out-of-pendula location per height. Note: An inclination is not allowed for piles.



Inclined columns are allowed in a building model only. Therefore the described input fields are not available in a norm slab input.

B 8.9.2 Tab sheet 'Support' of the Column Dialog



Settlement stiffness S_{dz}

The spring stiffness has the dimension [force/displacement]. It is determined by the program as follows:

$$S_{dz} = \frac{f_E EA}{h}$$

EA: normal stiffness of the column cross section; for composite sections $= \sum E_i A_i$
h: column height input in the 'Attributes' tabsheet of the column dialog
f_E: Sdz Factor (normally 1.0)

☞ *Columns normally have a higher grade concrete than slabs and walls and are also heavily reinforced. If this is not taken into account, there is a danger that their reaction forces are too small due to redistribution to neighbouring walls. This would lead to a dangerous underestimate of the risk of punching shear. If, in the analysis, one uses the slab concrete for the column cross sections, then their stiffness can be correspondingly increased proportionally by a factor* $f_E > 1.0$.

Spring support for the rotations of the tops of the columns S_{rx} und S_{ry}

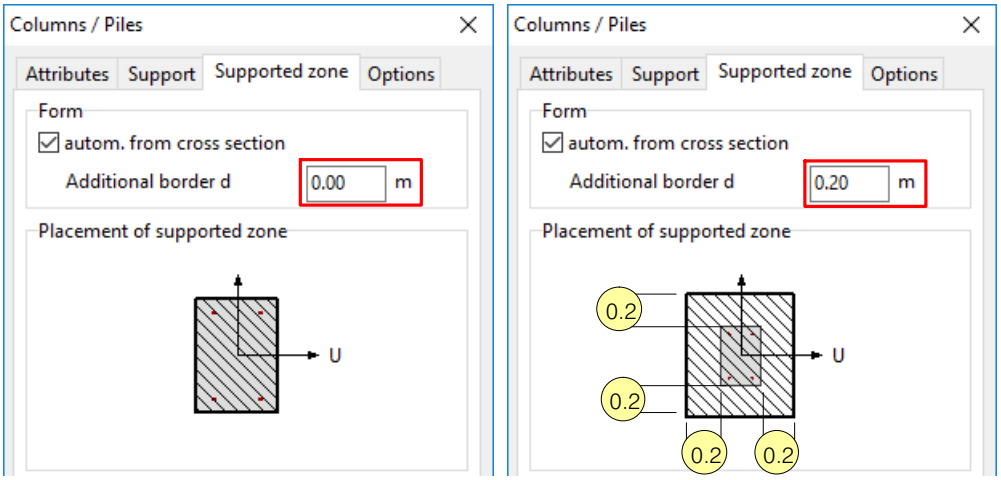
The spring stiffness has the dimension [moment/rotation]. It is determined by the program as follows:

$$S_{rx} = f_E \frac{4EI_x}{h} \quad ; \quad S_{ry} = f_E \frac{4EI_y}{h}$$ (corresponds to fixing the bottom of the column)

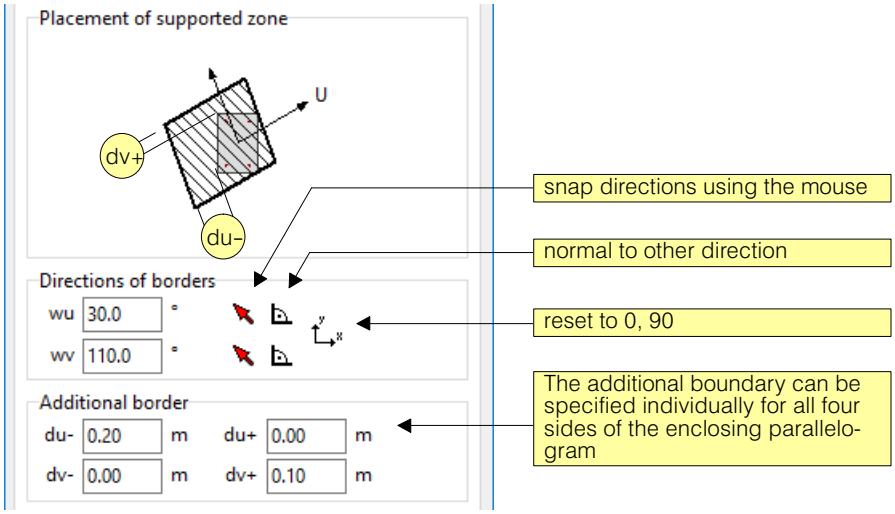
EI_x,EI_y: bending stiffnesses of the column cross section
h: column height input in the 'Attributes' tabsheet of the column dialog
f_E: Sdz factor (normally 1.0)

B 8.9.3 Tab sheet 'Supported zone' of the Column Dialog

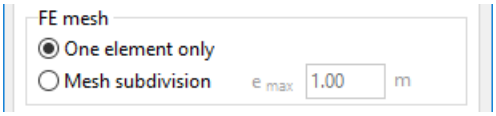
With the support models of type area and element support, the support area has to be input on which the slab above the column rests. It has either the shape of a rectangle of a parallelogram. If 'obtained automatically from cross section' and the value of the additional boundary equals 0.0 are specified, then the program chooses as the support area the rectangle enclosing the cross section in the input direction β' according to the 'Attributes' tabsheet. In the case of an input value for additional boundary, the rectangle is increased by the specified amount in all four directions:



If 'obtained automatically from cross section' is deactivated, then the support parallelogram can be defined in the dialog part:



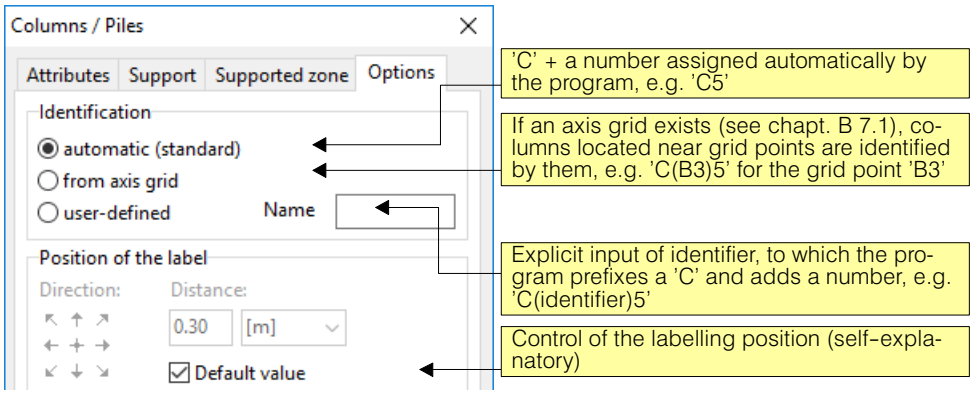
With the support model 'Element Support' the support area has a specific meaning. Its outline corresponds to the outline of an area-supported zone and is then a part of the FE mesh. One can choose whether an element should be derived from the support area or whether the area should be subdivided into several elements. This is specified at the bottom of the 'Area Support' tabsheet:



More information can be found in the description of the Element Support Model.

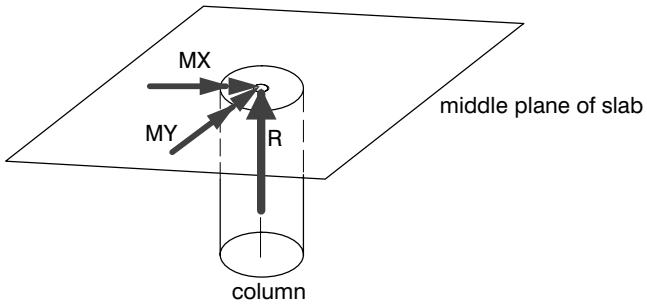
B 8.9.4 Tab sheet 'Options' of the Column Dialog

For identification purposes, in the various outputs the columns are provided with an identifier, with which they are labelled in the graphics. The possible settings are located in this tabsheet of the Column dialog:

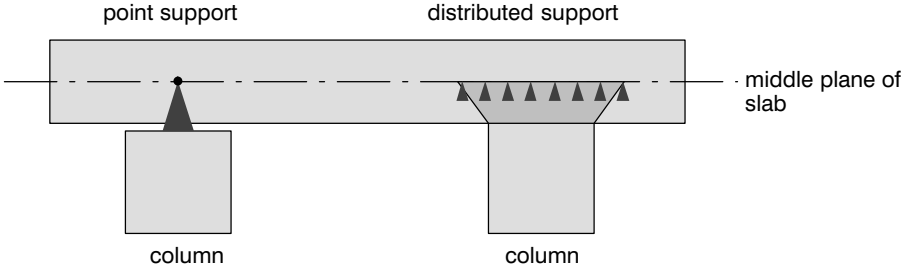


☞ *In the case of large outlines with regularly arranged columns it is recommended that one introduces a corresponding axis grid (see chapt. B 7.1) and in the above dialog for the column identifier 'according to axis grid' is specified.*

B 8.9.5 Support Models



From a column there is a transfer of a vertical reaction force R and, optionally, two reaction moments MX and MY. For the FE model the simplest way to proceed is to introduce a node in the middle plane of the slab at the point of action of the column axis and in this node the corresponding support conditions have to be formulated. This solution corresponds to the point support described below. However, it has two disadvanatges: Kirchhoff's plate theory, on which the FE model is based, cannot handle this case properly (singularities with concentrated load transmission), and the spread action zone of the reaction forces that in reality occurs is not taken into account, although it reduces the section forces.



CEDRUS provides three support models for columns, which are described in detail below.

Point-supported columns (= point supports)

According to Kirchhoff's plate theory the moments above a point support exhibit a singularity (infinite value). In the FE model this means that the peaks in the moments depend very much on the FE mesh and therefore are not of much use. The column dimensions do not enter into the model, i.e. there is no corresponding compensation. The concentrated spring to model the bending stiffness of a column is not compatible with Kirchhoff's plate theory in the sense that in the theory the plate would not be affected by this spring. In the FE model does have some influence, but depends greatly on the fineness of the mess.

In practice this model is only justified if the other models cannot be applied. This may be the case at the edge of the slab, if the cross section of a column overhangs the edge of the slab. At the edge of the slab, however, some of the abovementioned disadvantages are to some extent not relevant.

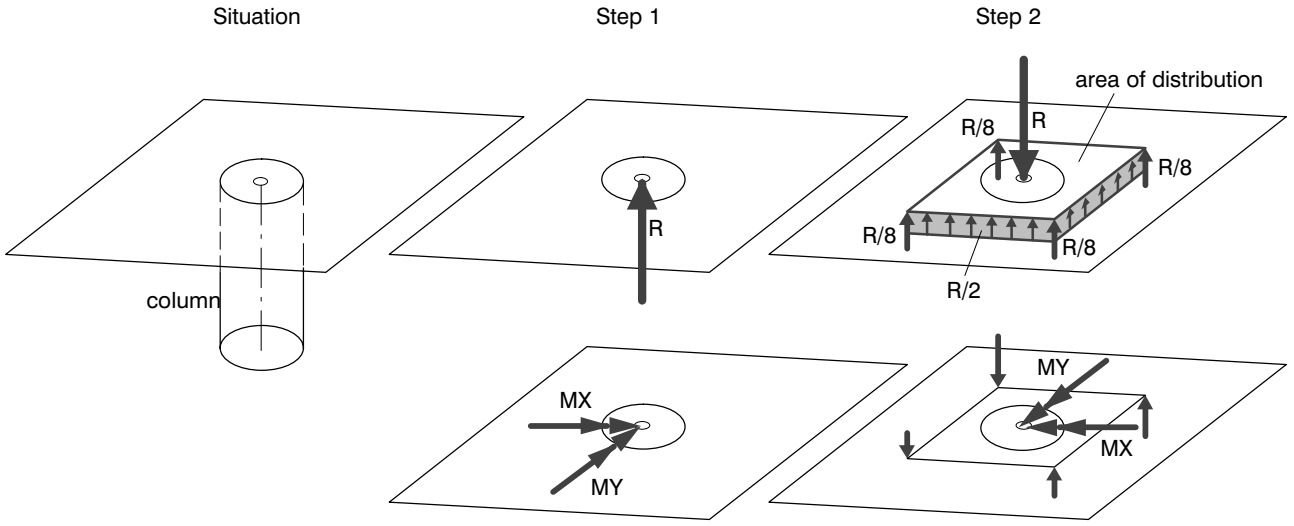
Area supports (= point support with area compensation)

The new model developed for CEDRUS has the advantages of the other two older models, without their disadvantages. The area support model therefore is recommended as the standard model to use.

With this model each loading is treated in two steps:

1) The loading is solved using a model with normal point supports. At every support the 3 reaction components RZ, MX, MY are obtained. If the slab can rotate freely over a column, the reaction moments MX, MY are zero.

2) A corrective loading is created which includes the loads over every column shown in the following figure. This corrective loading is added to the actual loading. Thus it involves a distribution of the column reactions that is similar to that with an area support and represents a residual stress state. The concentrated point support reactions of Step 1 are thereby neutralized.



Reactions for the model with point supports

Corrective loading in terms of residual stress state (forces in equilibrium)

The area of distribution is a rectangle or a parallelogram whose side directions can be specified. The size can be chosen by the user, with the condition that the centre of gravity of the column must lie within the area of distribution.

As is clear from the above table, this column model is usually advantageous, and thus it is suggested by the program as the standard one. The other models should only be used in special cases.

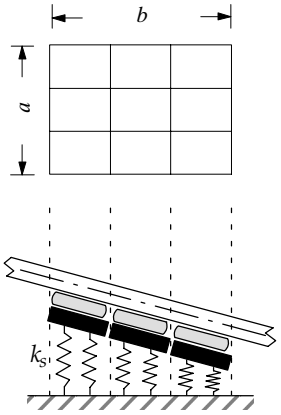
Element supports (= area supported elements)

Element-supported columns are modelled for the FE model with area-supported elements. The support area specified in the Column dialog in the tabsheet 'Support Area' forms an area-supported zone. The user has to decide if this zone is made up of a single finite element or if it should be distributed over several elements.

Element support with a single element is appropriate if a column does not have to resist moments. If, nevertheless, a bending stiffness is specified, then this is distributed automatically by the program in the form of rotational springs at the four corner nodes.

Element supports distributed over several elements cannot be avoided if their sides are subdivided by other geometrical input objects like material separation lines. They may also be desirable if one wants to have a fine FE mesh in the area above the column. But the subdivision into several elements means that such a column always exhibits a certain amount of bending stiffness, since each element acts as an independent support spring. With the value k_s for the modulus of subgrade reaction of the supported elements, the bending stiffness resulting from the area support for a rectangular column (see adjacent figure) can be estimated approximately as (requirement: the slab remains plane in the region of the column support):

Column support spread over several elements



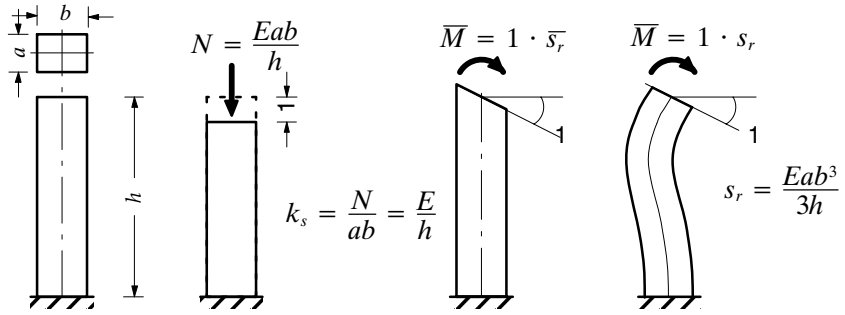
$$\overline{s_r} = \frac{ab^3}{12} k_s$$

If one determines k_s from the amount of settlement of the column (see following figure), then assuming that the support area is identical with the rectangular column cross section:

$k_s = E/h$ and thus $\overline{s_r} = \frac{Eab^3}{12h}$.

If the column cross section differs somewhat from the shape of the supported zone, the stiffness considerations can be adjusted accordingly

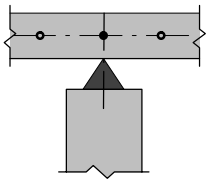
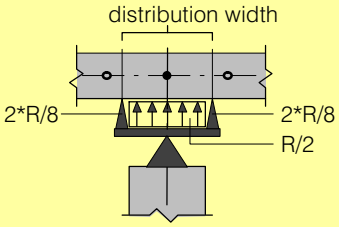
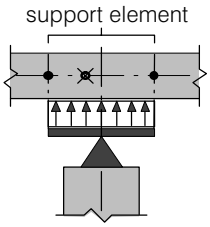
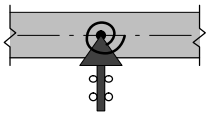
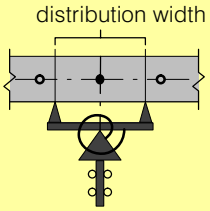
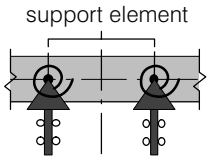
A column of the same dimensions fixed at its base (column on the right in the figure), on the other hand, exhibits a bending stiffness that is four times higher :



Since the stiffness may not be less than $\overline{s_r}$ this value represents a minimum input value. Higher stiffnesses, as in the case of a support with just one element, are dealt with by introducing rotational stiffnesses distributed uniformly over all the corner nodes of the column.

Element supports have scarcely any advantage over area supports and are mainly included to maintain compatibility with earlier versions of the program.

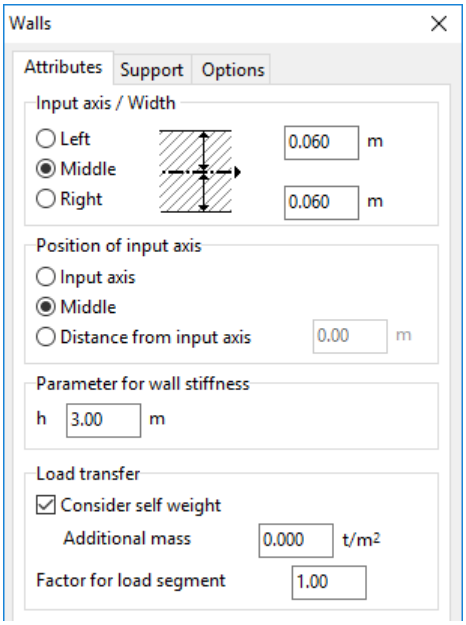
The following figure provides a concentrated summary of the support models available in CEDRUS:

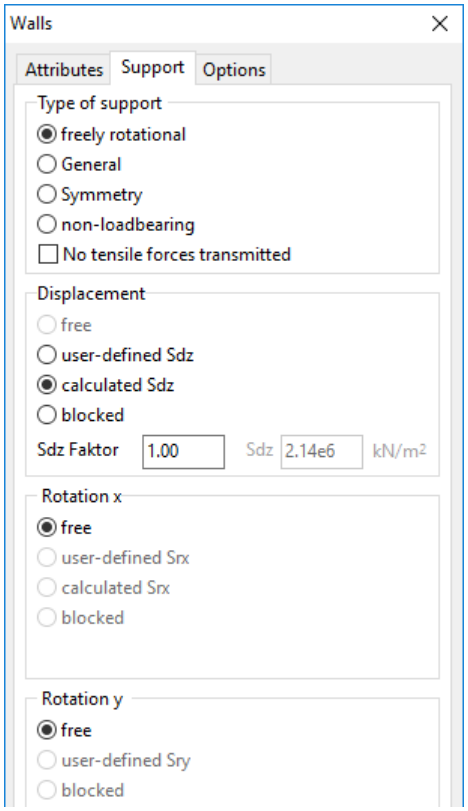
	Point support	Area support	Element support
column cross section	rectangle / circle / steel section / arbitrary FAGUS cross section		
support without bending stiffness with specified settlement stiffness FE mesh nodes: • obligatory ○ allowed ⊗ not allowed		 R/2 as constant reaction pressure R/2 concentrated in the corners	 constant reaction pressure (area-supported element)
takes bending stiffness into account			
⊕ advantages ⊖ disadvantages	⊕ no geometrical boundary conditions mesh point only in the axis ⊖ moment peaks (singularities with Kirchhoff's theory) ⊖ no area compensation	⊕ no geometrica boundary conditions mesh point only in the axis ⊕ no problems with singularities (moment peaks) ⊕ with area compensation ⊖ slightly greater computational effort (practically insignificant)	⊕ no moment peaks ⊕ with area compensation ⊖ condition for mesh, cannot be made finer ⊖ poor model for bending stiffness ⊖ column connection unsuitable for 3D model

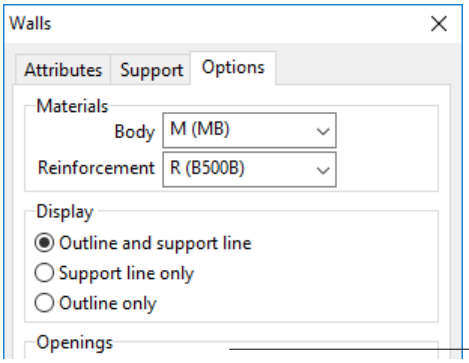
As can be seen from the table above, this support model normally has only advantages, and it is therefore standard proposed by the program. The other models should only be used in special cases.

B 8.10 Walls / Line Supports

Wall objects are double line polygons, which define the outline of the walls. They have the Attributes dialog shown below.







Part 'Openings' see below

In the FE model walls are introduced as line supports whose axes can be freely chosen within the given walls. These support axes are given lines in the FE mesh and are subject therefore to the conditions of minimum distance between each other and other structure objects. The wall outline only serves the purpose of visualisation and is not subject to any geometrical consistency conditions.

The settlement stiffness s_{dz} and the bending stiffnesses s_{rx} and s_{ry} of the walls are defined per unit length of wall 1 and can be input manually or determined by the program for the given wall thickness t, wall height h and modulus of elasticity E of the material. They are defined as follows:

s_{dz} $= Et/h$ (force per unit settlement 1)

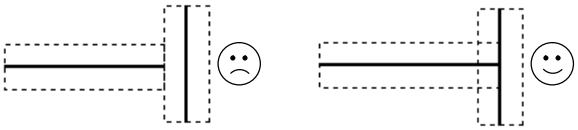
s_{rx} $= Et^3/3h$ (moment for a unit rotation 1 about the support axis)

s_{ry}: $= ?$ (moment for a unit rotation 1 normal to the support axis)

For s_{ry} the program does not determine any stiffness. This part of the stiffness may normally be freely chosen. Blocking it would mean that the settlement line along the wall at each node would have a horizontal tangent. In the case of a fine FE mesh the wall therefore can only settle uniformly as a rigid body, which is not what one normally wants.

B 8.10.1 Modelling Recommendations

- Blocked settlements are a poor modelling of reality and lead to very mesh-dependent and unrealistic reaction distributions with large erratic peaks. Normally, it is better to use the model 'Wall Stiffness'.
- It is important visually to have a clear picture with as many seamlessly connected wall outlines as possible. Attention should be concentrated on the support lines and the corresponding FE model:



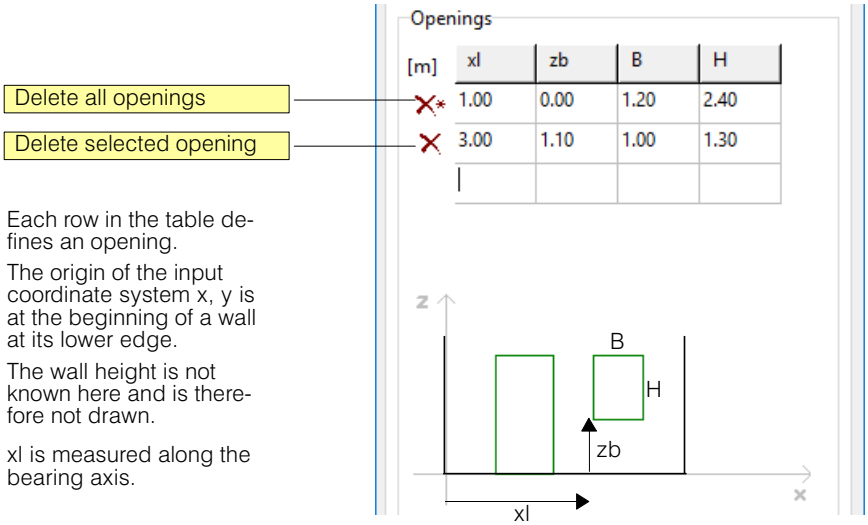
connect support lines as much as possible!

B 8.10.2 Direction of Wall Axis for Output of Reaction Moments

Reaction moments are vectors lying on the wall axis and pointing to the end of a wall segment. The end of a wall segment always has a bigger global X-coordinate than the start of the segment unless it is parallel to the Y-axis. In this case its end points to the positive Y-direction.

B 8.10.3 Wall openings

In order to obtain more precise wall weights and dimensions for building calculations, rectangular openings can be entered in the tab sheet options. The wall openings are defined in the local x-z coordinate system of the wall plane and are entered in a table in the tab sheet options of the wall dialog:



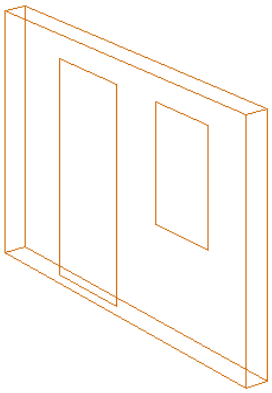
Each row in the table defines an opening.

The origin of the input coordinate system x, y is at the beginning of a wall at its lower edge.

The wall height is not known here and is therefore not drawn.

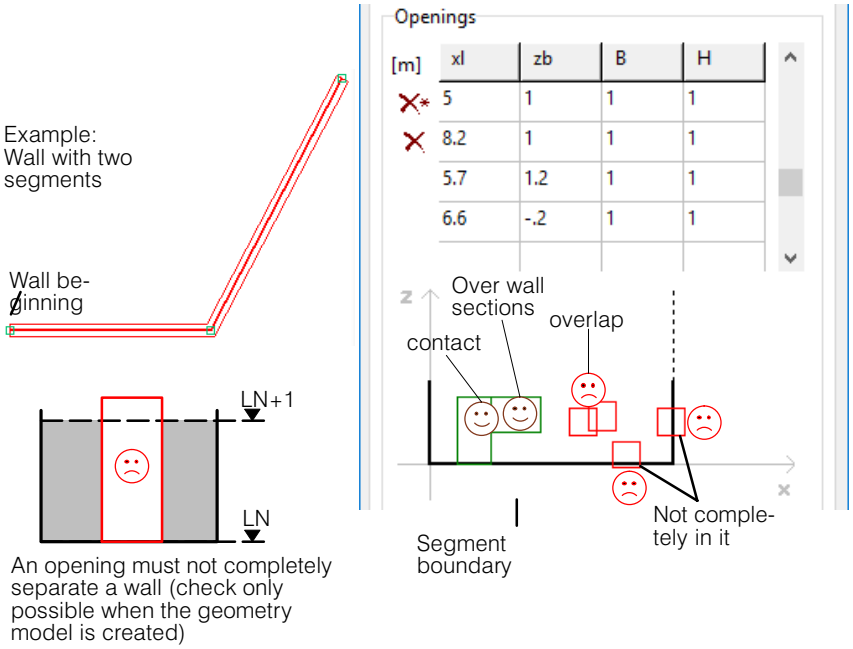
xl is measured along the bearing axis.

View in the building view



Each opening corresponds to a row in the table where the x- and z-coordinate of the lower left corner and the width and height are specified. The z-axis has its origin at the base of the wall and points upwards, while the x axis starts at the beginning of the wall and follows the wall bearing line. In the case of polygonal walls, the x-axis can have a kink on the wall corners. The entered openings are displayed in a preview. The preview shows the entered openings, the horizontal wall borders and the wall corners in the x-z wall plane.

Openings must be completely within the unwound wall plane and must not overlap. Incorrectly entered walls are drawn red in the preview.



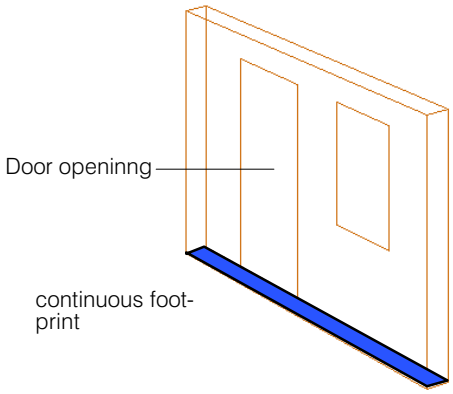
The corresponding line in the table is selected by clicking with the left mouse button within a drawn opening.

☞ *Wall openings are only possible in storeys of buildings, which is why the input fields described above are not visible in the wall dialog for simple slab calculations.*

Influence of the wall openings on the static models

Stacked slab model

Wall openings have an influence on the stacked slab model only insofar as the dead load and mass are correspondingly reduced. They have no influence on the static model, however. The footprint of the wall, which determines the load transmissionis not affected by a door opening:



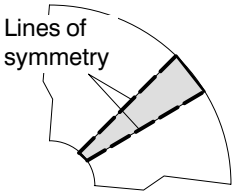
Space frame model

All member stiffness components of pillar with openings are reduced according to the opening-induced mass reductions. The stiffness reduction factor defined as the default for all pillars in the option dialog for the space frame model (see chap. E 3.3) is cumulated with the opening-induced reduction factor.

☞ *The stiffness reduction factors used in the generation of the STATIK model can be viewed in STATIK and, if necessary, modified.*

B 8.11 Lines of Symmetry

Lines of symmetry are special line supports, along which the slab is able to deflect, but cannot rotate about this axis. Symmetry conditions are of practical meaningfulness at slab edges, which is why lines of symmetry are also only allowed on outline lines.



Lines of symmetry are used to limit the modelling to parts of of slabs, whether due to genuine symmetries or by using a symmetry condition to obtain the best possible boundary condition for this purpose. One should note that for symmetry conditions the loads also have to be symmetric in their action.

An example of genuine symmetry is a circular plate - even if with the condition of a symmetrically acting load it is rather academic. Here it is sufficient to have the input of a segment with the corresponding symmetry conditions along the radial boundaries.

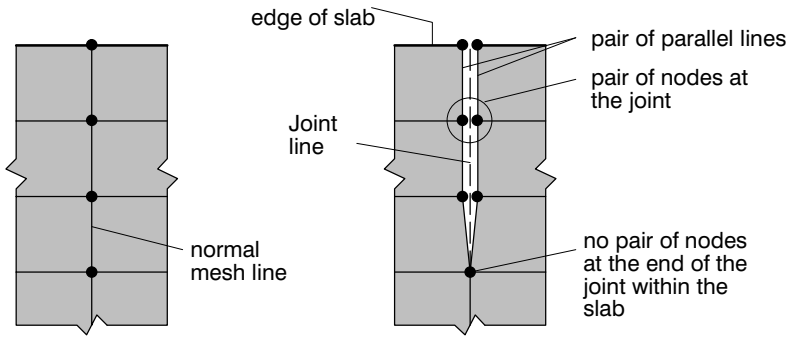
Lines of symmetry are not allowed for walls. If a wall is to act as a line of symmetry then the rotation about its support axis can be blocked. The user is responsible for halving the settlement stiffness.

Point supports on lines of symmetry are permitted. Their x direction, however, must coincide with the direction of the line of symmetry. The program then blocks the rotation about the x axis, but does not reduce the stiffnesses due to the symmetry. It is the responsibility of the user to specify the corresponding value. A simple possibility is to halve the modulus of elasticity of the column.

Point supports at the intersection of two lines of symmetry are also permitted. If they form a 180° degree angle then it is as if the line of symmetry were continuous (see previous paragraph). Otherwise the input direction of the column is of no importance and the program blocks both rotations. The user is once again responsible for the correct input of the settlement stiffness .

B 8.12 Joints

Joints are lines along which a slab is cut up. In this case the mesh nodes on the line of the joint are introduced as pairs of nodes, so that in one point there are always two opposite nodes on the pair of dividing lines.

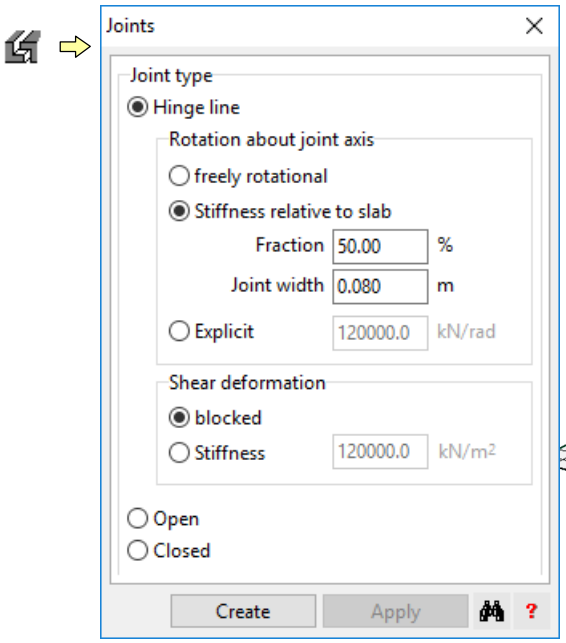


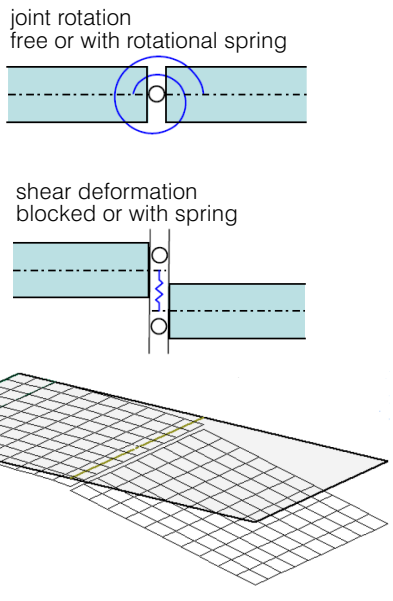
The program connects the pair of nodes according to the chosen type of joint. The following joint types are available:

a) **Hinge line** :
Different types of connections between the pairs of nodes are possible:
1) equal displacement Z (): makes a hinge line
2) additional rotational spring: makes an elastic hinge line
3) like 1 or 2 , but displacement Z not rigid but connected with a spring

b) **Open:**
No connection, i.e. the slab is completely separated along the joint line.
Restriction in building module: Separation lines don't influence the STATIK model, i.e. separated parts remain rigid concerning the membrane degrees of freedom.

d) **Closed:**
The pair of nodes are rigidly connected with one another, i.e. there is in fact no joint. This joint type can be useful when experimenting with different joint types and one can by this means profit fom the special results output for joints.





The stiffness of a rotational spring can be specified either directly as moment per unit length of the joint for a rotation of 1.0 [kNm/m/rad] or by defining a joint strip of a given width and stiffness (fraction in % of the rotational plate stiffness).

Joints are given lines in the FE mesh and thus they are subject to the conditions of minimum distance between lines and other structure objects.

B 8.12.1 Limitations

- Several joint lines can be joined in series, but **without forming branches**.

- Joint lines are not permitted at the edge of a slab (outline and openings).
- If a joint line lies exactly over a support, then depending on the support it is not defined which line of the pair of dividing lines has to be supported. For this reason joints are not permitted in the following cases:
  - hinge lines and spring hinge lines over supports with bending stiffness
  - dividing lines over supports
- Since, in the case of hinge lines with springs, the orientation of the nodes on the joint line has to correspond to the direction of the joint line, conflicts with supports may arise. For this reason joints are not permitted in the following cases:
  - spring hinge lines over point supports
  - spring hinge lines may not intersect with line supports including their end points
- Restriction in building module: Separation lines don't influence the STATIK model, i.e. separated parts remain rigid concerning the membrane degrees of freedom.

B 8.12.2 Loads on Joints

In the case of loads over joints, depending on the joint and load types, it is important to know on which line of the pair of parallel lines of the joint the load has to act (for example point load on dividing line). In all cases in which this is not clear a priori the load is halved and distributed between the two parallel lines. If one has to have the load on one side of the joint, the load can be moved correspondingly or placed unambiguously.

B 8.12.3 Results Output for Joints

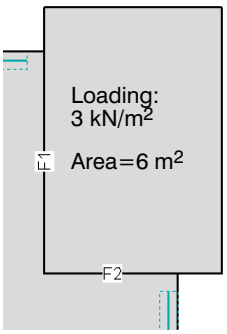
Joint lines, in the same way as sections, can be used directly for the results output. One can obtain **shear forces** and **moments** per unit length normal to the joint line.

Each joint line can be subdivided into sections of equal length, based on a given approximate section length. The results are then output for each section with constant mean values.

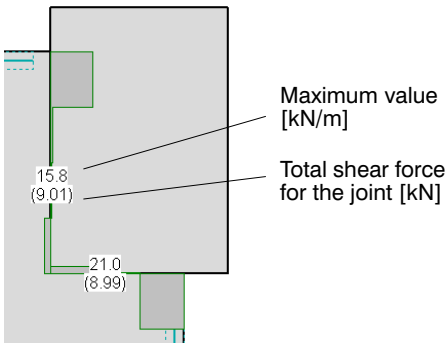
Comments on the results for joints

- No joint results are available at end points of joint lines that lie in the interior of a slab. For short joint lines ending in the interior of a slab it is important to make sure that the element mesh is fine enough. There must be several nodes along such a joint line to get acceptable results.
- The joint results are not determined in the same way as the section forces and the section results that depend on them, but the element nodal forces along the joint are used. This means, for example, that the integral the shear forces along a joint, which runs from one edge of the slab to another, is exactly the same as the sum of all loads and reactions acting on one side of the joint. Normally this is not the case.

Example:



Beam separated by the two joints F1 and F2



Shear force results:
Sum = 9.01+8.99=18.00
(corresponds exactly to the loading)

B 8.13 Remaining Buttons in the tab sheet 'Structure'

B 8.13.1 Legend of the Structure Data

All the structure data is recorded in tabular form.

B 8.13.2 Checking Structure Input

After completing the structure input it can be checked for errors.

B 8.13.3 Revert to Previous Structure

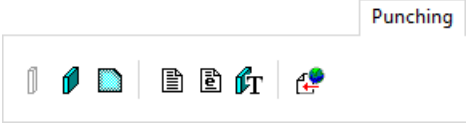
Reverts the structure to the unchecked state and deletes any calculated results. The entries in the CubusViewer, however, are not changed.

B 8.13.4 Import of Structure Data

CEDRUS can import structure data from different sources. The possible formats are available here.

B 9 Punching Shear

Punching shear objects are managed, modified and their punching shear resistances are determined in the tab sheet 'Punching'.



The input for the 'Punching Shear' tabsheet is described in chapt. B 9.2, and the import of punching shear results from other programs in chapt. B 9.3.

B 9.1 Treatment of Punching Shear in CEDRUS

With the aid of the punching shear verification one checks whether punching through the slab of columns, wall ends and corners or other concentrated forces is possible. The verification is positive (i.e. safe result) if the following condition is fulfilled:

$V_d \leq V_{Rd}$

V_d : dimensioning value of punching shear load
V_{Rd} : dimensioning value of punching shear resistance

The punching shear verification in CEDRUS is carried out in the following steps:

1. For each object endangering punching shear (columns, wall ends or corners, concentrated loads) a so-called **Punching Shear Object (PSO)** has to be activated or created, so that CEDRUS can carry out a punching shear verification for it. Each PSO contains all the necessary parameters for the verification, such as the load or reaction areas, the control perimeter, etc.
2. Thus the punching shear resistances can be determined and output in tabular form, together with all the parameters used.
3. On the basis of the chosen load superposition method, finally CEDRUS also calculates the punching shear loads and compares these with the resistances in the Punching Shear Verification Table.

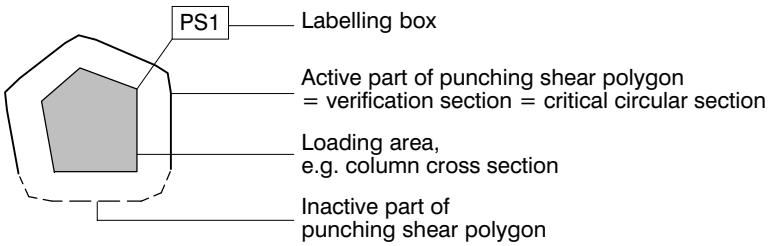
The first two steps are completed in the 'Punching Shear' tab sheet described here and Step 3 is carried out in the 'Results' tab sheet.

CEDRUS also permits the integration of dimensioning programs for punching shear strengthening of the corresponding manufacturers.

B 9.1.1 Punching Shear Objects (PSO)

Punching shear objects comprise the following elements:

- **Load application area:** closed polygon
- **Punching shear polygon:** closed polygon, consisting of active and possibly inactive parts. The active part of the punching shear polygon is the **Verification Section**.
- **Labelling box** with configurable content

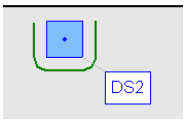


One differentiates between the following two types of verification:

- **Verification type Column:** The risk of punching shear comes from below from column and wall reactions
- **Verification type Load:** The risk of punching shear comes from above as a concentrated load, e.g. in the case of columns on foundation slabs.

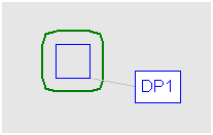
B 9.1.2 Input of Punching Shear Objects

To simplify the input, all columns and wall points always possess automatically a “sleeping” PSO, that only has to be activated and possibly modified. For the verification type Load and in special cases with the verification type Column a so-called free PSO must be created manually. The program recognizes three types of PSO:



Column PSO

Wall point PSO



Free PSO

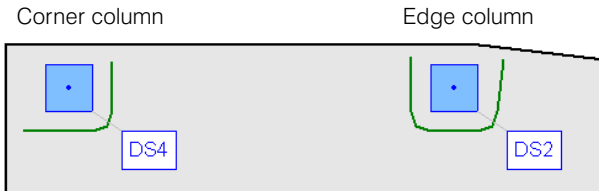
⚠ **Important limitations of automatically created PSO:** The program cannot always make a meaningful suggestion for the punching shear polygon (complicated slab boundary geometries, zones with different thicknesses, etc) and downstands are also not taken into account, so that **the proposed solution has to be checked in each case and if necessary corrected manually**. For this purpose, in the determination of the punching shear polygon one can choose the setting **'automatic' <> 'manual'** in the Properties dialogs of the PSO.

Column PSO

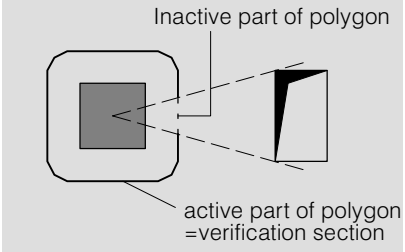
In the 'Punching' Shear tabsheet of CEDRUS instead of a column one sees a PSO. The punching shear polygon is only shown if the PSO is activated:

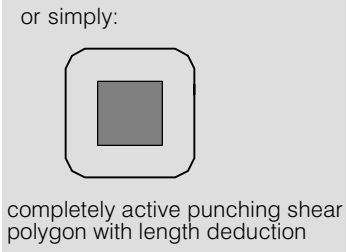
Inactive column PSO Activated PSO

The program tries to recognize whether it is a case of an inner, edge or corner column. If at a certain distance from the edge of the column no boundaries or openings are encountered, the column is assumed to be an inner column. Otherwise the opening angle of the punching shear polygon between the slab's entrance and exit provides a criterion to distinguish between edge and corner columns. The maximum length of control perimeter u is limited, depending on the code. It is compared with the length of a standard edge or corner column and, if necessary, trimmed back from the edge.



Deductions for openings, as sketched below, are not carried out by the program automatically. They can be either obtained by construction or simply input directly as a length deduction for the verification section.

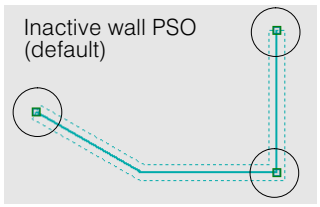


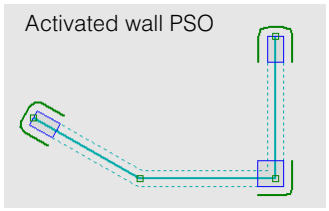


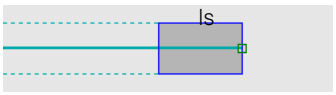
Wall PSO

In the 'Punching Shear' tabsheet wall points with inactive PSO are marked with a symbol. Activated PSO show their load areas and punching shear polygons:

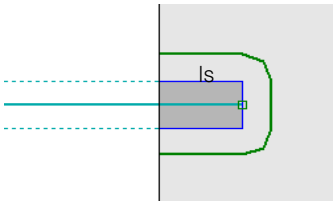
The wall PSO at the ends of the wall are, if possible, created by the program in two steps according to the following schematic:





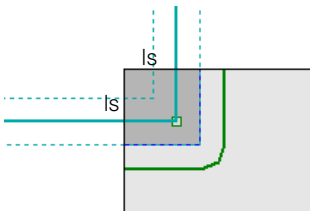


1) Introduction of an equivalent column at the wall end with the length ls or at the wall corner with ls x ls



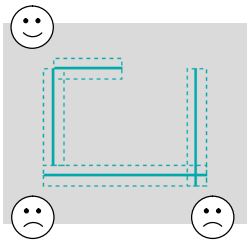
2) In the case of wall ends:
The extended column side within the wall is assumed to be like a slab edge and the punching shear polygon is constructed like an edge column.

In the case of wall corners:
The extended column sides within the wall are assumed to be like a slab edge and the punching shear polygon is constructed like a corner column.



Remarks:

- ls is code-dependent; with SIA262 ls = 1.5d (d = static depth of slab).
  However, it is at least as large as the wall thickness and can be changed by the user.
- The critical reaction force is given by the line support parts, which are located within the equivalent column. If, in the case of wall corners, the support line is located inside the wall, then ls may have to be extended.
- The punching shear load is the critical reaction force after deducting all loads within the punching shear polygon.
- In the case of obtuse-angled wall corners or special geometrical conditions the program does not introduce any PSO in the corresponding corner and end points. If required, one can introduce in such points a free PSO.
- In special cases a PSO is introduced, but it must be checked explicitly by the user and if necessary adjusted.



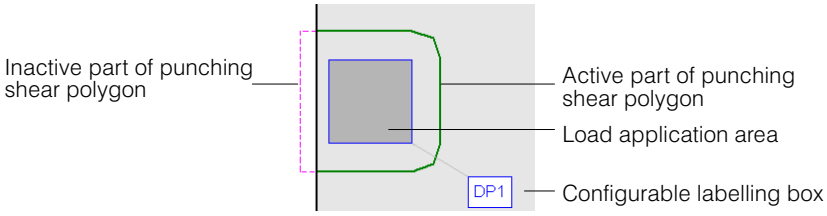
- If the ends of two different wall objects meet at a point, their support lines must have a common end point. Otherwise a PSO is not created.
- The code gives no specific information what one has to do in the case of wall ends and corners. Models that are automatically introduced by the program always have to be checked by the user.

Free PSO

If one wants to carry out the punching shear verification for a concentrated load, or for reactions, where no automatic PSO exists, a free PSO must be input. It consists, as other PSO, of the following elements:

B 9.1.3 Punching Shear Resistance

In most codes the punching shear resistance is given by the total shear force resistance along a so-called control perimeter u around the punching shear load. Thus the punching shear resistance depends mainly on u, the slab thickness and an “admissible” shear stress and is based on the concrete quality. The geometrical conditions can already be determined when the structure is input.



$$V_{Rd} = f(u, \tau, d_m)$$

u : length of the control perimeter

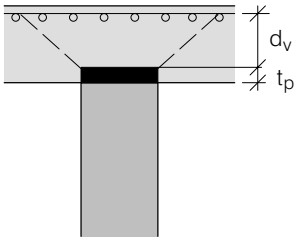
τ : characteristic value of the limit shear stress

d_m : static depth (mean)

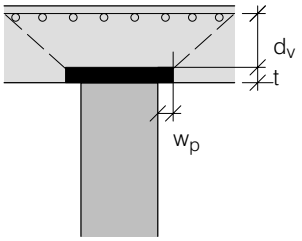
There are also further conditions, which are described for the individual resistance models.

Influence of a column head plate

A column head recessed in the ceiling reduces the static height. An overlap w_p of the head plate increases the length of the critical round section.



Weakening of the slab by an inner connection plate



Weakening of the slab, enlargement of the critical round section

Models to determine the resistance

For each PSO the model for the determination of the punching shear resistance can be chosen as follows:

0) PSO inactive: no punching shear verification is performed (default).

1) control perimeter (without punching shear reinforcement)

2) control perimeter with punching shear reinforcement

3) resistance is given

The models are described below. But first we want to comment on the required bending resistances and on the correction values for the punching shear resistances specifiable by the user.

Required bending resistances

In the case of the abovementioned Models 1 and 2, according to the code for the punching shear resistance to be achieved there are minimum conditions for the bending resistances. In all codes supported by CEDRUS the required bending resistance m_{Rd} depends on the existing punching shear load V_d:

$$m_{Rd} \geq \eta \, V_d$$

m_{Rd} : minimum required dimensioning value of the bending resistance allowing the required shear capacity to materialize

η : moment factor, depending on type of column - inner column, edge column, corner column, η = 0.125 .. 0.5)

In CEDRUS this specification is reformulated, in that with the user-defined reinforcement content ρ firstly a bending resistance and then from it the dimensioning value for a maximum punching shear load $\boldsymbol{V_{d,max}}$ is calculated:

$$V_{d,max} = \frac{m_{Rd}}{\eta}$$

The bending resistance m_{Rd} is estimated assuming a rectangular concrete stress distribution as follows:

Reinforcement area: $A_s = \rho\, b\, h$ (b: width of slab strip, h: slab thickness)

Thickness of compression zone: $x = \frac{A_s f_y}{b f_c}$ (rectangular compressive stress distribution)

$m_{Rd} = (d_m - \frac{x}{2}) A_s f_{sd}$ (f_{sd}: < resistance factor for steel, d_m: mean static depth)

In the case of skew boundaries and/or for rotated output directions the plastic resistances in the corresponding directions parallel and normal to the boundary are transformed.

Correction values for the punching shear resistances

For all models there is in addition a correction value $\boldsymbol{V_{Rd}^+}$, with which the user, depending on the model, can subsequently correct the determined or input dimensioning value of punching shear resistance either up or down. One possible application would be the resistance correction for prestessing cables. If, however, the prestress loading with the deviation forces comes in the load specification for the punching shear verification, the influence on the punching shear load is automatically taken into account and no correction is necessary.

Model 1: control perimeter

With this model the punching shear resistance is given solely by the resistance along the control perimeter. It is denoted by $\boldsymbol{V_{Rd1}}$. Thus the critical dimensioning value of the punching shear resistance is given by

$V_{Rd} = \min(V_{Rd1}, V_{d,max}) + V_{Rd+}$

Model 2: control perimeter with punching shear reinforcement

The punching shear resistance for the existing shear reinforcement depends on the code and the product and is determined according to the manufacturer's data. The corresponding value denoted by $\boldsymbol{V_{Rd}'}$ must be input by the user. To limit the concrete stresses according to the code there is a limit value of the punching shear resistance, which is determined by the program and is denoted by $\boldsymbol{V_{Rd2}}$:

SIA262: $V_{Rd2} = 2.0\, V_{Rd1}$
EC2: $V_{Rd2} = 1.6\, V_{Rd1}$ (1.6 = country-specific value)

Thus the critical dimensioning value of the punching shear resistance is

$V_{Rd} = \min(V_{Rd}', V_{Rd2}, V_{d,max}) + V_{Rd+}$ where $V_{Rd}' \geq V_{Rd1}$

Model 3: resistance is given

In the case of strengthening in the form of **steel constructions** the resistances ***VRd'*** supplied by the manufacturer are input in the program. Here there are no special limitations.

Thus the critical dimensioning value of the punching shear resistance is

$V_{Rd} = V_{Rd}' + V_{Rd+}$

B 9.1.4 SIA262 - Specifics

$$V_{Rd,c} = k_r \cdot \tau_{cd} \cdot d_v \cdot u \qquad \text{Eq. (57)}$$

u : Length of the control perimeter (at a distance d/2 from the supported surface), reduced by the factor k_e (eq. 56) for taking into account the moment transfer from the column to the slab

τ_{cd} : Design value of the shear stress limit
d_v : Effective static height (d_x+d_y)/2
k_r : Coefficient acc. to equation 58

$$k_r = \frac{1}{0.45 + 0.18 \cdot \psi \cdot d \cdot k_g} \quad \leq 2 \qquad \text{d in [mm]} \qquad \text{Eq. (58)}$$

$$\psi = 1.5 \cdot \frac{r_s}{d} \cdot \frac{f_{sd}}{E_s} \cdot \left(\frac{m_{sd}}{m_{Rd}}\right)^{3/2} \qquad \text{Eq. (59)}$$

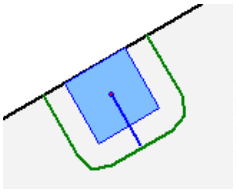
ψ : Slab rotation according to the above formula
d : static height
k_g : Coefficient for consideration of the maximum grain size (acc. to paragr. 4.3.3.2.1)
r_s : Distance between PSO and zero point of moment in the direction considered (x or y)
m_{sd} : Mean value of the bending moments on the colunm strip width
m_{Rd} : Mean value of the bending resistance by the column strip width

For the determination of y, i.e. of rs and msd, three approximation stages are provided in the code, of which stages 2 and 3 are implemented in CEDRUS.

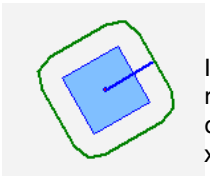
Approximation level 2 (applicable with regular flat slabs with spans of 0.5 to 2.0):
m_{sd} is estimated according to pararaph 4.3.6.4.7 on the basis of the punching load and its eccentricity. r_s can be set to 0.22 * span.

Approximation level 3 (if 2 is nat applicable):
r_s and m_{sd} are determined by a elastic slab calculation according to paragraph 4.3.6.4.8.

The directions referred to as the punching directions below x ', y' correspond with internal columns the output directions x, y (= reinforcement directions). In the case of boundary columns, x' is assumed to be perpendicular to the boundary and, in the case of corner columns, parallel to one of the boundaries. The punching direction x' assumed by the program is displayed in the tab sheet 'Punching'.



Boundary column: punching direction x' perpendicular to boundary



Internal column with rotated output directions x'=x

The determination of ψ is carried out in the punching directions x' and y', wherein the larger value is decisive.

Slab thickness and static heights are taken from the values entered in the material box (slab thickness, edge distances).

The geometrical reinforcement contents (ρ_x, ρ_y) and, in the case of approximation level 2, the two spans $(l_{x'}, l_{y'})$ and the eccentricities ($e_{ux'}$, $e_{uy'}$) must be entered for each punching object.

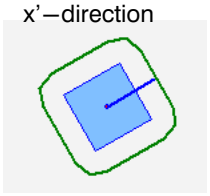
⚠ The reinforcement contents are defined in the output directions and the spans and eccentricities in the punching directions.

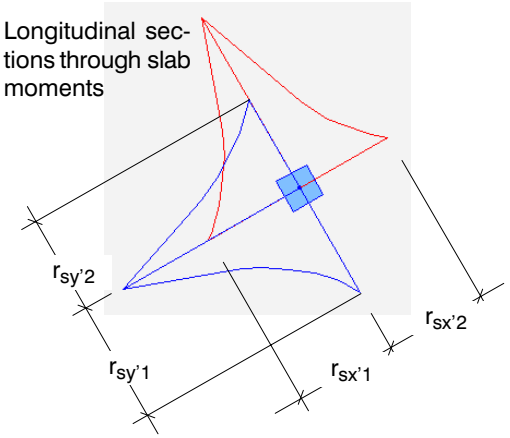
For consideration of rigid column connections according to paragraph 4.3.6.2.4 is with which the geometrically determined length of the control perimeter for multiplying the reduction factor k_e. The appropriate value must be entered in the punch-through dialog.

To Approximation level 3

While in the approximation levels 1 and 2 the values r_s and m_{sd} for the determination of the slab rotation ψ of hand-given formulas are estimated, their determination at the approximation level 3 is carried out by means of a elastic finite element analysis.

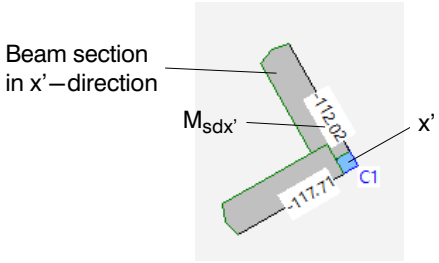
r_s is the distance of the location of zero moment from the center of the punching object in the direction of the punching direction under the relevant punching load case. The load case can be selected for each punching object from the list of all given loads and result combinations. In most cases, the loading 'dead load' or a combination of 'dead load' and 'distributed payload' is likely to be appropriate. In order to determine the values rsx' and rsy', CEDRUS introduces a longitudinal section in these directions and determines the zero crossing of the slab moment in the corresponding directions. For checking, a graphic is output for each punching object in the following way:





⚠ The **r_s** values are required in the formula for ψ and for determining the plate strip width b_s. Since r_{i1} and r_{i2} are generally not equal, the question arises as to which value is to be expected. In the current program version, the mean value of r_{i1} and r_{i2} is used in both cases.

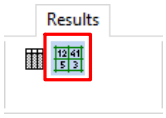
In order to determine **m_{sd}** beam sections of the width b_s between the edges of the supported zone are introduced and the reinforcement moments (i.e. taking into account the drill moments) are determined for the limit state specification used in the punching dimensioning. The larger of the two values is used at the sections end. There is also a control graphic for this:



M_{sd} : Total moment in the strip
m_{sd} : Average moment

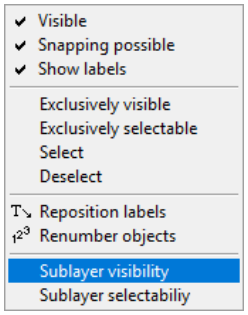
$m_{sd} = M_{sd} / b_s$

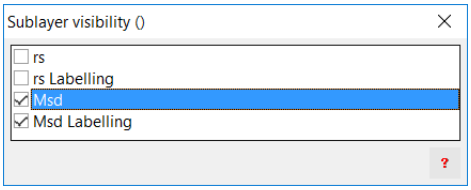
☞ *Result presentation in the program*



If the result output 'Numerical Graphics' is activated, two result layers are created. In the first layer, the beam sections with the limit values of the dimensioning moments are displayed graphically. The second layer shows the fulfillment factors for the punching detection for all PSO.

Using the context menu on the first layer, the total moments can also be displayed:





Interpretation of the output resistance values V_{Rd} and V_{Rd-e}

If V_{Rd} acc. equation (57) for given loads and given V_d is determined, the ratio $\alpha 1 = V_{rd} / V_d$ can not be regarded as a conformance rate. If one understands that the loads can be increased by a1 in order to reach the existing punching resistance, this is not correct. For increased loads, the value for m_{sd} used in equation (59) also increases, and the punch-through resistance decreases.

If, on the other hand, the factor $\alpha 2$, by which the loads can be increased until the dependent punching resistance V_{rd-e} is reached, is regarded as the conformance rate, this value probably indicates more about an existing safety. V_{rd-e} is referred to as an effective punching resistance in CEDRUS.
If $m_{sd} < m_{Rd}$, then $\alpha 2 < \alpha 1$.

If only the punching verification is to be provided, both $\alpha 1$ and $\alpha 2$ can be regarded as the conformance rate, since:
If $\alpha 1 > 1$ then also $\alpha 2 > 1$.

The punching resistance (approximation level 2) output in the tab sheet 'Punching' is always the effective punching resistance. Since the loads are not yet defined at this point, only V_{rd-e} can be determined.

In the tab sheet 'Results', on the other hand, both V_{rd-e} are output in the punching verification and this is done for both approximation levels 2 and 3 (see chap. B 13.2.5).

B 9.1.5 EC2 - Specifics

DIN EN 1992-1-1

Internal column $\beta = 1.1$ (Original value EN = 1.15)

For corners and endof wallsthe proposals from the DIN standard are used for all supported EN standards:

End of Walls $\beta = 1.35$
Corner of Walls $\beta = 1.2$

ENV 1992-1-1

The punching shear resistance without shear reinforcement (without normal force) amounts, according to the code EC2, to:

$$V_{Rd1} = u \cdot d_m \cdot C_{Rd,c} \cdot k \cdot \left(100 \cdot \rho_l \cdot f_{ck} \right)^{1/3} \geq \nu_{\min} \cdot u \cdot d_m > \beta \cdot V_{sd} \qquad [6.4.4\ (1)]$$

$$C_{Rd,c} = 0.18 \cdot \gamma_c \qquad [6.4.4\ (1)]$$

$$k = 1 + \sqrt{200 / d_m} \leq 2.0 \qquad [6.4.4\ (1)]$$

$$\rho_l = \sqrt{\rho_{lx} \cdot \rho_{ly}} \leq 0.02 \qquad [6.4.4\ (1)]$$

$$\nu_{\min} = 0.035 \cdot k^{3/2} \cdot f_{ck}^{1/2} \qquad [6.2.2\ (1)]$$

V_{Rd1} : dimensioning value of the punching shear resistance without shear reinforcement
V_{Sd} : dimensioning value of the total shear force to be resisted
u : length of control perimeter
d_m : mean static depth
ρ_l : mean reinforcement content
β : factor to take into account the effect of load eccentricity
automatic: inner column β=1.15, edge column β=1.4, corner column β=1.5
f_{ck} : characteristic uniaxial (cylinder) compressive strength

The punching shear resistance with shear reinforcement (without normal force), verified close to the column, amounts to:

$$V_{Rd2} = v_{Rd,max} \cdot u_0 \cdot d_m > \beta \cdot V_{sd}$$ [6.4.5 (3)]

$$v_{Rd,max} = 0.5 \cdot \nu \cdot f_{cd}$$ [6.4.5 (3)]

$$\nu = 0.6 \cdot (1 - f_{ck}/250)$$ [6.2.2 (6)]

$$f_{cd} = f_{ck}/\gamma_{Rc}$$ [6.2.2 (6)]

V_{Rd2} : dimensioning value of the punching shear resistance with shear reinforcement

V_{Sd} : dimensioning value of the total shear force to be resisted

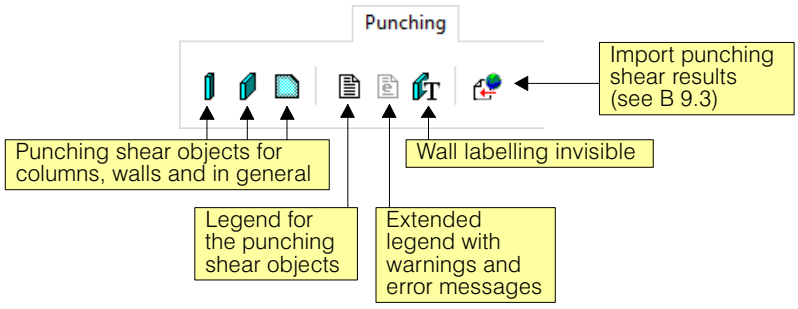
u_0 : circumference bordering on the column cross section, depends on column shape and position in the slab. [6.4.5 (3)]

f_{cd} : dimensioning value of the uniaxial compressive strength of the concrete

The circumference for which punching shear reinforcement is no longer required (u_{out}), is not verified.

B 9.2 Inputs in the 'Punching' Tab sheet

On the general treatment of punching shear in CEDRUS see section B 9.1,ff.

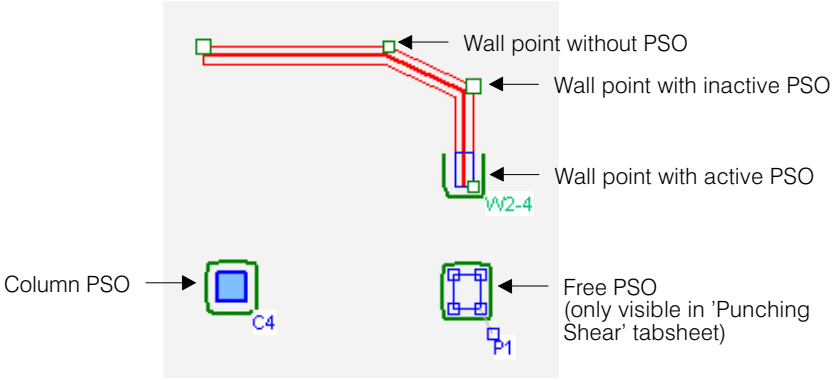


As described above, there are three types of **Punching Shear Object (= PSO)**:

- Column PSO, which are introduced by the program for each column
- Wall PSO, which are automatically introduced for each wall point
- Free PSO, which are introduced by the user

B 9.2.1 Presentation and Selection of PSO

The following figure shows how the PSO are shown in the 'Punching Shear' tabsheet :



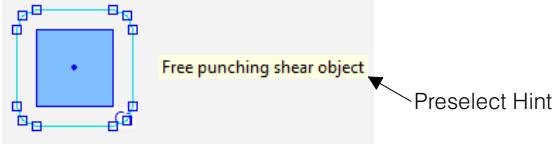
For this presentation the Graphics Editor button for the visibility of the object points must be activated (on the left side of the screen). When changing to the 'Punching Shear' tabsheet it is automatically activated. If the object points are not visible, then for the wall PSO first the wall has to be selected to be able to access the wall points to which the PSO is connected.

In order to switch off the wall labelling, should it disturb, this button in the 'Punching' tabsheet has also to be activated.

If one were to switch off the labelling for the layer properties for the wall, then the labelling in the PSO would also no longer be visible.

On the selection

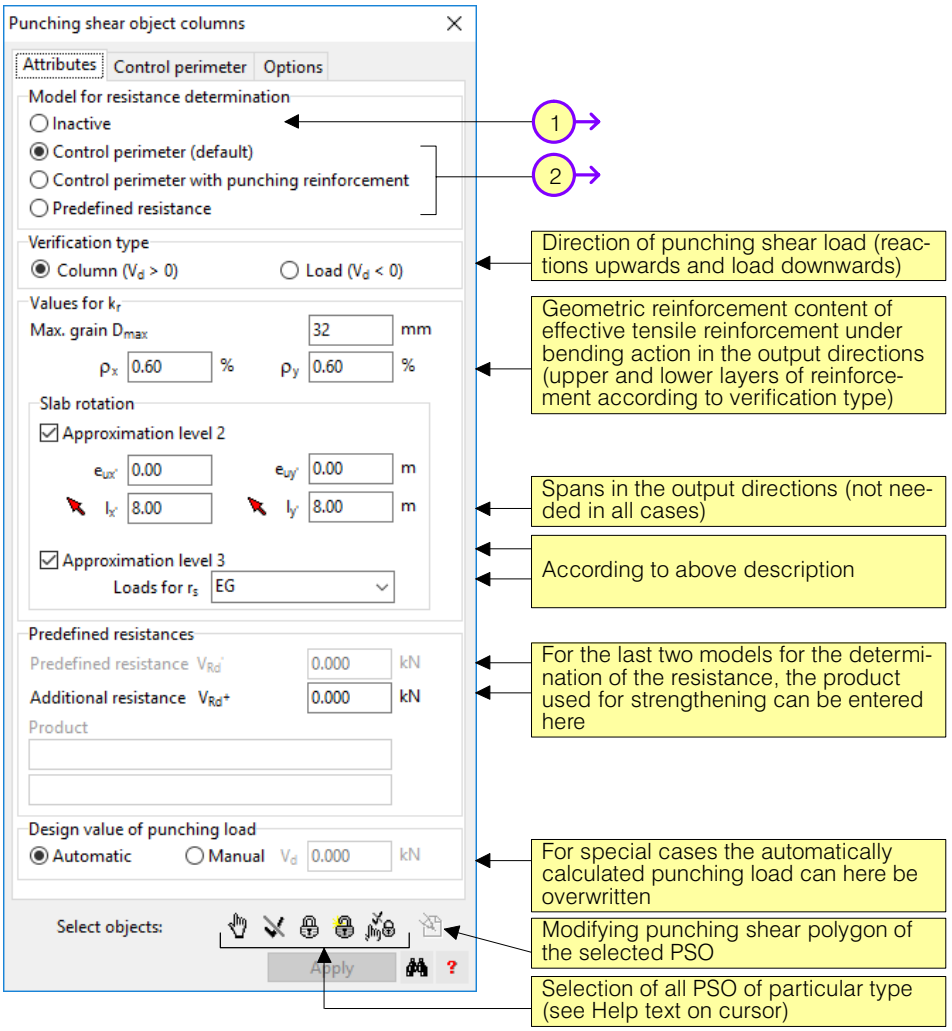
If a PSO dialog is open, then as is standard in the Graphics Editor, only objects of the corresponding type are selected. If one wants to select a PSO then one should pay attention to the Preselect Hint on the cursor, in order to get hold of the particular object desired:



B 9.2.2 PSO Dialogs

Each PSO type has its own Properties dialog with the corresponding button in the 'Punching Shear' tab sheet.

These dialogs are identical up to the tab sheet 'Wall Attributes', which occurs only in the wall PSO, and are described in detail below.



(1) The automatically generated PSO for column and wall points are always inactive by default. All PSO that one wants to include in the verification, must be activated manually

☞ *Since each PSO type has its own Properties dialog (e.g. column PSO), at any one time only PSO of the same type can be modified. If, e.g., all column PSO are activated, then one has to open the Column PSO dialog, select with <ctrl> + <A> all its objects, specify the common properties and apply.*

(2) In addition to the explanation in chapt. B 9.1 (page B-52) one specifies here when and which resistance model is to be selected:

0) inactive: for existing PSO that one doesn't want to include in the verification.

1) control perimeter: The punching shear resistance is determined assuming that there is no punching shear reinforcement or strengthening.

2) control perimeter with punching shear reinforcement: The punching shear resistance is determined according to the manufacturer's data and entered manually in the field *given resistance* V_{Rd}'. The effective resistance is displayed by the program with $V_{Rd} = \min(V_{Rd}', V_{Rd2}, V_{d,max}) + V_{Rd}^+$ (see page B-52).

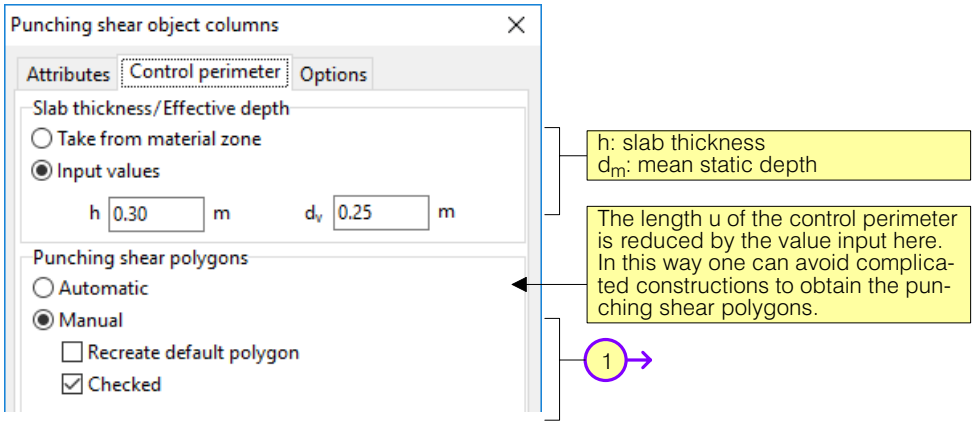
3) resistance given: This setting is selected if the punching shear resistance is known and one wants to include it (if necessary corrected by the amount V_{Rd}^+) without any other

conditions in the punching shear verification. The value is entered in the field *given resistance V_{Rd}'*.

One is automatically switched to this model if the punching shear resistance is imported automatically from other programs (see chapt. B 9.3).

Tab Sheet 'Effective perimeter'

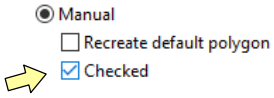
In the case of the above Models 1) and 2) also the settings in the tab sheet 'Effective perimeter' of the PSO dialog have to be considered:



(1) For all PSO the program creates a punching shear polygon on the basis of the selected code and the parameters specified in the dialog window. In its rough provisional shape this is a polygon with the specified distance to the load area polygon and with the setting **„Manual"** it is also left as it is. With the setting **„Automatic"** (default) the program tries to take into account the openings and slab edges in the neighbourhood in accordance with the code requirements. If this is not successful the program changes to "Manual". In each case the solution found has to be assessed visually by the user and if necessary the user has to change to "Manual".

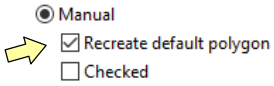
☞ *The program can only find a correct polygon automatically for the limited number of standard cases described in the codes. Many cases, however, do not apply to the models described. The polygon method in principle is not always meaningful or the punching shear action simply cannot in some circumstances be considered as an isolated phenomenon. In these cases engineering modelling by the user is the only possible way*

The setting **„Manual"** for an existing punching shear polygon means that its final shape is determined by the user by making the necessary corrections to the rough provisional polygon. For the modification of punching shear polygons see page B-61).



With the setting **„Manual"** the user may have to verify the correctness of the generated and possibly modified punching shear polygons by means of the control box **„Checked"**. Otherwise during the verification a corresponding error message is given.

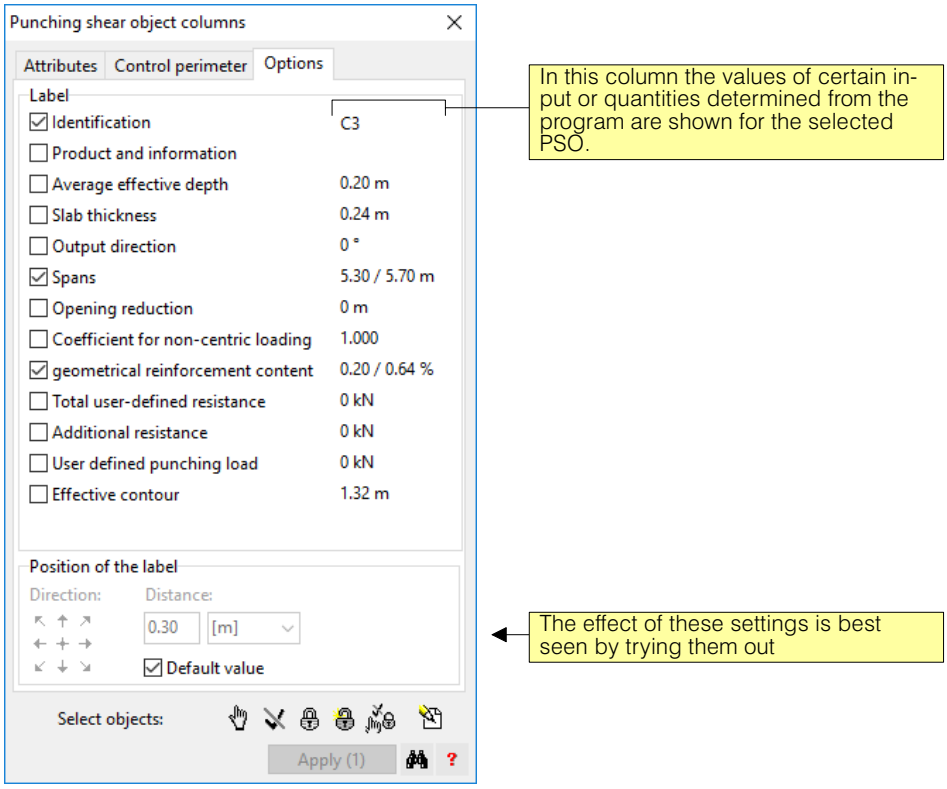
In the case of changes to the slab that could influence the punching shear resistance, the program assumes that a manual polygon is not necessarily correct anymore and "Verified" is switched off. These polygons therefore have to be checked once more for their correctness and then manually changed to „Checked".



After some changes to the slab, the punching shear polygons are automatically created again. If one wants to prevent this for some polygons that may have been modified manually, then this control box can be deactivated.

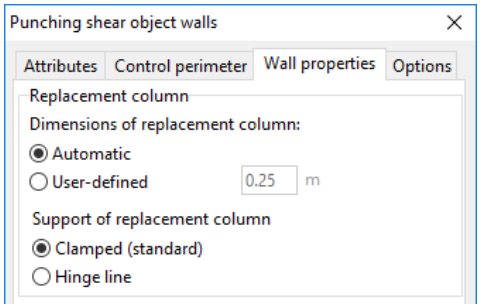
Tab Sheet 'Options'

In the tab sheet 'Options' of the PSO dialogs, the position and the amount of labelling of the PSO objects can be influenced. Particular values, which are of importance in the determination of the punching shear resistances, can be consulted interactively.



Tab Sheet 'Wall proprieties'

The PSO dialogs of wall points contain an additional tab sheet 'Wall Attributes':

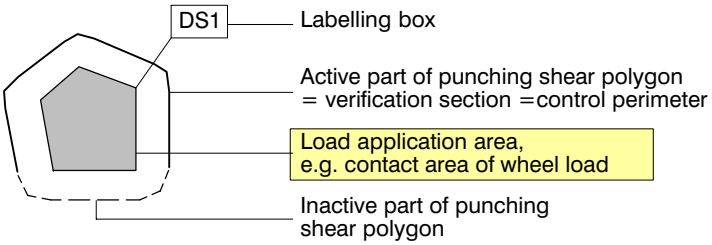


The treatment of the punching shear of wall points is carried out in CEDRUS by suitable placement of equivalent columns at these points (see Page B-49). The dimension of the equivalent column Is described there, that is normally determined by the program, can be overwritten here using the setting „User-defined“.

In the case of the equivalent columns normally introduced it is assumed that they possess bending stiffness. In special cases this can be changed.

B 9.2.3 The Input of Free Punching Shear Objects

With the introduction of a free punching shear object the polygon tool to input the **Load Application Area** starts automatically. We remind users here what constitutes a PSO:



It should be ensured that in the tab sheet 'Attributes' of the PSO dialog the correct verification type is specified!

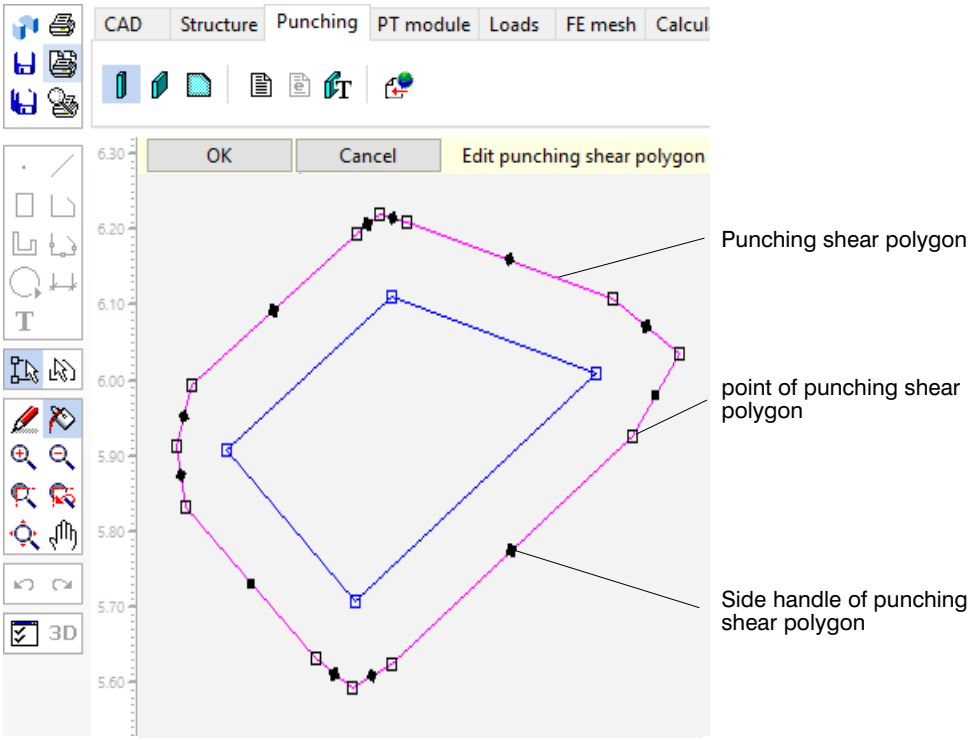
Based on the input load application area, the program creates automatically a punching shear polygon that is treated in the same way as the other PSO types.

B 9.2.4 Modifying Punching Shear Polygons

For each PSO a punching shear polygon is always generated automatically. It is a closed polygon with active and possibly also inactive sides. The control perimeter is given by the active sides.

Punching shear polygons can be modified by first selecting the PSO and then clicking on the button shown on the left.

CEDRUS then changes to a special editing mode to modify the punching shear polygon, which, after the modifications are finished, can be quit by clicking on [OK] or [Cancel].



The following actions are possible:

- Delete or move a selected end point using the context menu
- Select side handle and activate or deactivate the side using the context menu
- Select punching shear polygon and using the context menu adjust or introduce new corner points

⚠ The punching shear polygon may not be eliminated by deleting all corner points. If, on leaving the editing mode, it is not in a healthy state, the original polygon is recreated.

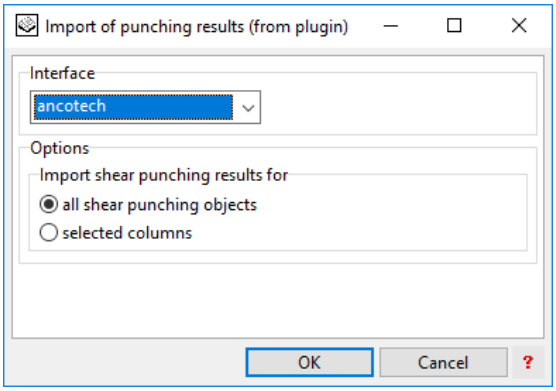
B 9.3 Import of Punching Shear Results

From CEDRUS dimensioning programs of the manufacturers of punching shear reinforcing can be called, whereby CEDRUS provides the third-party program with the necessary data (Export) and can also receive from this program the calculated results (Import).

A prerequisite for this is the correct installation of the corresponding third-party program, which has to be properly set up for this and its choice as a Punching Shear Plug-in program.

The import of results is only possible after calling the third-party program (in the 'Results' tabsheet), whereby the necessary calculations have to be carried out in the third-party program and the results exported.

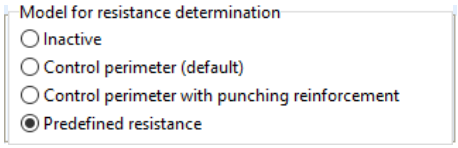
By clicking on the button in the 'Punching' tab sheet the following self-explanatory dialog appears:



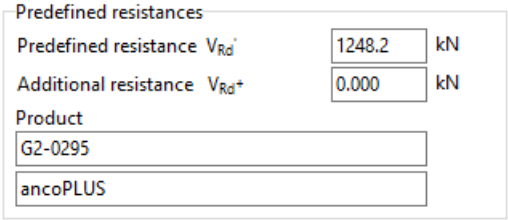
If one only wants to import selected PSO then the selection has to be made before calling the Import dialog (thus sometimes you need Cancel], so you have to select and make the call again).

The import is carried out only for those punching shear objects for which the third-party program has detected the necessity of a reinforcing measure. If this is not the case for an object, then its properties are changed as follows:

- The model for the determination of the resistance is changed to „Predefined resistance“



- Under „Predefined resistance“ the total resistance determined by the third-party program is input as well as in the fields „Product“ the name of the strengthening product also has to be input.



- Any DXF drawings of strengthening products are stored in the subfolder 'INP' of the current analysis folder under the name '*Name of punching shearobject*.DXF'.

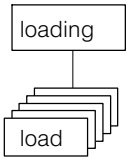
B 10 Loads

Actions and loadings are input and managed in the tabsheet 'Loads'.

B 10.1 Important Terms in Connection with Loads

Loadings

A loading in CEDRUS comprises an arbitrary number of single loads or load elements, like point loads, line loads and area loads. Loadings are the smallest unit for which results are created and which can be superimposed to obtain result combinations and for limit state values.



Examples:
- dead load (self-weight)
- live load Field 1
- vehicle in Position 1 (e.g. 4 wheel loads)

Load case

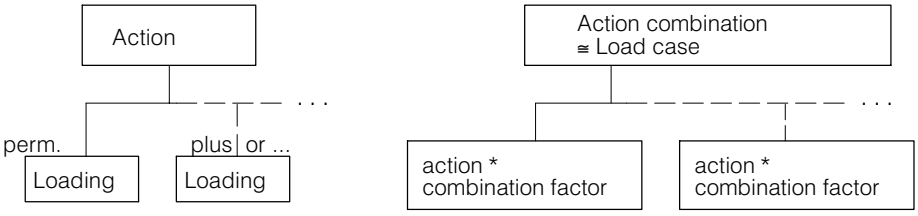
A load case according to the SIA code 260 *is characterized by a leading action and the simultaneously occurring accompanying action* (paragr. 4.2.6). Load cases, therefore, are the cobinations of actions that have to be investigated in dimensioning. The term load case only exists in this sense in the program. In the case of limit state specifications (see there) one speaks of action combinations and not of load cases, since the actions themselves may already be limit values.

Action

The very general term **action** is given a more restricted definition in the Cubus programs. In actuality an action consists of one or more loadings, which all belong to the same type of action, such as permanent loads, live loads of category A, snow loads, etc. Actions are the units from which the action combinations are formed. An action combination, for example, could be

1.35 * dead loads + 1.5 * live loads

and thus would correspond more or less to a load case in the above sense. The action combinations, however, involve in general somewhat more than a load case, since, e.g., the action 'Live Loads' does not have to be a fixed loading combination, but may represent a superposition rule for all possible live load positions

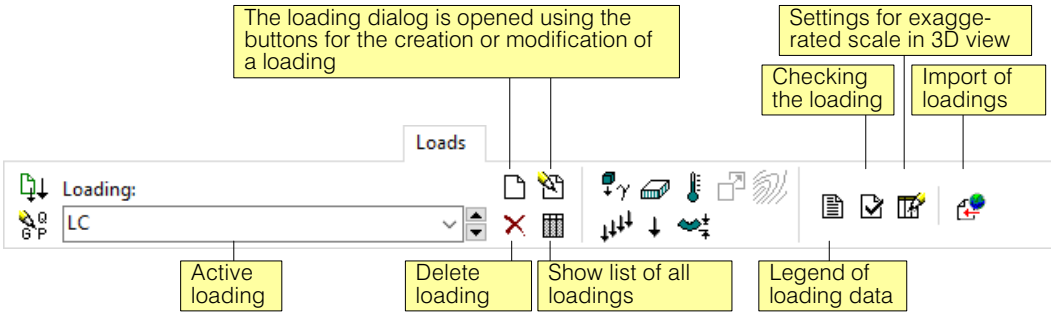


Limit state specification

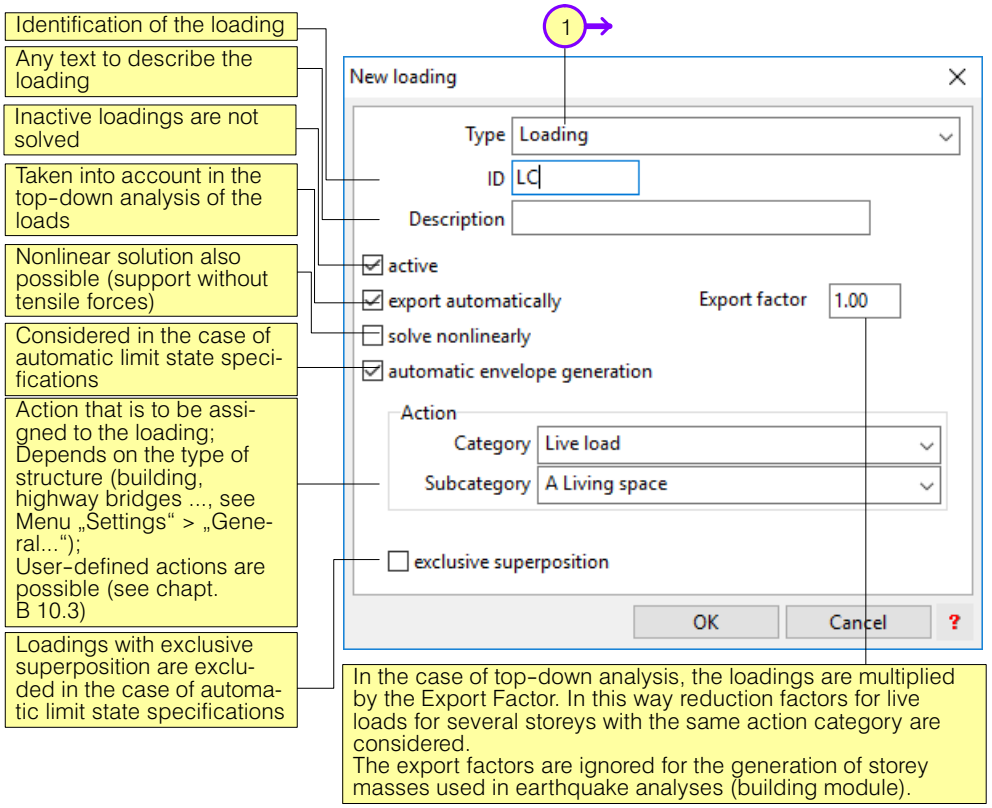
In general, for dimensioning there are several limit states and for each limit state several load cases or action combinations that have to be considered. The limit state specifications are a summary of all action combinations to be investigated for a given limit state (e.g. ultimate load). On their basis the limit values of individual action components for the corresponding limit state may be determined.

B 10.2 Loadings

Each loading has a number of properties, which are summarized in the Loading dialog.



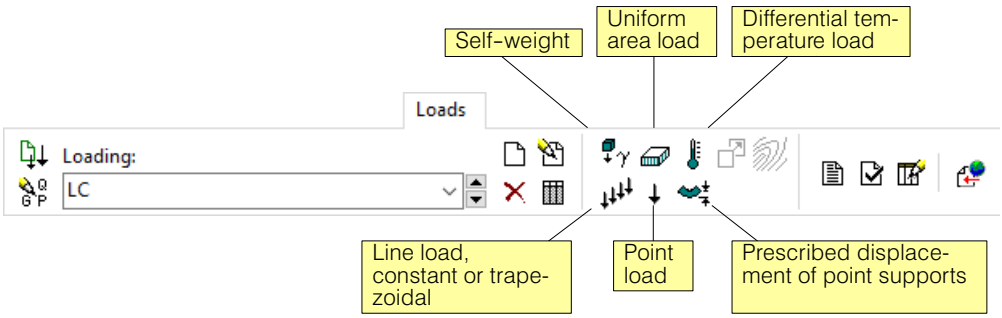
When creating a new loading the following dialog appears:



(1) Each time the type of loading must first be selected. Normally, this is the standard type „Loading“. The different types are described below in detail.

B 10.2.1 Loading Type 'Loading'

This is the standard type for the input of the following load types:



1) **Area loads**
   Rectangular or arbitrary polygon for:
   - Gravity loads (dead load, i.e. self-weight)
   - Uniformly distributed force
   - Differential temperature force

2) **Line loads**
   Constant or trapezoidally distributed
   - Forces
   - Moments (about load lines)

3) **Point (concentrated) loads**
   - Forces
   - Moments about x or y directions

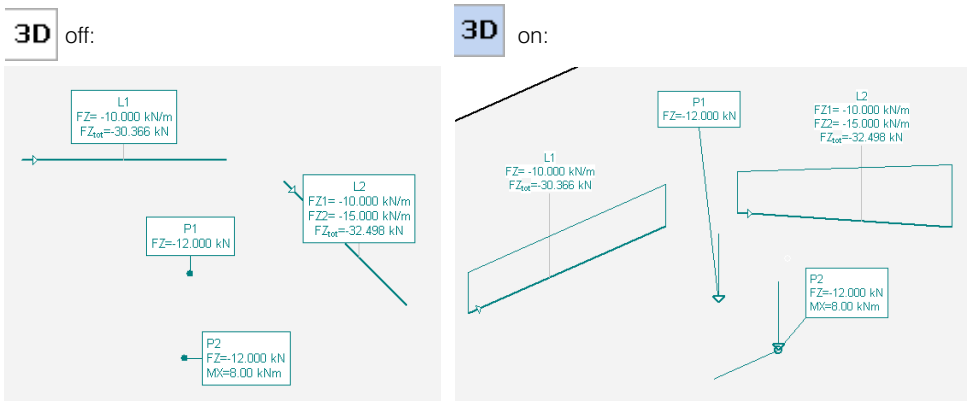
4) **Displacement of point supports (settlements)**

The following points have to be taken into account:

- The loads are input independently of the FE mesh.
- Only those load components are considered that are applied directly to the slab. Thus components that extend beyond the outline or that lie over openings, are not included in the analysis. This simplifies the input.
- In the case of point loads or area loads of limited extent it should be observed that in the computational model the actually effective extent depends on the fineness of the FE mesh. Broadly speaking, it is at least as large as the element dimensions.
- Prescribed displacements of point supports can only be requested for vertically blocked points of column supports. The load must be placed exactly in the middle point of the column.

3D-representation of loads

Point- and Lineloads have a special 3D representation:

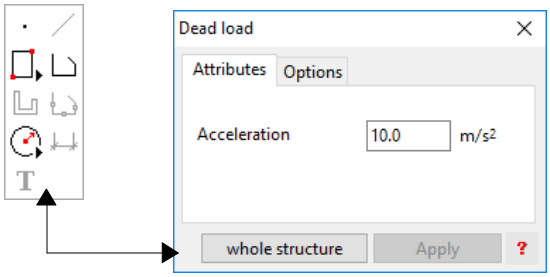


The scaling of the displayed load values can be influenced in the dialog called by the adjacent button. The settings are defined as follows:
The load value in its standard units [kN, m] is displayed as a length in [m], multiplied by the given scaling factor. A point load of 5 kN in direction Z for example is displayed as a line of the lengh 1.0 m in direction Z if the corresponding scaling factor is set to 0.2.

Area Loads

Self-weight

For compatibility reasons with other Cubus programs, the self-weight is defined as an inertia force due to a given acceleration. The acceleration ist preset to 10 m/s^2 (in Z direction) which results in an inertia force in (-Z)-direction corresponding the self-weight of the structure.

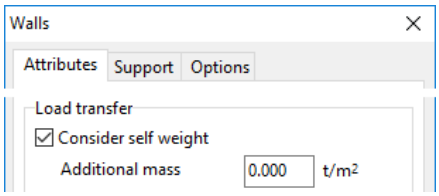


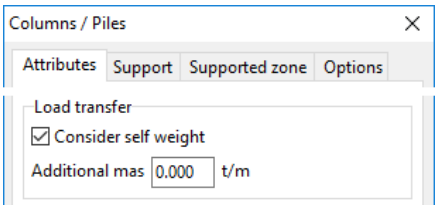
Normally you will introduce the dead load of the whole structure. For that you click on the button [whole structure] and then anywhere on the plate to put down the load symbol.

As an alternative to the [whole structure] button you may use the graphics editor tools to draw an area, where the self-weight should act exclusively.

☞ ***Columns an walls**: The self-weight of these vertical elements is needed for the calculation of the load transfer in a building. It is calculated based on the material specified for the body of these objects and can be increased by the specification of additional masses in their dialogs (e.g. for plastering or covering).*

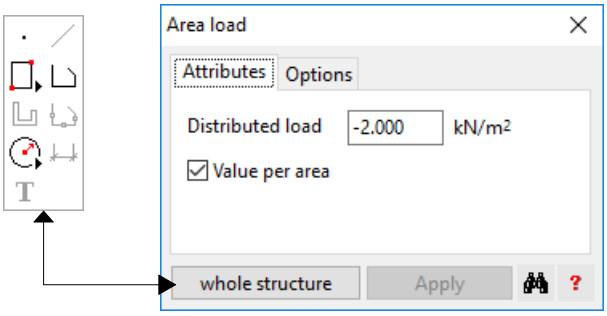
The following statements are only valid for calculating the load transfer outside the building module:
The weight of the columns and walls are always introduced in the self-weight loading. Preconditions are:
1) in the dialog for introducing the self-weight of the slab the button [whole structure] has to be used
2) the boxes 'Consider self-weight' must be checked in the dialogs of the vertical element





☞ *The check boxes „consider self–weight“ are not active in the building module.*

Uniformly distributed area loads



The load value has to be in accordance with the check box „Value per area“. Is it not checked the given load ist distributed uniformly on the defined load area.

Choose the button [whole structure] if the load should act on the whole slab, and then click anywhere on the slab to put down the load symbol.

Otherwise use the available graphics editor tools to draw a load area.

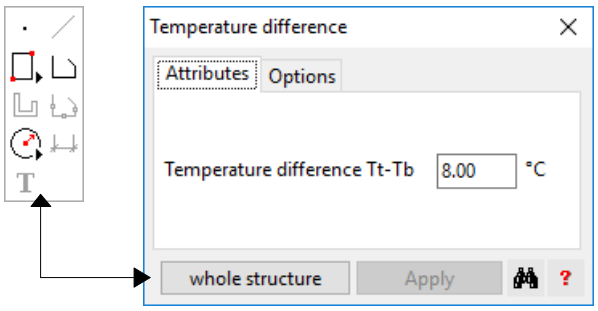
☞ *Drawn load areas may overlap the outline or openings of the slab. The load only acts on real slab area. If you specify the total load („Value per area“ unchecked), the load is distributed on the drawn area.*

Differential temperature loads

Temperature loads generate initial curvatures.

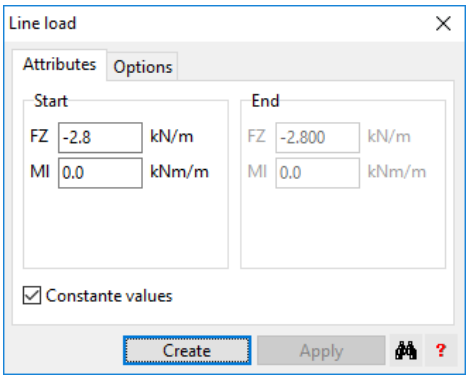
As input value the difference ΔT of the temperature between the upper and the lower surface of the plate is given in [°C].

With the plate thickness d and the temperature strain coefficient α (defined in the used material) the following curvature is generated: $\chi = -\frac{\Delta T \cdot \alpha}{d}$



This load can also act on the [whole structure] or on a drawn area.

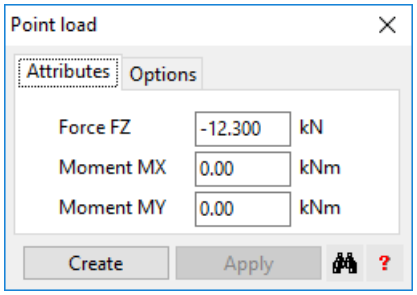
Line Loads



A distributed load FZ and/or a distributed moment MI can be specified. A positive moment turns clockwise around the input line. The direction of the input line is indicated by a small triangle.

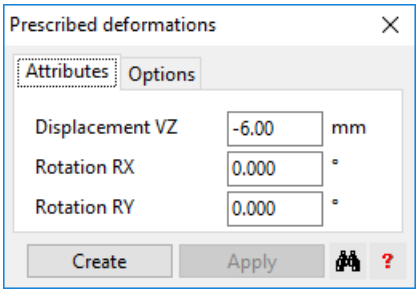
For specifying linearliy distributed loads the box 'constant values' has to be unchecked. The direction of the input line defines the start and end point of the load (see above).

Point Loads



Moments act in a vectorial sense around the global X- and Y-axes

Prescribed Displacements / Rotations



Prescribed displacements or rotations are only possible for point supported columns with the respective degree(s) of freedom blocked. Rotations act in de coordinate system defined by the column (by default in the global X, Y directions).

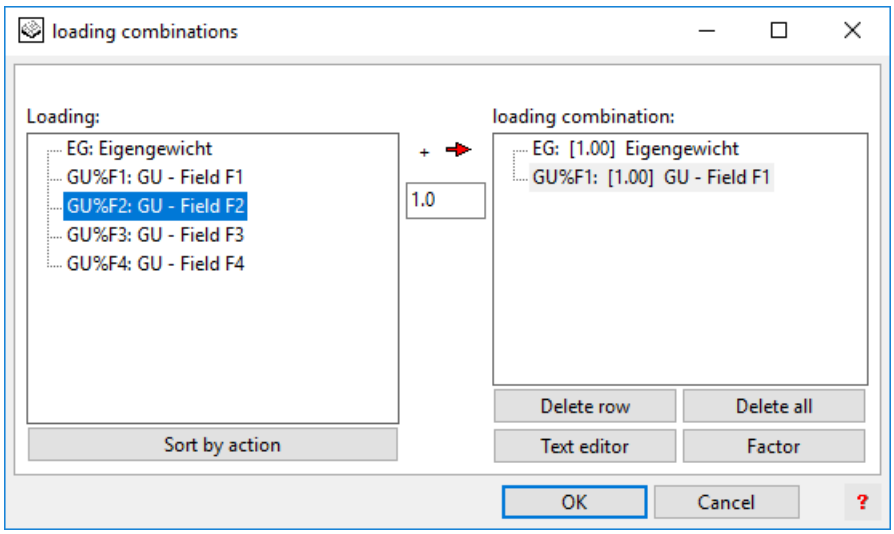
The load has to be placed exactly on the support point.

When the columns are rotated, the prescribed twists for X and Y are introduced in the rotated directions.

B 10.2.2 Loading Type „Combination“

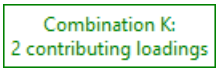
In this way a new loading can be defined as a combination of existing loadings of the type 'Loading'. The individual loadings can be assigned arbitrary factors. If one modifies a loading used in a loading combination, then this also influences the loading combination.

The following dialog appears after the introduction of a new loading combination:



By using the arrow butten the desired loadings from the list of the existing ones (left table) are transferred to the loading combination (right table) with the given factor. The rest of the the functions should be self explanatory.

The symbol shown for a loading combination looks as follows:



With a double click on this symbol or with 'Attributes' from its context menu the input dialog is displayed an can be modified.

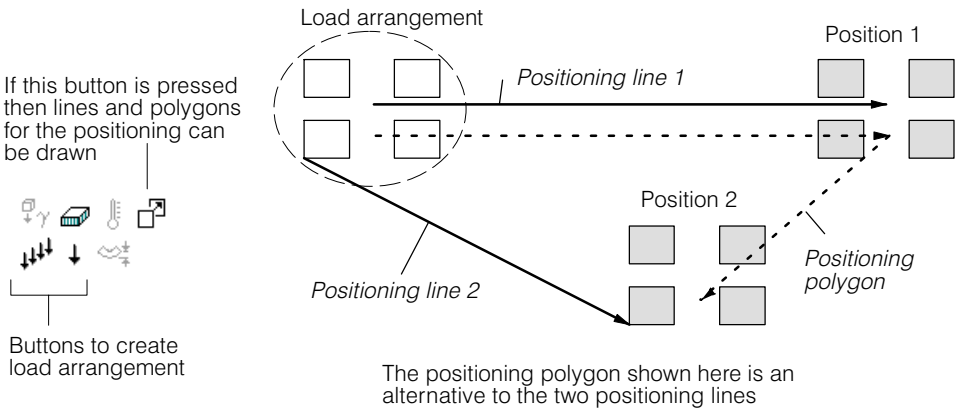
Loading combinations created in this way are used straightaway like normal loadings, except that they cannot be part of another loading combination.

☞ *It is possible to combine results of individual loadings (Results Combinations in the tab sheet 'Calculation'). If, however, a superposition of results is not allowed, as is the case with nonlinear analyses, then one has to work with a combination loading.*

B 10.2.3 Loading Type „Generator (Moving Load)“

A Generator Loading generates a series of loadings automatically, which are assigned an identifier of the form '*Gen*%*nr*'.
Gen = identifier of the generator loading, *nr* = number of the generated loading.

If a load arrangement (e.g. a vehicle) has to be considered in different positions, but acting only once simultaneously, then for each position a loading with an exclusive superposition is required. These loadings can be input individually or more easily created using the Generator 'Moving Load'. For this purpose one first inputs the **load arrangement** and subsequently the **positioning lines or polygons** for the individual positions. The generator then creates a loading automatically for each defined position.



The following points need to be considered:

- The input load arrangement can be placed anywhere. It does not necessarily represent a possible load position.
- Lines and polygons can be used for the generation of load positions. The beginning of a line or a polygon forms the handle point for the load arrangement. The line ends and/ or the polygon points are target points. The load arrangement is fixed at the handle point and set down at the target point (see above figure).
- Load arrangement and positioning lines can be corrected at any time. The loadings are regenerated automatically.
- The generator loading itself is not a real loading. It is not analysed and cannot be used in combination loadings, etc.

B 10.2.4 Loading Type „Generator (Unfavourable Pattern)“

A generator loading automatically generates a series of loadings, which are assigned an identifier of the form '*Gen*%*nr*'.
Gen = identifier of the generator loading, *nr* = number of the generated loading.

If the loading has to be considered to act unfavourably fieldwise, then it has to be subdivided into those load components (= loadings) that can act individually. These load components can all be input as individual loadings or more easily created with the generator 'Unfavourable Arrangement'. For each input load element (normally an area load for each field) this automatically creates a loading. The loads, however, have to be input manually. But if this can all be done in a loading, it is much simpler than if many individual loadings have to be input. This is especially so in the case of modifications.

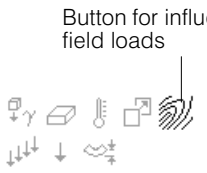
The following points need to be considered:

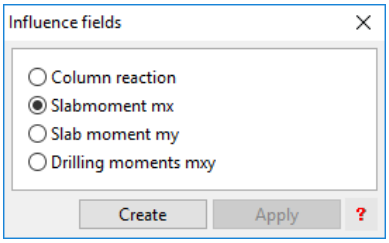
- The generator loading can be corrected at any time. The loadings are regenerated automatically.
- The generator loading itself is not a real loading. It is not analysed and cannot also be used in combination loadings.

B 10.2.5 Loading Type „Influence field“

For each influence field (available for slab moments mx, my, mxy and reactions of columns supported over an area) a loading of the type „Influence Field” has to be created. These are special loadings that cannot be combined.

By clicking on the only active load element button the following dialog appears, in which the the influence quantity is specified and then the influence point can be set using [Create].





The following points need to be considered:

- In the analysis the program moves the influence point to the centre of the element, in which the input point lies.
- Theoretically at the influence point the influence field exhibits a singularity. To obtain acceptable results the FE mesh in the region of the influence point has to be relatively fine.
- By introducing a unit point load or a unit support displacement the influence fields for point displacements or point reaction forces can be obtained.

B 10.2.6 Loading Type „Export Combination“

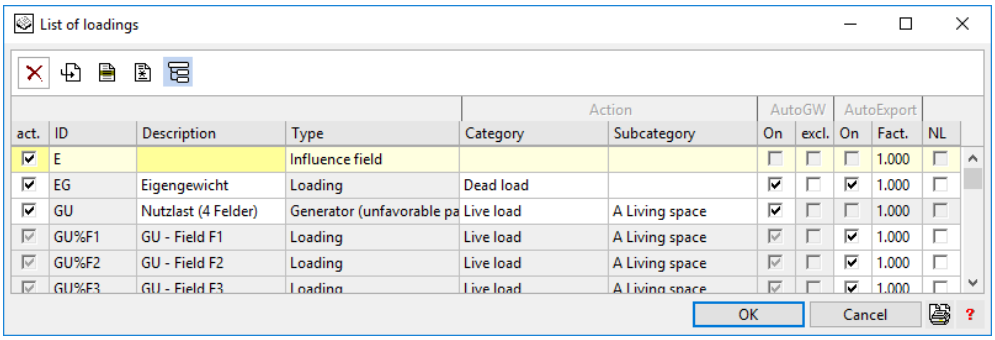
Alternatively or in addition to the automatic export combinations these can also be defined by the user. Export combinations are used in connection with the top-down analysis of loads (see there).

B 10.2.7 Loading Type „Mass Distribution“

Each dynamic analysis (see there) is based on a mass distribution, which has to be input as a dead load of the type 'Mass Distribution'. One can have several mass distributions, whereby each dynamic analysis can be assigned one. Combinations are not possible.

B 10.2.8 Loading List

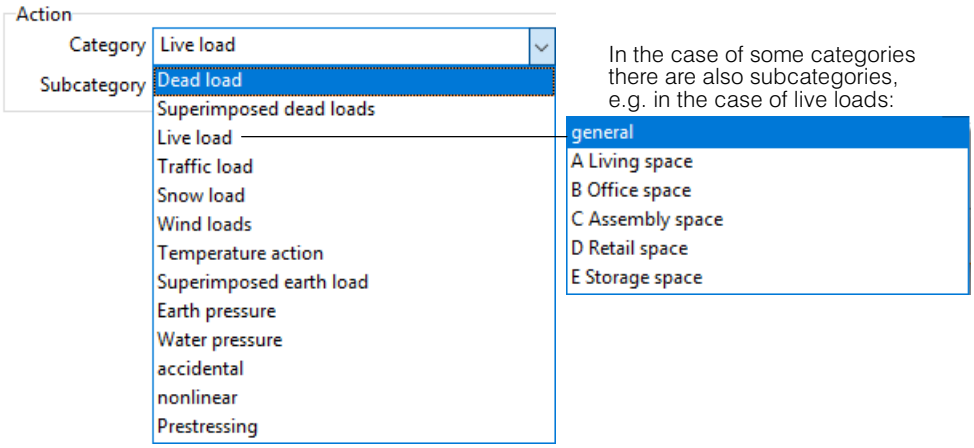
The loading list gives an overview on all specified loadings with their most important properties:



B 10.3 Actions

The term is described above under B 10.1. In CEDRUS, actions are only important in the creation of limit state specifications (see there).

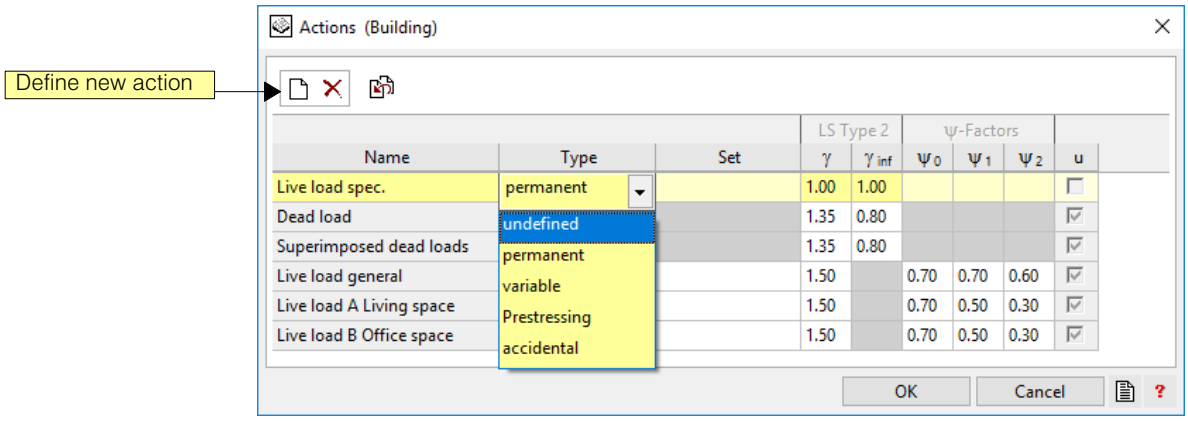
Each loading has to be assigned to an action. The standard catalog of predefined actions, which is given in the loading dialog, depends on the code and the type of structure (can be specified in the menu „Settings“ > „General...“). The selection list shown below comes from the SIA code and the structure type Building.



Action Dialog

Each action category used in an analysis is entered into a list by the program. This list, if necessary, can also be extended by self-defined actions. It is called using the following button

and looks as follows:



The column „Name“ normally contains the name of the action category followed by the name of an optional subcategory. For user defined categories it contains the name of the subcategory.

Predefined action categories have a given action type. For user defined categories the type can be chosen from „undefined“, „permanent“, „variable“, „prestressing“ and „accidental“. A type dependent set of load or combination factors are assigned to each action category. These factors shown in the action list are used to build up the limit state specifications (design situations).

To create a new, user defined action category.

To remove a marked category from the list. This is only possible, if the category is not referenced by loadings anymore. The function is only needed for user defined categories, since the others are removed automatically as soon as they are not used anymore.

Load and combination factors can be modified, not only for user defined actions but also for standard ones. With this button, standard actions can be reset to their original values.

Creats a print entry with the content of the dialogue.

With the column „Set“ the concept of action groups of some design codes is supported (see chapt. B 12.2.4).

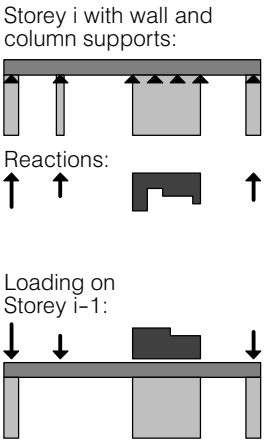
B 10.4 Load Transfer to Several Storeys

B 10.4.1 Description of Problem

☞ *A checklist for the procedure of dealing with load transfer is to be found on Page B-78.*

☞ *The top-down analysis of the loads is automated in the Building Module of CEDRUS. The procedure described here only applies to individual slabs, that are not integrated in a building model. The general description of the top-down analysis, however, is valid, with the exception of the effect of the shear forces in the wall, to a large extent also for the Building Module.*

The top-down analysis of loads is one of the main problems in the analysis of buildings. It is a question of the dimensioning of vertical elements, the foundations and also of floor slabs, if vertical elements are supported directly on the floor slabs. The whole procedure is in fact very complex, and one needs fairly simple models to obtain useful results with a reasonable expenditure of time. The following list gives some important points that have to be considered:

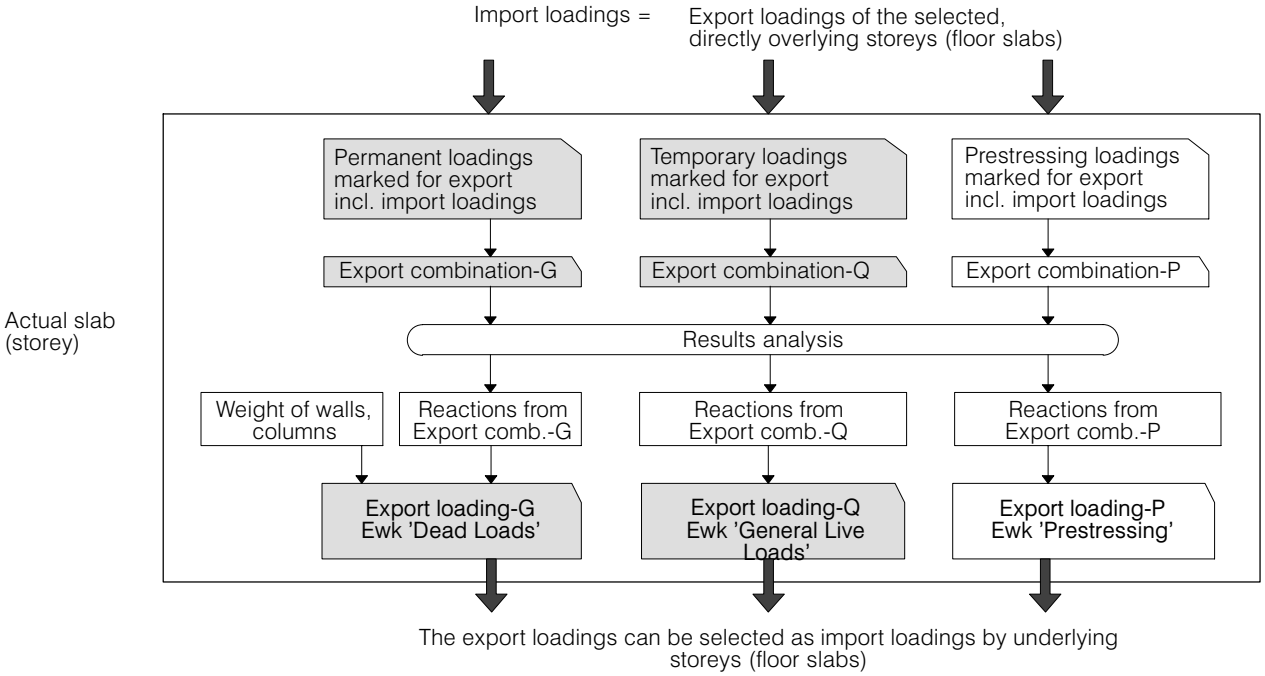


- Why does one not just make an FE model of the whole building, introduce the loads and with the pressing of a button we get the desired results?
  The procedure, in principle, is possible, but places high demands on the user and is only justifiable in exceptional cases. Such a model is computationally very time-consuming and often only provides an apparent accuracy. It is very sensitive to stiffness assumptions, especially in the case of tall buildings (see also other points below) and without complex extensions takes no account of the cracked state or the special properties of masonry, of long-term effects and of the construction method, but considers the building as though it were constructed in one single step. Also with the connections between the walls and the floor slabs simplifying assumptions have to be made. Regarding 'accuracy' ,therefore, normally one can hardly expect more than by considering individual storeys and load transfer from storey to storey, as one is used to when working conventionally, and how it is also implemented in CEDRUS.
- Due to the moments in the wall, shear forces are also formed, which are compensated in the floor slabs in such a way that their sum in two directions disappear. These shear forces can influence the stress distribution along the foot of the wall considerably.
- In the case of an irregular arrangement of the supporting vertical elements, large redistributions can result due to the slab, which are unrealistic and due to cracks and long-term effects they are partially reduced. One can avoid this problem if the stiffnesses of the vertical elements in relation to that of the slab are assumed to be sufficiently large.
- The self-weight of the walls is normally dealt with by means of a constant line load. Whether this model is allowed in a particular case has to be decided by the user.
- Normally, it is sufficient for the load transfer, besides the permanent loads to consider a uniformly distributed live load over the whole of the slab. Whether it is necessary to consider different load positions or vehicle loads is up to the engineer to decide.
- The load cases that are decisive for dimensioning are composed of several actions, which are weighted separately. In the case of buildings it is usually sufficient to consider dead loads and independent of them a variable action. For example, with snow loads on the roof and live loads on the floor slabs of different categories, one might be tempted to perform unnecessarily complicated analyses.
- The codes permit reduction factors for the live loads for groups of storeys, e.g. more than two storeys with the same live load category.

B 10.4.2 Implementation in CEDRUS

CEDRUS is a slab program, i.e. the top-down analysis of loads is carried out storey-by-storey by importing the loads from the overlying storey and exporting the loads to the underlying storey.

How load export and load import is organized, is illustrated by means of the following scheme and discussed in detail in the two following chapters.



B 10.4.3 Load Export

The load export, i.e. the transfer of the support forces of a slab to walls and columns as well as the dead loads of walls and columns to the underlying floor slabs is carried out via **export combinations**. This is a case of special combination loadings, whose reactions produce an export loading, which can be imported by other slabs as a loading.

The user has to decide which loadings with which factors are included in an export combination. Due to the different treatment in dimensioning of different actions, one also separates the loads from storey-to-storey according to the actions for the load transfer. For each action a corresponding export combination is required. In order not to make the process overly complex, normally one limits oneself to one action for the permanent loadings and one for the variable loadings and thus one works with two export combinations. The automatically created export combinations are also based on this simplification. Manually one is free to choose.

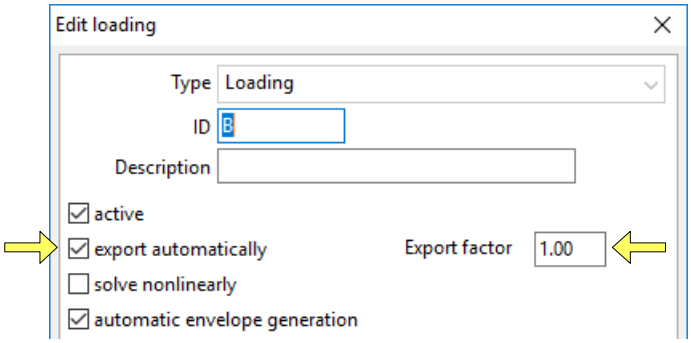
To treat the self-weight (dead loads) of walls and columns see below in the section *Calculation of Export Loadings*.

Automatic export combinations

CEDRUS automatically assumes the following export combinations:

- **'!Exp-G'** for the permanent loadings. It is assigned to the action 'Dead Load'.
- **'!Exp-Q'** for the variable loadings. It is always assigned to the action 'General Live Loads' zugeordnet.
- **'!Exp-P'** for the prestressing loadings. It is always assigned to the action 'User-defined Prestressing'.

For each loading one can specify whether it should be automatically exported and, if yes, with which factor (see the shown excerpt from the Loading dialog). Permanent loadings are thereby assigned to the export combination '!Exp-G' and variable ones to the export combination '!Exp-Q'.



Import loadings (see below) are also activated by default for the automatic load export. If they are assigned to an action of the automatic export combinations, they come in the corresponding export combination. Otherwise, an export combination with the corresponding action is automatically created.

All automatically created export combinations appear in the loading list.

Automatic export combinations are only supported for the structure type 'Building'.

Reduction factors in the case of storey groups of the same live load category

The codes permit reduction factors for variable loadings, if several storeys of the same live load category lie under one another. These reduction factors are a property of the individual loadings (export factor in the above dialog) and by default are set to 1.0 (i.e. no reduction). It is the responsibility of the user to specify the appropriate reduction factors (values <= 1.0) in the storeys where reductions are permitted. CEDRUS takes these reduction factors into account in the generation of the automatic export combinations.

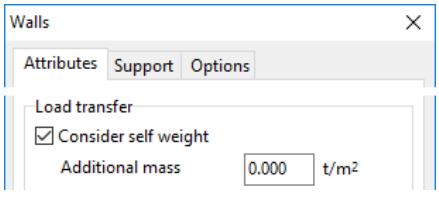
Manually created export combinations

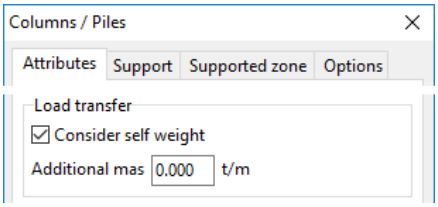
In the creation of a new loading it can be assigned to the type 'Export Combination'. It then becomes a combination loading and can be assigned to any action and any arbitrary combination of loadings. Manually created export combinations therefore are treated in the same way as automatically created ones.

Determination of the export loadings

Export combinations are normal loadings for the current floor with the exception that for for them, apart from the automatically determined reactions, no results are available.

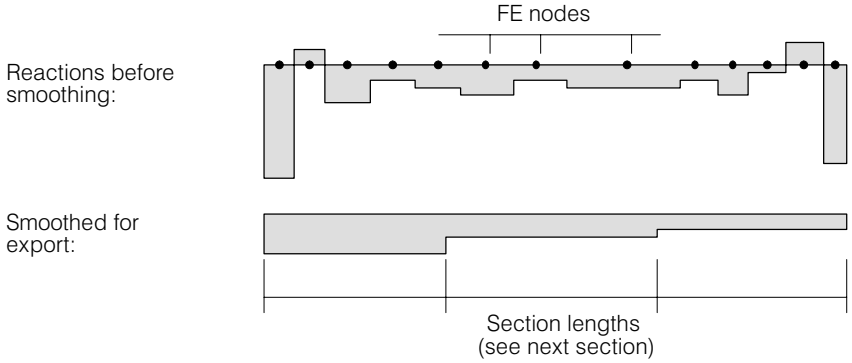
The loads to be exported are obtained from the calculated **reactions** for the export combinations as well as from the **self-weight of the walls and columns**, provided that in the export combination there is a loading with the load „self-weight of the whole structure“. The walls and columns however have to be specified accordingly ('Attributes' tab sheet of the Properties dialogs):





The self-weight of the walls and columns are given by their dimensions and materials, whereby there is the possibility of specifying an additional mass in [t/m^2] or [t/m] for cladding or the like.

Since wall reactions can vary greatly from node to node and during transition through the wall they are undoubtedly smoothed out, an assumption must be made for their distribution. For this distribution one can use the reactions determined in the individual sections, that are also used for the normal Reactions output.



The forces at the bottom of a wall, however, do not depend on the reaction forces on the top of the wall but also on the shear forces that come from the slab. Normally, in the case of buildings with several storeys these shear forces have the effect that the forces in the lower storeys tend to constant values for each wall. With the selection of section lengths this tendency can be influenced.

Section lengths

By default the program selects the wall height for the section lengths. These section lengths can be specified individually for each wall by inputting a factor for the wall height. A factor of 0.5 therefore would lead to a section length of 0.5 h (h = specified wall height).

The specified section lengths are only checked approximately by the program, since wall segments are divided into sections of roughly the same length and in addition section boundaries must be in the middle of two FE nodes. Therefore the smallest possible section length is also dependent on the FE mesh.

Nonlinearly determined export combinations

If one wants to avoid tensile forces in the load export, then one has to input the corresponding support accordingly and allow the the export combination to be analysed nonlinearly (setting in Loading dialog).

Export of export loadings

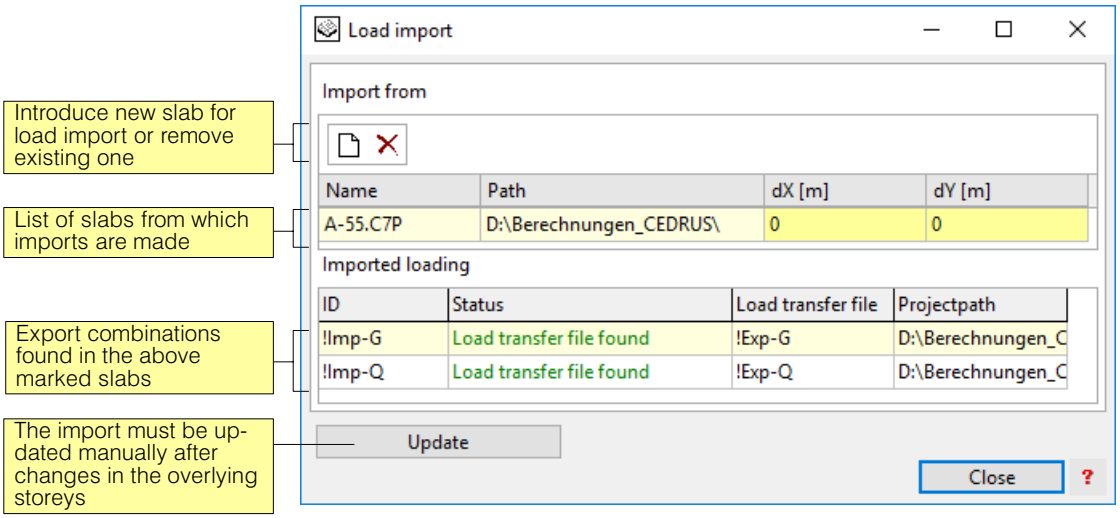
The export loadings created from the export combinations are ready for transfer to the next storey as soon as any results for the currently valid structure are requested. After modifications of the strucure or loadings that participate in the export, the results to be exported have to be created once again

☞ *Modifications in storey i therefore affect storeys that use the exported loadings (independent storeys), since the exported results are deleted after changes have been made and if need be are recalculated.*

B 10.4.4 Load import

The load import is managed by means of the Load Import dialog, which is opened by clicking on the following buttond

and looks as follows:



The analyses (slabs), from which the export loadings have to be imported, may be selected. They may result from one or several analyses (sources). If the specification has been made, the import, if necessary, i.e. after modifications to the exporting slabs and their reanalysis (by requesting any result), can be initiated at any time by the press of a button [Update].

The import loadings have their own options, which cannot be changed by the user.

Different live load actions in a storey

In a building with different live load categories and perhaps still with snow on the roof, one has to deal with a series of actions that one does not want to transfer separately from the roof to the foundation and treat separately in each limit state specification. For this reason all variable loadings, that are automatically exported, are summarized in a single export combination of the action „General Live Loads“.

Therefore, as variable loadings normally only those of the action 'General Live Loads' are imported. In the current analysis, however, there will be live loads of at least one other category, i.e. at least one additional live load action. Thus in the limit state specifications required for the dimensioning of the slab there will be no problems with several live load actions if, automatically all live load actions of buildings are summed together in a set called „Live Load Set“ (on limit state specifications and sets see chapt. B 12.2).

☞ *Summing several live load actions together in one set (with a given single action group in which all participating actions have the factor 1.0) means that in the automatic creation of action combinations in a limit state specification all these actions occur together either as a leading or as an accompanying action.*

B 10.4.5 Checklist for the Load Distribution

The checklist applies to the use of the automatically created export combinations. It is to be understood as a short instruction and assumes that the above described basic concept for load distribution has been understood.

For the floor slab which passes on loads (load export)

1) In the **List of Loadings** ('Loads' tab sheet), check whether all the desired loads for the automatic load export are marked (column „AutoExport" / „On") and whether they are assigned to the correct actions.

   Hint on the wall and column weights: *These are exported together with the loading that contains the self-weight (input as dead load over the whole slab).*

   In the list of loadings the two following export combinations must also be present:

   - !Exp-G for permanent loadings
   - !Exp-Q for variable loadings

   If you want to analyse them nonlinearly, then you can mark that in the column 'NL'. The properties of the support have to be specified accordingly.

   If you have imported loads from the overlying floor slab, the two import loadings also have to be listed:

   - !Imp-G for permanent loadings
   - !Imp-Q for variable loadings

2) **In order to create the load export files, a solution has to be obtained.** This happens automatically with the request for any result in the 'Results' tabsheet. Therefore, if you want the export files to be recreated after any relevant changes, you have to initiate the solution process.

3) After solving you can check the export combinations numerically including the sum of the loads (in the load legends) and graphically (by activating the corresponding loading layer).

For the floor slab which receives loads (load import)

1) Call the Load Import dialog ('Loads' tab sheet).

2) Create, if this has not yet been carried out, a load import.

   This involves a connection to an analysis, from which loads are to be imported (a row in the above list of the dialog) and the accompanying list of the imported loadings (at the bottom of the dialog).

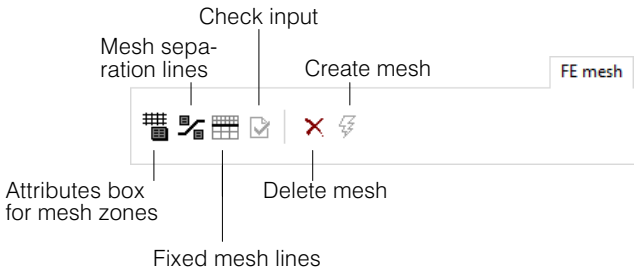
3) If the settings are correct, click on [Update], to obtain the current loadings specified in the import.

   This must be carried out manually after every change in the analysis from which loads are imported.

4) The imported loadings can be checked again, graphically immediately by selecting the corresponding import loading (in the list field of the loadings) and numerically using the legends of the loadings. The values of the sums of the loads are only shown there however after a solution has been obtained.

B 11 FE Mesh

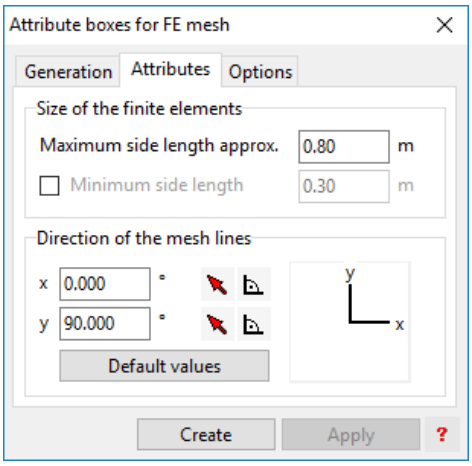
This tabsheet contains the tools to create the FE mesh.

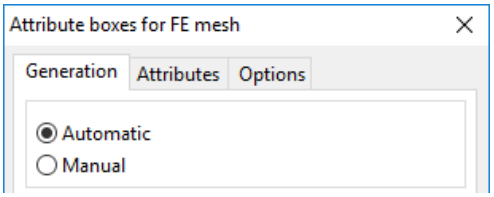


An important part of the modelling for a FE analysis is to choose a suitable mesh, such that sufficiently accurate results are obtained with a reasonable expenditure of effort in the input, computation and data storage. There is no simple recipe. One factor is the quality of the elements (see chapt. 2.1). Futher, in an FE analysis it is always important to know which results are of primary interest. To want all results in great detail and as exact as possible according to slab theory is uneconomic and also not always necessary (e.g. infinitely large moments at point supports and reentrant corners).

In CEDRUS the FE mesh is created automatically a zone at a time by specifying certain parameters. The zone boundaries are defined by polygon lines (mesh separation lines), which are also themselves mesh lines and thus are subject to the the conditions regulating separation distances for the structural objects. Automatic mesh generation can also be deactivated in individual zones. In these zones the mesh is input manually using fixed mesh lines.

The attributes box for mesh zones contains the parameters that influence the automatic mesh generation within a mesh zone. Its dialog looks as follows:





Maximum side length: With this, the most important parameter, the fineness of the mesh is determined.

Minimum side length: The minimum element side length is normally determined by the features of the structure's geometry. If, in areas without default values imposed by the structure's geometry, one does not want if possible to go below a certain element side length, this can be done by specifying a value with this optional parameter. However, only values in the range 10-50% of the given maximum element side length are reasonable.

Direction of the mesh lines: The mesh is created using two families of lines in given directions, whereby all input structure objects have to be considered and modified. It is recommended, if at all possible, to select these directions orthogonal to one another.

The following points have to be taken into account:

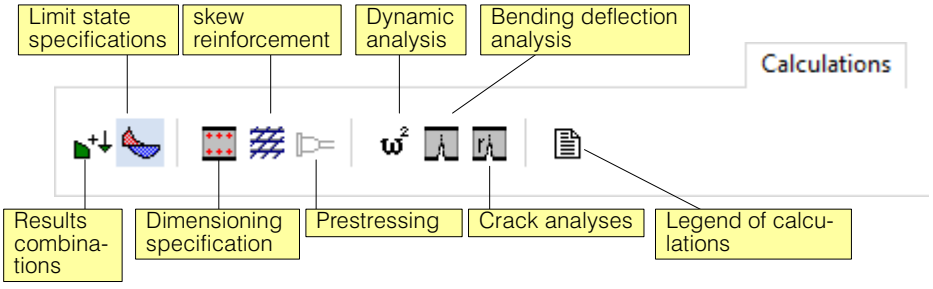
- With the input of a new slab the program automatically creates a mesh box. The mesh length thus defined (maximum side length) is not an intelligent suggestion. It has to be changed by the user for the actual conditions.
- The mesh length to be selected depends on how well the elements with their limited deformability can approximate the exact deformation state of a slab according to plate theory for a specific loading. In the region of a column, for example, the deformation state is much more complex under uniform loading, than in the case of a plate strip hinged on both sides.

  Thus the deciding factor is not in fact the thickness of the slab (although of course this influences the maximum field width), but the distance between critical objects (supports and possibly also point loads) in zones where one wants to have good results. As a rough guide on can say that between critical objects there should be at least 6 elements.

  If in doubt one can check whether the decisive results vary substantially with a refinement of the mesh. Spiky moment isolines in zones important for the results are an indication that the mesh is too coarse.
- The program automatically limits area supports to mesh zones, which are treated-automatically according to the input support attributes.

B 12 Analysis

In this tab sheet prepared specifications for special results output and special analyses are formulated.



B 12.1 Results Combinations

Often one wants to combine results of individual loadings.

If it is a case of determining extreme values of individual result quantities, for this we have the tool of **Limit State Specifications**, which is described in detail in chapt. B 12.2.

Results combinations, on the other hand are rules governing the fixed combination of the results of individual loadings. They contain a list of loading names with the corresponding factors. If, e.g., one wants to obtain the bending deflections under the total load in an analysis with the loadings dead load and live load, then one has to define a results combination, which can be accessed in the results output as for any individual loading.

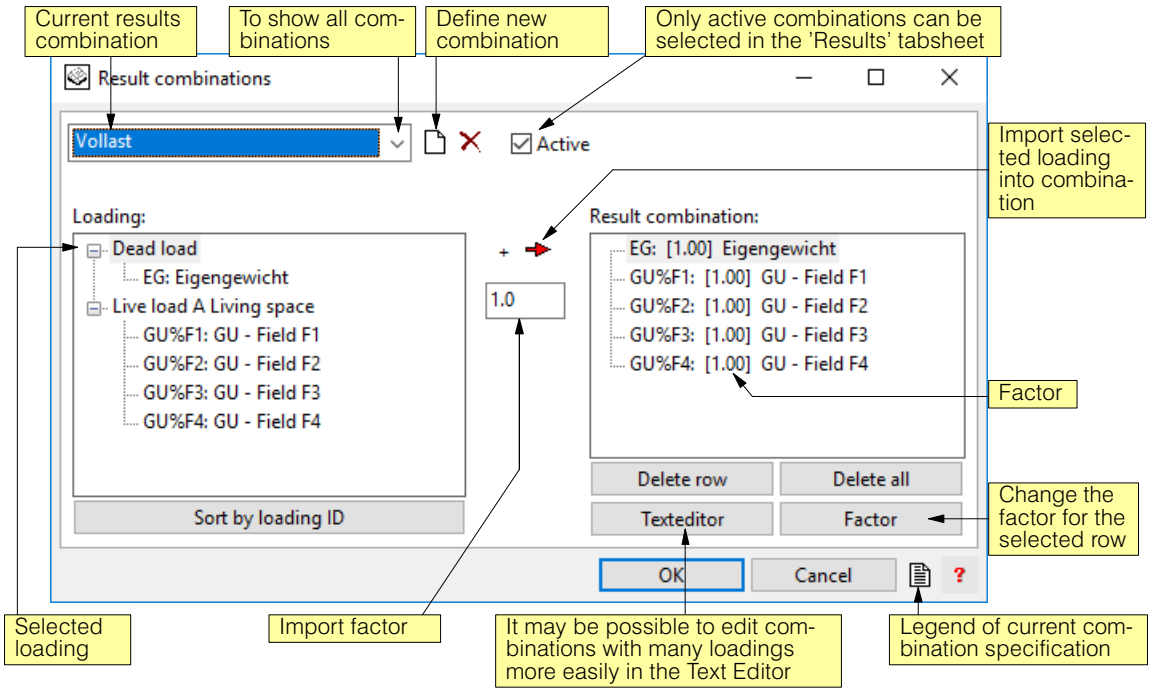
Difference between Results Combinations <> Combination Loadings

In the case of results combinations the results of individual loadings are superimposed. To superimpose results, however, in the case of results obtained from nonlinear analyses for individual loadings is not perimssible. In such cases the combination loadings are available (see Page B-68). With these, the loads of different loadings are combined to form a new loading.

If result combinations are permiited, the corresponding combination loadings give the same results. Thus it is a matter of preference which combination one uses. There are, however, special loadings (e.g. prestress loadings), which one cannot combine for combination loadings, which is why in such cases only results combinations are possible.

B 12.1.1 Input

The above buttons lead to the following dialog, which contains in the left table all combinable loadings and in the right table the combination specification:



B 12.2 Limit State Specifications

The following considerations employ the terminology of the code SIA 260, which agrees more or less with that of the EuroCode.

The aim of every structural analysis is simply the dimensioning of a structure. This is achieved using **limit states**, which among other things requires the choice of **design situations** with the corresponding **load cases**.

Each load case is characterized by a leading action and an accompanying action acting at the same time and consists therefore of a weighted combination of **actions**.

An experienced engineer can - at least for preliminary dimensioning - often restrict himself to just a few points and also without much effort recognizes the critical load cases for the investigated design situations.

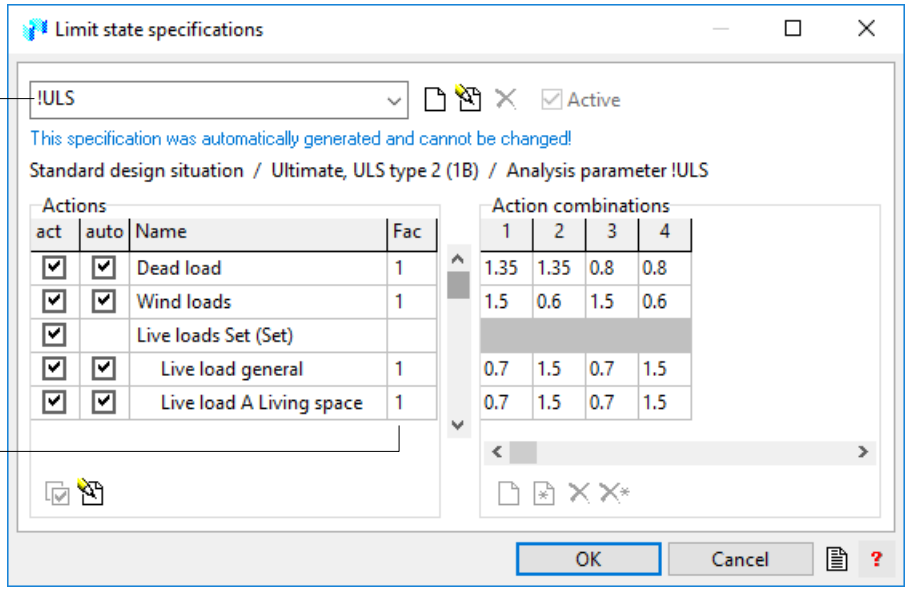
The strength of a program, however, lies in the systematic treatment of numerous sections or points. For many dimensioning tasks it is sensible to work with limit state values of actions, reactions or displacements. These are determined by the Cubus programs on the basis of limit state specification values that uniquely describe the combination rules for the individual loadings. How these limit state specifications are constructed is described below.

B 12.2.1 Overview of the Limit State Specifications

At the top level, the dialog for a simple limit state specification in the programs looks as follows:

List of existing limit state specifications with the management tools: Introduce, Modify and Delete limit state specifications

For each action there are selectable factors, with which the combination factors shown on the right still have to be multiplied Dynamic factors, e.g., can then be considered without having to modify the combination factors or even the loads.



The considered actions are listed on the left side of the dialog, while on the right are the investigated combinations of these actions with the corresponding combination factors. How these combinations came about is clearly seen here: the permanent actions „Dead Load" and „Superimposed Load" are investigated with the factors γ_{sup} (here 1.35) and γ_{inf} (0.8). In addition, we have the variable actions, of which once the leading action ($\gamma_Q = 1.5$) and the wind loads as accompanying action ($\psi_0 = 0.6$) and once the wind action as leading action ($\gamma_Q = 1.5$) and the live loads as accompanying action ($\psi_0 = 0.7$) are considered in turn. The load and accompanying factors that are used depend on

- the code
- the type of action (permanent, variable, accidental...)
- the design situation (persistent, transient, accidental)
- the considered limit state (ultimate load, serviceability)

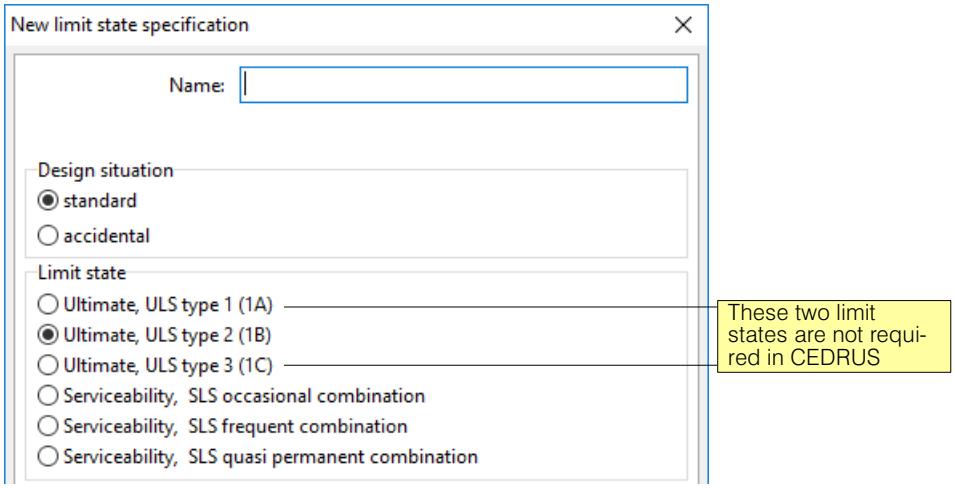
Design situation and limit state are selectable properties of a limit state specification (see Properties dialog of the limit state specification using).

The list of actions on the left side of the dialog is created automatically on the basis of the input loadings, each of which is assigned to an action. The right side with the action combinations can be automatically generated, but can also be arbitrarily defined by adding, deleting or modifying columns. On this automatic generation see chapt. B 12.2.3.

If no action sets (see below) exist, CEDRUS automatically creates the following limit state specifications:

- **!ULS** for the ultimate limit state (Type 2) for the standard design situation
- **!SLS(quasi-permanent)** for the serviceability limit state, quasi-permanent combination
- **!SLS(frequent)** for the serviceability limit state, frequent combination
- **!SLS(rare)** for the serviceability limit state, combination is rare occurrence

Besides, the user can input arbitrary, self-defined limit state specifications. The button shown opens the dialog, in which the design situation and the limit state are specified:



In the next chapter we will cover in detail the actions not yet dealt with.

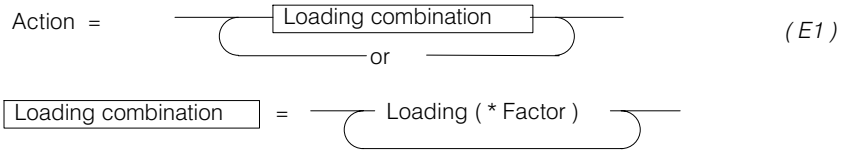
B 12.2.2 Actions

On the definition of actions see chapt. B 10.1 and B 10.3.

Action Specification

Part of the comprehensive definition of an action is the combination rule for the participating loadings denoted by **Action Specification**. Whereas permanent actions often consist of just one loading, for variable actions a complicated superposition is necessary. Consider, for example, the live load actions on a slab with several fields, where the loads are placed in the fields in unfavourable positions and where in addition a vehicle load has to be considered in various positions.

One can imagine as a simple combination scheme - here denoted by **E1** - the comparison of all possible loading combinations. This may be represented as follows in a flow chart:



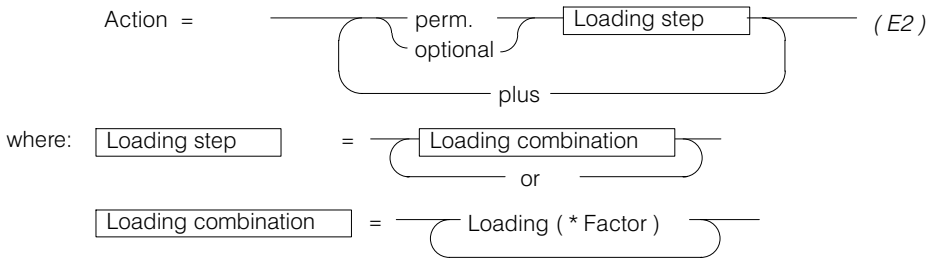
Example: Fieldwise unfavourably applied vehicle load on a three span beam

The 3 loadings A, B and C are applied in the individual spans.
The specification of the action 'vehicle load' according to the flow chart is E1:

A or B or C or AB or AC or BC or ABC

In the case of a five span beam one would have as many as 31 loading combinations to compare with one another. If, in addition, we had two more fields and let's say a vehicle load in n possible places, then the user would be quite overtaxed in his combinatorics skills and an unmanagable number of loading combinations would result.

However, with the extended **superposition scheme** - here denoted by **E2** - a compact and managable treatment of all possible loadings is possible. The simplest way to represent it is an a flow chart:



The scheme is based on an unconditional (permanent) or optional superposition of loading steps. In contrast to scheme E1 no fixed loading combinations are described. It is simply a question of an instruction, i.e. how the results of loadings have to be superimposed to construct the limit state values and how in the process the limit state values have to be constructed.

From the optional loading steps a resulting value for an extreme value at a point is only considered if it is decisive, i.e. the extreme value is increased absolutely, i.e. a positive value increases a maximum value and a negative one a minimum value.

In a simple case the loading step consists of a single loading or of a loading combination. It may also consist, however, of a series of loadings or loading combinations, of which at any one time only one can occur (e.g. different vehicle positions). In this case, in the evaluation of the loading step in each point there is a minimum and a maximum value of the considered result quantity.

The specification for the above example according to scheme E2 is:

A plus B plus C

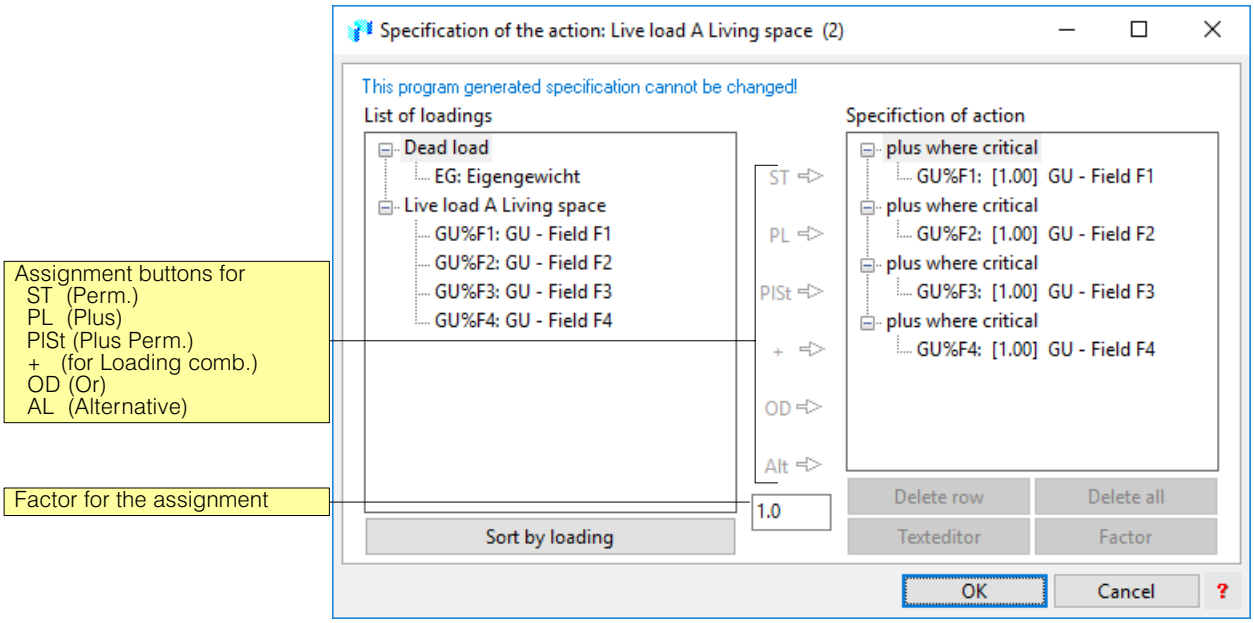
In the case of a five-span beam with the loadings in each span A, B, C, D, E with an additional vehicle in 9 positions (a, b, c, .., i) the specification is:

A plus B plus C plus D plus E plus a or b or c or d or e
or f or g or h or i

The user can specify any arbitrary number of actions following the superposition scheme E2.

☞ *The scheme E1 is contained as a special case in E2 (without 'PERM.' and 'PLUS(PERM.)').*

In the programs these specifications are made in the dialog shown below, which requires no further commentary.



Not described was the possibility known from earlier programversions of alternative superpositions (bottom button). Instead of this possibility, which leads to long and confusing specifications, there is the new concept of „Action Sets", which are discussed below.

Automatically generated action specifications

The program automatically creates for each action an action specification according to the following simple rules:

Permanent actions: all loadings belonging together are summed. Thus, such an action consists of a fixed loading combination.

Variable actions: Each loading, which is assigned to a variable action, is also given at input the superposition attribute, 'additive' or 'exclusive'. **Additive loadings** (a1, a2,...) can act simultaneously, **exclusive loadings** (e1, e2,...) exclude one other (e.g. vehicle in different positions). These actions are constructed according to the superposition scheme E2 described above as follows:

a1 *plus* a2 *plus* a3 *plus* . . . *plus* e1 *or* e2 *or* e3 *or* ...

In cases not covered by this simple rule, the specification must be carried out manually with the help of the dialog shown above.

B 12.2.3 Action Combinations

The action combinations to be investigated are on the right side of the limit state specification dialog.

Automatic creation of action combinations

Depending on the design situation and the limit state, for which a limit state specification is foreseen, the action combinations can be automatically created.

They are constructed according to the following scheme (code-dependent), whereby it is assumed that an action as a leading action is always defined in the same way, as with an accompanying action. The correctness of the created action combinations must be checked in each case by the user and if necessary the specific requirements have to be modified.

SIA 260:

Standard Design Situation, Limit State Ultimate Load

$$E_d = E\,(\,\gamma_G G_k\,,\,\gamma_P P_k\,,\,\gamma_{Q1} Q_{kl}\,,\,\psi_{0i} Q_{ki}\,) \qquad (4.4.3.4)$$

γ_G : load factor for permanent actions
G_k : char. value of the permanent action
γ_P : load value for prestressing actions
P_k : char. value of the prestressing actions
γ_Q : load values for variable actions
Q_k : char. value of variable action
ψ_0 : reduction factor for variable actions

Remark: Only considered for a variable accompanying action $\psi_{0i} Q_{ki}$ (the critical action).

Accidental Design Situation, Limit State Ultimate Load

$$E_d = E\,(\,G_k\,,\,P_k\,,\,A_d\,,\,\psi_{2i} Q_{ki}\,) \qquad (4.4.3.5)$$

A_d : dimensioning value of the accidental action
ψ_{2i} : load factor for the accompanying action i

EC/DIN:

Basic Combination (without Accidental Loading), Limit State Ultimate Load:

$$S_d = \sum \gamma_{G,j} \cdot G_{k,j} + \gamma_{Q,1}\left[Q_{k,1} + \sum_{i>1} \psi_{0,i} \cdot Q_{k,i} \right] (+\,\gamma_p \cdot P_k)$$

Accidental Action Combination:

$$S_{d,A} = \sum G_{k,j} + A_d + \psi_{1,1} \cdot Q_{k,1} + \sum_{i>1} \psi_{2,i} \cdot Q_{k,i}\,(+\,\gamma_p \cdot P_k)$$

G_k : char. values of the permanent loads (actions *dead loads, superimposed loads)*
$Q_{k,1}$,$Q_{k,i}$: char. value of the first or other variable actions
P_k : prestressingng
A_d: dimensioning value of the accidentental action
γ_G : partial safety factors for pemanent actions (1.35 and 1.0)
γ_Q : partial safety factors for variable actions
γ_P : partial safety factors for actions due to prestressing
ψ_0, ψ_1, ψ_2 : combination values for rare, frequent and quasi-permanent actions

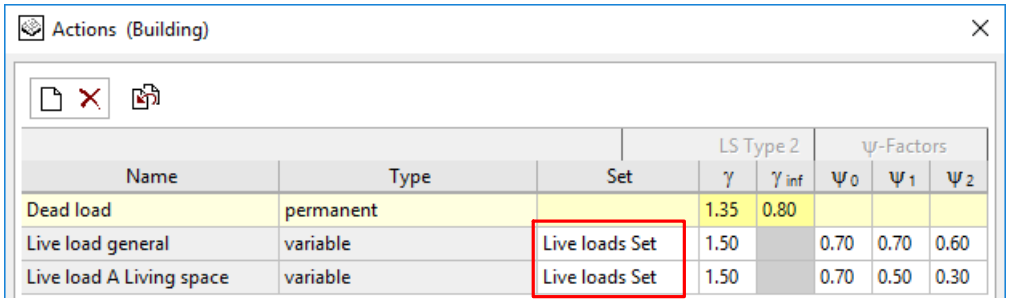
B 12.2.4 Limit State Specifications with Action Sets

Several actions may be combined to form an **Action Set** (e.g. several load models for a bridge). Such actions are not accessed individually in building the action combinations, but the set appears as a whole in a combination (e.g. as leading or accompanying action).

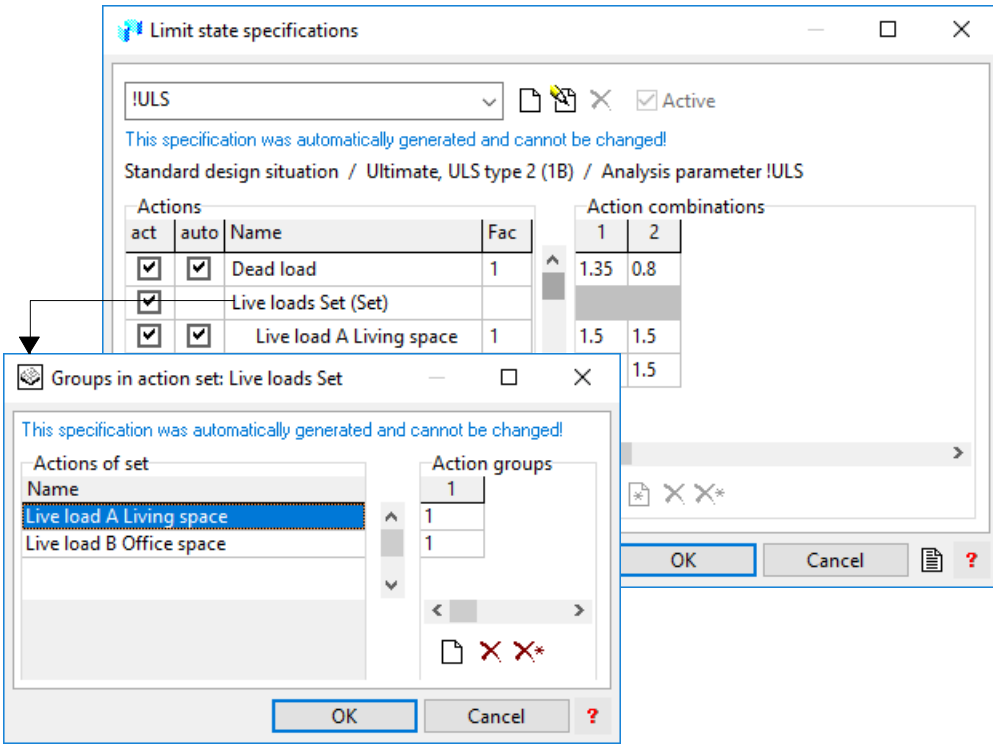
The codes speak in this connection of **Action Groups.** This term is also supported in the Cubus-programs in this way. By it is understood the combinations in which the actions of a set have to be investigated.

Which actions should be combined to form a set is specified in the Actions dialog. All that have the same name in the column set together form an action set.

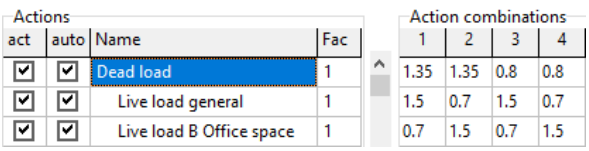
In CEDRUS, the sets are used above all in the following case:



In the top-down analysis of loads, in order not to produce an inflation of actions, the import combination with the variable loads is always assigned to the action 'General Live Load'. In the slab under consideration, however, one will have other action types, e.g. 'LIve Load Office Floors'. All these live load actions would normally appear together as a leading or accompanying action in the automatically created action combinations of a limit state specification. This is just what happens, when these actions together form a set. For this set only the one given action group is necessary, in which all participating actions are input with the factor 1.0. In the participating dialogs this looks as follows:



If the two live load actions do not form a set, the automatically generated action combinations would look as follows:



Since both live load actions are normal actions, each one is considered once as leading action in combination with the other as an accompanying action.

B 12.2.5 Limit Values of Nonlinearly Determined Results

With the use of loading combinations or 'plus' in the action combinations, the results are superimposed that are determined for the participating loadings. Such superpositions, however, are not permitted in nonlinear analyses.

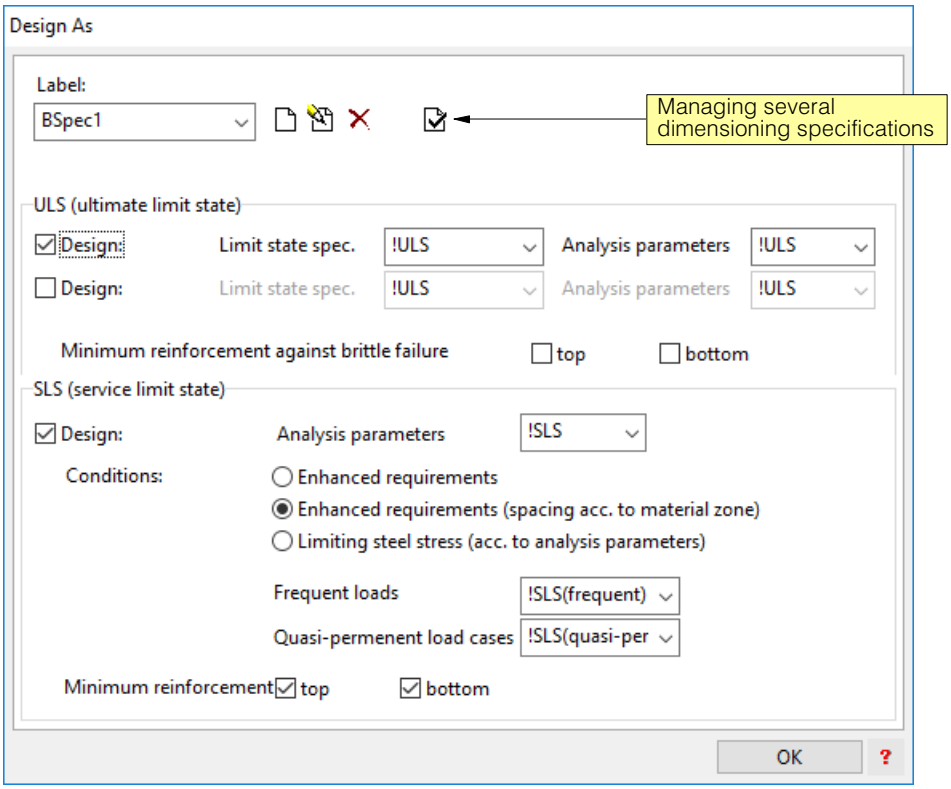
In order, in such cases, nevertheless to obtain useful limit values, the following points have to be considered:

- One has to restrict oneself to a single action, so that in the limit state specification no action combinations are produced.
- In the specification of the single action one may only use the superposition scheme E1, i.e. only 'B1 OR B2 OR B3 ...' , whereby the Bi may only be individual loadings.

  To still be able to work with the input loadings, CEDRUS provides **Combination Loadings**. Hereby the loadings are combined with the required factors to form a new loading before the analysis starts, which is then in the same as a normal loading (see Page B-68).

B 12.3 Dimensioning specification for orthogonal reinforcement

The button opens the following dialog, which specifies the criteria for the dimensioning of standard orthogonal untensioned reinforcement:



The dialog makes available the two dimensioning criteria 'ULS' and 'SLS' that are subsequently described in detail, which can be activated individually or together.

If both criteria are chosen, they are worked through in sequence and the maximum values of the maximum values of reinforcement are output.

The dimensioning is performed exclusively for bending action. It is based on the conditions

$E_d \leq R_d$ for the ultimate limit state (ULS) or

$E_d \leq C_d$ for the serviceability limit state (SLS)

with

E_d: effect of action, i.e.
dimensioning value of the effects of actions (ULS) or
the critical effect of an action (SLS)

R_d, C_d : resistance, i.e.
dimensioning value of the resistance (R_d)
C_d: serviceability state (C_d)

☞ *The dimensioning in the case of punching shear is carried out independently of that for the bending dimensioning (see chapt. B 9).*

Effect of Action

The E_d are the limit values of the reinforcement moments (see page B-114) on the basis of a limit value specification, which has to be selected to accord with the limit state.

Resistance

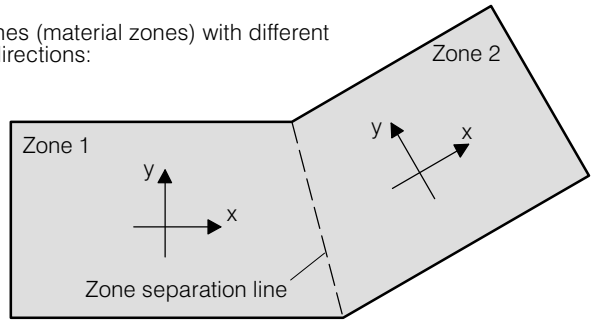
In the determination of the resistance the following are input:

- **Dimensioning type**: the dimensioning of the reinforcement can be carried out in the results points of the slab (where the results are calculated) or for beam sections. The corresponding explanations are given in the following sections.
- **Slab thickness and slab material** (properties of the material zones)
- **Reinforcement layers and reinforcement materials** (properties of material zones)
- **Analysis parameters:** This includes all other parameters required for the determination of the resistance, such as limiting strains, resistance factors, etc. The description is given in a subsequent section.

B 12.3.1 Dimensioning in the Results Points

With dimensioning in the results points at the centres of all elements and nodes the minimum required reinforcement cross section is determined for each upper and lower reinforcement layer in two directions x, y at right angles to one another. The x direction (output direction) may be different from zone to zone (property of the „Material Zones"). In the case of the reinforcement cross sections it is a question of cross sectional areas per unit length normal to the reinforcement direction.

Slab zones (material zones) with different output directions:

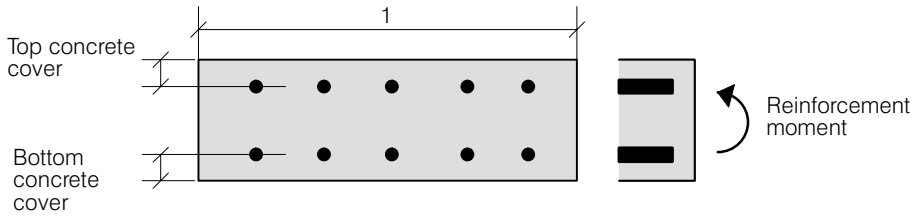


Reinforcement directions = output directions

If the tensile reinforcement is not sufficient by itself a compression reinforcement is introduced. Although, for example, there may not be a positive reinforcement moment over a column, it is nevertheless possible to output a lower reinforcement layer.

Basic Reinforcement

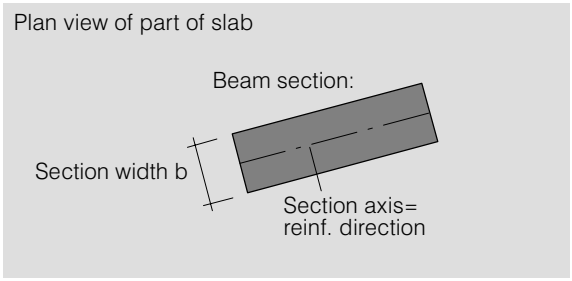
With the properties of the material zones a basic reinforcement can be input for the four reinforcement layers. In the output of results only the additional required reinforcement cross sections are given.

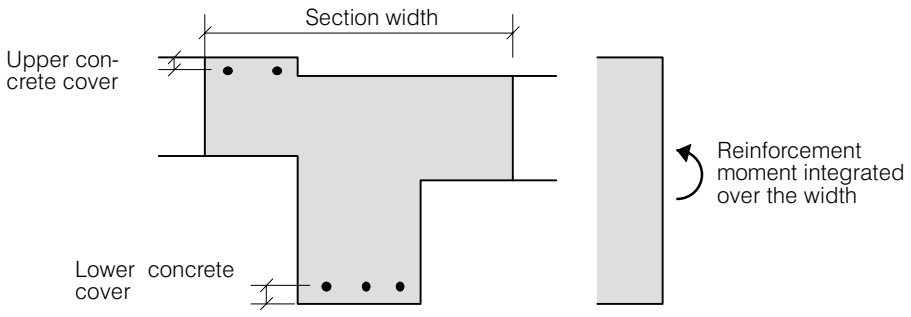


The amount of cover is a property of the material zones

B 12.3.2 Dimensioning for Beam Sections

In the case of beam sections the dimensioning is carried out for an upper and a lower reinforcement layer in the direction of the axis of the reinforcement. The stresses are integrated over the width of the section and the strip is dimensioned as a beam. It is assumed that there is sufficient slab resistance normal to the direction of the section. The sections can be in any direction.





If a beam section at a point stretches over several zones (as shown), then the reinforcement layers are given by the zones with the highest top edge or the lowest bottom edge.

The values of concrete cover defined in the material zones are used. If the beam section does not lie in either the X or Y directions, the concrete cover is obtained by interpolation of the input values of concrete cover.

The dimensioning of the beam sections is also carried out exclusively for bending action (without increasing the values to take into account shear forces). If necessary, compression reinforcement is also dimensioned.

For the determination of the reinforcement moments the program works with the section force components of the individual loadings integrated over the width of the section. The limit values of reinforcement moments are determined from these section force components, i.e. the reinforcement moments integrated over the width of the section are not used.

For dimensioning purposes **downstands** are treated as beam sections. The input effective widths of a downstand on the left and right of the downstand axis determine the width of the beam section corresponding to the downstand.

Basic reinforcement

For the properties of the material zones, basic reinforcements can be entered for the four reinforcement layers. These basic reinforcements have no influence on the beam section output.

B 12.3.3 Analysis Parameters

Analysis parameters are the parameters that are combined in so-called analysis parameter sets **(AP)**, which are necessary to be able to carry out resistance analyses for given cross sections (resistance factors, limiting strains, etc.).

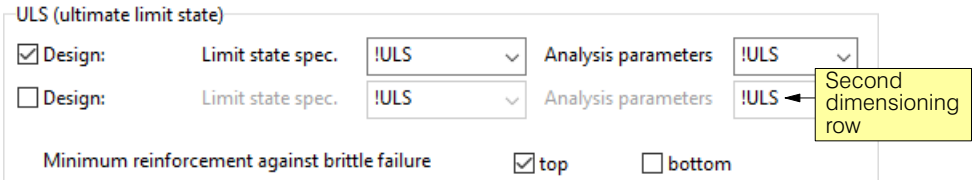
For each supported code the following two not modifiable AP are predefined:

- !ULS : for the ultimate load limit state
- !SLS : for the serviceability limit state

In addition, user-defined AP with arbitrary specified parameters can be input.

The management of the AP is carried out in the menu „Settings“ > „Analysis Parameters“.

B 12.3.4 Dimensioning for the ULS (Ultimate Load State)



As stressing values the limiting values of the reinforcement moments are used, which are given by the selected limit state specification.

In the selected AP as limit state criterion „Limiting Strains“ must be specified. Thus the limit state is reached when the strains in the tensile reinforcement or the compressive stresses at the edge of the compression zone reach the values specified in the AP.

Second dimensioning row

In the case of manually input dimensioning specifications a second dimensioning row can be activated below a ULS . If it is activated, then in addition dimensioning is carried out for the limit state specification selected here. This may be useful, e.g., in the case of additional dimensioning for earthquake action (accidental design situation, ultimate load state).

Minimum reinforcement

The minimum reinforcement for the ULS is meant to prevent brittle failure if a crack is formed, i.e. it has to be dimensioned for the cracking moment. If the switch for the minimum reinforcement is activated (separately for top and bottom reinforcement), then in the dimensioning this is taken into account where tensile stresses arise as a result of bending action.

SIA code: According to Section 5.5.3.2 the minimum reinforcement for slabs must be specified for the requirements of serviceability. Regarding prevention of brittle failure Table 16 prescribes f_{ctd} as the tensile concrete stress for the determination of the cracking moment. For the determination of resistance Fig. 31 also specifies a limiting stress in the reinforcing steel of 435 N/mm^2, which corresponds to a value of f_{yd} of a B500. In the same way, for the resistance the value of f_{yd} for the particular steel is used.

EC2, DIN1045-1: The cracking moment is determined for a concrete strength of f_{ct} and for the resistance one works with a limiting stress f_{yd} in the reinforcing steel.

B 12.3.5 Dimensioning for the SLS (Serviceability Limit State)

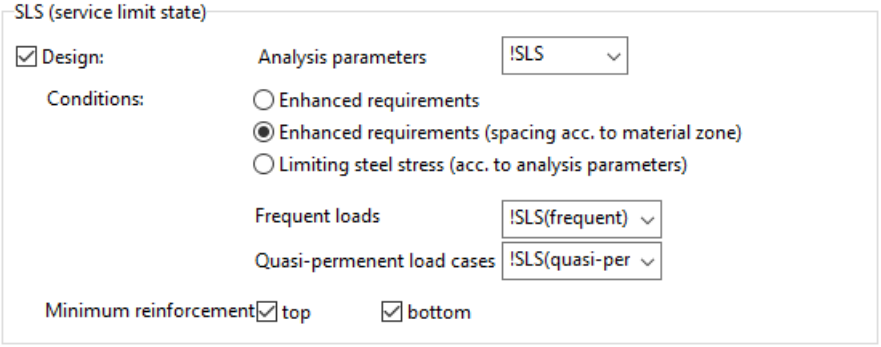
The serviceability limit is a bending resistance, which is determined on reaching the maximum value of the stress in the tensile reinforcement. The selected AP must be specified correspondingly. Whereas in the SIA code limits for the tensile stresses in the steel are specified directly, in codes based on the EC these are given indirectly using the maximum permissible crack widths.

Depending on the condition, the stressing is given by the limiting values of the reinforcement moments for frequent and/or quasi-permanent load cases. The corresponding limit state specifications have to be used.

The permitted maximum value of the tensile stress in the steel depends on the code and/or the specified condition. In the case of some conditions the concrete cover (i.e. distance from the edge to the reinforcing bars) is required. This is specified in the properties for the material zones.

The part of the dialog for the SLS is code-dependent:

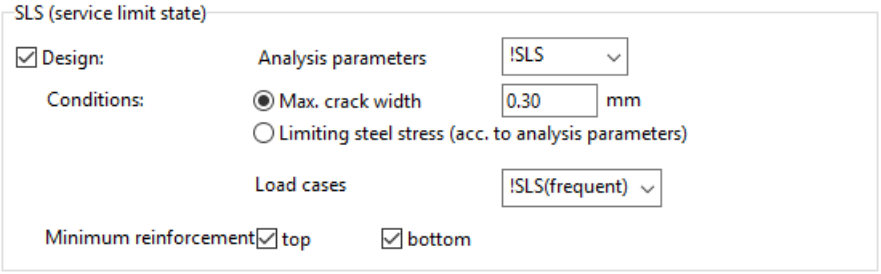
SIA-code:



The permissible steel stresses are obtained from Table 16 and Fig. 31 of SIA262. The **minimum reinforcement** corresponds to that obtained for dimensioning for ULS (see above).

With the condition 'Limiting Steel Stress' the corresponding value given in the AP is used.

EC2, DIN1045-1:



The serviceability verification is carried out indirectly using a limiting concrete cover or a limiting diameter of the reinforcing bars. The user prescribes a maximum crack width w_k (e.g. 0.3mm) and together with the distance between the steel bars the admissible steel stress is determined from the corresponding tables:

EN 1992-1-1: 2004:	Table 7.3N
DIN 1045-1:	Table 21

☞ *This procedure is described under 'Limiting the Crack Width without Direct Analysis'. The results can only be compared approximately with those of an analysis of the crack width. The codes mention however: "... if the minimum reinforcement according to. ... is complied with, exceeding the crack width is unlikely ...".*

With the condition 'Limiting Steel Stress' the corresponding value given in the AP is used.

Minimum reinforcement

If the switch 'Minimum Reinforcement' (separately for the top and bottom steel layers) is activated, the minimum reinforcement is taken into account in the dimensioning as follows:

According to EN 1992-1-1:2004: Section 7.3.2 resp. DIN1045-1: Section 11.2.2 for the minimum reinforcement:

$$A_{s,min} = k_c \, k \, f_{ct,eff} \, A_{ct} / \sigma_s$$

k_c: factor to take into account the stress distribution and the change of the inner lever arm z in the transition from State I to State II. For pure bending $k_c = 0.4$.

k: factor to take into account nonlinearly distributed concrete tensile stresses, which lead to a decrease in constraint (factor depending on slab thickness, see codes).

$f_{ct,eff}$: concrete strength:
Normally $f_{ct,eff} = f_{ctm}$ or less, if the crack formation is expected to take place before 28 days. By means of the analysis parameters it is possible to define a corresponding reduction factor.

A_{ct}: concrete area in tension (State I)

'thick' slabs: A range of action is defined for the reinforcement, which is defined by the following edge distance of reinforcement: 2.5 d_1 or (h-x)/2 (for drawings see codes).

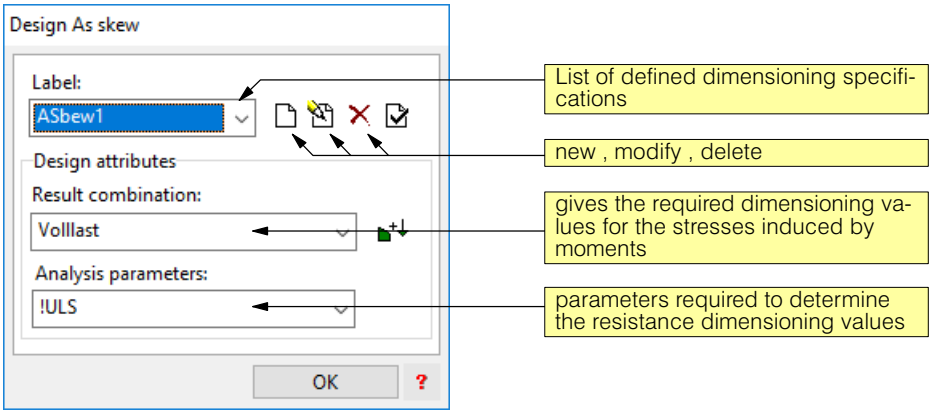
In DIN 1045-1 or DAfStb Issue No. 525 + DAfStb 466 there are additional rules (in figure h_{eff}/d_1 depends on h/d_1), which are also taken into account in CEDRUS .

σ_s: admissible steel stress (the same value is used as in a SLS analysis under static loading). With the condition „Limiting Steel Stress“ the steel stress specified in the AP (analysis parameters) is used.

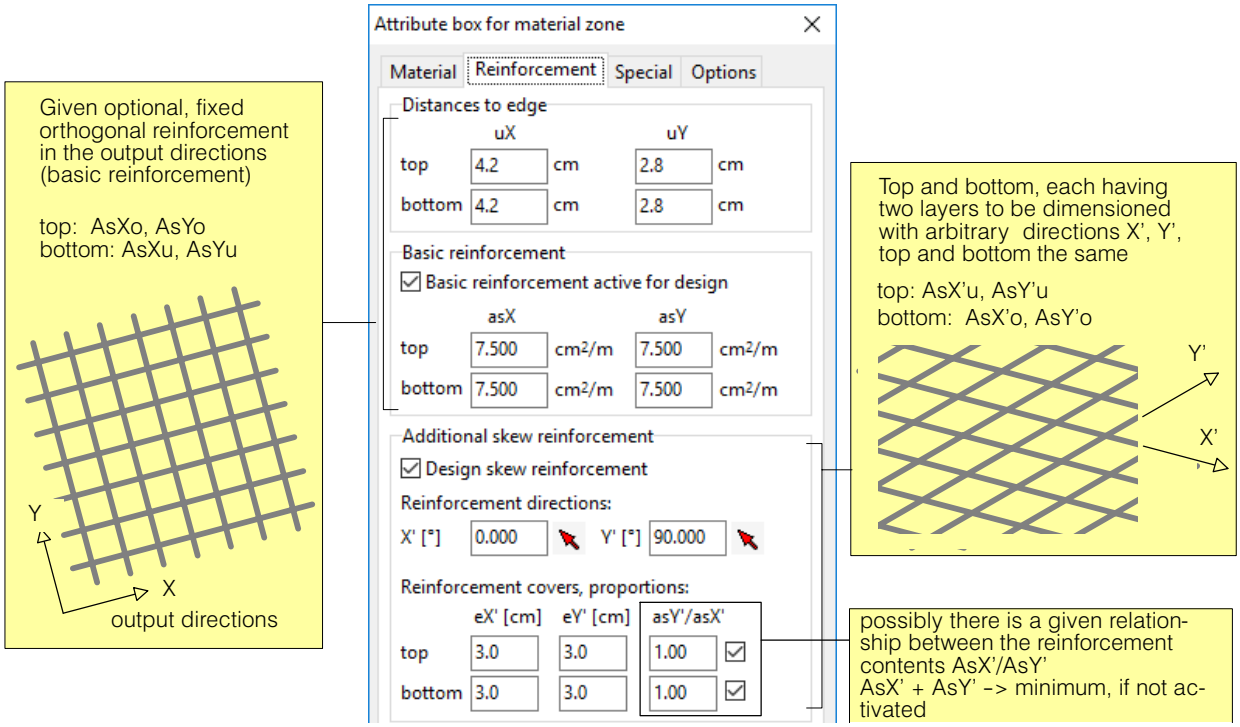
B 12.4 Dimensioning Specification for Skew Reinforcement

B 12.4.1 Input

The shown button in the tab sheet 'Calculations' leads to the following dialog, in which the dimensioning specifications for skew reinforcement are input and managed:



The zonewise configuration of existing and still to be dimensioned reinforcement layers is a property of the material zones. The input is made in the attributes box for material zones in the tab sheet 'Reinforcement', which looks as follows:



B 12.4.2 Basis of Dimensioning

The dimensioning is only carried out for the ultimate load state. The basis is given by the generally recognized normal moments yield criterion, according to which the slab cross section yields, if the external moments Mn of the slab just reach the plastic resistance Mpn in a section with the normal direction n.

The ultimate limit state is verified, if the following condition is fulfilled for each direction n:

$$M_n \le P_n \qquad M_n \ge -N_n$$

where:

M_n: dimensioning values of the stresses induced by moments in the section with the normal direction n

P_n, N_n: dimensioning values of the slab resistance (positive) for a positive or negative bending action in the section with the normal direction n

Formulated in the orthogonal x-y coordinate system the following **nonlinear yield criterions** result:

$$P_x - M_x \ge 0$$
$$P_y - M_y \ge 0$$
$$(P_x - M_x)(P_y - M_y) - (M_{xy} - P_{xy})^2 \ge 0$$
$$N_x + M_x \ge 0$$
$$N_y + M_y \ge 0$$
$$(N_x + M_x)(N_y + M_y) - (M_{xy} - N_{xy})^2 \ge 0$$

where:

M_x, M_y, M_{xy} : given slab section forces in the x-y coordinate system

P_x, P_y, P_{xy} : slab resistances transformed into the x-y coordinate system for the bottom tensile reinforcement

N_x, N_y, N_{xy} : slab resistances transformed into the x-y coordinate system for the top tensile reinforcement

Under the assumption of a constant lever arm of the inner forces for each reinforcement layer and the possibility of superposition the following relationships apply for the transformations of the slab resistances for the (skew) reinforcement into the orthogonal x-y coordinate system:

$$P_x = \sum_i P_i \cos^2 \phi_i \qquad N_x = \sum_i N_i \cos^2 \phi_i$$

$$P_y = \sum_i P_i \sin^2 \phi_i \qquad N_y = \sum_i N_i \sin^2 \phi_i$$

$$P_{xy} = \sum_i P_i \sin \phi_i \cos \phi_i \qquad N_{xy} = \sum_i N_i \sin \phi_i \cos \phi_i$$

where:

P_i : resistance of the i-th bottom reinforcement layer (positive)

N_i : resistance of the i-th top reinforcement layer (positive)

ϕ_i : angle of the i-th reinforcement layer with respect to the x axis

The resistances P_i and N_i, respectively, of a reinforcement layer i calculated for pure bending in the rectangular cross section corresponding to the input materials, edge distances to reinforcement (concrete cover), analysis parameters and the provisions of the selected code.

B 12.4.3 Section Forces for Dimensioning

The limit values of reinforcement moments used in the dimensioning of orthogonal reinforcement cannot be used here, since they are based on a linearized yield criterion. Thus the dimensioning of skew reinforcement layers for a fixed superposition of section forces is based on different loadings (denoted in the program by Results Combination). The combination factors must correspond to the ultimate load state (structural safety) and the desired design situation.

B 12.4.4 Analysis Parameters

Also the analysis parameters required for the determination of the bending resistances have to correspond to the limit state of ultimate load and the desired design situation.

B 12.4.5 Apportioning Criterion

The solution of the sought reinforcement cross sections is not unique, i.e. it involves a relationship between AsX'u and AsY'u or between AsX'o and AsY'o.

To obtain a unique solution, one can choose between of the two following conditions (separately for top and bottom reinforcement layers):

1) The sum of the skew reinforcement areas should be minimum (in each point)

   AsX' + AsY' -> Minimum

2) The ratio of the two reinforcement cross sections is given by

   AsX' / AsY' = λ = given

The conditions can be prescribed differently for each zone.

B 12.4.6 Verification of Concrete Compressive Force

There is no automatic dimensioning of compressive reinforcement. On the other hand, for the principal directions of stresses it is verified that the concrete does not fail, i.e. that the maximum compression for the specific analysis parameters is not exceeded.

For this purpose the dimensioned cross sections of all reinforcement layers are transformed into the principal directions of stressing (same formulas as for resistances), and each is calculated with a concentrated reinforcement top and bottom, each time in the centre of gravity of all layers. If the verification conditions are not fulfilled, this is indicated by the program.

B 12.5 Specifications for Dimensioning with Prestressing

In the case of prestressed slabs, the tendons can be considered to act on the resistance side in the dimensioning of the untensioned reinforcement. This type of dimensioning is only possible for beam sections and requires a special Specification dialog. The corresponding button (marked above) is only active if tendons were input.

For the input of tendons the Prestressing Module is required (Option V). For a description of the Prestressing Module including the above button see the respective Section of the manual.

B 12.6 Dynamic Analysis

With this module the natural vibrations, consisting of the eigenfrequencies and the associated vibration modes of a slab for arbitrary mass distributions, can be determined.

For natural vibration analyses the Dynamics Module is required (Option D). For a descriptioof the Dynamics Module including the above button see the respective Section of the manual.

B 12.7 Specification of Bending Deflection Analyses

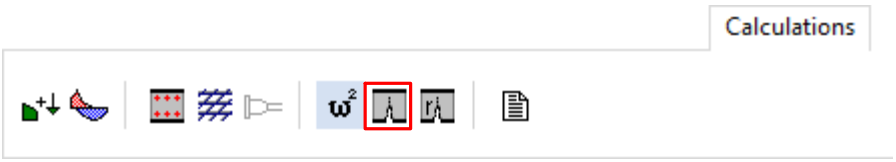
Bending deflection analyses taking into account cracking and long-term effects represent a very difficult task, for which there are hardly any exact methods, no matter how complex and time-consuming they are. Very many parameters and boundary conditions play an important part, which in an actual case are also very difficult to obtain in their entirety.

Thus one often depends on estimates of bending deflections. With the method described here CEDRUS provides two simple methods, one based on SIA 262 and the other on the Eurocode.

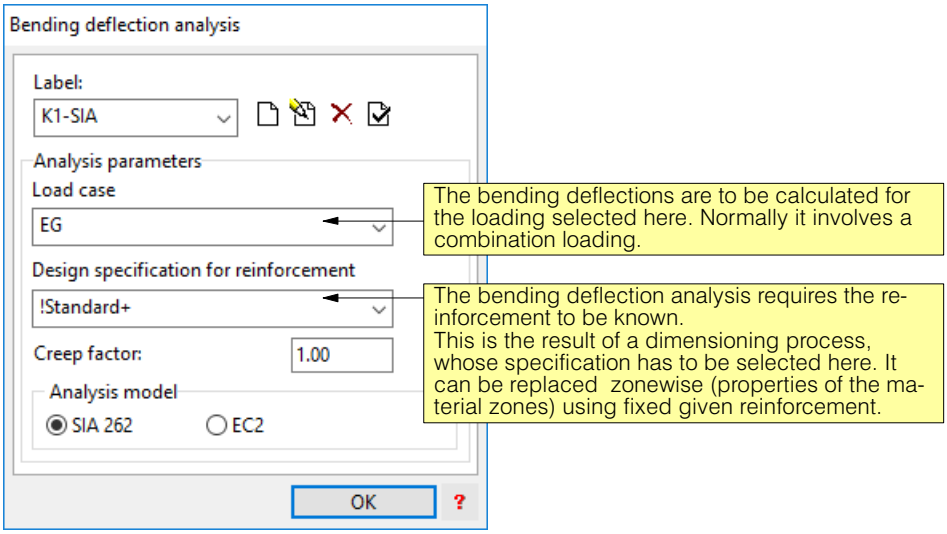
⚠ The normal force from any prestressing loads is not taken into account. The deflections are thus somewhat over-estimated in the cracked state. With the structure type 'slabs with normal forces', the deflection calculation is not available.

In the following there is a description of the input data and then of the analysis methods.

B 12.7.1 Input Data

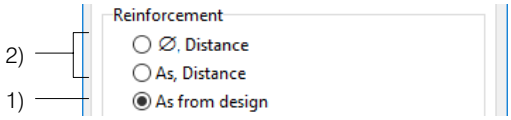


The button shown, in the tab sheet 'Calculations', opens the following dialog, in which the specifications for bending deflection analyses are input and managed:



Regarding the reinforcement

The bending deflection analysis depends on the reinforcement, i.e. it should be based on the reinforcement actually installed. There are two ways of specifying the reinforcement, which can be selected in the material zone in the associated Attributes Box (see B 8.4):



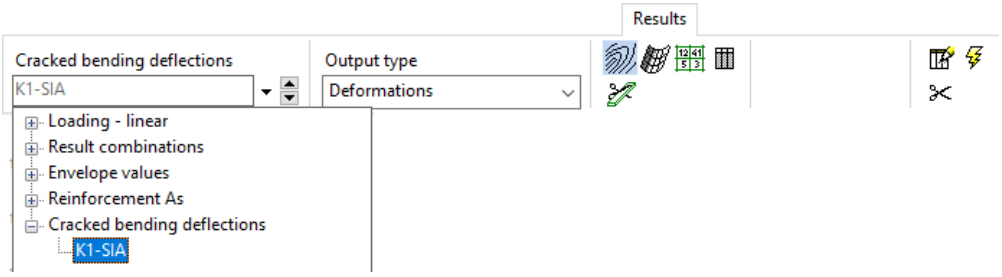
1) Setting **„As from design“**: In this case the dimensioned reinforcement is used that is based on the dimensioning specification selected in the above dialog. This reinforcement corresponds to the minimum required cross section at every point and cannot therefore be rearranged. Especially in the region of columns or walls the installed reinforcement may differ considerably from the dimensioned reinforcement. The advantage of

this type of input is minimum effort with the input. The downside, however, is that often the bending deflections are substantially overestimated.

2) The reinforcement cross sections for the four layers can be input as constant values for each material zone (settings **„∅, Distance"** or **„As, Distance"**). To input the reinforcement actually installed in this way is practically impossible for the whole slab. It is sufficient to restrict oneself to a few critical zones.

⚠ If one selects the dimensioned reinforcement, it can happen that the compression reinforcement requirement in a point is unrealistically large with a value > 3% , so that the formulas on which the algorithm is based are no longer appropriate. The program then issues the error message 'Compression Reinforcement > 3% in Element ...', and one has to check the dimensioning results and one may have to select in this zone the second type of input (Type 2).

The results can be accessed in the Results Tree ('Results' tab sheet) in the branch „Cracked bending deflections analysis":



B 12.7.2 Method based on SIA262

Standard default values (Section 4.4.3.2.2 ff)

For the uncracked state ($M_d < M_r$):

$$w_{c\varphi} = f_u \, w_c \qquad \text{(Formula 86)}$$

$$f_u = (1 + \varphi)$$

where:

w_c : homogeneous deflections, short-term
$w_{c\varphi}$: homogeneous deflections, long-term
φ : creep coefficient
M_d : dimensioning value of bending stresses
M_r : crack moment, reached with an extreme tensile strain at edge of f_{ctd}
f_{ctd} : dimensioning value of concrete strength

For the completely cracked state without normal force:

$$w = f_g \, w_c \qquad \text{(Formula 87)}$$

$$f_g = \frac{1 - 20\rho'}{10\rho^{0.7}}(0.75 + 0.1\varphi)\left(\frac{h}{d}\right)^3$$

whereby, in addition to the above:

ρ : geometrical reinforcement of the tensile zone
ρ' : geometrical reinforcement of the compressive zone
h : slab thickness
d : effective depth

Method

Given: CEDRUS slab
loading B, for which the deflections have to be determined
creep coefficient φ
reinforcement (ρ, ρ' in each point), input either as the result of dimensioning or zonewise

Sought: slab deflections for the state of cracking and long-term effects

The method is based on the indirect proportionality of deflections w and slab stiffness s.

$$w \sim \frac{1}{s} \quad \text{resp.} \quad s \sim \frac{1}{w}$$

The above formula provides factors f for deflection. If one multiples the slab stiffness by 1/f, with a new slab analysis one obtains directly the desired deflections. This is exact if the same factor f applies to the whole slab, which is the case, e.g., if the slab exhibits no cracking at all.

If the factor is variable, because for example some zones are not cracked and others are or because the reinforcement varies, then with the corresponding stiffness adjustments in the various zones there are redistributions of the sectional forces and the solution therefore is no longer exact. Here, however, not exact does not mean wrong or not useable, but with regard to reality there is an internal stress state that depends on various factors, such as load history, which strictly speaking cannot be analysed.

Thus the method has been implemented as sketched out above and comprises the following steps:

1) Determination of the reinforcement. This is normally the result of dimensioning (see chapt. B 12.3). In special cases, however, it can also be given explicitily for individual zones (properties of the material zones).
2) Calculate the section forces for the loading B, for which we want to determine the deflections.
3) From these find out where the slab is cracked.
   On crack moments see SIA262 4.4.1.3: $f_{ctd} = k_t * f_{ctm}$; $k_t = 1/(1+0.5t)$; t = h/3
4) From the crack moments one can also see if the tensile zone is at the top or at the bottom (for ρ and ρ', respectively)
5) Based on 1),3),4) for each element the deflection factors f_e are determined and from them the stiffness factors $s_e = 1/f_e$. This is carried out assuming orthotropic conditions, i.e. separately for M_x, M_y and M_{xy}.
6) The slab deflections are reanalysed with the changed element stiffnesses. The element stiffnesses are based on the modulus of elasticity of the material classes used and the stiffness factors s_e determined above.

   In addition to the deflections the associated crack pattern is also output.

☞ *For the linearly calculated deformations the E-moduli for the stiffness of the slab are taken from the properties of the chosen concrete material taking into account any stiffness correction factors specified in the material boxes. For the the bending deflection analyses, however, the E-moduli of the material classes are used without any correction factors.*

B 12.7.3 Method based on Eurocode 2

Standard default values

$$\alpha = \varsigma\, \alpha_{II} + (1 - \varsigma)\, \alpha_I = f\, \alpha_c \qquad (7.18)$$

$$\varsigma = 1 - \beta \left(\frac{M_r}{M_d}\right)^2 \;;\; \varsigma = 0 \text{ für } M_d < M_r \qquad (7.19)$$

where:

α : deformation quantity (bending, rotation, curvature).
In this method of analysis the α values now represent curvatures

α_c : homogeneous curvature, short-term

α_I : curvature in State I (homogeneous), with long-term effects
$\alpha_I = \alpha_c\,(1 + \varphi)$

α_{II} : curvature in State II (cracked state)

φ : creep coefficient
f : curvature factor α/α_c
ζ : distribution factor according to EC2
M_d : dimensioning value for bending action
M_r : crack moment, reached with an extreme (edge) tensile strain of f_{ctd}
f_{ctd} : dimensioning value of concrete strength
β : factor to take into account the loading duration
1.0 for individual short-term loading, otherwise 0.5 (in the program fixed at 0.5)

Method

The method is based on the same idea as for the SIA code, i.e. on the indirect proportionality of deflections w and slab stiffness s.

$$w \sim \alpha \sim \frac{1}{s} \quad \text{resp.} \quad s \sim \frac{1}{\alpha} \sim \frac{1}{w}$$

The difference to the SIA method is simply the way in which the stiffness factors are determined.

Outline of steps:

1) Determination of the reinforcement. This is normally the result of dimensioning (see chapt. B 12.3). In special cases, however, it can also be given explicitly for individual zones (properties of the material zones).
2) Calculate the section forces for the loading B, for which we want to determine the deflections.
3) For each element (index e), determine (orthotropic, i.e. separately for M_x, M_y and M_{xy})

   - the curvature α_{ec} (homogeneous, short-term) $\alpha_{ec} = \frac{12 M_d}{E h^3}$

   - the curvature $\alpha_{eI} = \alpha_{ec}\,(1 + \varphi)$

   - the curvature α_{ell} for M_d. This is carried out using FAGUS procedures taking into account the reinforcement determined in step 1).

   - of $\varsigma_e = 1 - \beta \left(\frac{M_r}{M_d}\right)^2$

   - the total curvature for M_d: $\alpha = \varsigma\, \alpha_{II} + (1 - \varsigma)\, \alpha_I$

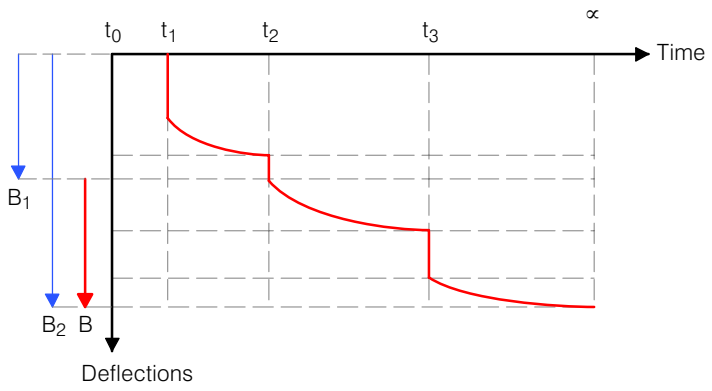
   - the curvature factors $f_e = \alpha/\alpha_c$ and the stiffness factors $s_e = 1/f_e$

4) The slab deflections with the modified element stiffnesses are recalculated. The element stiffnesses are based on the modulus of elasticity of the material classes used and the stiffness factors s_e determined above.

☞ *For the linearly calculated deformations the E-moduli for the stiffness of the slab are taken from the properties of the chosen concrete material taking into account any stiffness correction factors specified in the material boxes. For the the bending deflection analyses, however, the E-moduli of the material classes are used without any correction factors.*

B 12.7.4 Take note at bending deflection verification

- It is particularly difficult to provide a suitable value for the creep factor, since a load is composed of long-term components which are used at different points in time as well as from short-term active parts. Often, only a rough estimate is possible.
- If a partial load, e.g. Only for variable loads or loads which are used after the installation of technical equipment, the result is usually obtained only from the difference between two calculations:



Time:	
t_0:	Manufactur of slab
t_1:	Effect of dead load
t_2:	Imposed loads
t_3:	Variable loads
Calculs:	
B:	Result = B2 - B1 (Example)
B_1:	Calculation t_2
B_2:	Calculation t_3

B 12.8 Specification of Crack Analysis

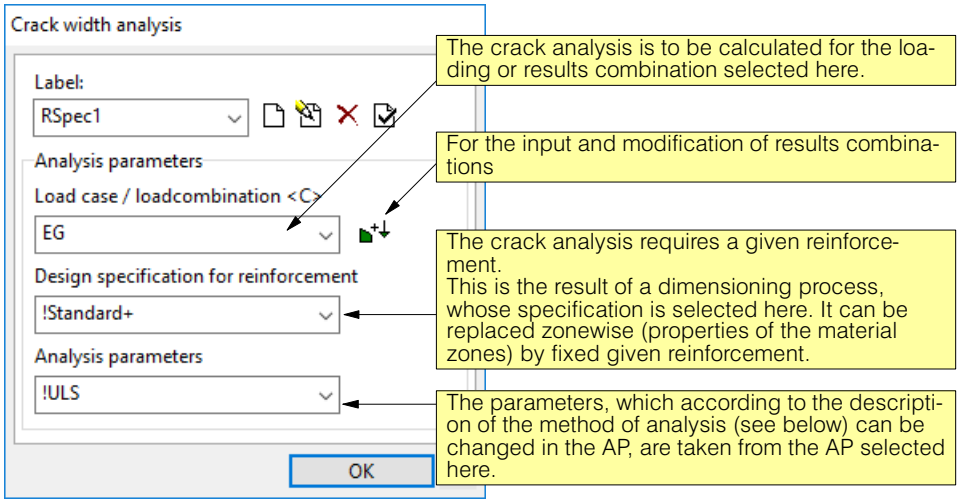
With crack analyses for the given loading or results combination the crack widths and the distances between the cracks are determined. The required reinforcement for this is the result of dimensioning (specifying of the corresponding dimensioning specification), whereby it can be overwritten zonewise for fixed reinforcement cross sections. Besides the reinforcement cross sections the spacing between the bars or bar diameters also required.

Below is a description of the input followed by the method of analysis.

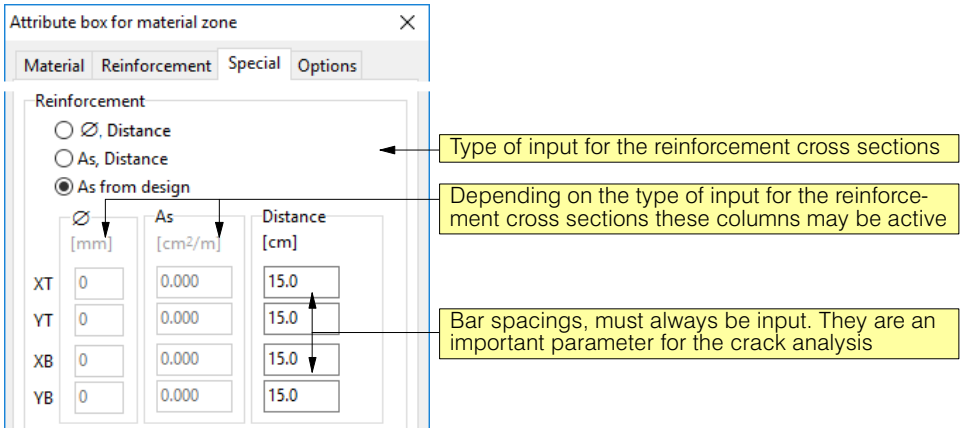
B 12.8.1 Input

Specification dialog

The button shown in the tab sheet 'Calculations' leads to the following dialog, in which the specifications for crack analyses are input and managed:



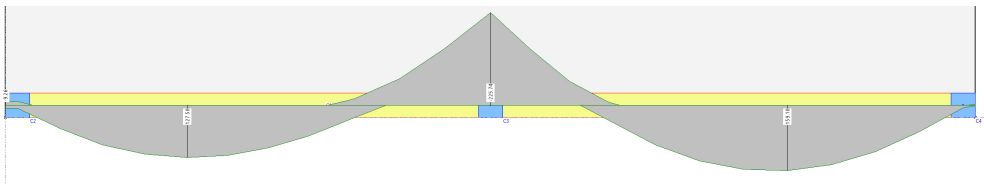
Regarding the reinforcement



⚠ The „As from design" setting in the attribute box for material zone dialog must be understood as a quick setting for determining the required diameter. For the crack calculation, the diameter of the reinforcement is required (see formulas on the following page), which is why you have to change to „Ø, Distance".

In the following tabsheet of the Material Boxes this reinforcement can be overwritten for each material zone. The additionally required bar spacings are also input here.

Explanation to the choice: The program works with the necessary values at the selection „As from design" at each place. Since distributed reinforcement is used in practice, however, the crack calculation must also be carried out with this selected reinforcement.



B 12.8.2 Method of Analysis

The determination of the crack widths and the distance between cracks is carried out according to the codes DIN 1045-1:2001-07 and EN1992-1-1:2004. For all other codes (incl. SIA) the verification is carried out according to EN1992. (Materials taken from the national codes, but using the procedure described below).

The **crack width** according to EN1992 is given by (similar procedure using DIN):

$$w_k = s_{r,max}\,(\epsilon_{sm} - \epsilon_{cm})$$

where

$$(\epsilon_{sm} - \epsilon_{cm}) = \frac{\sigma_s - k_t \frac{f_{ct,eff}}{\rho_{p,eff}}(1 + \alpha_e \rho_{p,eff})}{E_s} \geqslant 0.6\,\frac{\sigma_s}{E_s}$$

σ_s : steel stress

k_t : = 0.4 (for long-term load action);
the value can be changed in the Analysis Parameters

$f_{ct,eff}$: mean value of the effective tensile strength of concrete

$f_{ct,eff}$: $= f_{ctm}$ or smaller, is expected before 28 days

The reduction factor of 1.0 given in AP (Analysis Parameters) can be modified there, if necessary.

$\rho_{p,eff}$: reinforcement content (in the zone "$A_{c,eff}$" according to the figure)

α_e : ratio $\frac{E_s}{E_{cm}}$

The **spacing between cracks** is:

$$s_{r,max} = k_3\,c + k_1\,k_2\,k_4\,\phi/\rho_{p,eff}$$

with

$k_1 = 0.8$ for reinforcing bars with good bonding
(can be modified in AP)

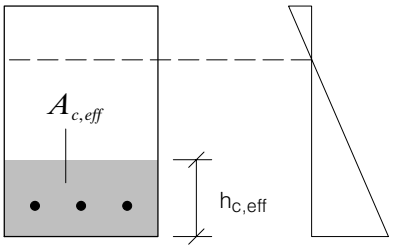
$k_2 = 0.5$ (for pure bending)
$k_2 = 1.0$ (for pure tension)
$k_2 = (\epsilon_1 + \epsilon_2)/2\epsilon_1$ (intermediate)

From the national code appendices

(fixed):

$k_3 = 3.4$ (modified in AP)
$k_4 = 0.425$ (modified in AP)

ϕ = mean bar diameter in $A_{c,eff}$



Reinforcement:

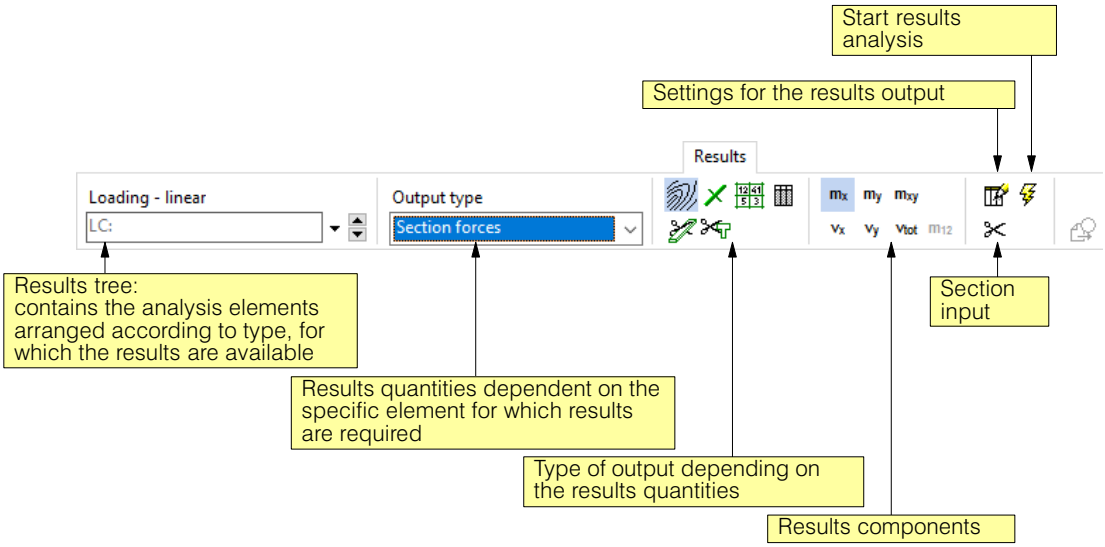
The reinforcement to be used in the analysis is defined by the hatched area in Figure 7.1 (EN1992). The zone boundary is taken as a straight line parallel to the neutral axis with the effective height $h_{c,eff}$ from the edge of the tension zone. According to the code the height is

$$h_{c,eff} = minimum\,[\,2.5(h - d)\,;\,(h - x)3\,;\,h/2\,]$$

B 13 Results

B 13.1 Organization of the Results Output

The output of any result is prepared in the tab sheet 'Results', which is structured as follows:



B 13.1.1 Starting the Results Analysis and Output

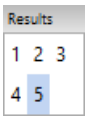
By clicking on the button to start execution, the results are calculated according to the settings in the 'Results' tabsheet and displayed. Besides these settings the following two settings have an influence on the output of results, both of which are carried out in the Menu „Representation“:
- Units / Decimal places
- Colours / Line type in the graphics output.

Certain results can be exported for use in foreign programs.

Results are output either in tabular or graphical form and can be printed directly or added to a print list (Print Entry).

All outputs included in the print list are managed with the CubusViewer and can be viewed there at any time and printed. Calling the CubusViewer is by means of the marked button. For details on the CubusViewer see the corresponding manual.

Graphical Outputs

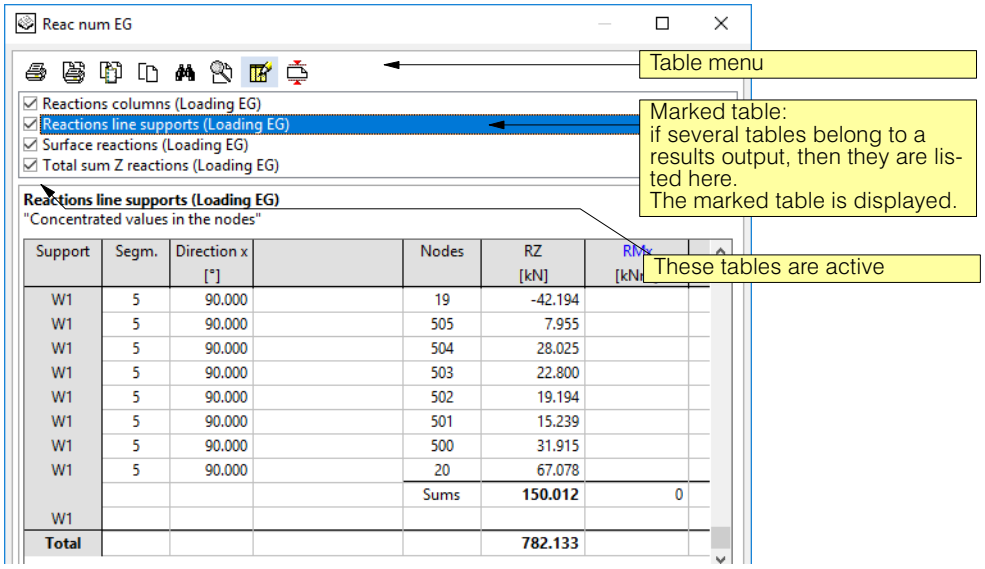


Graphical outputs are usually performed directly in der graphics area of the CEDRUS Window, but sometimes also in a separate window. In the case of outputs in the graphics area for each output a new layer is given in the Layer Group Results. The layer buttons created for this purpose are identified using consecutive numbers.

The final maximum 6 Results Layers since starting the program or since making changes to the structure are always available in the corresponding Layer buttons.

Several **Results layers can be overlain**, by activating further layers while pressing the <Shift> key.

Tabular Output

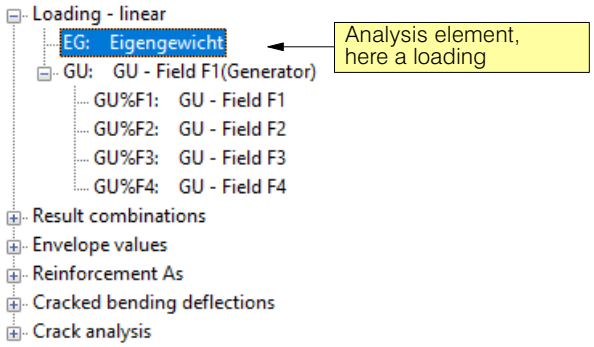


The Table menu provides the following functions:

- Direct printing and making a print entry
- Copying the active table to the Windows Clipboard
- Copying the marked part of the table to the Windows Clipboard
- Searching for a piece of text in the marked table (not active in Page View)
- Page View: all active tables are displayed
- Dialog with further display options
- Compact presentation: certain tables can be displayed in a compact mode with less detail. This mode can be activated here.

B 13.1.2 Results Tree

The following figure shows the Results Tree with its possible branches. The first branch with the linearly analysed loadings and those with the limit values are usually always available. The others are only shown if the corresponding specifications were made.



Firstly, the element to be analysed is always selected, i.e. the one for which one requires results. The other possible specifications in the 'Results' tab sheet depend on this selection.

B 13.1.3 Output Quantities, their Type of Presentation and Components

Depending on the selected analysis element in the Results Tree the following four output quantities with their accompanying types of presentation (described below) and components may be selected:

- **Loadings - linear / nonlinear, results combinations:**

Deformations		no component selection
		D_z R_x
Reactions		R_z R_m
		no component selection
Section forces		m_x m_y m_{xy} v_x v_y v_{tot}
		m_x m_y m_{xy} v_x v_y v_{tot} m_{12}
		v_{tot} m_{12}
		no component selection
		m_l m_n v_l v_n
		M V
Stresses		σ_{xt} σ_{yt} τ_{xyt} σ_{xb} σ_{yb} τ_{xyb} σ_{12b}
		σ_{12t} σ_{12b}
		σ_{xt} σ_{yt} τ_{xyt} σ_{12t} σ_{xb} σ_{yb} τ_{xyb} σ_{12b}
		no component selection
		σ_{lt} σ_{nt} σ_{lb} σ_{nb}
Results at joints		m_n v_n

- **Limit values:**

Deformations		$D_{z<}$ $D_{z>}$
		no component selection
		D_z
Reactions		R_z R_m
		$r_{z<}$ $r_{z>}$
		no component selection
Reinf. moments		ma_{xt} ma_{yt} ma_{xb} ma_{yb}

m_{axt} m_{ayt} m_{axb} m_{ayb} m_{at} m_{ab}

no component selection

m_l m_n

M V

Max. shear forces

no component selection

Results at joints

m_n v_n

Punching shear verification.

no component selection

- **Reinforcement As (standard):**

As-cross sections

a_{xt} a_{yt} a_{xb} a_{yb}

a_{xt} a_{yt} a_{xb} a_{yb} a_t a_b

no component selection

a_l a_n

no component selection

As and lamella c.s.

no component selection

- **Reinforcement As (prestressing=resistance):**

As-cross sections

no component selection

- **Reinforcement As skew:**

As-cross sections

a_{xt} a_{yt} a_{xb} a_{yb}

a_{xt} a_{yt} a_{xb} a_{yb} a_t a_b

no component selection

- **Dynamic analyses > Table of eigenvalues:**

Table of eigenvalues

no component selection

- **Dynamic analyses > eigenvalues, mode shapes:**

Deformations

no component selectionl

D_z R_x

- **Bending with cracking:**

Deformations

no component selection

D_z R_x

- **Crack analyses:**

Cracks

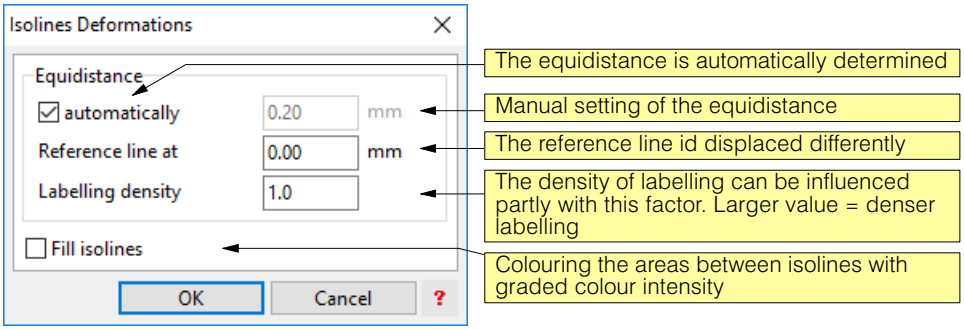
x y max

no component selection

B 13.1.4 Types of Presentation and the Corresponding Settings

Output of Isolines

A Results component is output in the form of contour lines with equidistance of contour level. The corresponding dialog „Preferences Output“ looks as follows:



☞ *Colour, line type and line thickness of the positive and negative isolines as well as the reference line can be specified in the menu „Representation“ > „Colour / Line Types“.*

With the visibility of the sublayer in the corresponding Results layer the labelling of the isolines can be hidden.

Numerical-graphical output

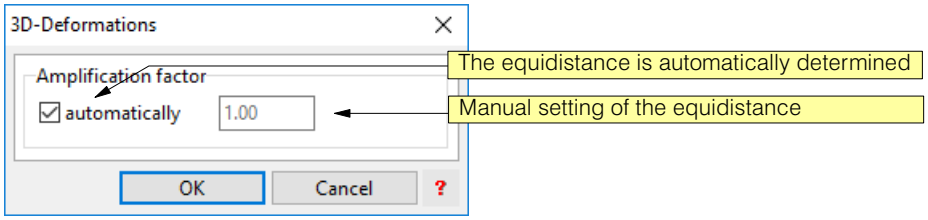
The numerical values are written directly into a figure with the FE mesh as the background. Parameters cannot be specified.

☞ *The colour of the positive and negative values corresponds to the colour of the isolines output.*

With the output of the element results (e.g. moments), to reduce overwriting, only the results at the centre of the elements are displayed. With the visibility of the sublayer in the corresponding Results layer the nodal results can also be made visible.

Three dimensional output

Isometric display of the undeformed and the deformed slab. The viewing angle can be specified in the dialog 3D View. As parameter the exaggerated scale factor can be input:



Output of principal values / Cracks output

Figure with crosses representing principal values. As parameter the exaggerated scale factor for the size of the crosses can be input (same as 3D output).

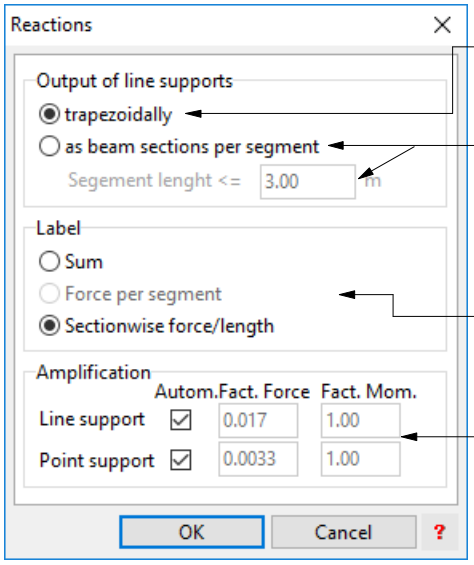
☞ *Colour, line type and line thickness of the contour lines of the principal value graphics corresponds to the settings for the isoline output (see menu „Representation" > „Colour / Line Types").*

Tabular output

See above!

Output of lines and point reactions

For loadings, combinations and limit values a figure with the point and line reactions in the 3D view is created. For the display the following parameters can be specified:



*) Trapezoidal over the whole support (with no limit values)

*) Subdivision of line supports into sections of equal length <= section length, output in beam form per section, section length 0 = node-like output of reactions in beam form (beam width=1/2 side length of neighbouring elements

Labelling for beam–like output with line supports (single support output: always force)

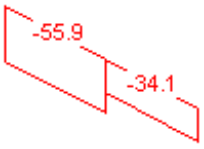
Exaggerated scaling factors for the graphical display of line support and single support (columns)

*) For the transformation of the nodal reactions of line supports into beam or trapezoidally distributed reactions only the nodal reactions in the Z direction are considered, no reaction moments).

Presentation examples with corresponding settings:

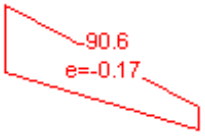
beam–like per section

Labelling:
sum per support



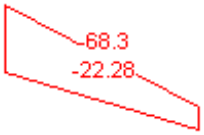
beam–like per section

Labelling:
sectionwise force/ length



trapezoidal

Labelling:
sum per support



trapezoidal

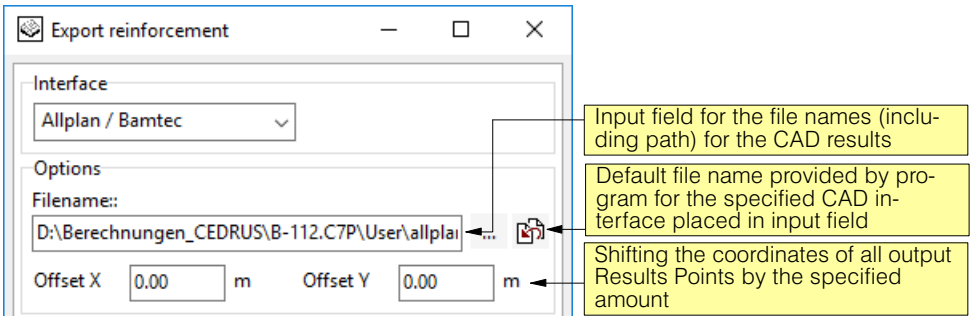
Labelling:
sectionwise force/ length

☞ *Colour, line type and line thickness of the positive and negative isoline reaction graphics as well as the reference line can be specified in the menu „Representation" > „Colour / Line Types".*

With the visibility of the sublayers in the corresponding Results layer the labelling of the reactions can be hidden.

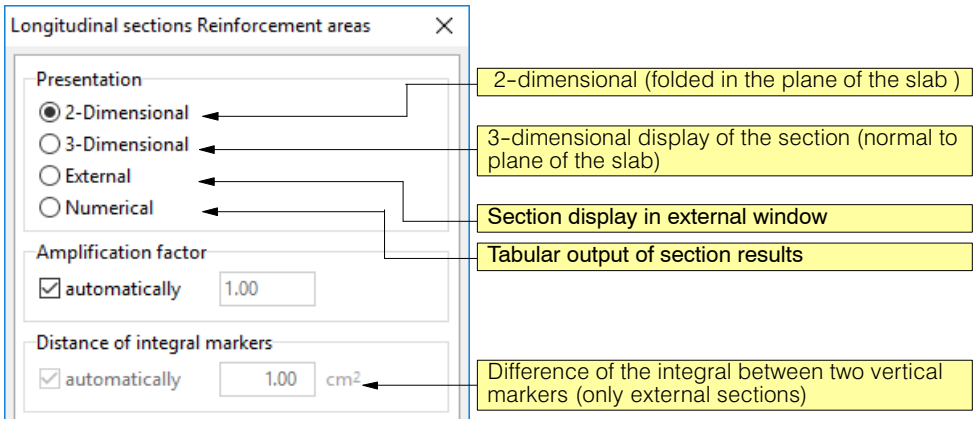
Output of results for CAD programs

With this option the results (reinforcement cross sections) in the output format of the specified CAD interface can be written to a file. The corresponding dialog „Export reinforcement" looks as follows:



Section results

Various results can be displayed in predefined sections that have to be input (see „Section Input"). The dialog „Preferences Output" gives the following options:



☞ *In the case of dimensioning limit values and reinforcement these section results can only be determined in zones, whose output direction coincides with the section direction or is normal to it.*

Beam section results

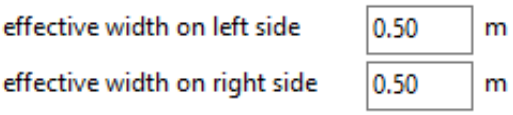
For beam section results the same sections (see „Section Input") are used as for normal section results. However, it is not the result value on the section axis that is output, but the integral of the required result component over the section width. Beam section results, therefore, are only possible for results that are defined per unit length and thus can be integrated over the section width.

The section width is a section property that is not used for normal section results. One speaks of beam sections because for this task a "strip" or "beam" is cut out of the slab. The section axis lies in the middle of the section boundaries. The section strips may partially extend over the edge of the slab boundary.

The dialog „Preferences Output" provides the same options as with sections (see above).

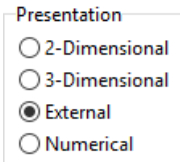
☞ *Beam sections are especially suited to determining the reinforcement in column strips, since the reinforcement moments are determined using integrals over the section width. Thereby the singularities over point supports due to the simplifications in the plate theory, which lead to strongly mesh-dependent peak values in the FE results, are smoothed out.*

Downstands function in the results output automatically as beam sections and can also be selected, if results in beam section form are available. The left and right sides of an imaginary cut of a downstand section (and thus its width) are defined in the Downstand dialog by the following two values:



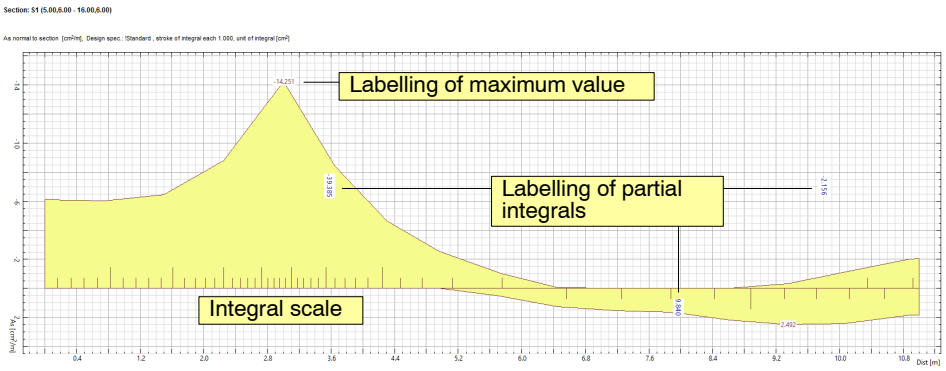
These two values of the effective width are used exclusively for the beam section results used here. They are measured from the downstand axis and may not be smaller than half the corresponding downstand width.

To dimension the reinforcement in beam sections see also chapt. B 12.3.



External section displays

With the setting „External" in the dialog „Preferences Output" the section results are output in a separate window in the following way:

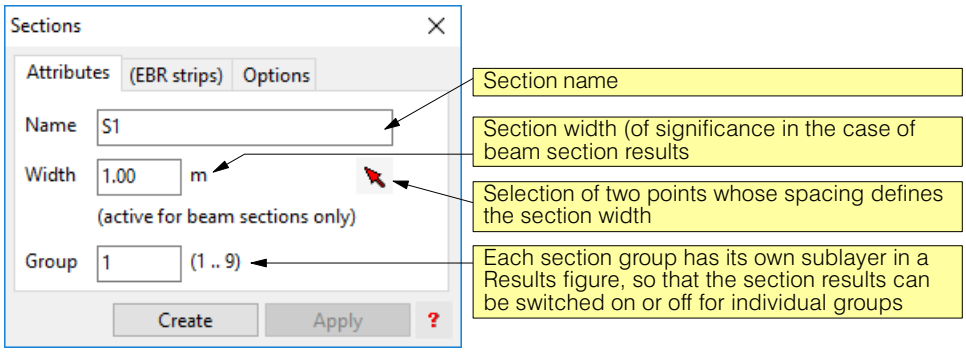


With the external section display for section results of section forces, reinforcement moments and reinforcement for components normal to the section there is an integral scale.

The increment (Delta) of the integral is constant from interval to interval. The value of Delta is given in the title (here 1 cm^2). In this way the integral between two points is determined by counting the interval markers. The scale begins again with each zero crossing. Further, the integral of each contiguous section is labelled between boundaries or zero crossings.

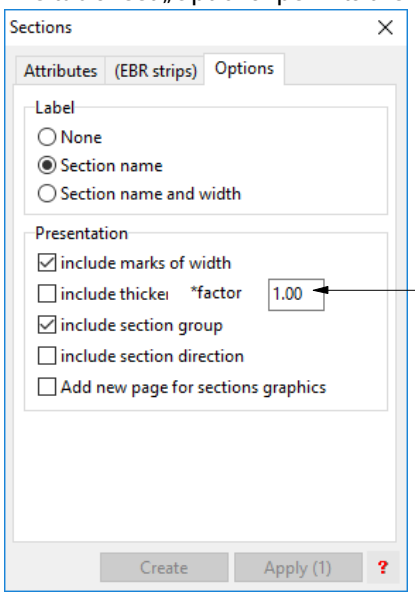
B 13.1.5 Section Input

With this button in the 'Results' tab sheet the following dialog is opened with whose help sections for section and beam section results are input:

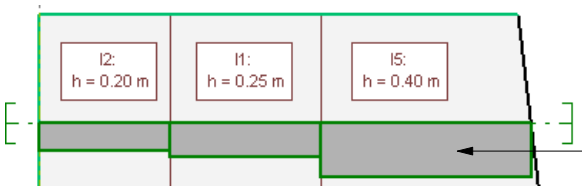


The tab sheet „EBR stripes" contains input for the analysis of glued strip reinforcement.

The tab sheet „Options" permits the following settings to be made:



Showing the slab thickness along the section axis (see example below)
(scale exaggeration factor 1.0: slab thickness is drawn to the same scale as the slab dimensions)



Representation of the slab thickness along the slab axis

B 13.2 Standard Results

Under Standard Results all results are understood that can be created with the Base Module of CEDRUS .

B 13.2.1 Raw Results

The solution algorithms of CEDRUS create the following raw results for each input loading:

- Nodal displacements
- Reactions for the supported nodes and elements
- Section forces at the centre of the element in global coordinate directions
- Section forces at the corners of elements in global coordinate directions, which are summed to give mean values in the different zones and are not available individually

These computed raw results are stored in binary form and serve as the basis for determining most result types in the desired form of presentation. They are all calculated automatically, as soon as the first result is requested

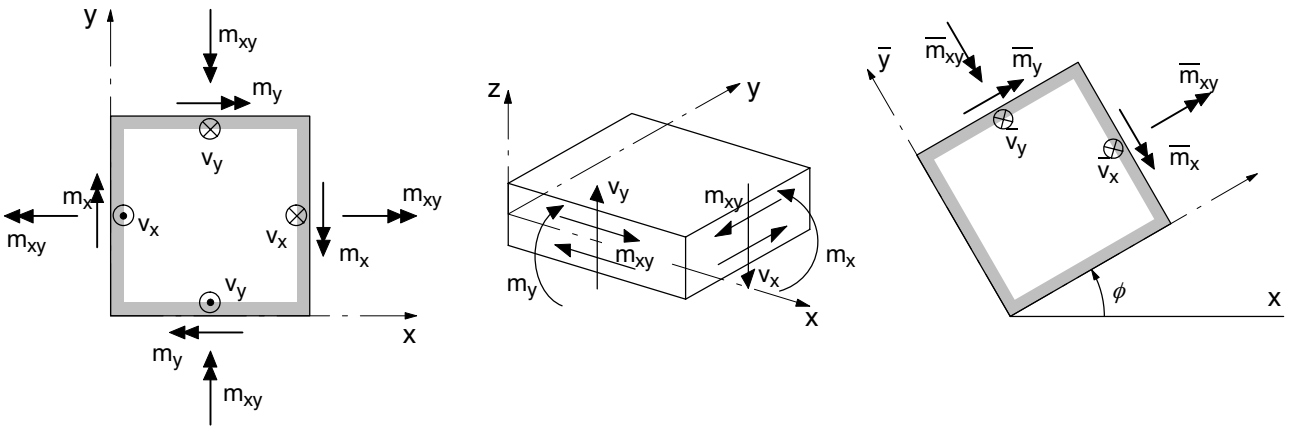
B 13.2.2 Result Quantities for Loadings and Results Combinations

Deformations:

With deformations it is a case of settlement and rotations about the x and y axes in each node of the FE mesh. The rotations of unsupported nodes are output in the global coordinate system. In the case of point and line supports the x direction of the input object is adopted and the y direction is normal to it. The x axis of line supports points in the support direction.

Section forces:

The slab section forces comprise the moments m_x, m_y, m_{xy} and the shear forces v_x, v_y. The following figure shows their action on an infinitesimal plate element:



The output of the slab section forces is carried out for each zone (material zone) with definable output directions. The transformation formulas are as follows (Mohr's cirle for moments and vector transformation for the shear forces):

$$\bar{m}_x = m_x \cos^2\phi + m_y \sin^2\phi + 2m_{xy} \sin\phi \cos\phi$$

$$\bar{m}_y = m_x \sin^2\phi + m_y \cos^2\phi - 2m_{xy} \sin\phi \cos\phi$$

$$\bar{m}_{xy} = -(m_x - m_y) \sin\phi \cos\phi + m_{xy}(\cos^2\phi - \sin^2\phi)$$

$$\bar{v}_x = v_x \cos\phi + v_y \sin\phi$$

$$\bar{v}_y = -v_x \sin\phi + v_y \cos\phi$$

$$v_{tot} = \sqrt{v_x^2 + v_y^2}$$

Depending on the form of output the principal moments or the maximum shear forces are also output.

Edge stresses:

One can also obtain the above moments-shear forces corresponding to the edge stresses in the output directions as well as the principal stresses.

Reactions:

The reactions are output sorted according to node or element depending on the individual supports. In the case of the graphical output of the line supports there is the possibility of presenting the nodal reactions section-by-section by defining a desired section length.

Export loadings:

(See also chapt. B 10.4)

Here it is a question of export combinations, which are transformed into loadings for transferring the loads. These loadings are used in the underlying storey but can be inspected in the current analysis, however, for checking (Loadings legends in the 'Loads' tabsheet) after their analysis has been carried out automatically by requesting any result.

B 13.2.3 Results Quantities for Limit State Specifications

Deformations:

The limit values of vertical displacements may be obtained with the corresponding rotations. One should note that displacements have a negative sign if they are downwards and that thereby normally the lower limit values (minimum values) of the displacements are of interest.

Reactions:

The limit values of the reaction components of columns, line supports and the reaction pressures of the elements of area-supported zones may be obtained.

Reinforcement moments:

The slab section forces are superimposed according to the specific superposition rules on reinforcement moments specified in the limit state specifications.

As reinforcement moments in a point we mean the four moments needed to determine the upper and lower slab reinforcment in two orthogonal directions.

If the directions of the principal moments for all considered loadings coincide with the output directions x,y (can be constrained using material with low drilling resistance), then one could sum all positive and all negative moments of the participating loadings and dimension for the limit values thus obtained.

In the frequent case of isotropic material, however, the stresses have arbitrary principal directions for which loading and dimensioning for the accumulated $m_{xmin/max}$ and $m_{ymin/max}$ would err greatly on the unsafe side. One requires a method, therefore, which permits superpostion of the results of several loadings and provides safe values for dimensioning in two given, orthogonal directions. In CEDRUS, use is made of the well-known linearized plasticity conditions:

$$ma_{xb} = Max\,(m_x + m_{xy},\ m_x - m_{xy})$$
$$ma_{xt} = Min\,(m_x + m_{xy},\ m_x - m_{xy})$$
$$ma_{yb} = Max\,(m_y + m_{xy},\ m_y - m_{xy})$$
$$ma_{yt} = Min\,(m_y + m_{xy},\ m_y - m_{xy})$$

wobei:

ma_{xb}	reinforcement moment for the lower layer in the output direction x
ma_{xt}	reinforcement moment for the upper layer in the output direction x (negative values)
ma_{yb}	reinforcement moment for the lower layer in the output direction y
ma_{yt}	reinforcement moment for the upper layer in the output direction y (negative values)

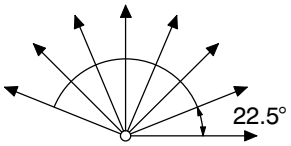
The output directions may be defined in the properties of the material zones.

☞ *For the very efficient construction of the limit states in CEDRUS by superposition of the results of the individual loadings a linearized yield condition is necessary, as described above.*
If one wants to work with nonlinear yield conditions (see chapt. B 12.4), using combinatorics one has to construct from the limit state specification all possible loading combinations. Then one could determine for all these loading combinations the section forces and from them the required reinforcement and thus find the maximum required values. This method, however, is rather inefficient, since from typical limit state specifications one would quickly produce thousands of loading combinations. This is especially the case when one has to consider the unfavourable loading positions fieldwise.

Maximum shear forces:

The maximum shear forces for a particular loading are calculated using $v_{tot} = \sqrt{v_x^2 + v_y^2}$ and due to the fact that we are dealing with a nonlinear condition and for this maximum value there is a corresponding direction, which can be different for each loading, they cannot be superimposed directly to form limit values. By constructing all possible loading combinations beforehand it would be possible to obtain the limit values. However, this would be very inefficient (cf. above note on reinforcment moments).

To obtain nevertheless useful values for such limit values, CEDRUS uses the following method:



For each loading we do not determine the maximum shear forcea, but the shear force in all shown eight directions.

In these eight directions in each case the maximum value (all positive values) and the minimum value (all negative values) are updated in constructing the limit values and finally, of the sixteen values carried forward the the largest absolute value is output as the limit value.

In this way the superposition of limit values is possible and one obtains a result that is usually sufficiently accurate. In the case of numerical output the directions corresponding to the limit values are also shown.

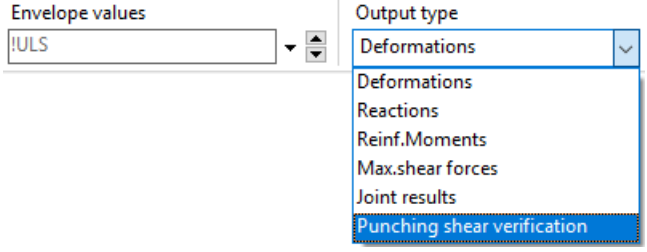
B 13.2.4 Required Reinforcement

Orthogonal reinforcement in the output directions is determined on the basis of reinforcement moments, as described above. The dimensioning is based on a dimensioning specification, as described in chapt. B 12.3. All details regarding dimensioning are to be found there.

For skew reinforcement there is a special dimensioning method described in chapt. B 12.4. Here nonlinear yield conditions are used. Thus one cannot work with the reinforcement moments described above.The algorithm is based on a fixed loading combination.

B 13.2.5 Punching Shear Verification

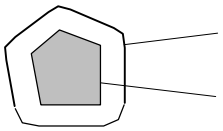
It is available as an output quantity for limit state specifications, if active punching shear objects are present.



The verification of punching shear, therefore, as far as loads are concerned, is based on a limit state specification. How the punching shear load is determined from it, is explained below. The required input for determining the punching shear resistances is made in the 'Punching Shear' tabsheet. The treatment of punching shear in CEDRUS is described in detail in Ch. B 9 .

Punching shear loads

The dimensioning value of the punching shear loads are determined by the program on the basis of the selected limit state specification (for the ultimate load state). This involves calculating the minimum and maximum force that acts on the piece of slab cut out on the basis of the punching shear polygon. The force components for the loadings considered are the sum of the loads and reactions acting within the punching shear polygon.



The sign of the maximum absolute values has to correspond to the punching shear type specified in the punching shear object (column=positive, concentrated load =negative).

If prestressing loadings are also present in the limit state specification , then the action of the deviation forces within the punching shear polygons is also taken into account.

Example of an output table

PUNCHING SHEAR VERIFICATION

for limit state specification !ULS

Id	Typ	l_{Reac} [m]	V_{Reac} [kN]	V_{Load} [kN]		V_d [kN]	V_{Rd} [kN]	V_{Rd-e} [kN]	AL	Verif. type
C1	Clm		504.499	-4.475		500.023	650.380	590.121	2	ok
							612.922	566.535	3	ok =0.94*NS2
C2	Clm		152.102	-2.392		149.710	252.715	201.580	2	ok
							353.046	266.715	3	ok =1.40*NS2
C3	Clm		184.705	-3.000		181.705	368.457	275.682	2	ok
							473.172	349.042	3	ok =1.28*NS2
C4	Clm		67.305	-2.599		64.706	113.861	86.910	2	ok
							306.765	221.348	3	ok =2.69*NS2
C5	Clm		83.443	-2.599		80.844	138.973	112.340	2	ok
							160.367	126.751	3	ok =1.15*NS2
C6	Clm		160.994	-2.392		158.602	210.304	189.868	2	ok
							228.305	202.271	3	ok =1.09*NS2

Typ : Verification for column (Clm)/load(Ld) (Punching shear load positive/negative)
l_{Reac} : Wall segment length for load reduction and reaction calculation in punching analysis
V_{Reac} : Reaction proportion of V_d
V_{Load} : Load proportion of V_d
V_d : Dimensioning value of punching shear load correspondingly Nachweistyp: $V_d = V_{Reac} + V_{Load}$
V_{Rd} : Dimensioning value of punching shear resistance acc. code
V_{Rd-e} : design value of eff. punching resistance (V_d increased until $V_d = V_{Rd-e}$)
AL : Approximation level for SIA code

Supplementary table for SIA Code and approximation level 3

Supplementary table for approximation level 3

Id	rs Bel	x'/y'	m_{sd} [kN]	m_{Rd} [kN]	b_s [m]	r_s [m]	ψ [mrad]	k_r
C1	EG	x'	-69.3	118.0	1.96	1.22	6.83	1.425
		y'	-77.0	127.3	1.96	1.40	8.14	1.332
C2	EG	x'	-6.4	29.0	1.00	0.67	1.06	2.000
		y'	-34.2	78.1	0.78	1.04	4.66	1.699
C3	EG	x'	-7.8	40.0	1.00	0.67	0.71	2.000
		y'	-46.1	127.3	0.83	1.23	3.33	1.746
C4	EG	x'	-7.7	26.6	0.80	0.53	1.29	2.000
		y'	-7.4	29.0	0.80	0.53	1.06	2.000
C5	EG	x'	-39.4	78.1	0.80	1.05	5.78	1.609
		y'	-23.6	85.8	0.80	1.04	2.30	1.929
C6	EG	x'	-48.1	86.4	1.00	0.98	6.30	1.570
		y'	-27.9	78.1	0.93	1.09	3.59	1.797

rs Bel : Loading or result combination for evaluating of the rs values
x'/y' : considered punching direction

It can happen that V_{Reak} is much smaller than the usually output reaction limit values. This may be because for each loading that is investigated, to obtain V_{Reak} it has to be checked whether the load transfer is greater than the reaction. If this is the case then this loading is not critical and the investigation is not pursued further. Such a situation may arise, e.g., in the case of import loadings, when an imported column load is not completely transferred,

but due to the column being too flexible it is redistributed over the slab to neighbouring walls.

Integration of third-party program for the punching shear verification

The dimensioning of punching shear strengthening depends on the product. It can be performed using third-party programs supplied by the manufacturer, which are combined with CEDRUS as plug-ins.

This necessitates the correct installation of the corresponding third-party program, which has to be adapted accordingly.

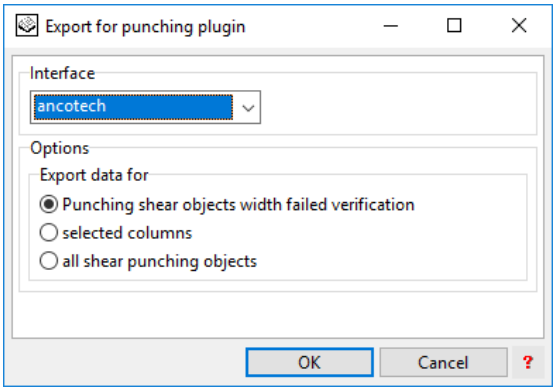
☞ *CEDRUS must be already installed before the third-party program is installed. Reinstall it in any case as a precaution.*

CEDRUS passes on the necessary data to the third-party program (Export) and can also receive calculated results from this program (Import).

Export to third-party program

To call the third-party program, for the required form of output the button shown on the left has to be selected and then . If several third-party programs are installed a Selection dialog appears.

The third-party program receives all necessary data, such as topology, geometry, loads, materials and reinforcement for the punching shear objects to be investigated using an interface file created automatically by CEDRUS. To specify the punching shear objects to be investigated the following self-explanatory dialog appears:



To use the third-party program one requires the manufacturer's instructions. If one wants to import the results determined there into CEDRUS, in the third-party program you have to export them to CEDRUS using a special command in the third-party program.

Importing results from a third-party program

The import of results is only meaningful for those punching shear objects that require strengthening. For these objects the third-party program provides

- the identification of the required strengthening product
- the correspondingly calculated punching shear resistance

The import is not automatic. It is initiated in the Properties dialog for the punching shear objects in the 'Punching Shear' tabsheet The imported results are used in the punching shear objects and reported in the punching shear verification of CEDRUS.

Vol. C CADview & CADextract

C 1 Introduction

CADview and CADextract are special designed functions in CEDRUS with which one is capable to extract efficiently and directly the data of a floor plan from a CAD drawing and use it for the analysis of a structure in CEDRUS. In this way the effort for the input of the structural model can be considerably reduced. Both the CADview and CADextract functions are an optional module to the basic module of CEDRUS.

CADview

Using the CADview function for 2D CAD drawings, the data for a floor plan, if it is in the dxf or dwg format can be imported into CEDRUS as a background drawing. The imported floor plans can be shown as background drawings or hidden away and are very convenient for inputting the structure (e.g. to read off coordinates).

In the import processes all existing information stored in a CAD drawing such as Layers, text, colours, fill styles and lines can be read in and managed together as a background drawing. This is fundamentally different to the use of the dxf import function in the basic module 'DXF-Lines' (main menu 'File'>'Import'), with which only the lines from a dxf file can be imported as normal Graphics Editor objects.

☞ *A valid CADview licence is necessary not only to import but also to display CAD drawings.*

CADextract

Using the CADextract function, both the data for the outline of a floor slab and the data for structural members can be extracted from a CAD drawing and converted directly into CEDRUS structural elements (outline, voids, walls and columns).

A common feature of most CAD drawings encountered in practise is that usually they are best appreciated only when printed. Generally, a direct use of the electronic data of a CAD drawing for the purpose of modelling structures is not possible. This may be due to various reasons: The CAD drawing may not have a clear straightforward data structure, i.e. the assignment of the drawing elements to the Layers is not correct or contains serious inaccuracies in the drawings, e.g. in the case of overlapping drawing elements. The CADextract function is subdivided therefore into several phases, which allow the user step-by-step to extract interactively from a CAD drawing both the geometry of the floor slab's outline with the openings and the wall and column geometries with the material properties. Subsequently the extracted data can be converted into CEDRUS structural elements.

☞ *Both a valid CADview licence and a valid CADextract licence are necessary for extracting floor plan data from a CAD drawing!*

C 2 CADview

C 2.1 Introduction

With the CADview function, within a CEDRUS analysis one or more CAD drawings can be imported and displayed as background drawings. In general, suitable for an import are 2D CAD drawings in the dxf or dwg format. Here, basically all the information in a CAD drawing, i.e. Layers, colours, fill styles, text and lines are read in. Thanks to optimized algorithms detailed floor plans, e.g. architect's plans, can be drawn and displayed accurately and in detail very quickly.

☞ *For every CAD drawing a specific drawing file (of type .cvg) is created and saved together with the project data.*

In the following section C 2.2 firstly the management of the imported CAD drawings into a CEDRUS project is described together with, in section C 2.3, the import procedure for importing a CAD drawing into a CEDRUS project.

C 2.2 Management of CAD drawings

By clicking on the button on the left in the 'CAD' tabsheet of CEDRUS in the management window for background drawings (see also section B 7.2) is opened. Fig. C-1, for example, shows such a window. The most important data with regard to management is shown in rows in a table for each imported CAD drawing: The first column contains the type of background drawing and the second column the name of the project-specific cvg-files. In order to be able to control the visibility of the background drawings separately, these are stored individually in their own Layer using the Graphics Editor. The designation of this Layer is automatic following the pattern D1, D2, D3,... and is listed in the third column. The values in the last column show the source of the CAD drawings, i.e. in each case the complete name and path of the original files.

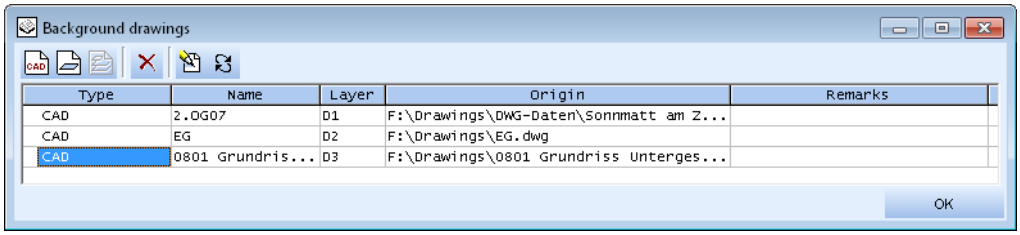
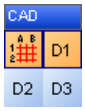


Fig. C–1 *Management of background drawings*

By clicking on the button on the left in the management window for the background drawings the background drawing from a CEDRUS project can be deleted.



The general visibility of a CAD drawing used as a background drawing is governed by the Layer belonging to the CAD drawing, which is located in the CAD Layer group of the Graphics Editor.

The display of the drawing elements of a CAD drawing is changed by means of the settings within the Layer properties window (Fig. C-2).

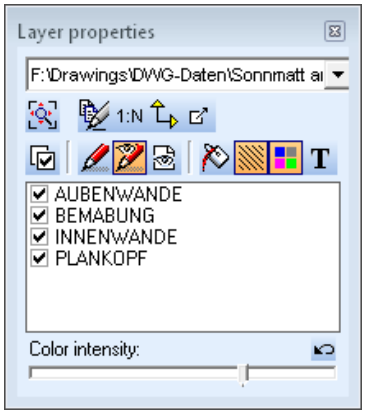


Fig. C–2 *Settings of the Layer properties*

This window can be opened either by clicking on the adjacent button in the management window for the background drawings (see Fig. C-1) or via the context menu of the corresponding Graphics Editor Layer button. In principle, the visibility of the Layer attributes, e.g. colours, fill styles, text, etc., can be set using the settings of the buttons directly above the Layer lists. In addition the following functionalities are available:

By clicking on the button on the left in the Layer properties window the active zone for centreing the CAD drawing (press the key F3) can be specified by a rectangle.

By clicking on the adjacent button the drawing elements of a background drawing can be copied into a Layer of the User group. The newly created Layer has the same name. In this Layer the elements of the background drawing can be edited as individual Graphics Editor objects. Only the selected Layers of the background drawing are copied into the Layer lists (see Fig. C-2). In the case of a large number of drawing elements the copying process can take a long time. Further, it should be observed that a large number of copied objects can slow down work both in the Graphics Editor and also when loading and saving project data.

The scale of the CAD drawing can be changed.

The point of origin of the drawing which has coordinate values of zero can be set by picking a point.

The CAD drawing can be moved by defining a translation vector.

☞ *If more than one CAD drawing is managed it is convenient to orientate the CAD drawings relative to each other.*

The colour intensity of the background drawing can be regulated using the slider at the bottom of the Layer properties window. With the slider it is possible to fade in and fade out the CAD drawing gradually, i.e. continuously without steps.

C 2.3 Importing a CAD drawing

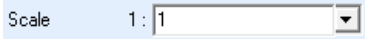
With the adjacent button in the management window for the background drawings (see Fig. C-1) the CADview function for importing a CAD drawing is started. Firstly a dialogue window opens to select a dxf or a dwg file. The selected CAD drawing is loaded and displayed in the CADview window, which is subdivided into various areas.

Fig. C-3 shows, by way of an example, the CADview window with a loaded CAD drawing.

The CAD drawing is shown in the display area of the CADview window taking into account the selected Layer and coordinate settings. The visible part of the CAD drawing may be changed using the toolbar buttons on the left .

Setting the CAD drawing scale

In the import of a CAD drawing into a CEDRUS project, besides the actual input of the drawing elements, a suitable scale for the floor plan has to be set. Inputting the scale is necessary, since in the CAD files usually only the coordinate values without information regarding a dimension are stored. In the corresponding selection field, at the bottom of the CADview window, the scale of the floor plan can be set.



As an aid to inputting the correct information, in the display area the dimensions of the visible part of the drawing are shown to the defined scale.

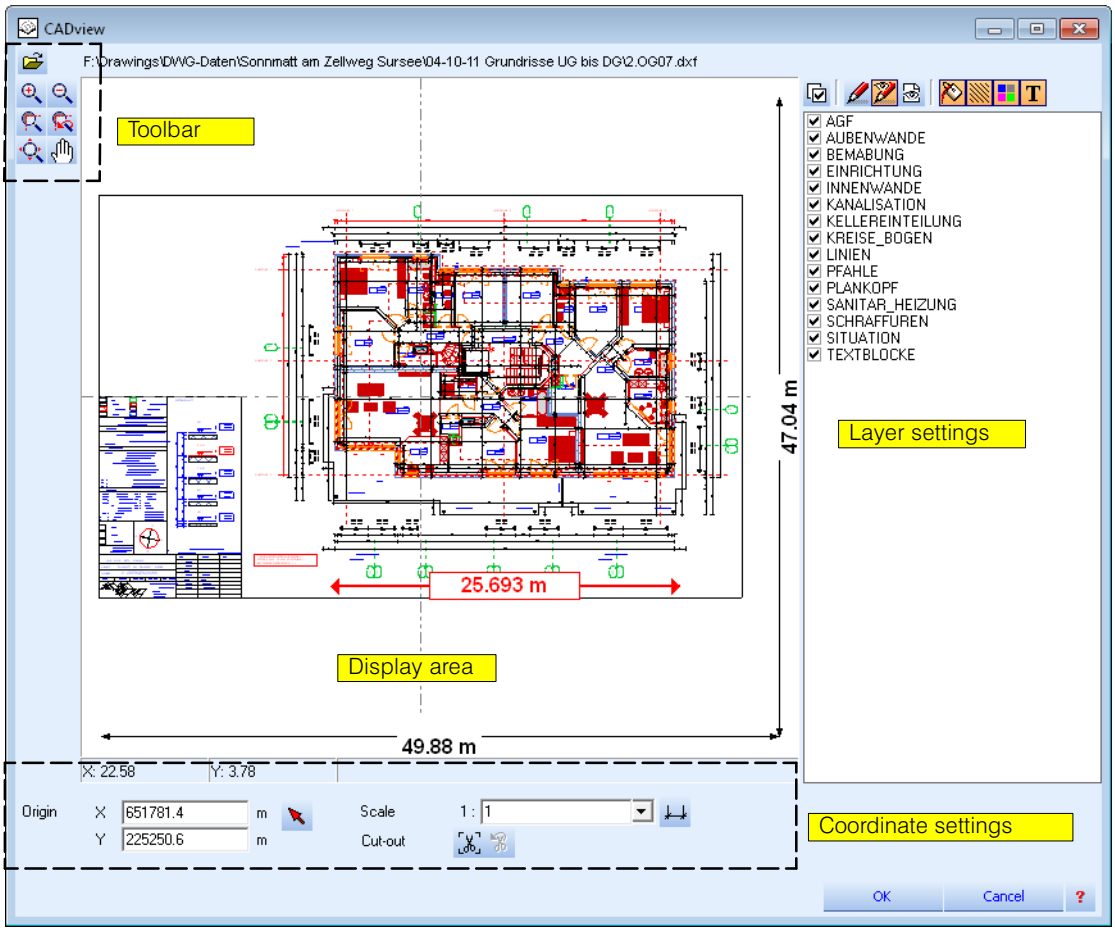
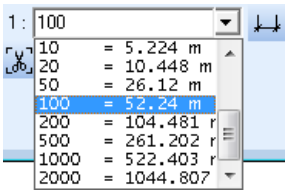


Fig. C–3 *CADview window(importing a CAD drawing)*

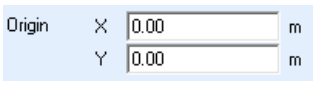
Additionally a dimension line, which acts as a reference line, can be entered in the display area. The input is done graphically using the left button. The dimension line together with their length according to the defined scale of the CAD drawing are visualized in the display area in thick red. In addition the length of the dimension line is shown for each listed scale in the comobobox.



☞ *The scale of an imported CAD drawing can be changed subsequently in the Layer properties window (Fig. C-2).*

Defining the position of the CAD drawing

Besides the input of the correct scale an appropriate origin of coordinates has to be selected. The origin can be set either directly by inputting the coordinates or by selecting a point using the mouse.



The selected origin is used as a reference point for the position of the background drawing in the CEDRUS coordinates system.

In general the origin of coordinates used in the CAD drawing is in an unfavourable position that can lead to rather large coordinate values and result in undesirable numerical inaccuracies in an FE structural analysis. Besides, with the selection of a suitable origin of coordinates, the work is simplified, especially if several CAD drawings have to be managed in a project.

☞ *The origin of an imported CAD drawing can be changed subsequently in the Layer properties window (Fig. C-2).*

Defining a rectangular cut-out

Using the adjacent button a rectangular cut-out of the CAD drawing can be defined. All drawing elements which are outside the rectangle are discarded. Lines and fill styles which intersect the outline of the rectangle are trimmed accordingly.

Once the cut-out has been created the new scale of the CAD drawing is 1:1.

The scale of the CAD drawing must be set correctly before creating a cut-out to ensure a proper cutting.

The cutting can be cancelled subsequently using the adjacent button.

Setting the layer properties

On the right side of the CADview window the Layers can be selected for the import. In the selection list this is done by activating or deactivating the corresponding check box.

A reduction of the Layer to the essential Layer and Layer attributes for the input of the structure is advantageous in the case of a very detailed drawing, since the number of imported drawing elements has a significant influence on the speed at which the background drawing is displayed.

Above the Layer list, the Layer attributes like colours, fill styles, texts, etc., are defined that have to be imported.

By clicking on the adjacent button in the CADview window (see Fig. C-1) the CAD drawing corresponding to the given settings can be imported as a background drawing.

Only the visible Layer and Layer attributes of the CAD drawing are imported into the CADview window!

C 3 CADextract

C 3.1 Introduction

Previously imported floor plans with CADview can be semi-automatically analysed using the CADextract function. For a given storey both data for a floor slab's geometry, which contains an outline and voids, and data for structural members of walls and columns can be extracted from a CAD drawing. In general the nature of these two procedures are independent of each other. In the following they are called outline detection and structural member detection. However within the CADextract function the two procedures are linked to each other and have to be processed step-by-step in the mentioned sequence. After the outline detection and structural member detection the extracted data can be transferred to CEDRUS as structural elements. Thus the CADextract function consists of three main steps, which are described briefly in the following sections:

- Outline detection,
- Structural member detection (walls and columns),
- Final check and export of the extracted data.

In section C 3.4 it is described how to prepare a CAD drawing within a CAD program in a way that the CEDRUS input can be earsily generated with CADextract.

C 3.1.1 Outline detection

The outline detection is based on the lines of the CAD drawing. Starting from an automatically builded hull the desired outline of the slab can be formed interactively by inputting constraint lines with the mouse.

The area inside the outline of the slab defines a material zone with a constant thickness and a material of the slab. Additionally voids can be defined graphically within the material zone.

In detail the outline recognition is executed in two steps:

1. Restricting line segments of the CAD drawing which are used in the automatic process of building the starting hull,
2. Detecting the slab's outline interactively, inputting voids graphically and defining the material zone.

C 3.1.2 Structural member detection

The structural member detection involves several steps: The basic step is the automatic detection of the cross section's geometry of walls and columns. The detection is based on the analysis of filled areas of the CAD drawing. In the process, the geometry of a detected wall segment is described by the outline of the wall's cross section and the corresponding wall's axis with a constant thickness. The geometry of a detected column is described analogously by the outline of the cross section in a corresponding principal axis system. Material properties can be assigned to the detected wall segments and cross sections of the column for a subsequent structural analysis in CEDRUS. This involves both the input of the material itself and also the designation load-bearing or non load-bearing. In addition, interactively wall openings between similar wall segments can be closed, i.e. filled up.

The detailed procedure for structural member detection is carried out in six successive steps:

1. Selecting drawing elements of the CAD drawing to be used,
2. Detecting wall segments,
3. Assigning material properties to wall segments,
4. Closing wall openings,
5. Detecting column cross sections,
6. Assigning material properties to the columns.

C 3.1.3 Final check and export of extracted CAD data

The CADextract function is finalized with the opportunity to export the extracted data to CEDRUS, which means that the outline together with the voids of the slab (including the material zone) and the cross sections of walls and columns can be converted to CEDRUS structural elements.

C 3.2 Basics of handling

In this section some basic information how to use the CADextract function are given. The next section contains a detailed description of each step of the outline detection and structural member detection.

Starting CADextract

The CADextract function can be started by activating the adjacent button. Fig. C-4 shows

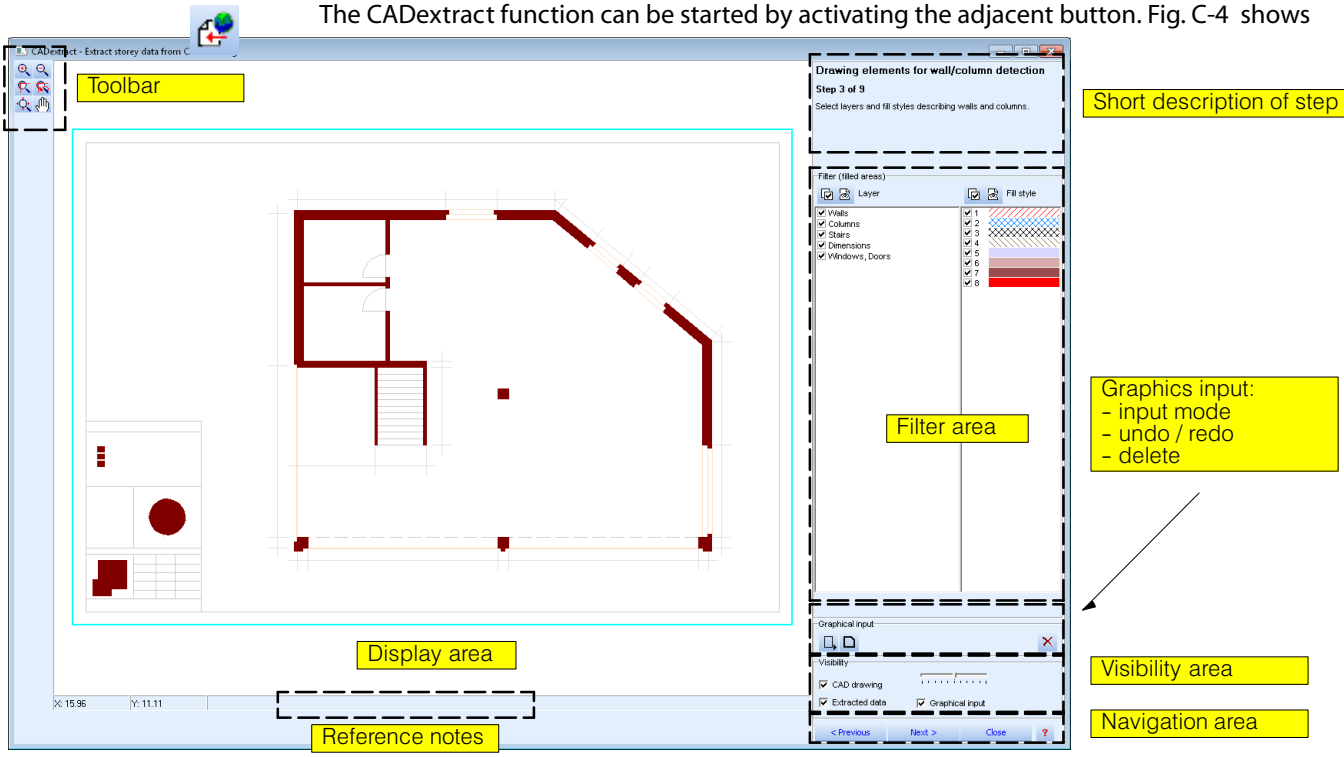


Fig. C–4 *Description of the CADextract window*

the general features of a CADextract window, which opens if the following conditions are fulfilled:

- With the CADview function (see Section C 2) a CAD drawing must already have been imported. If several background exist, it must be ensured that only one of them is visible.
- For wall detection, only visible Layers of the active background drawing are taken. These can be selected appropriately beforehand.
- For the outline detection and structural member detection only the visible Layers of the active background drawing are used. Those have to be set previously. Die CADextract function starts only if the visible Layers contain line segments.

Display area

In middle of the CADextract window there is the drawing or display area. Here the active CAD drawing acting as a background drawing, the line segments used in the process of the outline detection, the cross section areas extracted in the process of structural member detection and symbols for the interactively executed graphics input are displayed.

With the aid of the adjacent toolbar buttons the size of the visible area can be changed (zooming, moving, centreing etc.).

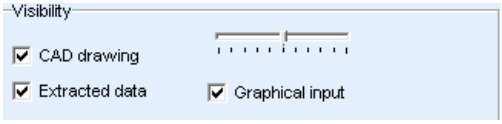
Below the display area there are reference notes pertaining to detected drawing elements where the mouse is positioned.

On the right side of the CADextract window there are areas for controlling and executing the individual phases of structural member detection. As a help, in the upper part there is a short description of the actual step.

Visibility

Above the navigation area is the visibility area. With the current settings the representation in the display area is controlled.

Depending on the CAD drawing, in some cases there may be a large number of drawing elements. To simplify the work, the visibility of the CAD drawing, of the extracted data and the graphics input executed during the outline detection and structural member detection can be faded in or out independently of one another. To improve user comfort in using the CAD drawing as a background drawing, in addition the brightness of the CAD floor plan can be changed using a slider.



Step sequence

In total the CADextract function consists of nine successive steps which correlate with the previously mentioned steps of the outline detection, structural member detection and export of the extracted CAD data. The steps mentioned in C 3.1.1 and C 3.1.2 have to be carried out according to the listed sequence, i.e. it is not possible to jump arbitrarily from one step to another. However, it is possible to go back to the previous step.

If the active background drawing does not contain filled areas, the process of the structural member detection cannot be executed. In this case the corresponding steps mentioned in C 3.1.1 are skipped.

The navigation between the steps is by means of the adjacent buttons located on the lower right of the CADextract window.

On leaving a step, i.e. by changing to the next or to the previous step, all the graphics inputs and settings executed in the current step are saved and the saved state of the new step is restored. On leaving a step the Undo/Redo lists of the current step are deleted.

The CADextract function can be terminated at any time by means of the button [Close] in the navigation area of the CADextract window.

Graphics input

In the different phases of structural member detection graphics input can be performed interactively. Depending on the current step, this can be done pointwise, linewise or areawise.

The buttons to activate one of the graphics input modes have the symbol of an arrow. An activated mode remains active as long as it is turned on, until it is explicitly turned off or by activating another mode. To ensure an easy handling in some input modes the object snapping is activated. In general only the objects which are **visible** in the display area can be snapped.

The executed graphics inputs are displayed symbolically in the display area in the form of a red filled circle. If, due to changed settings, graphics input in the case of the repeated execution of a step can no longer be carried out, then this unsuccessful graphics input is symbolized by white filled circles.

For each separate step the graphics inputs can be deleted individually by activating the delete mode (button on the left with an arrow), the graphics inputs to be deleted are deleted by clicking on the red and white input symbols, respectively.
In addition, all executed graphics inputs of a particular step can be deleted simultaneously (button without arrow).

For each separate step the graphics inputs are managed in an Undo/Redo list. By clicking on one of the adjacent buttons one can undo or redo the graphics inputs.

Filters

Above the graphics input area is the area where the filters of a particular step are shown. Basically the task of a filter is to reduce the amount of data extracted from a CAD drawing by restricting the analysis to a particular step-dependent area. In this way the restrictions, depending on the type of filter, apply in different ways:

- Analysis parameters are input either in text fields or by specifying them or setting them in an editable list field.
- In general Layers and fill styles also represent filters. These are managed in selection lists.
  Besides the individual selection of an entry it is possible, with the button shown on the left situated above the selection list, to select or deselect all entries in a selection list.
  For the Layers and fill styles, in each case there is a separate preview mode, which can be activated with the button on the left.

☞ With a change in a filter setting the current phase of wall detection is automatically updated and the display area is adjusted accordingly.

C 3.3 Introductory example

In this section the functionality of the individual phases of outline and structural member detection are illustrated with the aid of an introductory example following the sequence of steps listed in section C 3.1.

Loading the introductory example

▶ Load the introductory example by selecting in the main menu 'Help' > 'Load example' > 'CADextract C3'.

The example project already contains an imported CAD floor shown in Fig. C-5. One can also see the structural elements of an overlying storey, which are grouped in the CAD drawing by means of the Layer 'Walls', 'Columns', 'Windows,Doors', 'Stairs' and 'Dimensions'.

▶ In the basic module, by clicking on the adjacent button, start the CADextract function with all Layers of the CAD drawing (see also section C 3.1).

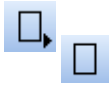
Step 1: Drawing elements for outline detection

In the majority of cases not all line segments of the CAD drawing are needed for the process of outline detection. Mostly only line segments of certain Layers and line types are needed. Once the CADextract dialog has been opened the program analyzes all line segments of the CAD drawing and stores them layerwise. The number of lines used in the process of outline detection can be restricted with so-called filters. In this sense the detected Layers and line types of the CAD drawing act as filter. Here a line type consists of a colour and a style. Besides the outline detection can be restricted to a certain area of the CAD drawing by defining a perimeter.

In the present introductory example the Layer 'Stairs', 'Dimensions' and 'Windows, Doors' as well as the blue and brown line styles are not needed.

▶ Set the filter settings for the Layers and line styles according to Fig. C-6.

To exclude the outer thermal insulation from the outline detection in an easy way it is advantageous to define a perimeter for the outline detection.



▶ Activate the graphic input mode of a rectangular perimeter for outline detection using one of the adjacent buttons. Following define a rectangle with two mouse clicks: The first one at the upper left corner of the wall, the second one at the lower right corner of the column. Afterwards the display area of the CADextract window should look as shown in Fig. C-7.

☞ *To exclude the inner line segments from the outline detection it is possible to define a band along the perimiter. The band has a constant width and is visualized in a grey style in the*

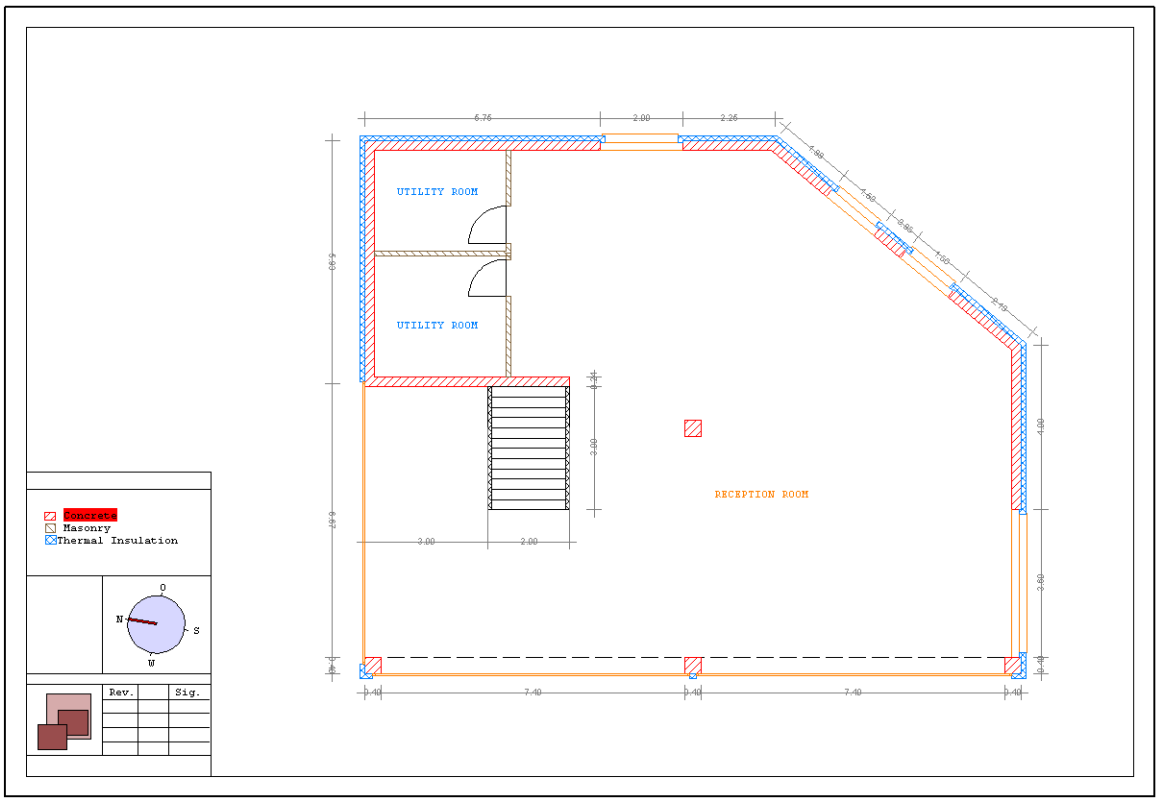


Fig. C–5 *CAD drawing of the introductory example*

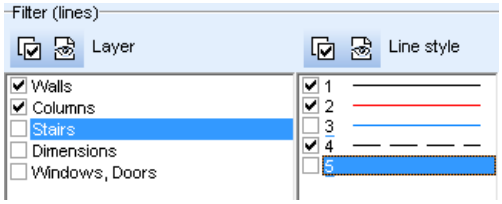
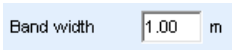


Fig. C–6 *Introductory example: Filter settings (Step 1)*



display area of the CADextract window. Fig. C-8 shows the definition of such a band. The process of outline detection only takes the line segments inside the band into account. The width of the band can be set using the adjacent text field.

☞ *All line segments which are used in the process of outline detection (next step) are highlighted in the display area of the CADextract window. The display is updated whenever a setting has changed or a graphic input has been done.*

▶ Proceed to the next step using the button on the left.

Step 2: Outline of the floor slab

In the second step a hull, which is shown in the display area of the CADextract function, is generated automatically in respect of the line segments. In the case of the introductory example the auto-generated hull resulting from the settings and input of the first step is shown in Fig. C-9.

In general the hull consists of several parts which are visualized in form of thick red and light red lines:

- A red line of the hull indicates a part of the hull which is either congruent with line segments of the CAD drawing or corresponds to a polyline which consists of line segments connecting the two end points of the red line of the hull.
- A light red part of the hull indicates that the hull is interrupted in this area, i. e. a closed connection between the end points of the light red line does not exist.

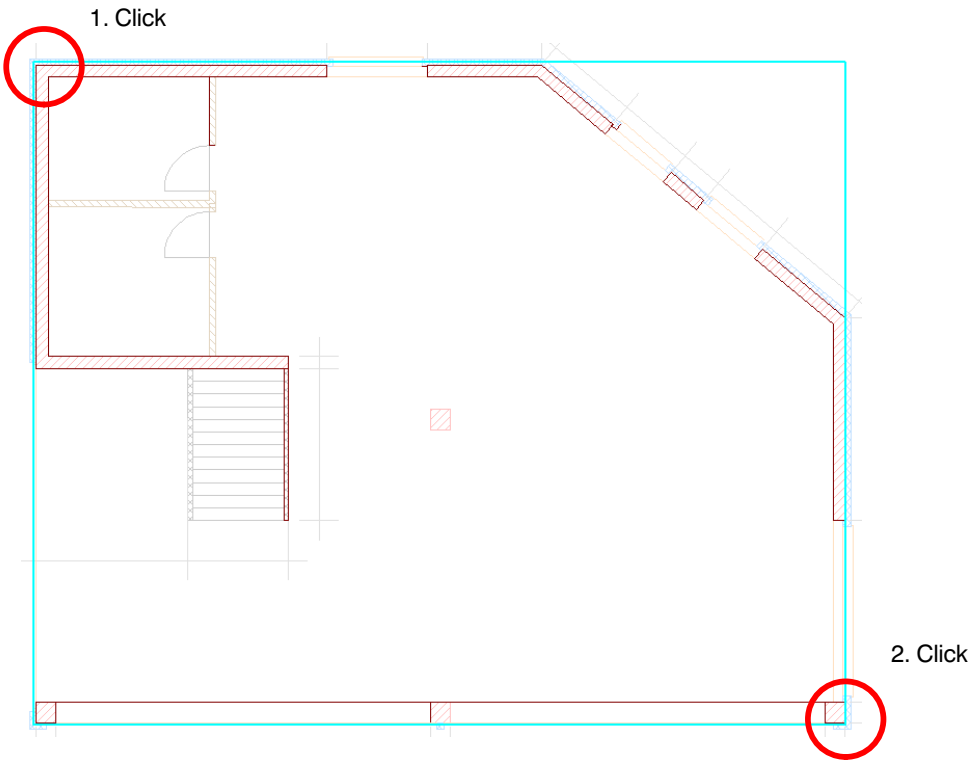


Fig. C–7 *Introductory example: Restricting the number of lines for outline detection (Step 1)*

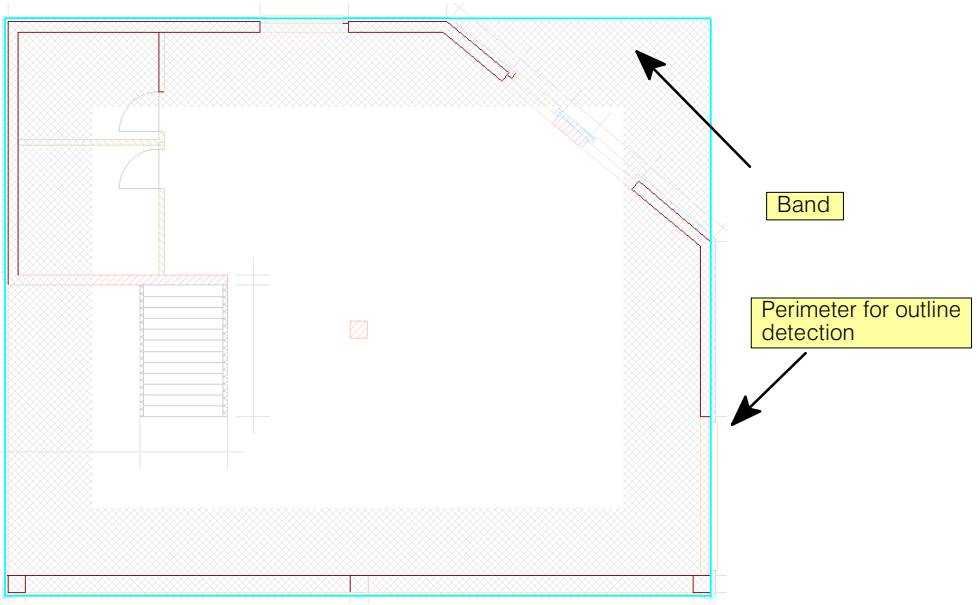


Fig. C–8 *Illustration of the band with a constant thickness belonging to the perimeter for outline detection*

☞ *The preview mode is activated when the mouse pointer is moved to the vicinity of a light red part of the hull. The polylines starting from the end points of the light red line of the hull are highlighted.*

The auto-generated hull can be adjusted to the desired outline of the slab by graphic inputs. For this purpose specific line segments can be excluded which should not be used in the process of the outline detection. In addition constraint lines can be defined which are incorporated in the hull.

▶ Exclude the line segment of the thermal insulation which disturbs the upper area of the hull. For that purpose activate the graphic input mode for excluding lines using the adjacent

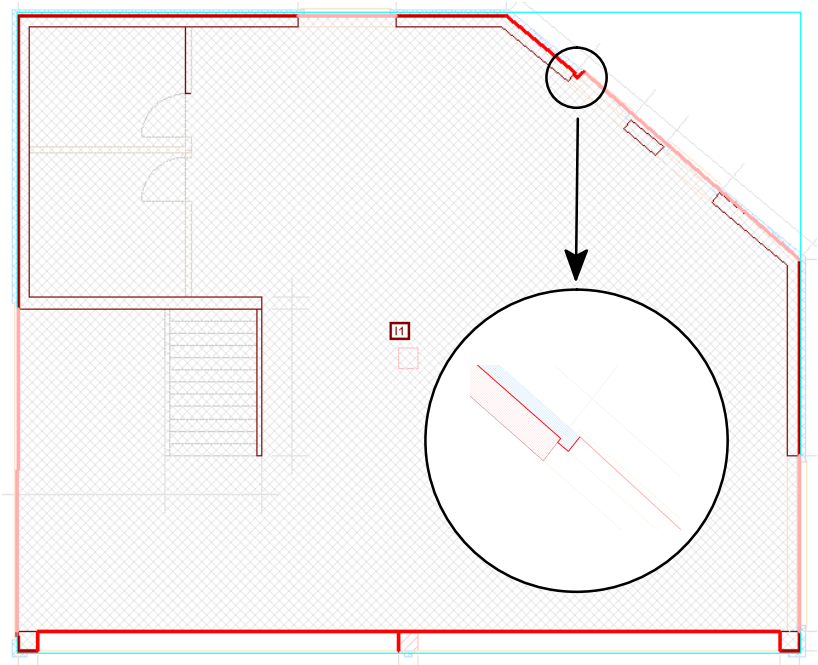


Fig. C–9 *Introductory example: Auto–generated hull (Step 2)*

button and click on the specific line with the mouse pointer. The hull is then adjusted accordingly and has the shape as shown in Fig. C-10.

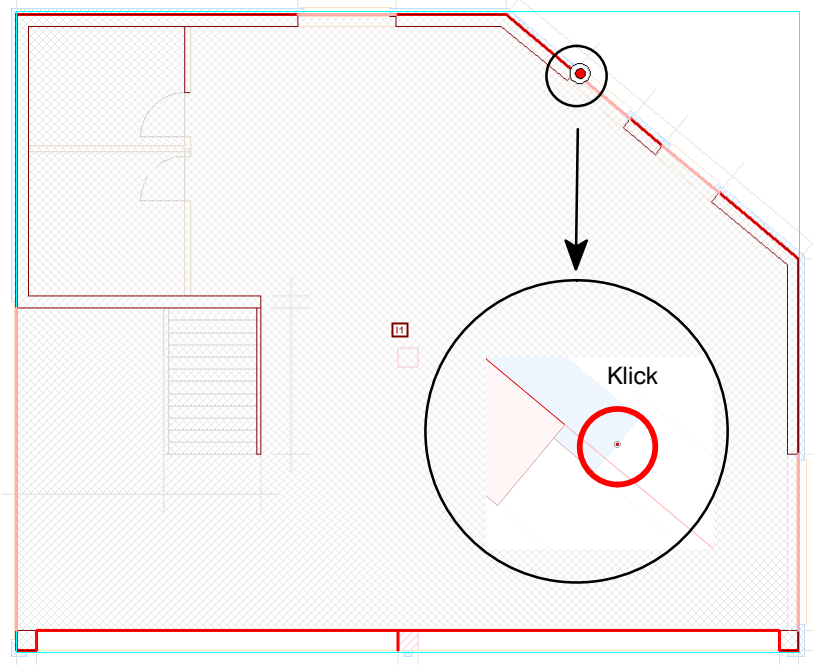


Fig. C–10 *Introductory example: Excluding a line segment (Step 2)*

▶ Modify the lower red part of the hull to a straight line. For that purpose activate the graphic input mode for defining constraint lines using the adjacent button. Define a constraint line with two mouse clicks from the outer edge of lower left column to the outer edge of the lower right column. As a result the hull is adjusted automatically and has the shape according to Fig. C-11.

Now the definition of the outline is completed. Subsequently voids can be created graphically by defining closed polylines inside the outline. In principle the procedure is the same as described before in the context of defining constraint lines. The difference is that the lines of a void must not touch or intersect the outline.

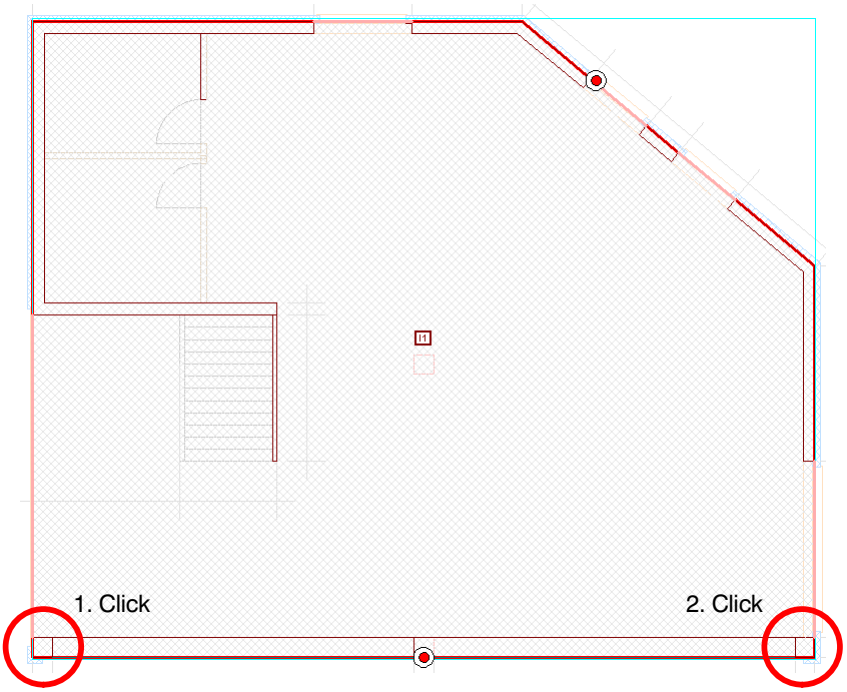


Fig. C–11 *Introductory example: Adding a straight constraint line (Step 2)*

▶ Activate the graphic input mode for polylines using the adjacent button. Define the perimeter of the void at the staircase railing according to Fig. C-12.

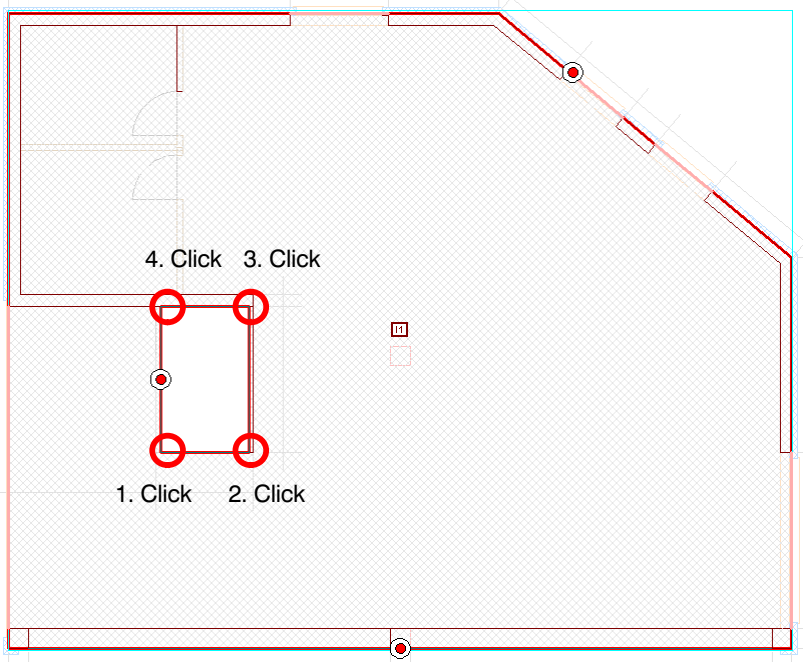
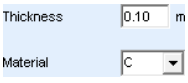


Fig. C–12 *Introductory example: Definition of a void at the staircase railing (Step 2)*

Together with the outline of the slab a material zone with a constant thickness and a material is introduced at the same time. Analogously as in the 'Structure' tabsheet of CEDRUS a material is uniquely defined by setting an attribute box (I1) inside the outline of the slab. Initially the position of the attribute box is placed automatically. Subsequently it be can replaced manually using the adjacent button.

☞ *The material zone is visualized in a grey hatched style in the display area.*



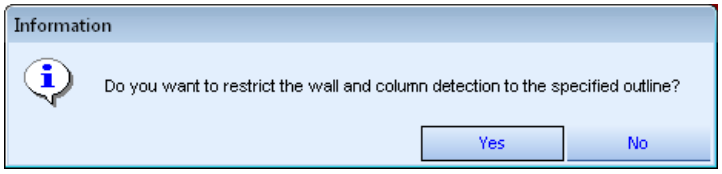
The attributes of the material zone (thickness and material) can be set using the adjacent text fields.

☞ *The outline detection is carried out automatically after each graphic input that has been made and the display is updated accordingly.*

Step 3: Drawing elements for structural member detection

By proceeding to the third step the program analyses all filled areas of the CAD drawing and saves them layerwise. When the third step is started for the first time, the process of structural member detection can be restricted directly to the area inside the defined outline of the slab.

▶ Confirm the subsequent dialog with [Yes].



All filled areas used in the process for structural member detection together with the perimeter can be seen in the display area as shown in Fig. C-13.

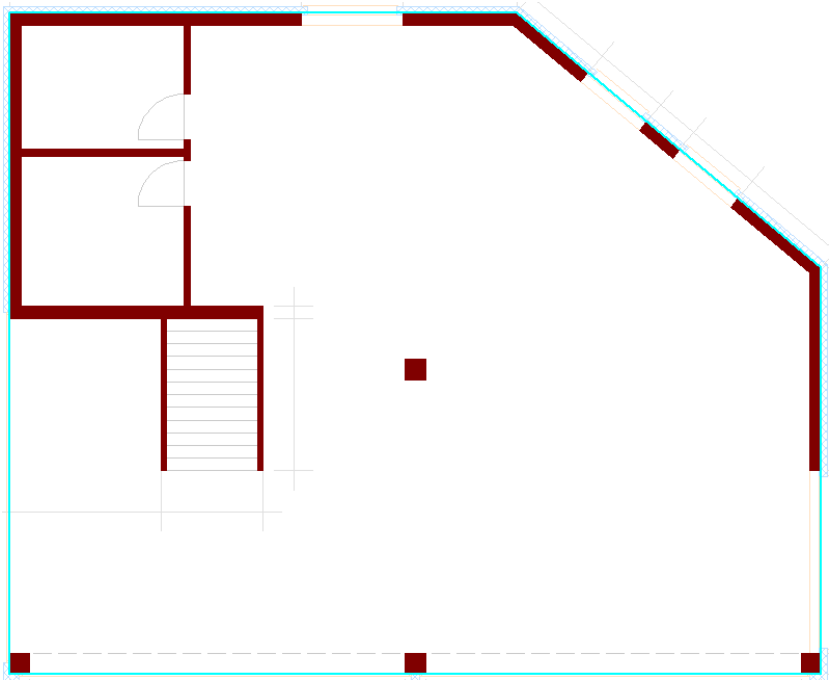


Fig. C–13 *Introductory example: Restriction of the filled areas for the process of structural member detection (Step 3)*

If the perimeter has been modified or deleted the detected outline of the previous step can be resumed and used as the perimeter for the process of structural member detection by using the adjacent button.

Alternatively a rectangular or polygonal perimeter can be defined manually. The proper graphic input mode is activated by using one of the adjacent buttons.

The number of drawing elements which are used in the next step during the structural member detection can be restricted in the same way as in the previous outline detection process by using filters. Both the detected Layers containing filled areas and the detected fill styles serve as filters. In this context a fill style of a drawing element is characterized by a fill colour and a fill pattern.

In the present introductory example the Layer settings and fill style settings shown in Fig. C-14 can adopted for the process of structural member detection.

☞ *All filled areas used for structural member detection are highlighted in the display area of the CADextract window. In the case of a change in the settings or of the graphics input the presentation is automatically updated.*

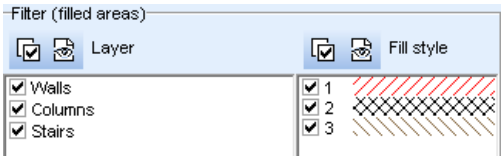


Fig. C–14 *Introductory example:Filter settings (Step 3)*

▶ Proceed to the next step using the adjacent button.

Step 4: Wall detection

The fourth step is wall detection. The filled areas are taken from the third step and are analysed. The filled areas automatically detected as walls are drawn red in the display area of the CADextract window and also the corresponding wall axes. Wall candidates, i.e. areas that cannot be clearly identified as walls, are displayed in light red and without an axis.

Likewise, as in the third step, it is possible to restrict the areas to be considered. The following serve as filters:

- Layers and fill styles: The further restriction to particular Layers and fill styles can be meaningful, if the drawing is clearly structured and the drawing elements can be separated from one another, e.g. if columns and walls were drawn in separate Layers. In the introductory example the Layers 'Columns' und 'Stairs' are no longer needed for wall detection.

▶ Set the Filter settings for the Layers and fill styles according to Fig. C-15.

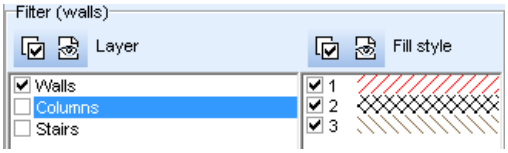


Fig. C–15 *Introductory example: Filter settings (Step 4)*

- Minimum wall segment lengths: Filled areas whose principal axis is shorter than the defined minimum wall segment length are not detected as walls. In the introductory example one can work with the setting 0.5m (see Fig. C-16).
- Wall thickness range: The minimum and maximum permissible wall thickness can be defined separately by directly inputting a value or alternatively an automatically detected wall thickness is selected. In the introductory example the right staircase railing is located in the Layer 'Walls' and thus is also detected as a wall. To exclude it, the minimum wall thickness must be set to , e.g., 12.5cm (see Fig. C-16).

▶ Set the filter settings for the wall segment length and the thickness range according to Fig. C-16.

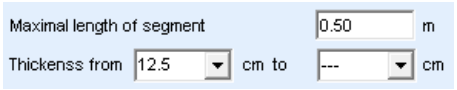
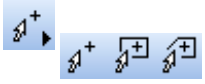


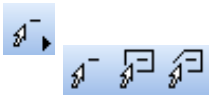
Fig. C–16 *Introductory example: Filter settings (Step 4)*

Fig. C-17 shows the detected wall segments in the introductory example, which result from the described filter settings according to Figs. C-15 and C-16.

In addition to the automatic wall detection, wall candidates can be converted into walls manually. This is done by activating a graphics input mode using the buttons shown below:



Likewise, walls incorrectly detected can be subsequently excluded. This can also be done by activating a graphics input mode using the buttons shown below:



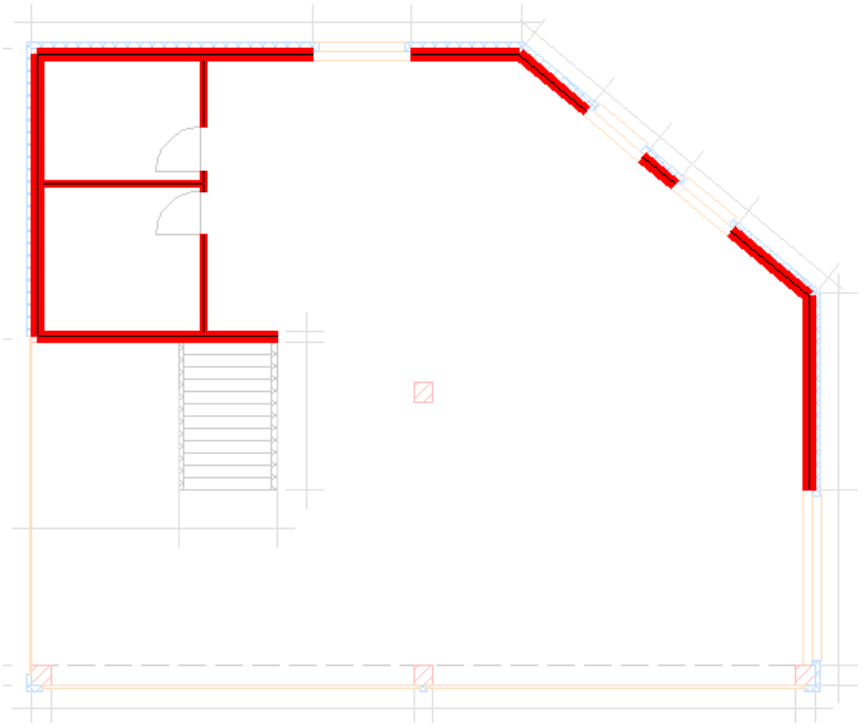


Fig. C–17 *Introductory example: Wall segments (Step 4)*

⚠ Note, all surfaces that are not detected as walls in this step or were manually converted into walls can no longer be defined as walls in the subsequent steps! Likewise, for subsequent column detection only the areas not designated as walls are considered, i.e an area detected as a wall in this step can no longer be converted into a column!

Next > ▶ Proceed to the next step using the button shown on the left.

Step 5: Material assignment for walls

In this step material properties are assigned to wall segements detected in the previous step. This is done by defining wall types.

A wall type consists of the characterization of the material (concrete, masonry, timber etc.), the property 'load-bearing/non load-bearing' and filter settings for automatic wall segment assignment. The filter attributes of a wall type consist of a thickness range (minimum and maximum wall thickness) and a fill style.

The defined wall types are shown in the CADextract window in rows in a list. Fig. C-18 shows, e.g., such a list.

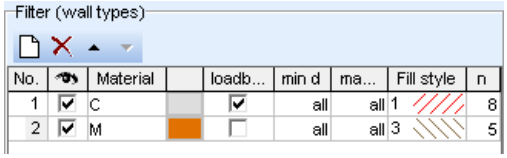


Fig. C–18 *Introductory example: Filter settings (Step 5)*

The first column contains the Type-No., the second column the visibility of the wall type. The material can be selected from a list in the third column. In the fourth column the associated material colour is shown. In the display area of the CADextract window the wall segments are drawn in the corresponding material colour of the assigned wall type. The attribute 'load-bearing/non load-bearing' is placed in the fifth column. Columns six to eight describe the filter settings for the automatic wall segment detection: The thickness range of the wall type is defined in columns six and seven, the fill style in column eight. (Besides selection from the list, the fill style can also be selected graphically (grabbed) from the display area.) The last column of the type list contains the number of wall segments assigned to a wall type.

☞ *The number of all wall segments not assigned to a wall type is shown below the type list. These wall segments are drawn in the cyan colour in the display area of the CADextract window.*

In the introductory example the load-bearing outer walls are assigned the material concrete and the non load-bearing inner walls the material masonry.

▶ Set the filter according to Fig. C-18, such that the display area of the CADextract window is set according to Fig. C-19.

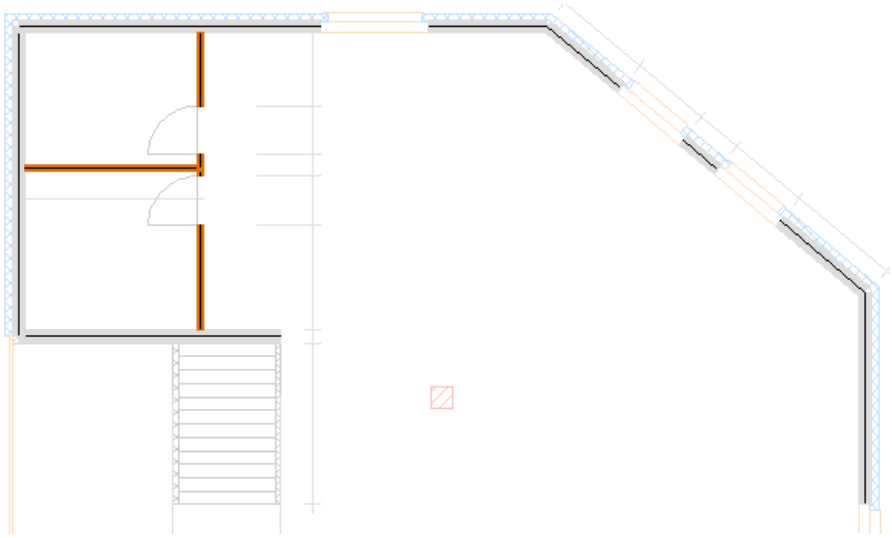


Fig. C–19 *Introductory example: Display area (Step 5)*

The sequence of defined wall types is decisive for the assignment of a wall segment to a wall type. Since the automatic assignment of the wall segments to a wall type is based solely on the filtering across the thickness range and the fill style, in principle it is possible that a wall segment can belong to several wall types at the same time. To exclude this possibility, a wall segment that matches the description of several wall types is assigned to that type that occurs first in the list. This automatic type assignment can be overruled by assigning a specific wall type to a wall segment manually.

After activating the graphics input mode with the adjacent button, by clicking on a wall segment this is assigned to the active wall type in the type list.

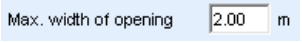
Note, all walls not assigned to a wall type in this step are omitted completely and not included for export to CEDRUS!

▶ Proceed to the next step using the button shown on the left.

Step 6: Closure of wall openings

After assigning material properties to the walls, in the sixth step wall openings between wall segments of the same type can be closed. The wall types are drawn in the display area in the material colour corresponding to their wall type. The closure of wall openings is carried out by connecting wall ends along a line input in the graphics input mode.

As an aid to assessing the size of the openings a maximum opening length can be defined in the corresponding text field.



Automatically detected wall openings, which are smaller than the prescribed maximum opening length, are visualized in the display area using a thick red line (see Fig. C-20). In the introductory example the upper openings of the exterior wall can be closed.

▶ Activate the graphics input mode by clicking on the adjacent button and close the upper openings by inputting the corresponding lines with two mouse clicks for the start and end points along the openings according to Fig. C-20.

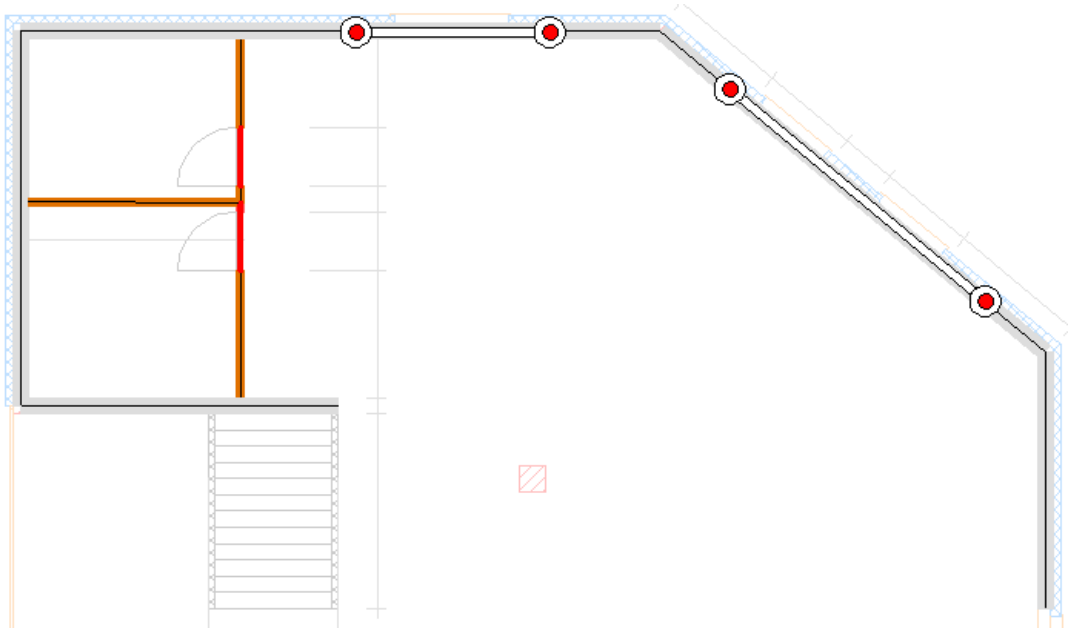
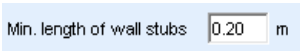


Fig. C–20 *Introductory example: Display area (Step 6)*

⚠ Note, the detected openings are not automatically closed! If this is necessary it must be done manually.

After closing the wall openings in this step also wall stubs, i.e. wall ends that are shorter than the minimum wall segment lengths (see Step 2), are filtered out. The minimum length can be defined in the corresponding text field.



☞ *Wall stubs that are smaller than the defined minimum length of a wall stub are not displayed.*

In the introductory example there are two wall stubs in the door area of the interior walls.

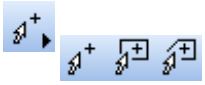
⚠ Wall stubs that are not connected interactively with other wall ends, are rejected when moving to the next step and completely removed!

▶ Proceed to the next step using the button shown on the left.

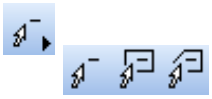
Step 7: Column detection

All drawing elements not detected as walls and all unrejected drawing elements are considered for column detection in this step. As in the first and second steps, it is possible to restrict the detection to specific Layers and fill styles. For each candidate column the cross section dimensions of the enclosing rectangle are determined with reference to the principal axis system. By selection or direct input of the minimum and maximum cross section dimensions the column detection can be restricted. The cross sections detected as columns are drawn in green in the display area of the CADextract window.

As in the second step, automatically detected column candidates can be converted into columns. The following buttons are available to activate the corresponding graphics input mode:



In addition, specific detected column cross sections can be rejected. The corresponding buttons of the graphics input mode are:



▶ In the introductory example, reject the left staircase railing as a column, by activating the adjacent button in the graphics input mode and clicking on the railing. Fig. C-21 shows an excerpt from the corresponding display area in the CADextract window.

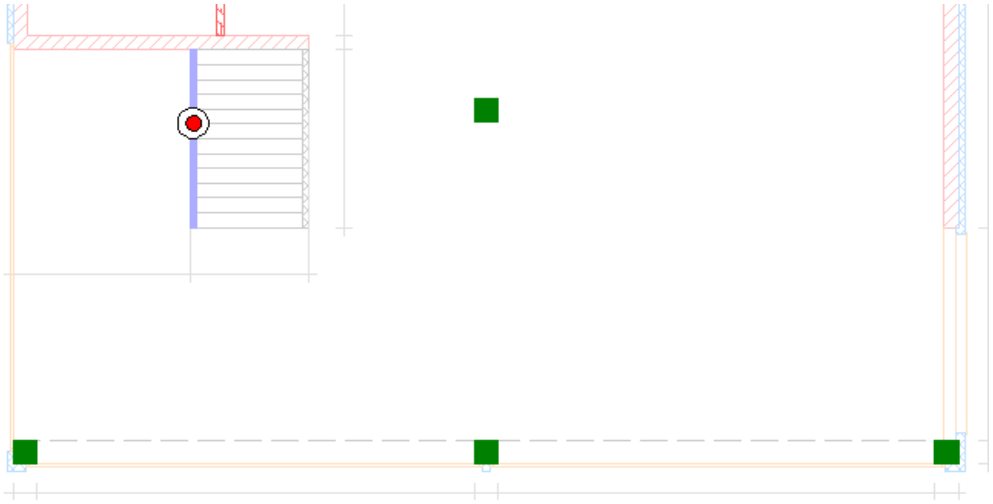
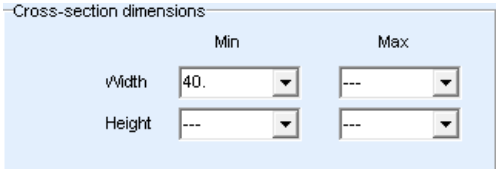


Fig. C–21 *Introductory example: Column detection (Step 7)*

☞ *Excluding the railing can also be achieved if either the Layer 'Stairs' is deselected in the filter settings or the range of the column dimensions is restricted, e.g. with the following settings:*



Only the highlighted green filled areas are available for subsequent use in this step as column cross sections!

▶ Proceed to the next step using the button shown on the left.

Step 8: Material assignment to columns

A material is assigned to each of the column cross sections defined in the previous step. In contrast to a wall type the definition of a column type is restricted to just the material (concrete, steel, timber, etc.) and to the filter attributes for the automatic type assignment of the column cross section to a type.

The procedure for type assignment is in principle the same as in the fifth step : The defined column types are presented in rows in a list. The column type filters are the minimum and maximum permissible dimensions of the outline rectangle of a column cross section.

As in the fifth step it is possible to assign column cross sections manually to a column type.

In the introductory example the automatic material assignment of the four columns is left unchanged. Fig. C-22 shows the corresponding filter settings.

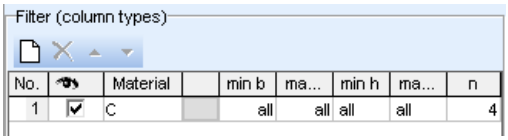


Fig. C–22 *Introductory example: Filter settings (Step 8)*

In general all column cross sections, to which no type is assigned in this step, are neglected!

▶ Proceed to the next step using the button shown on the left.

Step 9: Final checking and exporting the extracted data

The last step serves as a final check of the extracted data which is exported to the 'Structure' tabsheet of CEDRUS. In the display area the definied material zone and the extracted walls and columns of the CAD drawing of the previous steps are shown. Here one can switch between a representation in the classical CEDRUS structure presentation and a representation in the material colours. Optionally the geometry of the material zone including the attribute box and/or the geometry of the walls and columns respectively together with the assigned material properties can be exported.

Fig. C-23 shows a detail of the display area of the CADextract window at the end of the structural member detection of the introductory example.

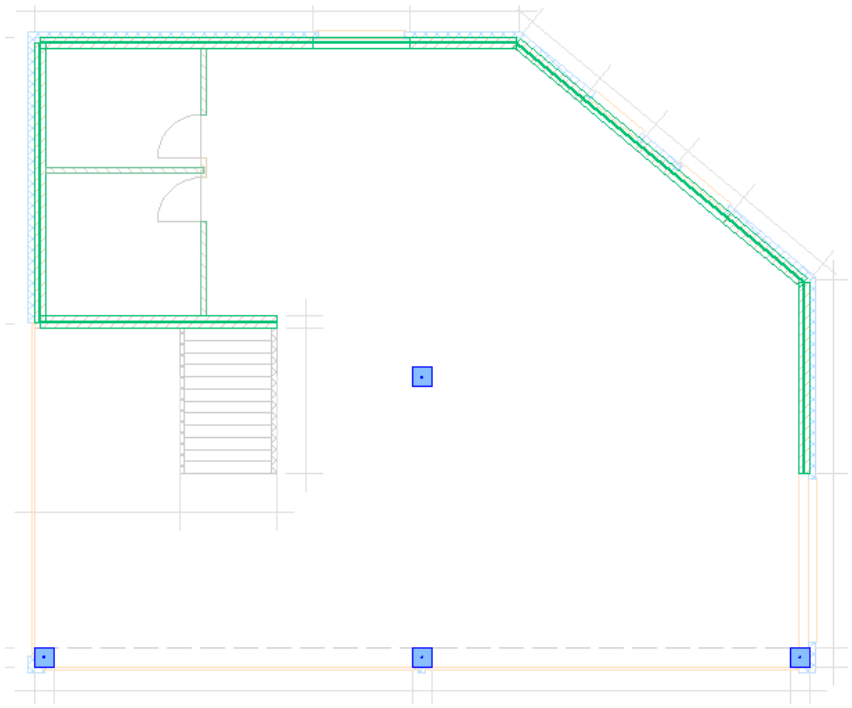
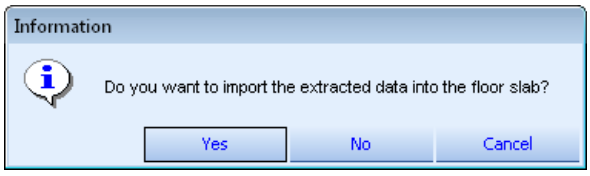


Fig. C–23 *Introductory example: Checking and export (Step 9)*

OK

▶ Close the CADextract function by clicking on the adjacent button in the CADextract window. To import the extracted data, select the button [Yes] in the dialogue window.



⚠ In the process of transforming into CEDRUS structural elements, column cross sections that touch each other and are of the same type are merged to form a single cross section!

C 3.4 Design of a CAD drawing for an efficient work with CADextract

The effort one has to invest working with CADextract mainly depends on the quality of the CAD drawings. CADextract is very powerful in detection and filtering drawing elements. But an experienced CAD user might find more convenient ways built in commercial CAD programs to carry out the graphical clean-up process which is often necessary. For this reason it might be appropriate if an user who is familiar with the CAD program designs or redesigns the CAD drawing in such a way that the user of CEDRUS is able to use CADextract with less effort to generate CEDRUS structural elements.

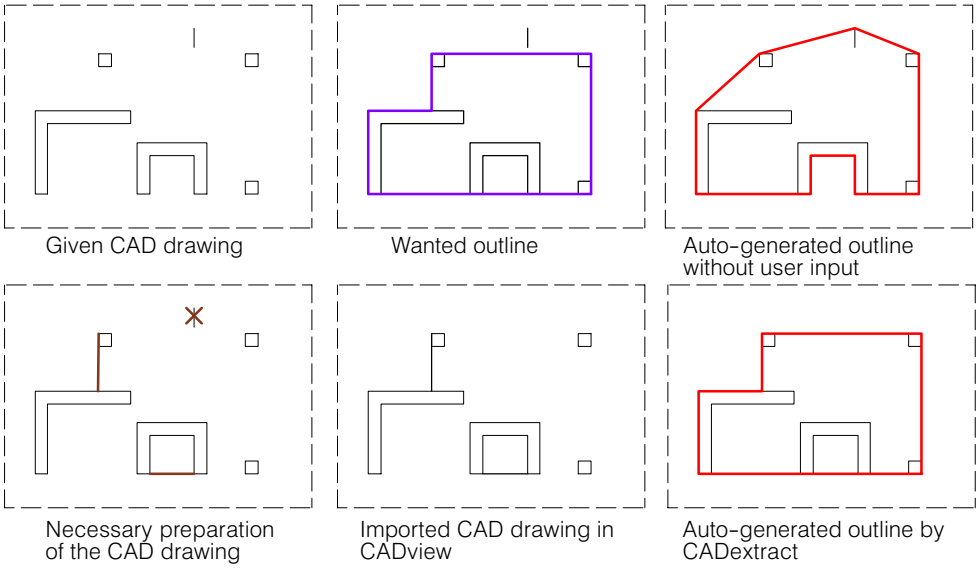
The recommended way consists of the three following steps: preparation, actions aimed for outline detection and actions aimed for walls and columns detection, respectively.

Recommended preparation

All Layers which are needless should be deleted (see below).

C 3.4.1 Outline detection

In the first step CADextract generates a convex hull of all visible line segments and line ends respectively. CADextract tries to find a closed polyline from the start to the end point of each part of the convex hull wich is not congruent with line segments of the CAD drawing. If a polyline can be build with the existing line segments of the CAD drawing the related part of the convex hull is replaced by the polyline.



Recommended actions

1) Generate a Layer named 'Outline'
2) Option1: Way 1: Copy all needed drawing elements into the Layer of the outline and correct the lines in such a way that CADextract detects the geometry of the wanted outline without any graphical input by the user.

   Option 2: Draw the wanted outline explicitly.

C 3.4.2 Structural member detection

The CADextract function is focused on the detection of filled areas as walls or columns. To ensure an efficient detection the following properties should be obvious in the CAD drawing:

- Walls and columns,
- Materials of walls and columns,
- Load-bearing or non load-bearing walls.

To distinguish the filled areas by the mentioned properties the drawing elements should be either placed in different Layers or they should have different fill styles.

Recommended actions

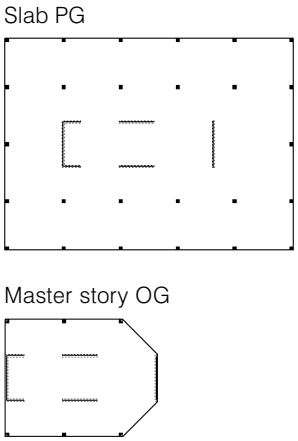
1) Delete all filled areas of the CAD drawing which do not represent walls or columns.
2) Rearrange the remaining filled areas in separate Layers in such a way that the criteria mentioned above are fulfilled.
3) Introduce filled areas for missing walls or columns.
4) Delete all wall stubs and similar drawing details which are not necessary in any case for the CEDRUS model.

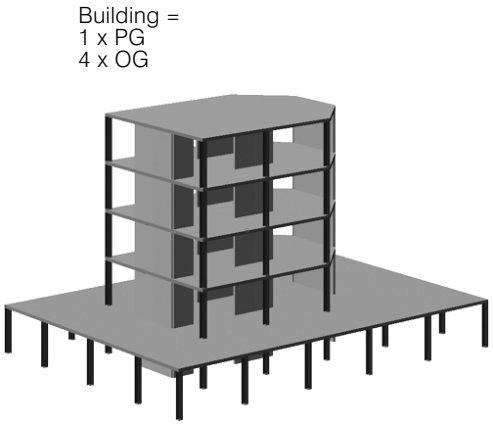
Vol. D Building Option G

D 1 Introduction

CEDRUS in its basic version is a user-friendly and efficient program to analyse individual reinforced concrete slabs. It reaches its limits, however, when it is important to consider the entire building, whether for the top-down analysis of loads or the transmission of horizontal forces due to actions like wind and earthquake.

Since in a CEDRUS analysis with the floor slab, columns and walls practically all the information of a building story is input, the logical step is to generate a geometric building model by stacking CEDRUS slabs.





The Cubus **Building Module** (for definition of terms see below) has been developed as an extension of the CEDRUS slab program. In the current version it consists of two program options:

- **G:** This enables the generation of the **geometric building model** as well as the automatic top-down analysis of **vertical loads** using the stacked slab model.
- **H:** Extends Option G. With this option **horizontal loads**, can also be taken into account, analysed top-down using the stacked slab principle. This option also allows the generation of various three-dimensional **frame models** for analysis with the program STATIK. These models contain complete STATIK input including loads and mass distributions, so that in STATIK one can begin directly with the earthquake analyses implemented there.

Definition of terms:

- **Building Module**: This is an extension of the program CEDRUS that allows entire buildings to be considered.
- **Geometric building model:** Here we strive to obtain as complete a description of a building as possible, primarily geometric, whereby questions of structural modelling are not yet considered.
- **Stacked slab model:** With this model both vertical and horizontal loads transmitted through walls and columns are determined from the top to the bottom of the building one story at a time, the results being passed on each time to the underlying story.
- **Frame models:** These are static models to deal with horizontal forces, appropriate to earthquake analyses using the program STATIK.

Example

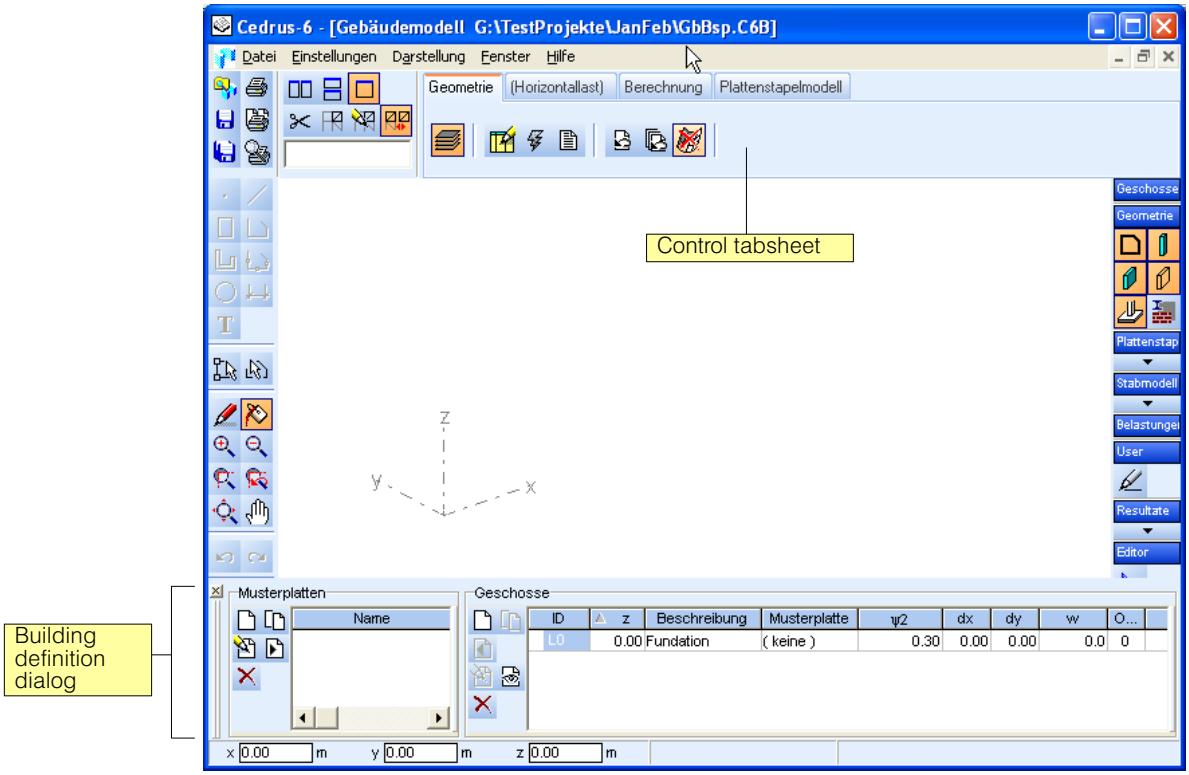
This part of the manual describes the functionality of the Building Module covered by Option G. The next part is concerned with the possibilities of Option H. A simple building example from A-Z on Option G is described in § A 2.3.

D 2 Starting the Program and Organisation of the Data

D 2.1 Starting the Program and the Building Window

CEDRUS handles two types of analysis, slab analysis (incl. shear walls) and building analysis. When starting a new CEDRUS analysis, whether from CEDRUS with 'File > New' or using then CubusExplorer, the corresponding analysis type (i.e. here Building Analysis) must be chosen. In the case of existing CEDRUS analyses, the analysis type cannot be changed.

After starting CEDRUS and opening a new or an existing building analysis the program window looks as follows:

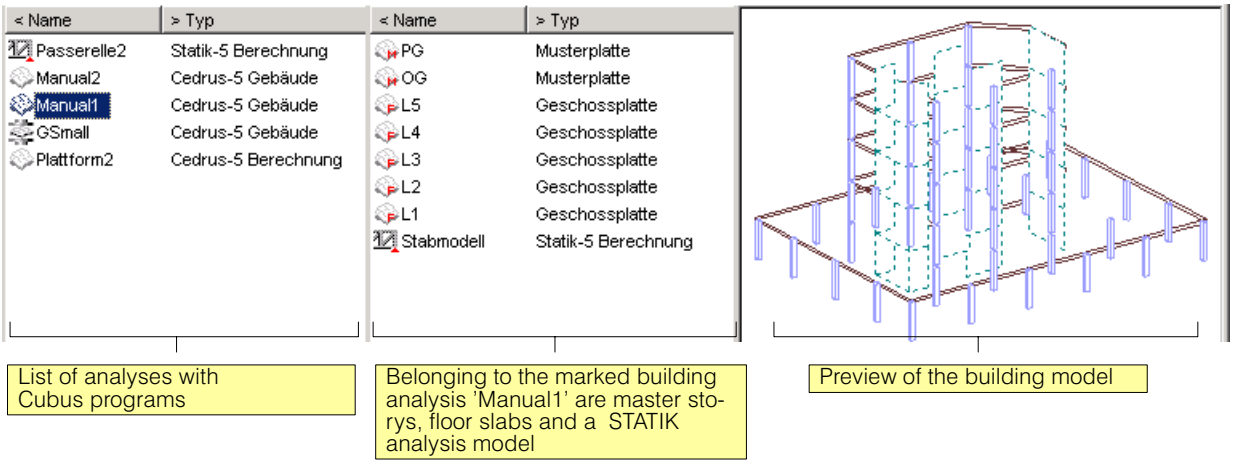


With the program the Building Document opens automatically (in the Building Window). It contains tabsheets typical of Cubus programs. Besides the Building Document, slab analyses for the building can also be opened, since the Building Module, like other Windows applications, is a multi-document application. How these document windows are used is described § D 2.3.

At the bottom of the Building Window is the Building Defintion dialog for the management and modification of **master storys** (left side) and the **storys** (right side) is located. These storys and master storys are normal CEDRUS slabs and therefore also called slabs and master slabs.

D 2.2 Organisation of the Data

A building analysis appears in the Cubus-Explorer, as the data of other Cubus programs, with its name in the list of analyses. In addition to the actual building data it also contains the data of the master storeeys, floor slabs and any generated STATIK analyses. The following figure shows an excerpt from the CubusExplorer, which shows these items:



The 'Sub-Analyses' shown in the middle table are not shown in the normal application of the CubusExplorer. If, however, this is called using the menu 'File > Import ...' , the sub-analyses are visible, so that they can be selected for import. Storys and master storys of a building, for example, can be imported into a normal CEDRUS slab analysis or into any such analysis for another building.

D 2.3 Handling Windows

Program, building and slab windows

When you start CEDRUS, the **CEDRUS Program Window** is opened. The program cannot be started over and over again, and therefore there is at most only one CEDRUS program window on the screen.

When opening the building analysis the **Building Window** is opened within the program window with the selected building analysis. Only one building analysis can be open at any one time. Closing the building window is equivalent to closing the corresponding building analysis.

CEDRUS is a multi-document application (operating system terminology), which means here that besides the building window several slab analyses can be open at the same time. Each of these analyses has its own document window, which we call the **Slab Window.**

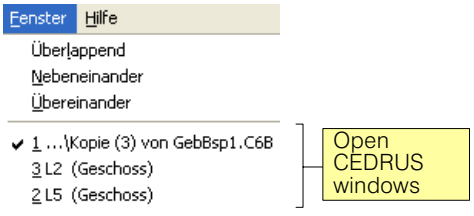
We describe in the following how one handles these windows.

Open, active and visible building and slab windows

First we want to define the terms open, visible and active for building and slab windows. It is assumed that CEDRUS, i.e. the program window, is open.

Open windows

To see if a window is open we can look in the menu 'Windows' :



Open windows normally remain open until they are closed by the user in the same way as other Windows applications. To close all slab windows at once, there is in the menu 'File' the function **'Close all slabs'**. An open window is not necessarily visible. How to make it visible is explained below.

In principle, one doesn't need to close open slab windows. There are, however, certain functions, such as deleting slabs, that can only be applied to closed slabs. In the case of large buildings it is recommended for safety's sake, however, to close slabs that are no longer needed (in the case of closed slabs one knows that they are saved), storage economy and the overview.

Active window

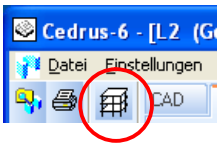
One of the open windows is always the active window, with which the user is directly communicating. Which window is the active one may be seen as follows:

In **full screen mode:** the visible window is the active one.

Note that when modifiying slabs in the full screen mode one should check every now and again the heading of the program window, to make sure that you are actually working with the desired slab:

In the **window mode:** The frame of the active window is different (depends on Windows version and settings).

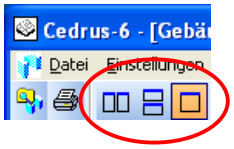
An open (possibly invisible) slab window can be activated in the usual Windows way, e.g. using the menu 'Windows'. To open or activate slabs CEDRUS provides various possibilities (see below under 'Change to a slab window).



To activate the building window from an active slab window there is a special button.

Visible

An open window doesn't always have to be visible. Controlling the visibility is managed basically in the usual Windows way. The activation (see above) of an invisible window also makes it visible. Also note the following possible settings:



In the building window there are buttons for the standard layout of the document window. Normally, and especially for less experienced users, the layout 'Maximized Window' (extreme right) is the easiest to work with. It ensures that the active window is always maximized, i.e. displayed in full screen mode,.

Change to a slab window

Changing to a slab window can simply mean to open it or activate a window that is already opened and, if necessary, to make it visible. There are a range of possibilities for doing this:

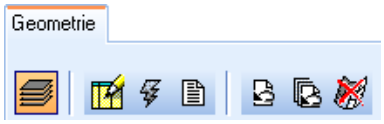
1) Open slab windows can be activated in the menu 'Windows'. The menu also contains the usual Windows automatic layout possibilities, like 'adjacent' or 'cascading' (i.e. on top of one another), which makes all open windows visible.
2) If you want to open and/or activate a slab analysis, then in the shown Building Definition dialog the desired slab is opened (click on an input field in the slab row) and then click on the button to modify/view the slab. This also applies to master and floor slabs.

   Still quicker is a double click on a field of the slab row.
3) A double click on an element in the 3D building model automatically opens/activates the floor slab, to which the element belongs

With this method of opening, always the floor slab is opened and not the master slab, on which it may depend.

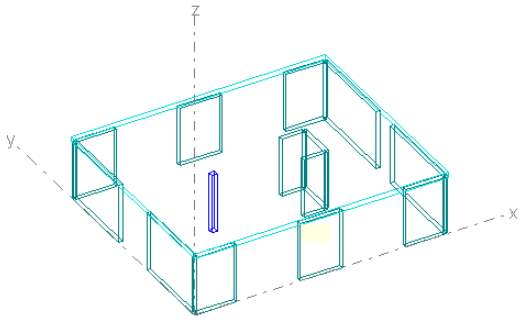
D 3 Geometrical Building Model

The geometric building model is developed in the tabsheet 'Geometry' of the Building Window:



D 3.1 Coordinate System

A building is defined in a Cartesian, right-hand x,y,z coordinate system:

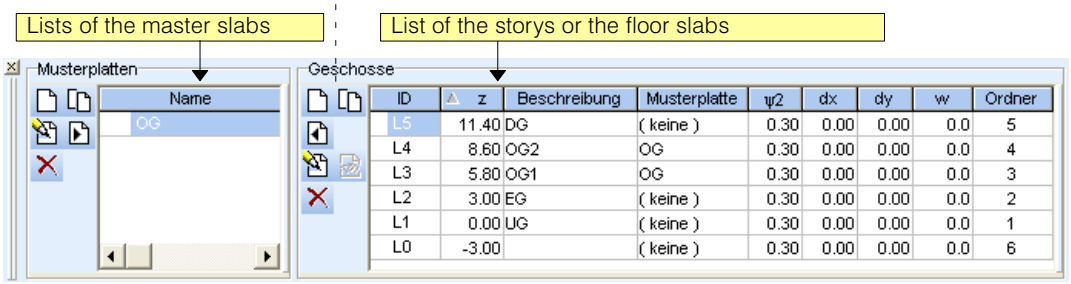


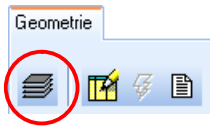
The z axis points upwards and x,y are the orthogonal horizontal directions.

The x,y coordinates of the floor slabs correspond to the standard x,y coordinates of the building. It is possible, of course, to move or rotate the coordinate systems of the floor slabs in the Building Definition dialog one story at a time.

D 3.2 Storys

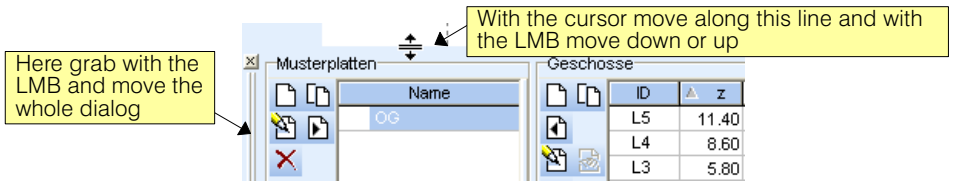
D 3.2.1 The building Definition dialog



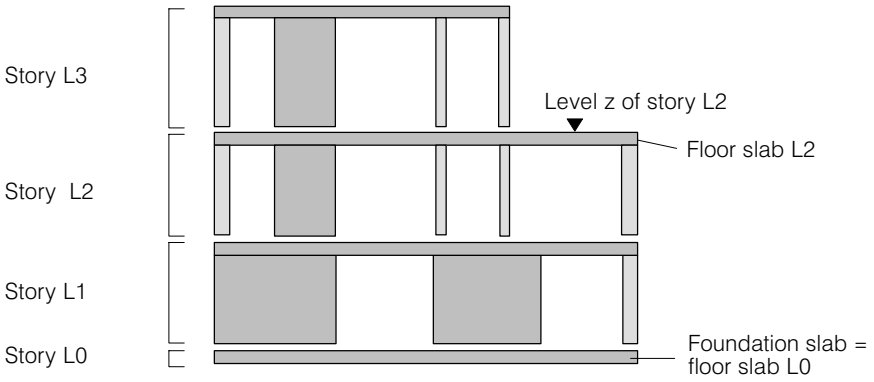


A building model is defined by the list of **storys** on the right side of the above Building Definition dialog**.** If the Building Definition dialog is not visible, it can be activated using this button shown on left in the building window.

The Building Definition dialog is docked (i.e. located) by default at the bottom of the program window. Its height can be adjusted and the whole dialog can, if necessary, be made available or docked elsewhere (usual Windows-type manipulation):



A story includes a floor slab, a level (i.e. height) and the other properties listed in the Building Definition dialog (see below).



The column and wall supports of the floor slab form the columns and walls (= wall pillar) of the story.

☞ *The floor slab can be missing in the case of the foundation story (no foundation slab) and then consists simply of an area support zone, possibly of unlimited extent.*

D 3.2.2 Master Slabs / Floor Slabs

In CEDRUS one differentiates between master slabs (also called master storys), which are listed in the left table of the Building Definition dialog, and floor slabs (also called storys) listed in the right table.

A floor slab may be a CEDRUS slab (structure and loads), or it may be a clone of a **master slab**. If it is a clone of a master slab, it **depends** on it and changes made in the master slab always affect the clones that depend on it. How the dependencies come about or are activated, is described below.

Working with **master slabs** makes sense if in a building there are several identical storys in regard to structure and loads. Such identical storys are then declared to be clones of a master slab. This has the advantage that one can see straightaway that in a building there is a group of identical storys, and any changes only have to be made in the master slab.

A building analysis can, to be sure, be performed without master slabs. Master slabs may exist, however, on which no floor slabs depend. If one wants, e.g., to remove an existing floor slab from the building model (perhaps provisionally), without losing it, then instead of deleting it, it can be moved to the list of master slabs.

Both master slab and floor slab analyses are in fact normal CEDRUS slab analyses.

D 3.2.3 The Story Properties in the Building Definition Dialog

ID: Story identification. It is automatically assigned according to increasing level z and may change if the level is changed.

z: Level of the upper edge of the floor slab in the building model. The level can be changed at any time as desired, but then a new story sequence may result.

Description: Description of story is free.

Master story: If the story depends on a master story, then its name is given here, otherwise it has 'None'. By clicking on an input field in the column 'Master story' a selection becomes active, from which one can change the dependence (adjacent figure).

If an independent story is made to depend on a master story, then its original content is lost.

If a previously dependent story is made independent of the master story using 'None', then it begins its independent existence as a copy of the master story.

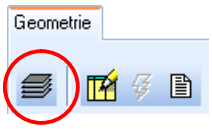
dx, dy: For a displaced position dx,dy of the story.

w: For a new positioning by means of a rotation by an angle w (about 0,0 of the slab).

Folder: In contrast to the ID, here we have an identification that is directly connected to a particular slab, which does not change if the sequence of floors changes. It corresponds to the name of the subfolder, in which the slab data is stored.

The folder name is only of importance to the user if he wants to import slabs from one building into another building. The CubusExplorer then shows not the story IDs, but the folder numbers. Thus here one has to be careful that one has the correct slab.

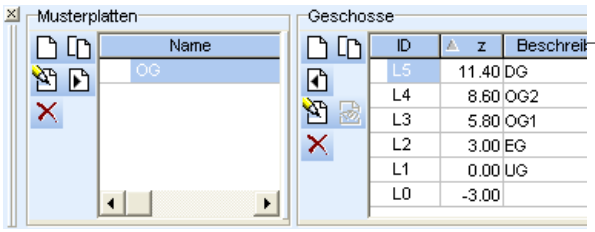
For a story that depends on a master story the structure and loads cannot be changed. If there are differences, e.g. different loadings, the corresponding storys has to be made independent of the master story before one can make changes in it.



The Building Definition dialog is only avaiable in the Building Window and can be activated using the button shown on the left.

D 3.3 Creating and Managing Storys and Master Storys

The crreation and management of storys and master storys is done in the Building Definition dialog.



Some of the buttons described below are available both for the master storys and the building storys. They always act, of course, only on the corresponding slab type.

With this button a new master story or story is created. In the process it is immediately opened and is ready for modification.

Storys can also be duplicated. One selects the story that has to be duplicated and then clicks on the button shown on the left.

This is a very useful function, since the various storys of a building often have much in common and the quickest way of creating a new slab is by modifying an existing one.

The copy created by duplicating a story is an independent story and does not depend on the original. Storys that depend on master storys cannot be duplicated.

In the case of identical storys (structure and loads) it is more efficient to work with master storys.

With this button the selected story (master or not) are opened or activated and can be modified as in a normal slab analysis. The same can also be achieved by double clicking on a field in the slab row. In view of the use of the slab in the building model, there is still data to be input, which is done in the tabsheet 'Building Element'.

This button is only active if an independent floor slab is selected in the story list. By means of it the selected floor slab and any available results can be viewed, but not modified (see also the previous button).

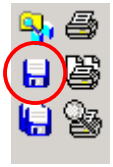
With this delete button the slabs marked in the corresponding list are deleted. Slabs that are still in the process of being modified, i.e. are open, cannot be deleted.

With this button a new dependent story is created from the selected master story. The automatically generated level can subsequently be adjusted to the given circumstances.

With this button a copy of an independent story can be added to the list of master storys. There is in this case no dependence between the story and the master story.

D 3.4 Modifying Storys

D 3.4.1 General



Storys are just normal CEDRUS slabs. When beginning to modify such a slab, a document window for a CEDRUS slab analysis is opened. The slab, as customary, can then be input or an existing slab can be imported using the menu 'File > Import > ...' and modified or extended. Older CEDRUS slabs and also those from other building analyses can be imported.

If storys depend on master storys, they can only be viewed, but not modified. If one wants to modify a dependent story, beforehand the dependence must be removed.

If slabs are input or modified, they always have to be saved, so that in the Building Module dependent parts are notified of it and can be updated accordingly!

Limitations in the slab input

Lines of symmetry are not meaningful in the building model. Therefore they are not accepted in the slab input.

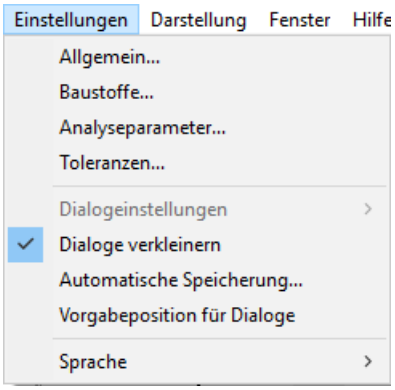
Extensions for wall and column input

Within the building model, additional attributes can be entered for the walls and columns in the corresponding object dialog: For a support, a skew can be specified that changes the geometry and load transfer of the building. For a wall, openings can be defined which influence the weight and the weight of the building. See also chap. B 8.9.1 for supports and chap. B 8.10.3 for wall openings.

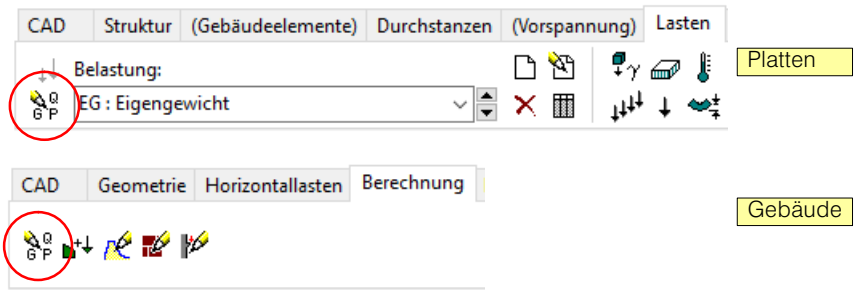
Common characteristics of all slabs

The following settings or data are centrally managed and apply to the entire building. They can be entered in the building window as well as in the individual slabs:

- Most of the inputs in the Settings menu, in particular building materials, analysis parameters and tolerances:

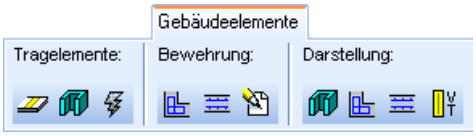


- The dialog for managing the categories of actions, which can be called up in the load register of the slabs, can also be reached from the Calculation tab of the building window:



Additional input for building

In regard to the building model and the analysis models derived from it, in some circumstances additional input is required. This is carried out in the tab sheet 'Building Elements':



Basically it is a matter of

- the input of slab strips, which have to be considered in a frame model, if needed,
- the properties of wall pillars (for this term see § D 3.4.3) and their grouping and
- any reinforcement of wall pillars required.

D 3.4.2 Slab Strips for a Frame Model

☞ *Slab strips are used in frame models (see page E-55).*

Besides downstands, which are always modelled as members in a frame model, slab strips with a chosen width can be input, which also have to be modelled as members of a frame model.

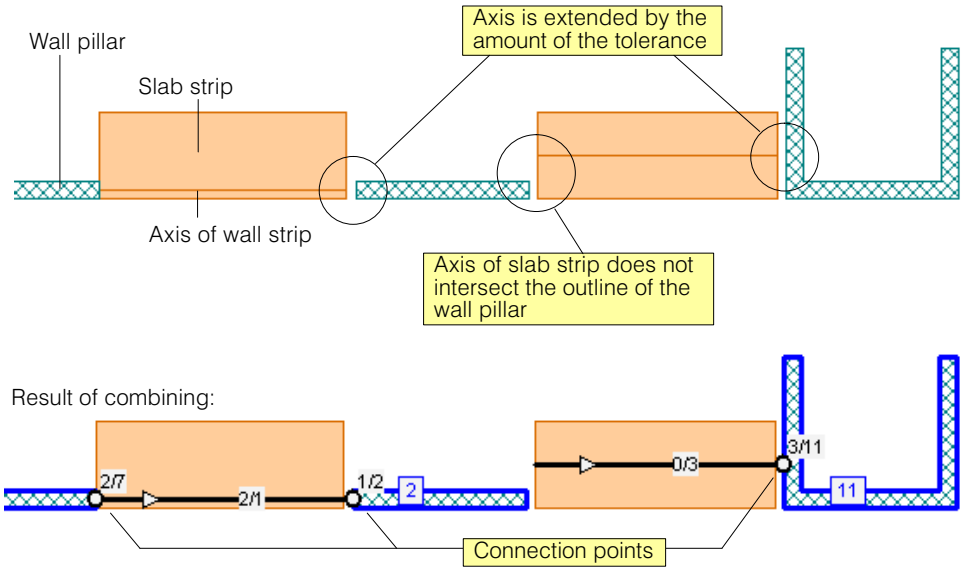
To design such slab strips, click on the button shown on the left.The following dialog appears, in which the width of the strip is specified.

After clicking on [Introduce] in the dialog the axis of the slab strip is drawn as a line.

Slab strips are only meaningful if they are supported on other elements and do not terminate at some arbitrary place in the slab field. These may be vertical elements (columns, wall pillars) or downstands or other slab strips.

Connection of slab strips and downstands to vertical elements

The program extends the axes of downstands and inputs slab strips automatically to meet the outlines of the vertical elements. The following figure shows two slab strips and how these are connected to the existing wall pillars:

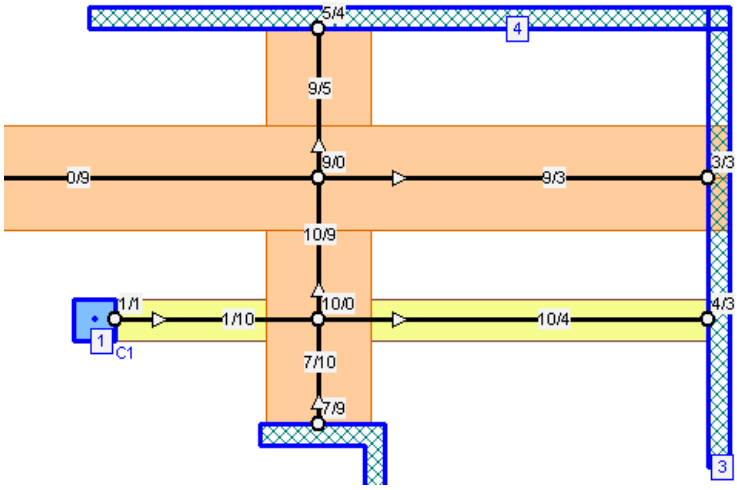


☞ *In the frame model there is a rigid connection between the connection point and the centre of gravity of the corresponding wall pillar or column (eccentric member connection)*

The result of combining (bottom of above figures) can be checked using the button shown. It creates a Results Layer, whose visibility can be controlled using the corresponding button in der Layer group 'Results'.

Connection between slab strips and downstands

The program also extends the axes of downstands and slab strips to interconnect with each other. The points of intersection become nodes in the frame model where the horizontal elements are connected.



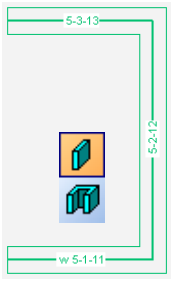
D 3.4.3 Wall Pillars

When inputting the slab the walls are input as **wall polygons**. A wall consists of one or more wall **segments** (polygon sides). Whereas the subdivision of the walls into wall polygons for the slab input is carried out according to criteria of input efficiency and has no influence on the analysis model, for the mode of action of the walls in the building model more information is needed. In the building model, therefore, there are only so-called **wall pillars**, which are derived as follows from the wall polygons:

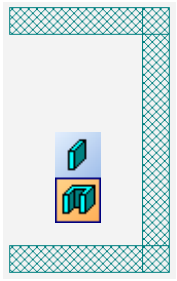
A wall pillar consists of one or more wall segments. The user determines which wall segments are grouped to a wall pillar. A wall pillar can be made up of wall segments belonging to different wall polygons.

By default, every wall segment is a wall pillar. For the arbitrary grouping and regrouping two functions are provided in the tabsheet 'Building Elements':

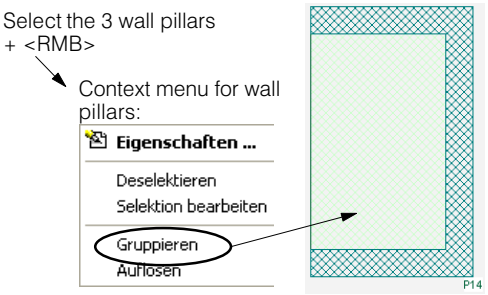
- **Grouping several wall pillars to form a new wall pillar:** One chooses the wall pillars to be grouped and then performs the selection in the context menu 'Group'.
- **Ungrouping an existing wall pillar**: The desired wall pillar is selected and in the context menu 'Ungroup' is selected. In this way all wall segments of the selected wall pillar become individual wall pillars.



Tabsheet 'Structure': wall consisting of 3 segments



Tabsheet 'Building Elements': thie 3 wall segments appear by default as 3 independent wall pilars



If necessary, several wall pillars can be grouped using the procedure shown here, i.e. merged to a single wall pillar. 'Ungroup' is the reverse function.

Properties of wall pillars

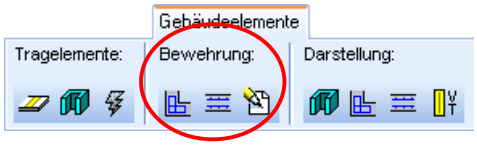
For the wall pillars there is the Properties dialog shown on right and the corresponding button in the tabsheet.

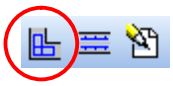
The interpretation of the properties depends on the analysis model to be created. The corresponding descriptions are given in Ch. D 5 for the stacked slab model and in E 3.2.2 for the frame model.

The wall pillar properties also include the **reinforcement**. This is needed for verification and dimensioning tasks and is described in the following chapter.

D 3.4.4 Reinforcement for Wall Pillars

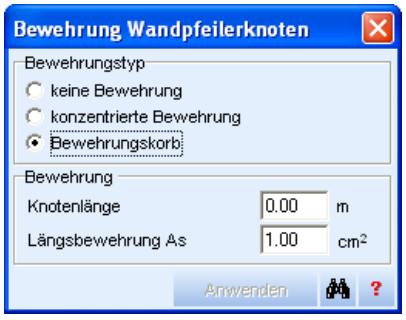
For the dimensioning or verification or reinforced concrete wall pillars, reinforcement and a reinforcement layout are needed.

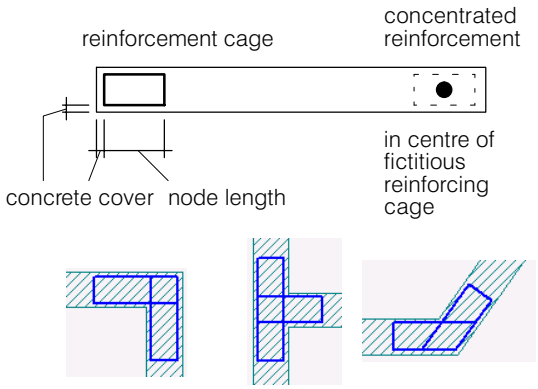




The reinforcement of a wall pillar consists of the following elements:

- **Wall pillar nodes**: At every free end, at every point of intersection and at every corner point of a wall pillar there is a wall pillar node. By clicking on the button on the left all wall pillar nodes are made visible and the following dialog appears





The assignment of properties is done in the usual way by (1) selecting the desired wall pillar nodes, (2) specifiying the dialog properties and (3) using [Apply].

There are two possible layouts:

- reinforcement cage: the given As of the longitudinal reinforcement is assumed to be distributed uniformly along the lines shown

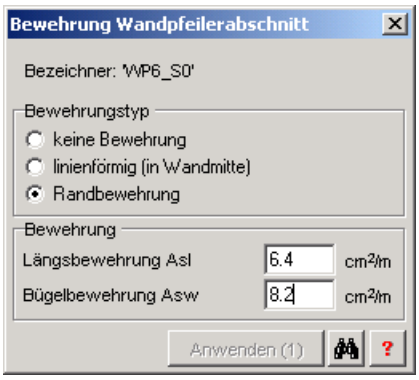
- concentrated reinforcement: the whole of the reinforcement is concentrated in one point in the centre of the fictitious reinforcement cage

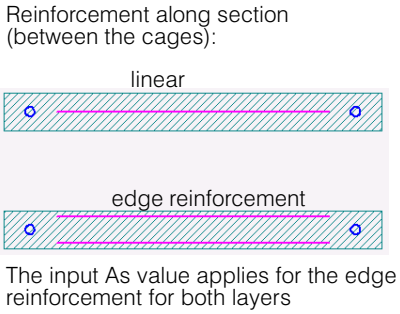
From the structural (statics) point of view both layouts are more or less equivalent, since within the framework of the models used both provide a good value for the total reinforcement in a node. Since both layouts do not correspond to any actual reinforcement layout, the choice is basically a question of user preference.

In the dimensioning of a wall pillar the total reinforcement of all its nodes increases in proportion to the input values, until the dimensioning criterion is fulfilled.

The **concrete cover** applies globally and is specified in this dialog:

- **Wall pillar sections:** Between the wall pillar nodes a uniformly distributed single layer or two layers of longitudinal reinforcement as well as stirrup reinforcement can be specified.

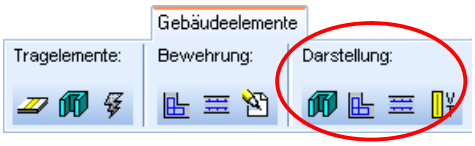




The reinforcement in the wall pillar sections is fixed and is not increased during dimensioning.

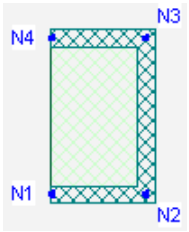
D 3.4.5 Visibility Control for the Wall Pillars

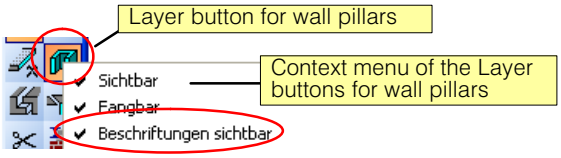
With the buttons shown in the following figure the visibility of some properties of the wall pillars can be switched on and off:



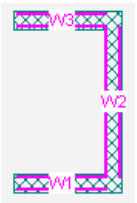
Using a circumscribing (i.e. enclosing) polygon for wall pillars to obtain better visibility of the particular wall elements grouped to form a wall pillar.

Visibility of the reinforcement in the nodes. The corresponding labelling is needed for the tabular outputs of the dimensioning results (STATIK) and is only visible if the visibility of the labelling of the wall piller layers is activated:





Visibility of the wall reinforcement. Regarding labelling, the same applies as for nodal reinforcement



Visibility of shear walls used for dimensioning (see FAGUS manual)

D 3.5 Creation of the Geometric Building Model

D 3.5.1 Assembling the building from storys

The geometrical building model is based on the storys listed in the Building Definiton dialog in the column 'Storys'.

The storys are sorted automatically according to their levels z and identified correspondingly with 'L0', 'L1', 'L2'... (Level). The story level is with reference to the top of the floor slab.

At a particular level, i.e. at one and the same level, there can be several storys. For example, there can be two building towers rising from the same underground garage or next to a normal story there may at the same level be a basement story (see below).

At the start of a new building analysis there is always a story generated by the program with the level 0.0, which simply consists of an area support box and thus serves as in infinitely long basement story without a foundation slab (see below). This story can be arbitrarily changed or even deleted. If there is a foundation slab, the story can be extended accordingly. The story levels are arbitrary and do not have to begin at 0.0.

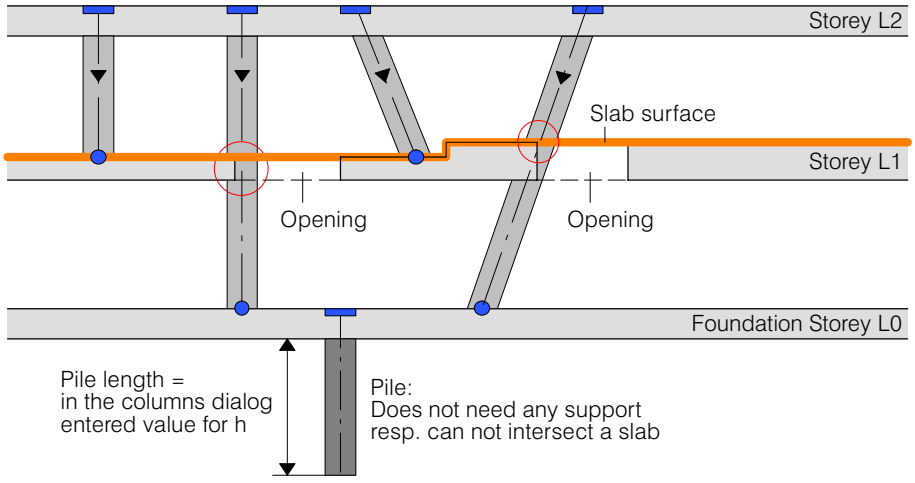
Support of wall pillars and supports on lower slabs

All vertical elements, i.e. Supports (but not piles) and wall pillars extend downwards from the slab in which they have been inserted, until their axis reach a slab on a lower level (see below for 'allowed slab types ...'). If no such slab is found, an error message is displayed.

Please note the following:

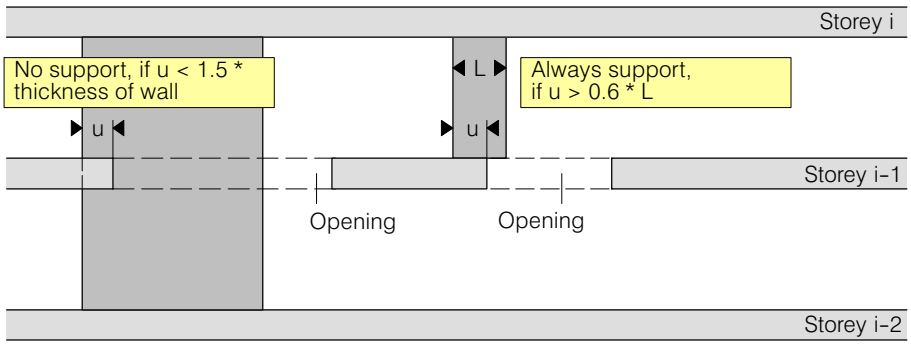
For columns and piles:

The support axis must meet the effective slab top surface:



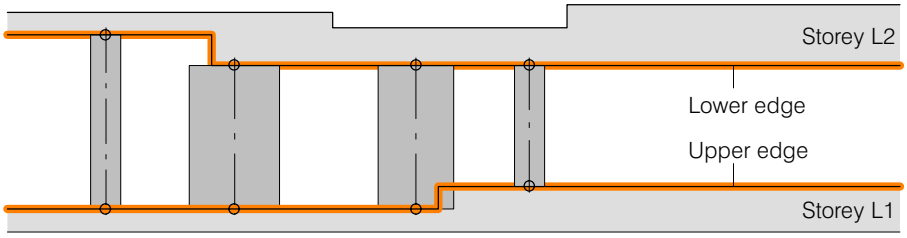
For pillars:

A non-grouped wall pillar must be brought into contact with a minimum length of its axis on a slab to catch it at its level. This minimum length is 1.5 * the thickness of the wall pillar, 0.6 * of its length for short wall segments. For grouped pillars must satisfy this condition at least one of its segments:



Heights, weights and masses of vertical elements

The heights of the vertical elements are determined by the intersections of their axes of gravity with the upper resp. lower edges of the storeys:

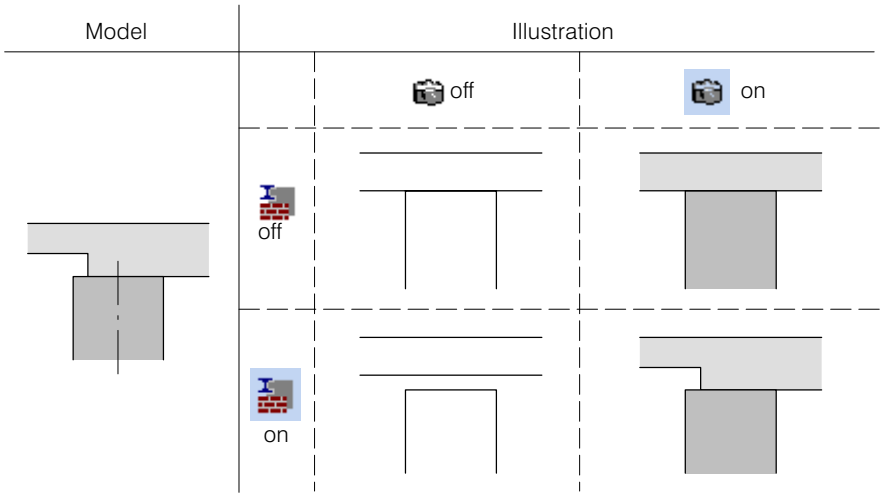


These heights are decisive for the self weights and masses of the wall pillars and columns, which play a role both in the geometrical building model, as well as in the stacked slab model and the space frame model. Piles start at the bottom edge of the basement storey. Their length is determined by the input value h in the columns dialog.

Any openings in wall pillars (see Chap. B 8.10.3) must be within their defined heights. They are taken into account during the weight and mass determination.

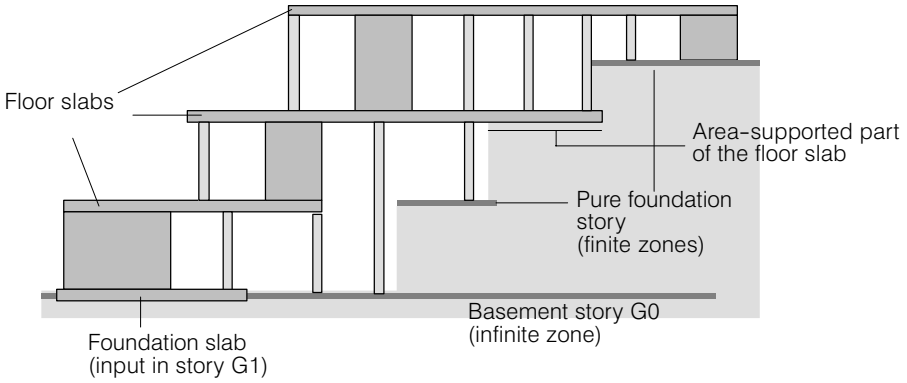
Illustration of vertical elements in the geometry model

The illustration of the heights of the vertical elements in the geometry model depends on the switch positions for rendering and building material presentation:



Nature of ground surface / basement storys

All vertical elements of a floor slab are supported by the underlying story. This may be a normal floor slab (possibly with elastic supports) or it may be a basement story that consists of an area support box and possibly of a finite zone (area support dividing lines). If a supporting story is not found, the program gives a corresponding error message.



The basement story G0 was input here as unlimited (only an area support box). One could also have input it as a finite support zone next to the foundation slab. The higher level basement storys, however, have to be input as finite zones, as otherwise all elements would be supported at the corresponding level.

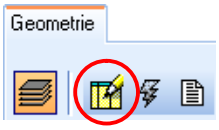
Permitted slab types in a building analysis

In a CEDRUS building analyis, floor slabs are accepted if they fulfil the following conditions in regard to structure input (otherwise when the program performs checks an error message is given):

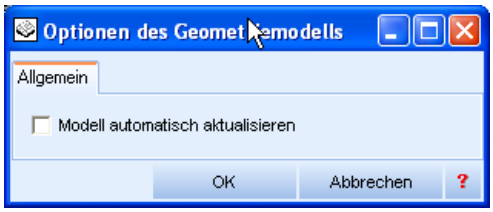
1) Normal analysable slabs, possibly with area support (partial or complete).
2) „Degenerated" slabs with only one area support box. Each additional input (with the exception of the next point) would not be accepted by the program, as long as the slab is not completely defined. Effect: foundation area of unlimited extent.
3) As 2), but with a finite zone. Effect: foundation area of limited extent.

D 3.5.2 Creation and Updating of the Geometric Building Model

The creation of the geometric building model built up of the storys in the Building Definition dialog is automatic or can be controlled by the user. An automatic creation or updating means that after each change and saving of a story and after every change in the story attributes (e.g. level) the building model is automatically updated. Since this requires some time in the case of large projects, normally user-controlled updating is recommended.



The required specifications are made in the dialog, which is opened using the button on the left:



The user-controlled creation or control of a building model is done using the updating (lightning) button. As soon as an existing building model is no longer the current one, the user is informed by means of a dimmed display of the old model and an active updating button. If a model has not yet been created, the building window is empty and the updating button is active.

Graphical documentation of the building

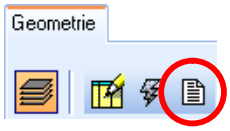
As soon as a valid building model exists, it is displayed in the building window.

The display can be controlled in the usual way. This includes

- The presentation tools of the Graphics Editor on the left side of the program window
- The layer switch of the group 'Storys' for the visibility of the different floor slabs and the walls and columns that support them
- The layer switch of the group 'Geometry' for
- slabs, columns and walls, non load-bearing walls, foundation zones, display of construction element type or material.

With the subsystems, the clipping box and the filter for the name of the structural element, however, there are new and very efficient control elements for the visibility of buildings. These are described in Ch. D 4.

Tabular documentation of the building



With the button shown, a tabular documentation of the entire building can be created. The following figure shows an example in condensed form (with button for 'Compact View' in the table header). In the non-condensed presentation, all slabs, walls and columns are listed individually.

Gebäudegeometrie

Switch for compact display, in which wall segment results are omitted

Geschosse

Geschoss	Beschreibung	Z-Kote [m]	abh. v. Mustergeschoss Geometrie	abh. v. Mustergeschoss Lasten	Bel.Fak. ψ_2	verschoben um dx [m]	dy [m]	Winkel [°]
L4	DG	8.60	-	-	0.300	0	0	0
L3	OG1	5.80	OG	OG	0.300	0	0	0
L2	EG	3.00	-	-	0.300	0	0	0
L1	UG	0	-	-	0.300	0	0	0
L0		-3.00	-	-	0.300	0	0	0

Flächen und Volumen

Geschoss	Bauteil	Baustoff	Höhe [m]	Flächen Platten [m²]	Flächen Wände [m²]	Volumen Platten [m³]	Volumen Wände [m³]	Volumen Stützen [m³]	Volumen Summe [m³]
L4				107.60	40.88	25.82	8.18	0.52	34.52
L3				120.00	51.80	28.80	10.36	0.52	39.68
L2				120.00	95.10	28.80	19.02	0.19	48.01
L1				120.00	151.20	31.20	30.24	0	61.44
L0				0	0	0	0	0	0
				467.60	338.98	114.62	67.80	1.24	183.66

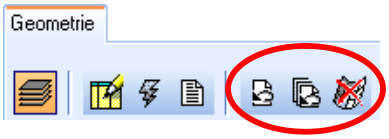
Flächen und Volumen nach Baustoffen

Geschoss	Baustoff	Flächen Platten [m²]	Flächen Wände [m²]	Volumen Platten [m³]	Volumen Wände [m³]	Volumen Stützen [m³]
	C	467.60	306.78	114.62	61.36	
	CC					1.24
	M		32.20		6.44	

Massen

Geschoss	Bauteil	Platten [t]	Wandsegmente Konstruktion [t]	Wandsegmente Zusatz [t]	Stützen Konstruktion [t]	Stützen Zusatz [t]	Summe [t]
L4		64.560	17.886	0	1.313	0	83.759
L3		72.000	20.726	0	1.313	0	94.038
L2		72.000	47.550	0	0.469	0	120.019
L1		78.000	75.600	0	0	0	153.600
L0		0	0	0	0	0	0
		286.560	161.762	0	3.094	0	451.416

Reset



The three following steps are available for resetting the model:

With this button any existing geometrical model is reset and the button to create the model is active again. This button is scarcely of any practical importance, however, because the model. if necessary (i.e. after changes). can be automatically reset by the program itself.

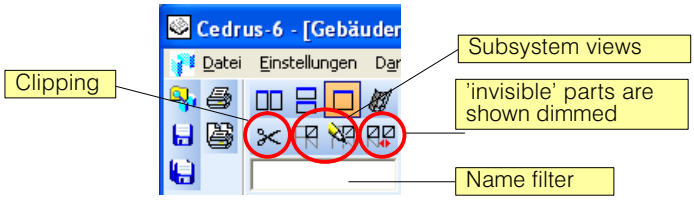
With this button the complete building model is reset. This too, if necessary, is done by the program automatically. After error situations this provides a fresh starting situation.

With this maximum step of resetting, the amount of data of a building analysis is greatly reduced. The program opens each existing slab and resets it, i.e. the FE mesh and any existing results are deleted.

⚠ This function must be used before a building analysis can be sent by Email. It is also useful for maintaining long-term archives.

D 4 Special Ways of Controlling Visibility

Beside the familiar methods of controlling the visibility of individual objects, such as zooming and layer switching, in the building window there are further possibilities for limiting the visibility, that are described in this chapter. These are clipping, subsystems and the name filter.



The user is also referred to the possibilities described in § B 7.2, of fading in the plan views (outlines) of other storys or CEDRUS slabs.

Invisible structural parts dimmed visible

Existing structural elements, which are invisible for the following reasons, can be displayed dimmed. This allows you to see immediately where the visible part lies in the entire structure.

Reasons for invisibility:

- Switched off storey layer
- Active Subsystem
- Active Clippingbox
- Activer Name filter

D 4.1 Subsystems

Subsystems (abbreviated in the following to SS) are one of the means of limiting the visibility of a large structure to selected structural elements. One can define as many SS as one wants. A structural element can belong to more than one SS.

☞ *SS are a means of controlling visibility and have no effect on geometry or any computational model for the building.*

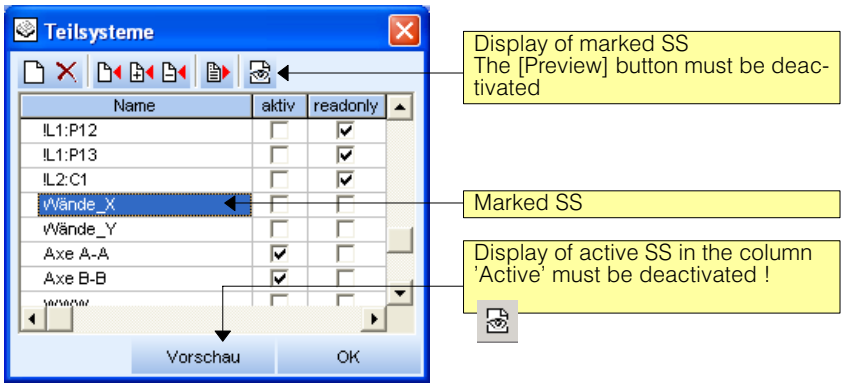
For calling the Subsystems dialog, in which the definition of SS and the selection of the active SS for visibility is made.

With this button one can switch between full view and Subsystem view. The subsystem view limits the visibility to the active Subsystems.

In the Subsystem view, one can show with this button the hidden elements in a dimmed way. This allows one can see immediately where the subsystem is located in the structure as a whole.

D 4.1.1 The Subsystems Dialog

The Subsystems dialog contains a list of the defined subsystems.



The Subsystems dialog includes the following functions, which are only active if the button [Preview] is deactivated:

- Define new subsystem.

- Delete marked SS.

- The current selected objects are assigned to the marked SS. Existing objects in the SS are removed beforehand.

- The currently selected objects are added to the marked SS.

- The currently selected objects are removed from the marked SS.

- The objects of the marked SS are selected (with<Shift>-key this process is additive)

- The visibility is limited to the marked SS. The button is only activated if [Preview] is switched off.

  If a SS is marked and the SS dialog is active, then the marking can be moved up and down with the arrow key, which allows a comfortable 'leafing through' the SS.

It is possible, in the case of changes to the structure, that existing subsystem definitions get muddled up (SS, e.g., can point to elements that no longer exist). Since automatic updating is not possible, the user has to check the SS and if necessary modify them.

D 4.1.2 Steps in Defining a New Subsystem

- Define new SS. The default name can be directly overwritten. The row with the new SS should now be marked.
- Select structural elements that you want to be included in the SS.

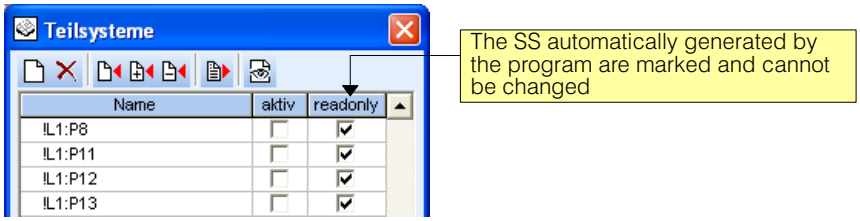
- With this button the selected structural elements now form the SS.

☞ *With* *and* *elements can be added or subtracted from the SS.*

If the SS dialog is closed, with this button the current subsystem view can be switched on or off. The current subsystem view consists of the active SS in the SS dialog.

D 4.1.3 Automatically-generated Subsystems

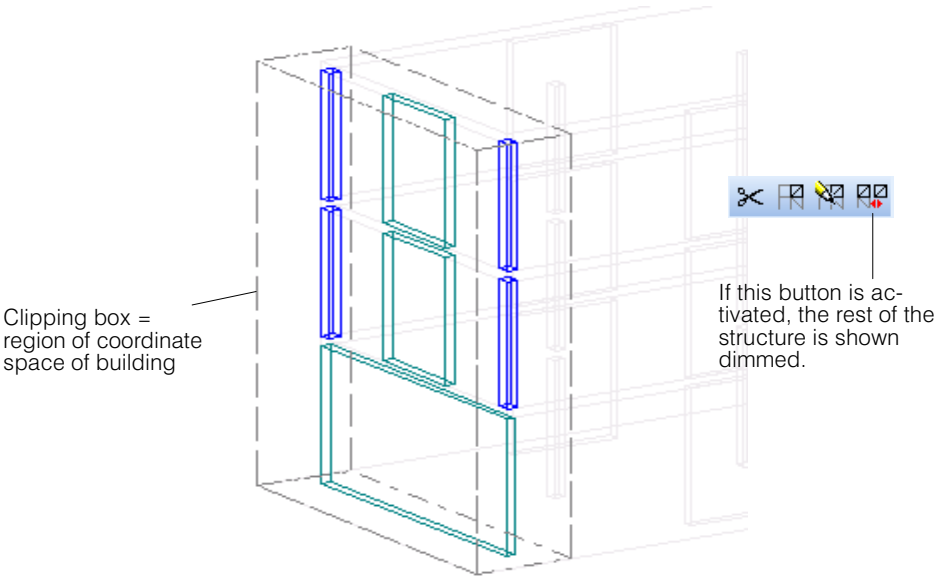
When creating a frame model the program automatically generates subsystems:

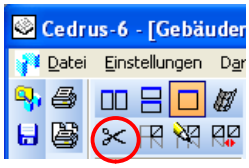


A SS is formed for each separate configuation of wall pillars and columns lying on top of one another, which is very useful for checking.

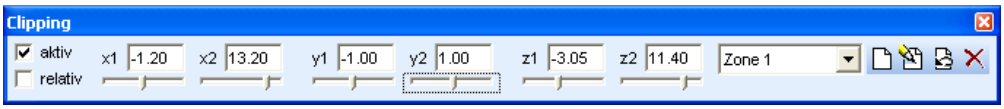
D 4.2 Clipping

Clipping allows the visibility to be reduced to a limited region of the building coordinates i.e like an excerpt).:





The clipping dialog, in which the dimensions of the clipping box can be specified, is opend with this button:



The origin of coordinates of the clipping box is defined in absolute coordinates.

The diagonal to the origin lying at the corner of the clipping box can be defined to be absolute or relative to the origin.

The values can be specified directly in the input fields or using the the slider beneath it. After clicking on a slider it can be operated very simply using the arrow keys.

Using this button the clipping box can be adjusted to the maximum dimensions of the structure.

The view of the clipping box is only limited if this button is activated.

The contents of the clipping box(es) remain saved when closing the Clipping dialog. One can define as many clipping boxes as one wants. For this purpose the following tools are available:

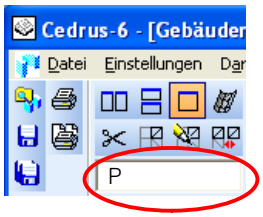
Selection field for the current clipping box.

Create new clipping box.

Change name of the current clipping box.

Delete current clipping box. If only one clipping box remains, it cannot be deleted.

D 4.3 Name Filter



The visibility of the structural objects can also be limited using a name filter. As soon as a character string appears in the input field shown on the left, only those objects are displayed that have this text string in their name.

In the example given, this is all objects that include a 'P', i.e. all wall pillars are shown.

It is also possible to input several character strings separated by a blank. For an object to be shown, at least one of the input character strings must appear in its name.

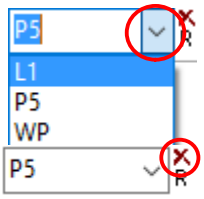
An important case, for example. is when a particular object is sought, e.g. 'L5:P3-1', which otherwise would be difficult to find.

☞ *How the individual objects are identified can be found out by switching on the labelling for the Structure Layer or individually by moving the cursor onto an object with activated 'Preselect' and 'Preselect Hint' in the dialog for the Input options (button in the tools of the Graphics Editor).*

Special functionalities

<Ctrl-F> This key combination is used to jump directly into the filter field.

<Enter> This saves an input in the filter field.



With this button or with <Ctrl-F> the stored entries are displayed for selection.

This empties the input field.

Regular expressions

If the first character in the input field is an '!' , the following text is interpreted as a *regular expression* for the text comparison. Regular expressions obey their own syntax and in IT they are a common way of formulating a character string to be sought or compared. Thus whoever is familiar with this formalism can use it here.

D 5 Stacked Slab Model

D 5.1 General Considerations regarding the Stacked Slab Model

The stacked slab model denotes an analytical model in which the effects of story loads and loads from overlying storys are determined at the bottoms of walls and columns and transferred top-down from one story to the next story:

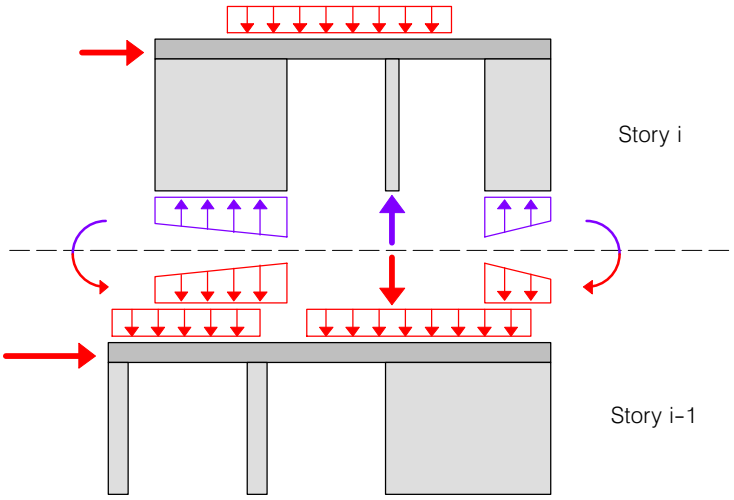
Horizontal loads cannot be treated in Option G of CEDRUS . Therefore, only vertical loads are considered, which are defined for the floor slabs.

The model is extended to horizontal story loads in Option H . These are input in the tabsheet 'Horizontal Loads' of the Building Window.

In the present part of the manual dealing with Option G, only vertical loads are treated. (D 5.3 and E 2.3 for a description of the program tabsheet 'Stacked Slab Model')

D 5.2 Stacked Slab Model with Vertical Loads

In the CEDRUS slab program it is possible to export reactions and weights of a slab's walls and columns and to import these into another slab. In this way vertical loads of a 'virtual' building can be analysed top-down manually (see also § B 10.4). In principle, one can also call this a stacked slab model.



In the building's stacked slab model this procedure has been automated and generalized.

D 5.2.1 Overview of Model

When performing a top-down analysis of the vertical loads according to the stacked slab model the program follows essentially the following steps for each floor slab:

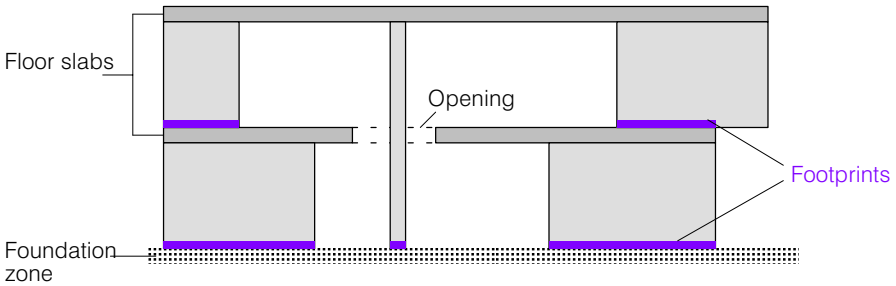
1) From the loadings on a slab (including imported loadings of upper storys) the export combinations Exp_G! for the permanent loads, Exp_Q! for the live loads and gExp_P! for any prestressing loads present are calculated automatically (see also Chapt. B 10.4, Page B-74).

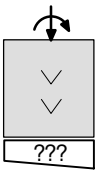
2) The reactions in the column and wall supports are determined for these export combinations. For each vertical element (columns and walls) these reactions can be concentrated in its axis. In general, a normal force N results and two momentes My, Mz.

3) For each vertical element the program determines on which of the underlying floors it rests. Thus the height of the element is known.

   ☞ *Not all walls and columns of a slab have to be supported by the same floor slab (see figure below). This is the first main difference of the Buildings module when working with individual slab analyses.*

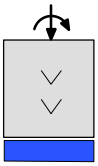
   In the case of walls, not only the level of the bottom of the element is determined, but also the footprint, i.e. over what area the walls are actually supported.





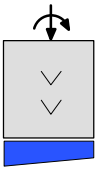
4) From the reactions at the top of an element (from Step 2) all the section forces are determined in the axis at its bottom end, whereby in the case of the export combination for permanent loads the self-weight of the element is also added.

   There are two models that can be selected by the user for determining the forces at the bottom of an element, which then apply to the whole building. As a default, the model 2 is set.



Model 1: forces constant at base
The total vertical base force in a vertical element is distributed over the support area

with a uniform pressure. In buildings having a more or less symmetrical configuration and regularly arranged walls this is often a very satisfactory model. If these conditions are not met, Model 2 is available, in which the shear forces in the vertical elements are also taken into account.



Model 2: forces not constant at base

Here moments and shear forces are taken into account in the vertical elements. In the first step an assumption is made regarding the variation of moments in the initially assumed fixed vertical elements. This gives the shear forces. These section forces at the bottom of the element we denote by S_0. In their totality the shear forces have to be in equilibrium with the total horizontal loadings in the floor. Since here we only consider the vertical forces in the building, the shear forces have to be zero. If this is not the case, the out-of-balance forces are applied to the rigid fixed horizontal floor membrane, no longer considered fixed, and distributed using an equilibrium model to the vertical elements. The section forces at the bottom of the elements resulting from this **compensation step** are now denoted by S_a. Thus the sum of the section forces at the bottoms of the elements are $S = S_0 + S_a$. They, i.e. the normal force and the moments, are distributed over the footprints, such that a linear pressure distribution results.

The base forces determined in Step 3 now have to be transferred to the corresponding floor slabs. This is achieved by means of imported loadings, which are created for the floor slabs involved.

a) Determination of the Reactions N, M for Exp_Q! for all wall pillars in the slab program, reduced to the pillar axis

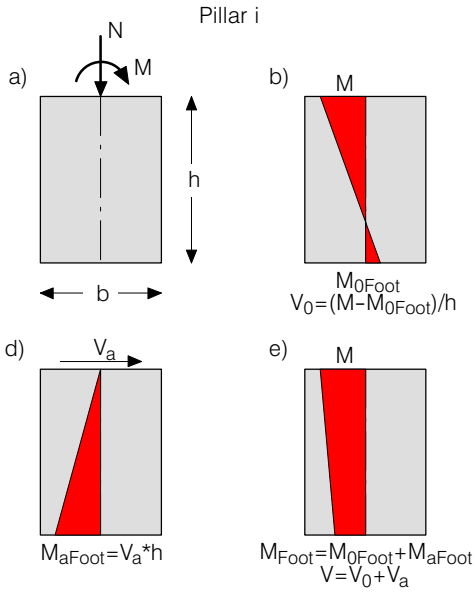
b) Assuming over moment per pillar (horizontally held, depending on b / h)
-> M_{i0Fuss} -> V_{i0}

c) $V_{tot} = \Sigma V_{i0}$
= fictitious storey holding force (unbalance force)

d) „Release horizontal support“, i.e. apply $-V_{tot}$ as a horizontal load on the storey (distribute stiffness on the wall pillar)
-> V_{ia} , M_{iaFuss}

e) Sum results of b) und d)

f) Spread foot forces linearly on footprints



The procedure described above relates to supports which are mounted not rotate freely in the basement slab input in the corresponding direction. The same applies to the transverse direction of wall disks, which are not freely rotatable around the wall axis.

Remarks:

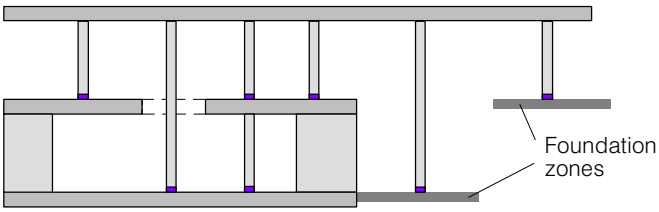
I) If there is no stable wall / column system when introducing the negative storey holding force (point (d) of model 2), the program displays a corresponding error message. Example: only walls in one direction, all freely rotatably mounted.

II) For both models, there are normally transverse forces in the vertical elements, which are not indicated in model 1. These transverse forces must be able to be transmitted from the rigid storey membranes. Corresponding verifications are not supplied by the program.

5) The foot forces determined in step 4 must now be transferred to the corresponding storey. This occurs in the form of import loads, which are generated for the affected storey.

D 5.2.2 Treatment of Columns

For the load transfer every column looks for its base, whereby it is lengthened downwards, till its axis meets a floor slab. It should be remembered that floor slabs may also be just foundation zones (of infinite or finite extent) :



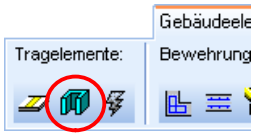
In the case of Model 1 from Chapt. D 5.2.1 it is not important how the columns are supported. In the case of Model 2, however, a column can resist moments and shear forces, if they were not specified as rotationally free in the slab input.

Skew columns

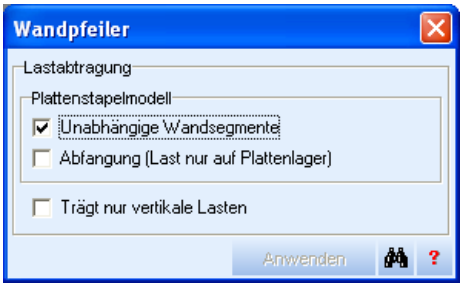
Skew columns (see page B-35) differ from vertical columns in that, in model 2, they transmit a horizontal force dependent on the vertical load and the skewness of the column to the storey membrane, thereby contributing to the fictitious storey holding force V_{tot} (see page D-24 at model 2, point c).

D 5.2.3 Treatment of Wall Pillars

Walls are called wall pillars in the building model. It may be a case of simple shear walls or groups of more than one wall. For their exact definition and input see Chapt. D 3.4.3.



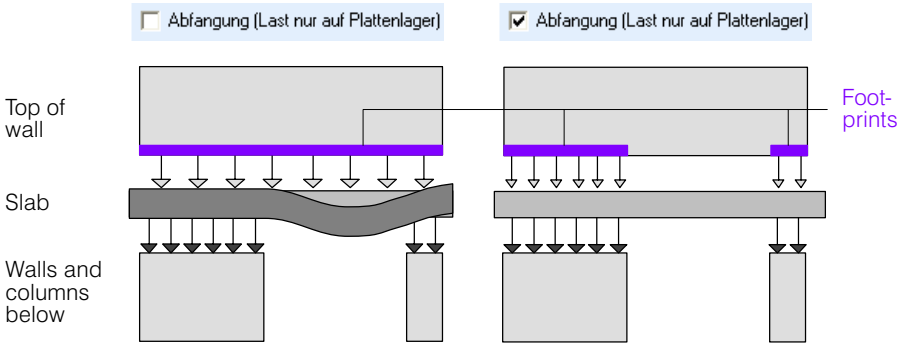
Among other things, the wall pillars possess properties that influence load transfer in the stacked slab model. These properties are summarized in a Properties dialog, which is called in the tabsheet 'Building Elements' of the story slabs using the button shown on the left (see also Chapt. D 3.4.3):



The settings have the following meanings:

Independent wall segments: If activated, every wall segment is treated individully, irrespective of whether it is part of a group of wall pillars or not. Otherwise the wall pillar is treated as a whole. In the case of Model 1, this leads to uniform pressures along the footprint of the whole wall pillar, whereas in the case of Model 2 a linear pressure distribution over the wall pillar is obtained.

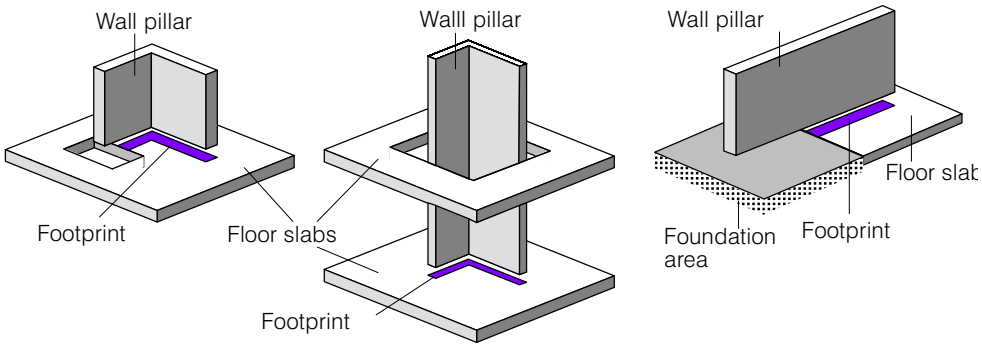
Support: If activated, the footprint is reduced to the parts of the slab which are supported. Thus such a wall pillar only transfers loads where it is directly supported:



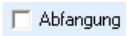
Carries only vertical loads: If activated the wall pillar in Model 2 is treated in the same way as in Model 1.

On the footprint of wall pillars

A wall pillar transfers its loads downwards via its footprint. The footprint always lies on a horizontal plane. This plane lies at the level where the wall pillar - or just a part of it - when extended downwards first reaches a supporting area. Supporting areas are a floor slab and, as special cases, finite or infinitely long foundation zones. The footprint is the area on which the wall pillar actually rests on the supporting area. If, in the same plane, a wall pillar rests both on a floor slab and on a foundation area, then the footprint is restricted to the slab part (the case shown on the right in the following figure). Sometimes such a wall has to be divided into two separate walls.



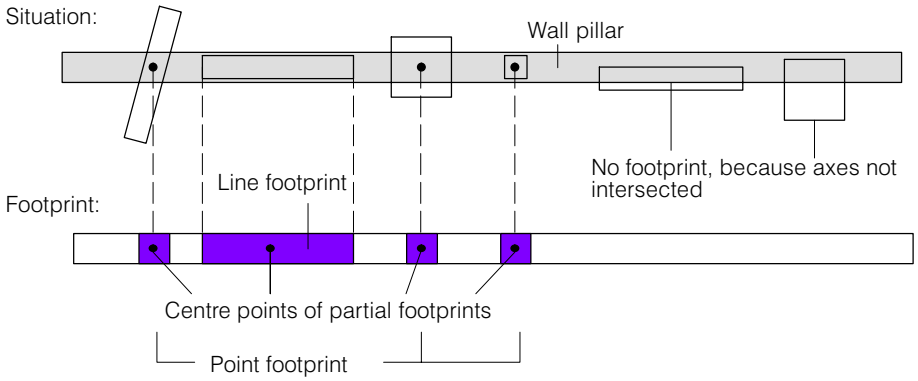
The setting under load transfer in the Wall Pillar dialog also has an influence on the definition of the footprint.



If the load transfer is not restricted to supported parts of the floor slab (standard case), then the footprint is equal to the contact area of the wall pillar with the slab.

☑ Abfangung

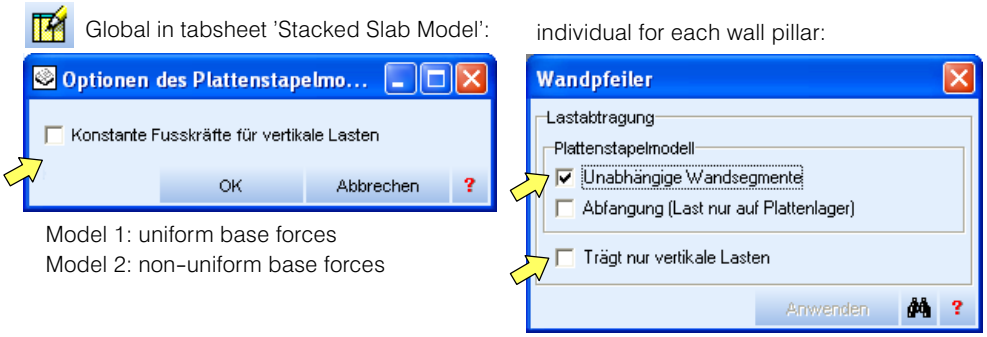
If the load transfer is exclusively through supported parts of the floor slab, the following figure shows how the footprint is determined. It shows a wall pillar (horizontal rectangle), which rests on various walls and columns. Footprints are only given by those walls and columns, whose axes lie on the wall pillar axis or intersect it (tolerance=2mm). In the case of obliquely cut walls and columns a quadratic footprint results:



Load distribution along the footprint

In Paragraph D 5.2.1 it is described how the forces at the bottom of the wall pillar are determined. Here we describe once more, with what distribution the wall pillars transfer their base forces.

As already mentioned, the force transfer is along the footprint of a wall pillar. The results depend on the following settings:



Model 1: uniform base forces
Model 2: non-uniform base forces

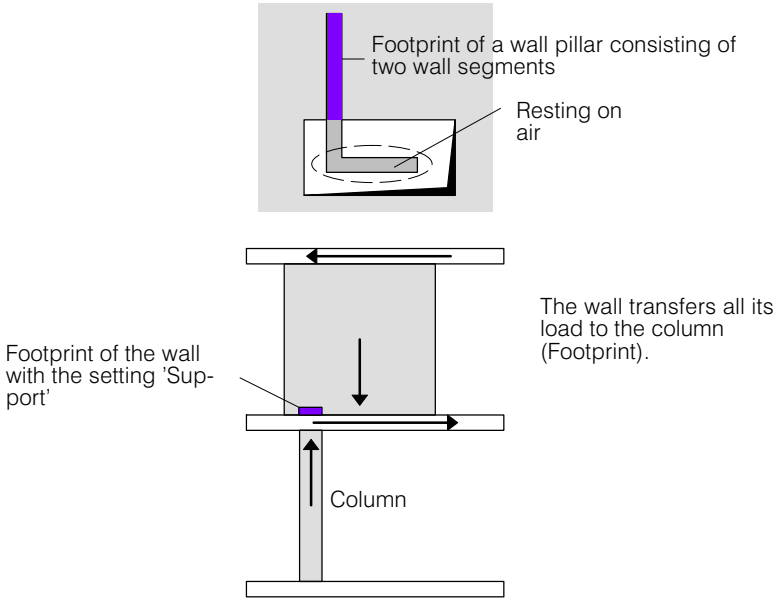
In the case of **Model 1** a uniformly distributed pressure is given under the footprint of a whole wall pillar or under the footprints of the individual wall segments, provided ☑ Unabhängige Wandsegmente is activated in the wall pillar properties (default setting).

In the case of **Model 2** the pressures are linearly distributed along the footprint of a whole wall pillar (or linearly along each of its wall segments with the setting ☑ Unabhängige Wandsegmente) . With the setting ☑ Trägt nur vertikale Lasten the wall pillar is treated as in Model 1.

☞ *If ☑ Unabhängige Wandsegmente is activated, here it doesn't matter therefore if there is a selected grouping of wall segments to wall pillars. However, a grouping can have an influence on which floor slab a wall pillar is supported. The following figure shows an incorrect situation, in which a wall segment that is part of a group forming a wall pillar cannot transfer its base force because it lies wholly over an opening. Without being part of a group this wall segment would find its own support in a lower-lying floor and transfer its force there.*

Point support

If a plane wall pillar (plane wall) with the setting ☑ Abfangung is only supported at one point (by a column or or an intersecting wall) we call this a point support. The following figure gives an example of such a situation:



Equilibrium here is only possible by means of compensating forces in the floor slabs. A prerequisite of such compensating forces in the floor slabs is the existence of sufficient walls in the story.

⚠ Walls with point supports can not carry horizontal forces. They must be set to 'vertical loads only' by the user, otherwise the program displays an error message.

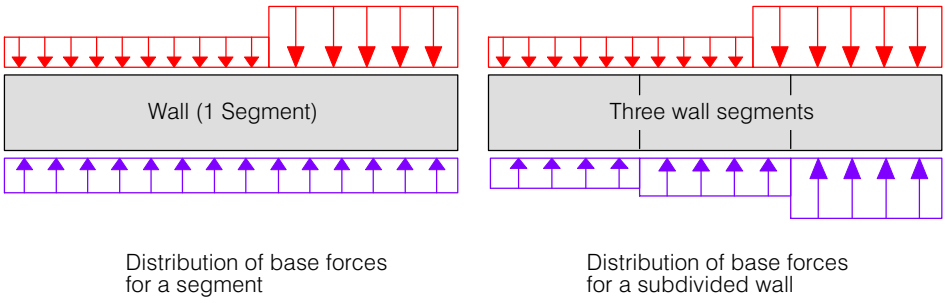
Model 1 (uniformly distributed base forces): Whether the model is admissible or not and sufficient walls are available to enable an adequate state of equilibrium, is left to the user's own judgement. In any case the setting 'Support' cannot be used.

Model 2: The upper wall carries a horizontal force dependent on the vertical load and the ratio of height to length of the wall to the storey membrane, thereby contributing to the fictitious storey holding force V_{tot} (see page D-24).

With the compensation step, by means of out-of-balance forces the upper wall does not resist any shear force (see Page D-25 with Model 2). If there are not enough other walls to take the out-of-balance forces, the program issues a warning message. In the lower story likewise - as in all storys - there must, of course, be a sufficient number of walls to ensure stability with respect to horizontal displacements.

Long sections of plane walls

If, with respect to the height, there are very long sections of plane wall that only consist of one wall segment and do not only in general have a uniform load distribution, the assumption of a uniformly distributed base force in Model 1 may be unsatisfactory. Even with the linear distribution of base force in Model 2 it is not necessarily better. In such cases it is better to subdivide a long wall into several segments:



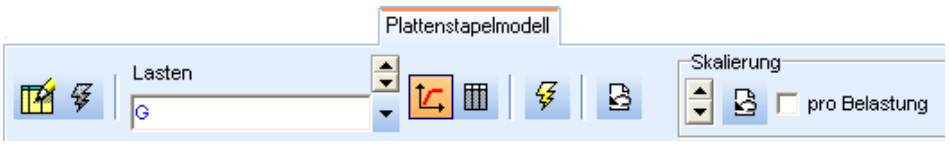
☞ *The Graphics Editor supports a subsequent subdivision of walls as follows:*
1) Selection of the wall

2) Context menu: *'Introduce Intermediate Points'*
(Reverse Function: 'Remove Intermediate Points)

D 5.2.4 Limits of the Load Transfer Model

As with every model, the stacked slab model described here has its limitations. It is the responsibility of the engineer to decide, in which cases further considerations or analyses are necessary.

D 5.3 Tab sheet 'Stacked Slab Model'



The creation and analysis of stacked slab models is carried out in the tab sheet 'Stacked Slab Model' (see also the corresponding § E 2.3 for Option H).

D 5.3.1 Creation of the Model

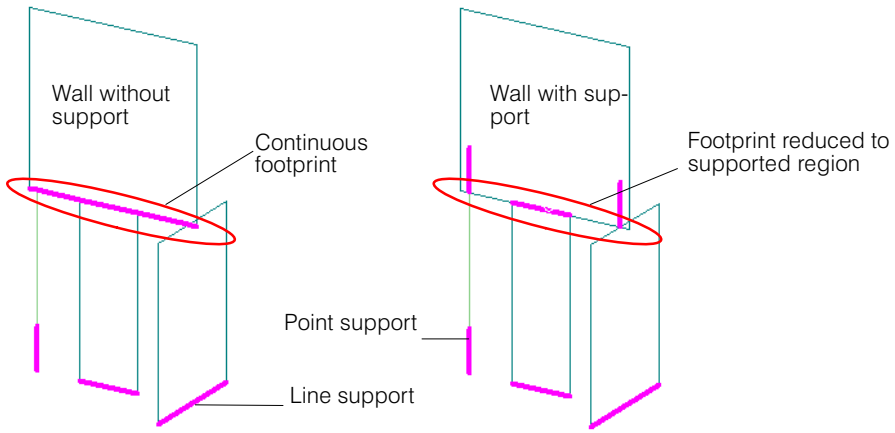
For the only option of the stacked slab model see Page D-28

If all storys are defined, the stacked slab model can be generated using the lightening button shown on the left.

In emergencies the model can be restored to its previous state using .

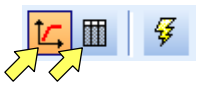
In the generation of the stacked slab model, the geometry of the building is investigated and it is determined how the loads are transferred (see § D 5.2). With the button shown the footprints, i.e. the points and lines where loads are transmitted, can be made visible.

Subsystem view of the building:



D 5.3.2 Starting the Analysis

After the successful creation of the stacked slab model and checking the method of load transfer for the walls and columns, the **analysis of the load transfer in the whole building** can be started with the button shown on the left. Thereby the requirement for the output of results in the individual storys is also fulfilled, because the imported loads have to be known, i.e. the import loadings have to be generated.

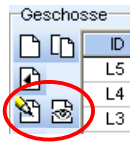


When the analysis has been succcessfully completed, results are immediately shown in the selected form (graphical or tabular) .

After relevant changes to the building, the stacked slab model must be renalysed before the corresponding results are available for the floor slabs.

☞ *The creation of the stacked slab model described in D 5.3 does not have to be done explicitly by the user. This step is automatic in the analysis of the model.*

D 5.3.3 Results in the Individual Floor Slabs



After the analysis of the stacked slab model the import loadings !Imp-G, !Imp-Q and if present !Imp-P are generated in all floor slabs and thus the results in each floor slab contain the loads from the top-down analysis in the stacked slab model. The floor slabs can now be analysed and dimensioned individually, in the same way as in the slab program.

☞ *If, after relevant modifications of the building slab, results are requested, without having first carried out a new analysis with the stacked slab model, the program issues a corresponding warning.*

D 5.3.4 Results in the Columns and Walls

The results in the columns and walls give an overview of the top-down analysed loads in the building. Whereas the slabs are often dimensioned without taking into account the horizontal loads, this is not reasonable in the case of walls. The wall results created with Option G under vertical loads alone serve above all for checking.

How the effects of horizontal building loads can be taken into account in the dimensioning of walls, is described in Part E of the Manual.

Results for ...

By default the following building loadings and combination loadings are created on the basis of the loadings input for the floor slabs, for which results can be obtained here:

- G: sum of all permanent loadings
- Q: sum of all live loadings
- Q0: $\psi_0 * Q$
- Q2: $\psi_2 * Q$
- P: prestressing loadings (if present)
- !GiQ: $\gamma_{G,inf} * G(+P) + \gamma_Q * Q$
- !GsQ: $\gamma_{G,sup} * G(+P) + \gamma_Q * Q$
- !GQ2: $G(+P) + \psi_2 * Q$

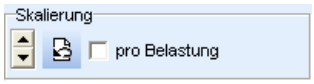
☞ *In the tabsheet 'Analysis' it is possible to define other combination loadings (e.g. G * 1.0 + Q * 0.6). These also appear in the Results selection field.*

Output of results

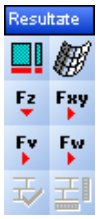
▶ In the Results selection field, first the loading or combination has to be selected, for which one wants to have results (e.g. 'G' for the permanent loads).

Graphically

The exaggerated scale of the diagrams and load arrows in the graphics can be controlled using the following self-explanatory tools:



To display the base forces in walls and columns in the Layer group 'Results' the following layer is provided:



Displaying the footprints of the stacked slab model (see D 5.3).

This Results Layer shows the bending deflections of the slab in the Building view. Normally, however, one would consider the deflections in the individual slabs.

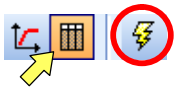
Vertical base forces in wall segments;
Labelling: Nu (= total vertical force)
Pointer on cursor: Nu, Mw(= moment about the major axis of the segment)

The following three layers concern horizontal base forces and are empty in the case of Model 1 (uniformly distributed bottom forces for vertical loads):

Horizontal force (sum of both components) at the base of the vertical elements in vectorial form.

Wall segments: base shear force along the wall segments
Columns: base shear force in the direction X of the slab coordinate system

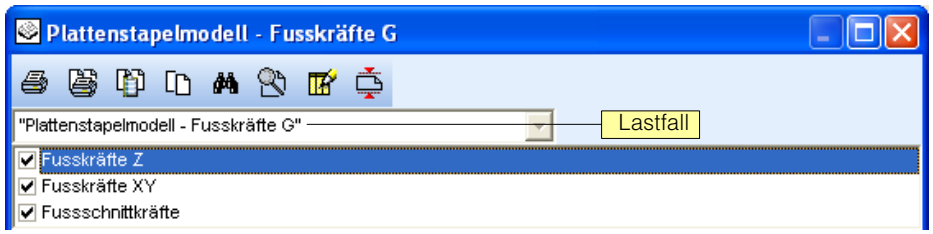
Wall segments: base shear force normal to the wall segments
Columns: base shear force in the direction Y of the slab coordinate system



Tabular

If the button for tabular output is activated, the tabular results are created using the lightening button.

The output is created for the specified load case (in the figure = G) and consists of the three tables listed below:



The table headings look as follows:

Fusskräfte Z

Geschoss	Bauteil	Wandsegmente				Wandpfeiler			Stützen	Fundation	Summe
		Fuss	fz_{min}	fz_{max}	Fz	fz_{min}	fz_{max}	Fz	Fz	Fz	Fz
		[%]	[kN/m]	[kN/m]	[kN]	[kN/m]	[kN/m]	[kN]	[kN]	[kN]	[kN]
L5	C1								173.8		
L5	C2								32.7		
L5	C3								32.7		

The Fz correspond to the quantities described above in the graphics output. Besides the total forces in each wall segment, in the tabular output the force per unit length fz_{min} and fz_{max} are also output (different values in the case of a trapezoidal distribution). The values for the wall pillars are also output, which in the case of grouped wall pillars are different from the segment values.

The following two tables contain above all horizontal base forces, which in the case of Model 1 (uniformly distributed base forces for vertical loads) are zero:

Fusskräfte XY

Geschoss	Bauteil	Wandsegmente		Wandpfeiler		Stützen		Fundation		Summe	
		Fx [kN]	Fy [kN]	Fx [kN]	Fy [kN]	Fx [kN]	Fy [kN]	Fx [kN]	Fy [kN]	Fx [kN]	Fy [kN]
L5	C1					0	0				
L5	C2					0	0				
L5	C3					0	0				

Here the components of the horizontal forces at the base in the global coordinate system of the building are output.

Fussschnittkräfte

Geschoss	Bauteil	Schnittkräfte						
		Nu [kN]	Vv [kN]	Vw [kN]	Mu [kNm]	Mv [kNm]	Mw [kNm]	
L5	C1	173.77	0	0	0	0	0	
L5	C2	32.72	0	0	0	0	0	
L5	C3	32.73	0	0	0	0	0	

The directions u,v,w are defined here as follows:

u: **vertical**

v: **wall segments**: in direction of wall segment
wall pillars (grouped): direction X of the slab coordinate system
columns: direction X of the slab coordinate system

w: **wall segments**: normal to direction of wall segment
wall pillars (grouped): direction Y of the slab coordinate system
columns: direction Y of the slab coordinate system

⚠ The story identification in the tables (1st column) corresponds to the floor slabs to which the listed columns and walls belong. Summarized and provided with summed values, therefore, is the output for the vertical elements of a story (sometimes resting on different floor slabs and foundation zones).

D 6 BIM-Schnittstelle

D 6.1 Einleitung

„Building Information Modeling“ (BIM) ermöglicht den Datenaustausch von dreidimensionalen, digitalen Gebäudedaten zwischen den beteiligten Planungspartnern. CEDRUS verfügt über eine BIM-Schnittstelle, welche es erlaubt, ein dreidimensionales, bauteilorientiertes Modell aus einem CAD-System (z.B. Architekturmodell) einzulesen und daraus ein Berechnungsmodell zu generieren. Mit der aktuellen Version von CEDRUS können 3D-Gebäudemodelle aus BIM-Dateien im IFC- oder Allplan-XML-Format importiert werden.

☞ *Voraussetzung für die BIM-Schnittstelle ist eine gültige Gebäude-Lizenz (Option „G“). Für Allplan-XML ist zusätzlich eine gültige AP Lizenz (Serviceoption) erforderlich.*

BIM-Modell ≠ Berechnungsmodell

Ein importiertes BIM-Modell kann nicht direkt für statische und dynamische Analysen verwendet werden. Dies hat mehrere Gründe:

- BIM-Modelle sind Volumenmodelle, welche die dreidimensionale Geometrie der Bauteile beschreiben. Ein Berechnungsmodell besteht hingegen nur aus aufeinander abgestimmten Flächen und Achsen, welche das statische System abbilden.
- Im BIM-Modell werden alle Bauteile geometrisch möglichst detailliert abgebildet, während im Berechnungsmodell nur die statisch wirksamen Strukturelemente zulässig sind. Diese sollten zudem geometrisch so einfach wie möglich definiert sein, damit die Resultate noch überprüft werden können.
- Kennzeichnend für den überwiegenden Teil der in der Praxis vorkommenden CAD-3D-Modelle ist, dass sie meistens nur in ausgewählter Plandarstellung oder als gerenderte Ansicht überzeugen. Die meisten CAD-Systeme garantieren nämlich keine konsistenten, dreidimensionalen Daten, d.h. sie lassen substanzielle Zeichnungsungenauigkeiten zu (z.B. Überlappungen). Konsistente Daten sind jedoch gerade die Grundvoraussetzung für ein brauchbares Berechnungsmodell.

Um ein vernünftiges Berechnungsmodell zu erhalten, muss die Geometrie der Bauteile aus dem BIM-Modell nachbearbeitet werden.

Das Modul BIMlink

Mit BIMlink verfügt CEDRUS über ein interaktives Programmmodul, welches den Datenaustausch via BIM-Dateien ermöglicht. Wie der Name impliziert, verknüpft BIMlink die Daten einer importierten 3D-BIM-Datei mit dem Berechnungsmodell von CEDRUS. Das Modul extrahiert die relevanten Bauteile aus den 3D-BIM-Daten und erzeugt ein geschossbasiertes Berechnungsmodell, dessen Geometrie nicht exakt dem BIM-Modell entsprechen muss, sondern dank Geometrieanpassungen realistische statische und dynamische Berechnungsresultate liefert. Dank der integrierten Update-Funktionalität kann das Berechnungsmodell bei Änderungen an der ursprünglichen BIM-Datei einfach nachgeführt werden.

Generierung des Berechnungsmodells

BIMlink verfügt über ausgeklügelte Algorithmen, welche die automatische Generierung des Berechungsmodells aus dem BIM-Modell erlauben.

Dabei werden u. a. folgende Anpassungen vorgenommen:

- Sich überlappende oder vollständig überdeckende Bauteile werden verschmolzen
- Übereinanderliegende Decken werden vereinigt und ev. als Deckenabsatz modelliert
- Wandoberkanten werden an Deckenflächen angepasst
- Flächen sich berührender Wände werden aufeinander abgestimmt
- Stützenachsen werden an Deckenflächen angepasst
- Wände und Stützen, welche auf keiner Platte stehen, werden an ihrem Fuss mit einem Fundament versehen
- Nichttragende Strukturelemente werden als Lasten modelliert

Durch diese automatischen Anpassungen kann der Nachbearbeitungsaufwand in der Regel erheblich gesenkt oder sogar gänzlich eliminiert werden. Trotzdem liegt es in der Verantwortung des Ingenieurs, das erzeugte statische Modell zu überprüfen und gegebenenfalls anzupassen.

D 6.1.1 Beschreibung der BIM-Schnittstelle

Die in CEDRUS implementierte BIM-Schnittstelle wird in diesem Handbuch an mehreren Orten beschrieben. In diesem Kapitel finden Sie neben einer allgemeinen Einführung in die Thematik BIM aus Sicht der Tragwerksberechnungen eine detaillierte Beschreibung des Moduls BIMlink. Im Kapitel LEERER MERKER finden Sie drei Einführungsbeispiele, welche Sie Schritt für Schritt durch typische Anwendungsszenarien führt.

D 6.2 Einsatz von CEDRUS in der BIM-basierten Planung

In der BIM-basierten Planung gibt es kein standardisiertes Verfahren zur Einbindung der Tragwerksplanung. Der genaue Einsatz von BIM hängt von den Prozessen im Planungsteam aber auch von den individuellen Abläufen innerhalb des Ingenieurbüros ab. CEDRUS trägt diesem Umstand Rechnung indem es dem Ingenieur ein flexibles Werkzeug zur Verfügung stellt, welches sich dem aktuellen Planungsprozess anpassen kann. Mit BIMlink kann der Ingenieur aus 3D-Modelldaten unterschiedlicher Qualität effizient ein Berechnungsmodell aufbauen, welches realistische Resultate liefert.

D 6.2.1 Extrahieren des Berechnungsmodelles

Der Umfang der aus dem BIM-Modell extrahierten Daten kann in BIMlink sehr pragmatisch der Qualität des BIM-Modelles und den Bedürfnissen der Planungsphase angepasst werden. Dies wird hier anhand von 3 denkbaren Szenarien illustriert:

Szenario 1: Architektur BIM-Modell direkt in CEDRUS einlesen

So ist es z.B. möglich und sinnvoll in der Vorprojektphase direkt das BIM-Modell des Architekten mit BIMlink einzulesen. Dies ist z.B. dann der Fall, wenn der Ingenieur möglichst schnell die Beanspruchung des untersten Geschosses ermitteln will, um die Fundation abzuklären. Denn obwohl das BIM-Architekturmodell in dieser Phase oft nur visuell überzeugt und viele fehlende Details und geometrische Inkonsistenzen aufweist, kann dank BIMlink trotzdem in relativ kurzer Zeit ein Berechnungsmodell generiert werden, welches die Fundationskräfte mit genügender Genauigkeit ermitteln kann. Da die Generierung der CEDRUS-Strukturelemente i.d.R. mit diesen Daten nicht gelingt, sollte man in BIMlink nur die Hintergrundpläne aus horizontalen Schnitten durch das 3D-Modell erstellen lassen. Die Struktureingabe der Geschosse kann daraus in CEDRUS recht effizient von Hand gemacht werden.

☞ *Bei BIM-Architekturmodellen empfiehlt es sich mit BIMlink die Geschossstruktur und nur die entsprechenden Hintergrundpläne ohne die CEDRUS-Strukturelemente zu generieren.*

Szenario 2: BIM-'Schalungsmodell' in CEDRUS einlesen

In einer späteren Projektierungsphase ist es z.B. möglich und sinnvoll, dass im Ingenieurbüro ein eigenes 3D-BIM-Modell konstruiert wird, welches u.a. für die Schalungsplanung verwendet wird. Dieses Model ist i.d.R. recht konsistent. CEDRUS kann aus diesen BIM-Daten einen grossen Teil des Berechnungsmodells automatisch generieren. Falls bei einem Geschoss die Generierung der CEDRUS-Strukturelemente scheitert, so erzeugt BIMlink automatisch den Hintergrundplan aus horizontalen Schnitten durch das 3D-Modell. Bei einigen Geschossen muss der Benutzer noch von Hand eingreifen, damit ein sinnvolles Berechnungsmodell entsteht.

☞ *Bei BIM-Schalungsmodellen können mit BIMlink die Geschossstruktur und die CEDRUS-Strukturelemente generiert werden.*

Szenario 3: BIM-'Berechnungsmodell' in CEDRUS einlesen

Natürlich ist es auch möglich, mit einem CAD-System ein BIM-Modell selbst aufzubauen, welches nicht der effektiven (Schalungs-)Geometrie entspricht, sondern die notwendigen, geometrischen Vereinfachungen des Berechnungsmodelles bereits vorwegnimmt. Mit solch einem Modell kann CEDRUS mit nur wenigen zusätzlichen Benutzereingaben (z.B. Lasten) ein Berechnungsmodell generieren. Dies entspricht aber nur der Verlagerung der Struktureingaben von CEDRUS ins CAD-System und bietet sonst keinen Mehrwert, da dieses BIM-'Tragmodell' nicht auch für die Schalungsplanung taugt. Dies ist i.d.R. nur selten effizient, da der Benutzer sowohl CEDRUS wie das CAD-System gut beherrschen muss und der Nachführungsaufwand steigt, will man die beiden Modelle immer synchron halten.

☞ *Die Konstruktion eines BIM-Modelles mit der vereinfachten Geometrie des Berechnungsmodelles lohnt sich i.d.R. nicht.*

D 6.2.2 Nachführung des Berechnungsmodelles (Update)

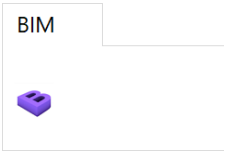
Im Laufe der Planung werden die 3D-BIM-Daten immer wieder verändert. Um diese Modelländerungen ins Berechnungsmodell übernehmen zu können, bietet CEDRUS eine Update-Funktion an. Dabei liest BIMlink die neue Version der BIM-Datei ein, vergleicht diese mit dem ursprünglichen Modell und stellt die Änderungen geschossweise grafisch dar. Der Benutzer muss nun entscheiden, ob die Änderungen für das Berechnungsmodell relevant sind oder nicht. Ist dies der Fall, so kann das Berechnungsmodell geschossweise auf zwei Arten nachgeführt werden:

- CEDRUS-Geschosseingaben automatisch nachführen:
  Dies ist dann sinnvoll, wenn nach der Generierung der Strukturdaten aufgrund des ursprünglichen BIM-Modelles nur wenig bis keine manuellen Korrekturen notwendig waren, um ein realistisches Berechnungsmodell zu erhalten.
- CEDRUS-Geschosseingaben manuell nachführen:
  Dies ist dann sinnvoll, wenn das neue BIM-Model sich nur wenig vom Original unterscheidet oder die automatische Generierung bereits beim ursprünglichen BIM-Modell mit sehr viel nachträglichen Korrekturen verbunden war. Bei der manuellen Nachführung sind die veränderten BIM-Elemente im Hintergrundplan grafisch hervorgehoben, was die Korrekturen sehr vereinfacht.

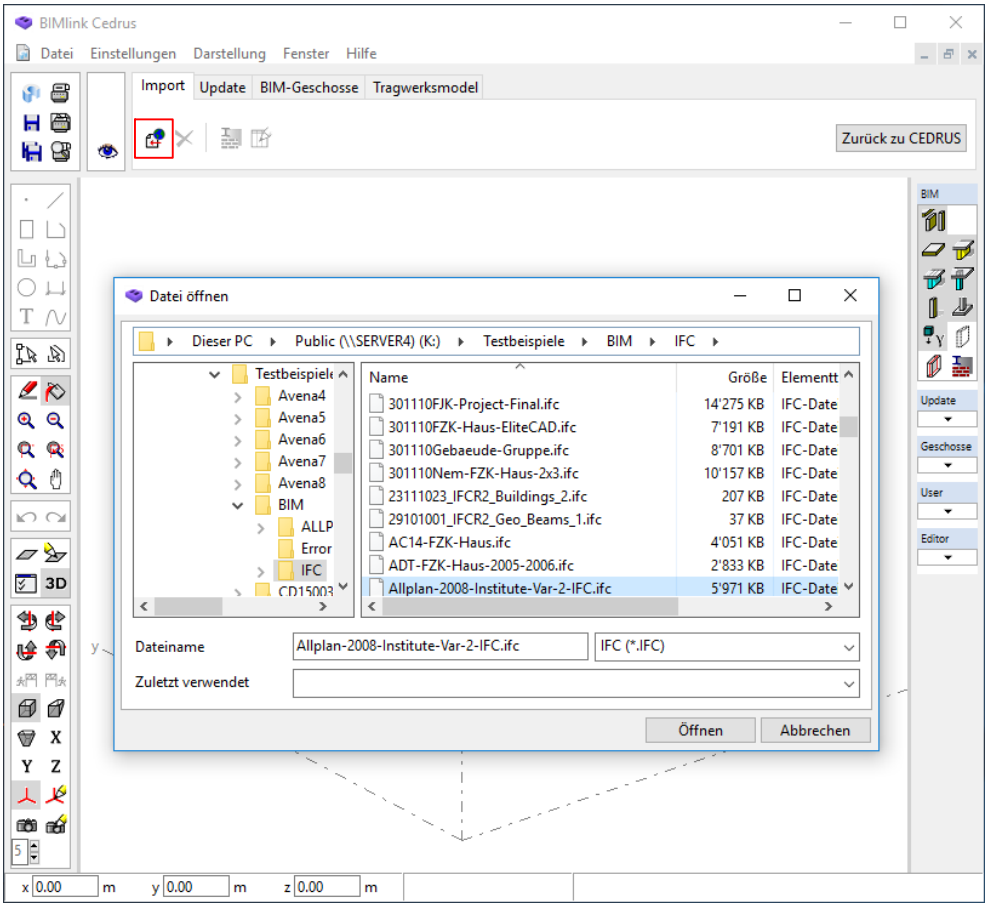
Welche der beiden Arten zum Zug kommt, kann vom Benutzer geschossweise definiert werden.

D 6.3 BIMlink

Die Integration der Cubus-Programme in den BIM-basierten, digitalen Planungsprozess erfolgt über das BIMlink-Fenster. Dieses lässt sich im Register 'BIM' einer Gebäudeberechnung mittels der Schaltfläche 'BIMlink starten' öffnen (s. Bild unten).

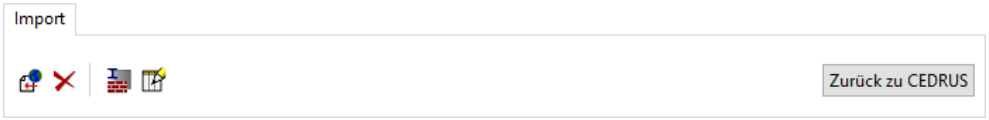


Das BIMlink-Fenster ist analog zum Hauptfenster des Cubus-Programmes ein Grafikeditor, welcher eine Zeichenfläche, ein Menu, ein Steuerregister, Zeichnungswerkzeuge und Layerschalter aufweist. BIMlink arbeitet in der 3D-Ansicht und verwaltet das 3D-BIM-Modell, welches von einer BIM-Datei eingelesen wird (s. Bild unten).



D 6.4 Das Register 'Import'

In diesem Register werden die 3D-BIM-Daten aus einer Datei im IFC- oder ALLPLAN-XML-Format eingelesen.



Mit der Schaltfläche 'Import' kann eine BIM-Datei eingelesen werden. Bereits vorhandene Daten werden dabei gelöscht.

Mit der Schaltfläche 'Zurücksetzen' werden alle vorhandenen BIM-Daten gelöscht.

In einer BIM-Datei sind die Materialdaten nur als Namen gespeichert. Für die Tragwerksberechnung müssen diesen Namen Baustoffe zugewiesen werden, was im Dialog 'Zuweisung der Baustoffe' erfolgt. Dieser wird mit nebenstehender Schaltfläche geöffnet.

Mit dieser Schaltfläche wird der Dialog 'Einstellungen Elementfilter' geöffnet (s. nächster Abschnitt).

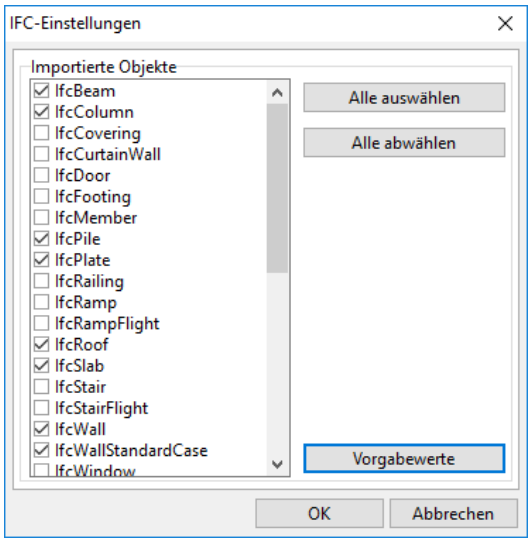
☞ *Zum Einlesen von BIM-Daten im ALLPLAN-XML-Format wird eine gültige Lizenz der Service-Option 'AP' (ALLPLAN) benötigt.*

D 6.4.1 Einstellungen IFC-Schnittstelle

Mit BIMlink können IFC-Dateien der folgenden Versionen eingelesen werden:

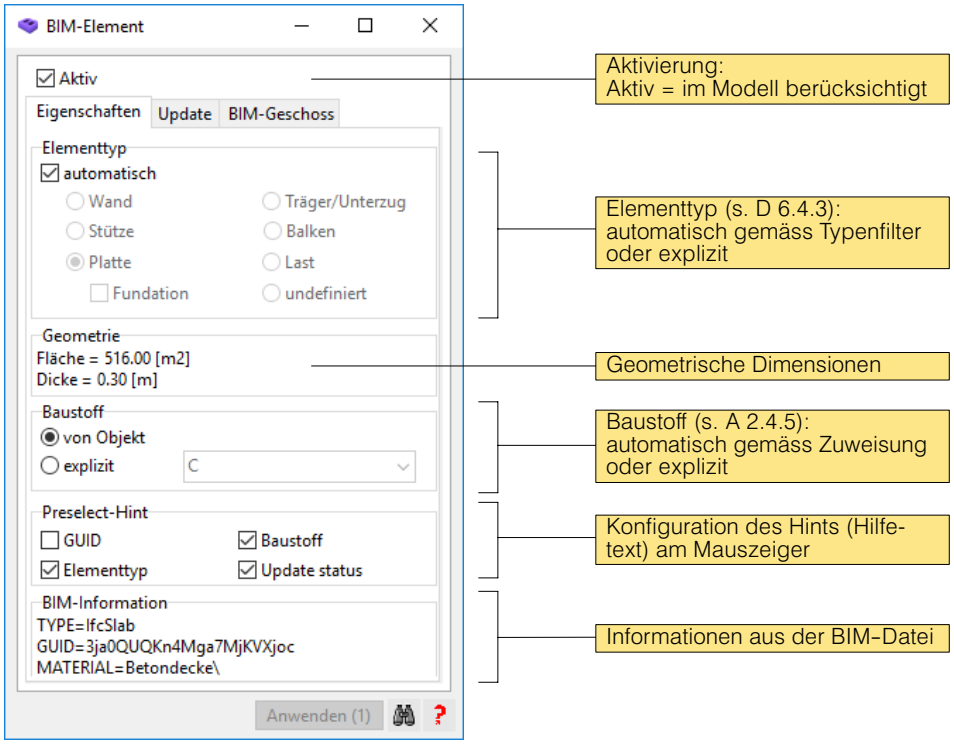
- IFC Version 2x3
- IFC Version 4

Die BIM-Datei eines Gebäudes im IFC-Format kann sehr viele geometrische Objekte enthalten, welche für die Tragwerksplanung irrelevant sind (z.B. Möblierung, technische Installationen). BIMlink liest aus diesem Grund nur einen Teil der 3D-Daten ein. Welche IFC-Datenobjekte eingelesen werden, lässt sich im Dialog 'IFC-Schnittstelle' einstellen, welcher über „Einstellungen“ > „IFC-Schnittstelle“ geöffnet wird. Diese Vorgabe-Einstellungen sind in der Regel genügend und sollten nur in Spezialfällen verändert werden.



D 6.4.2 BIM-Element

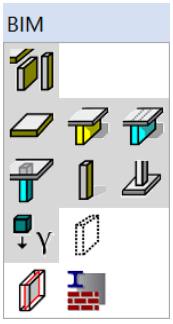
Beim Import wird jedes erkannte 3D-Objekt einzeln als ein sogenanntes BIM-Element in CEDRUS eingefügt. BIM-Elemente sind normale Grafikobjekte mit Attributen und können gelöscht oder bearbeitet werden. Die Werte der Attribute werden beim Import aus den Daten des BIM-Modells entnommen und können danach im Eigenschaftendialog verändert werden.



D 6.4.3 Elementtyp

Das wichtigste Attribut eines BIM-Elements ist der sogenannte Elementtyp. Er bestimmt die Modellierung im Berechnungsmodell. Dabei stehen folgende Elementtypen zur Auswahl:

Wand:	Modellierung als tragende, vertikale Wand
Stütze:	Modellierung als vertikale Stütze
Platte:	Modellierung als horizontale Platte (evtl. mit Absätzen)
Träger/Unterzug:	Modellierung als Unter-/Überzug einer Platte
Fundation:	Modellierung als Bodenplatte (mit Flächenlagerung)
Last:	Modellierung als zusätzliche Eigenlast auf die unten liegende Platte
undefiniert:	wird nicht ins Berechnungsmodell aufgenommen

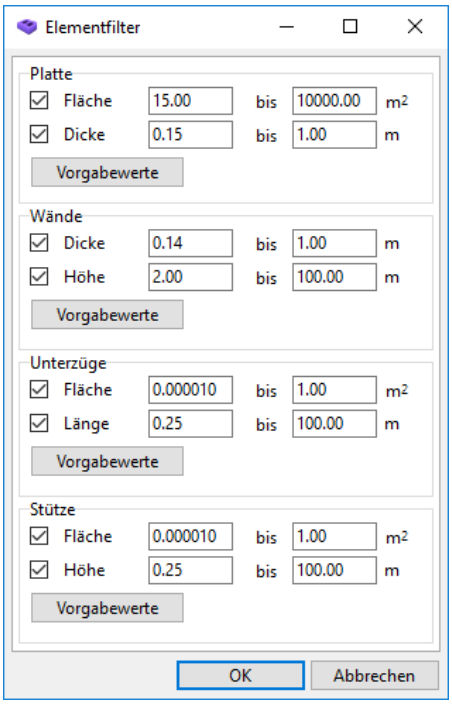


Mit der Standardeinstellung 'automatisch' wird der Elementtyp direkt aus den geometrischen 3D-BIM-Daten ermittelt, wobei nicht zuordnungsfähige Objekte den Elementtyp 'undefiniert' erhalten.

Um die importierten Daten überblicken und kontrollieren zu können, wird jeder Elementtyp in einer anderen Farbe gezeichnet. Dabei werden die gleichen Farbwerte wie im Gebäudemodell verwendet. Zudem kann die Sichtbarkeit der BIM-Elemente pro Elementtyp einzeln gesteuert werden, indem die entsprechende Layerschaltfläche ein- bzw. ausgeschaltet wird.

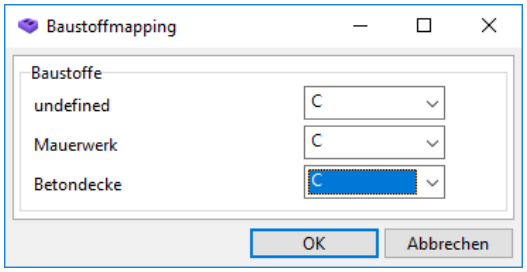
D 6.4.4 Elementfilter

Die für die Tragwerksberechnung relevanten BIM-Elemente werden aus der Masse der Eingelesenen herausgefiltert und einem Elementtyp zugewiesen. Nur geometrische Objekte, welche den Filtereinstellungen entsprechen, werden eingelesen. Damit können viele ungültige oder unnötige Objekte aus den 3D-BIM-Daten herausgefiltert werden. Mit nebenstehender Schaltfläche im Register 'Import' kann der Dialog zur Einstellung der Elementfilter geöffnet werden.



D 6.4.5 Zuweisung der Baustoffe

In einer BIM-Datei sind die Materialdaten als Namen gespeichert. Um eine Tragwerksberechnung zu ermöglichen, muss deshalb jedem Materialnamen ein CEDRUS-Baustoff zugewiesen werden. Dies erfolgt im Dialog 'Zuweisung der Baustoffe'. Dieser wird mittels der nebenstehenden Schaltfläche im Register 'Import' geöffnet.



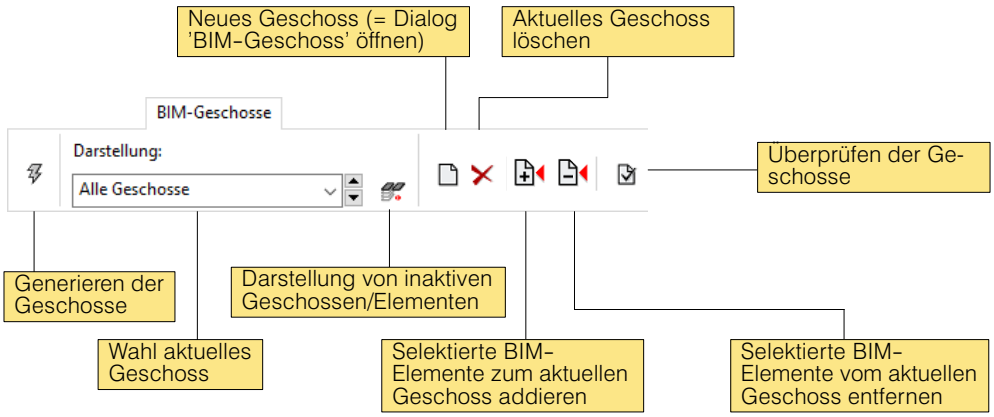
Im Dialog stehen links die vorgefundenen Materialnamen aus der BIM-Datei und rechts die entsprechenden Baustoffe für das Berechnungsmodell.

☞ *Es können nur Baustoffe zugewiesen werden, welche im Gebäude unter „Einstellungen" > „Baustoffe" aufgelistet sind.*

☞ *Der Baustoff lässt sich zudem im Eigenschaftsdialog für jedes BIM-Element einzeln verändern.*

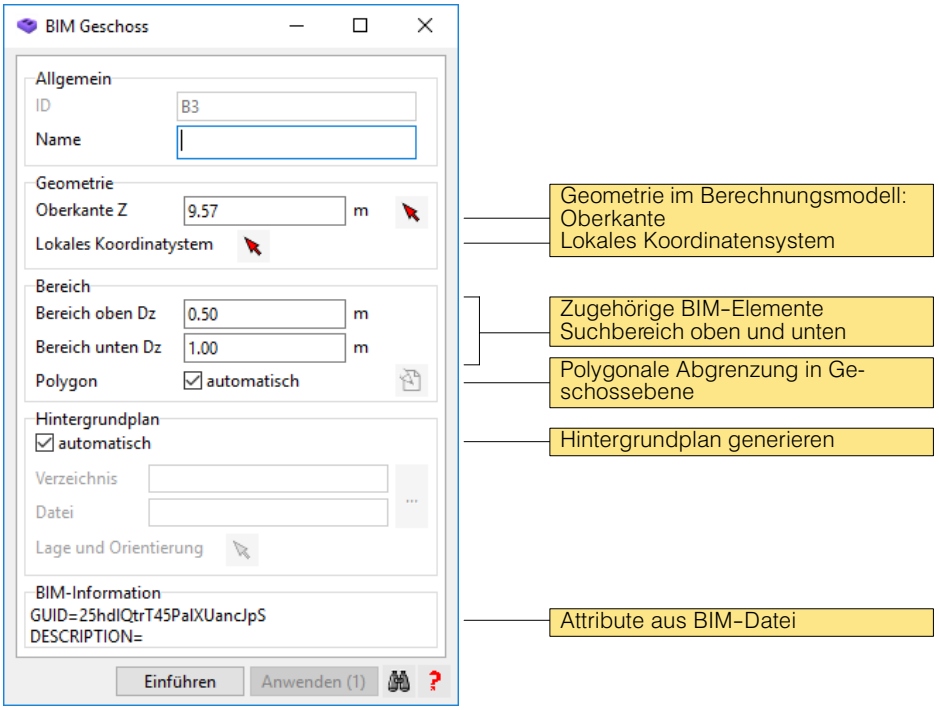
D 6.5 Das Register 'BIM-Geschosse'

In diesem Register geht es darum, die Geschossstruktur festzulegen, welche für das Berechnungsmodell verwendet werden soll.



D 6.5.1 Attributdialog 'BIM-Geschoss'

Jedes Geschoss ist ein Grafikobjekt mit ein eindeutiger ID ('Bx') und wird in der Grafikfläche als ebenes Polygon dargestellt wird, welches den Bereich der zugehörigen Elemente umschliesst. Alle Geschosse können mittels der -Schaltfläche im Register automatisch aus den 3D-Daten generiert oder von Hand mit dem Eigenschaftendialog 'BIM-Geschoss' eingeführt werden. Dieser sieht wie folgt aus.



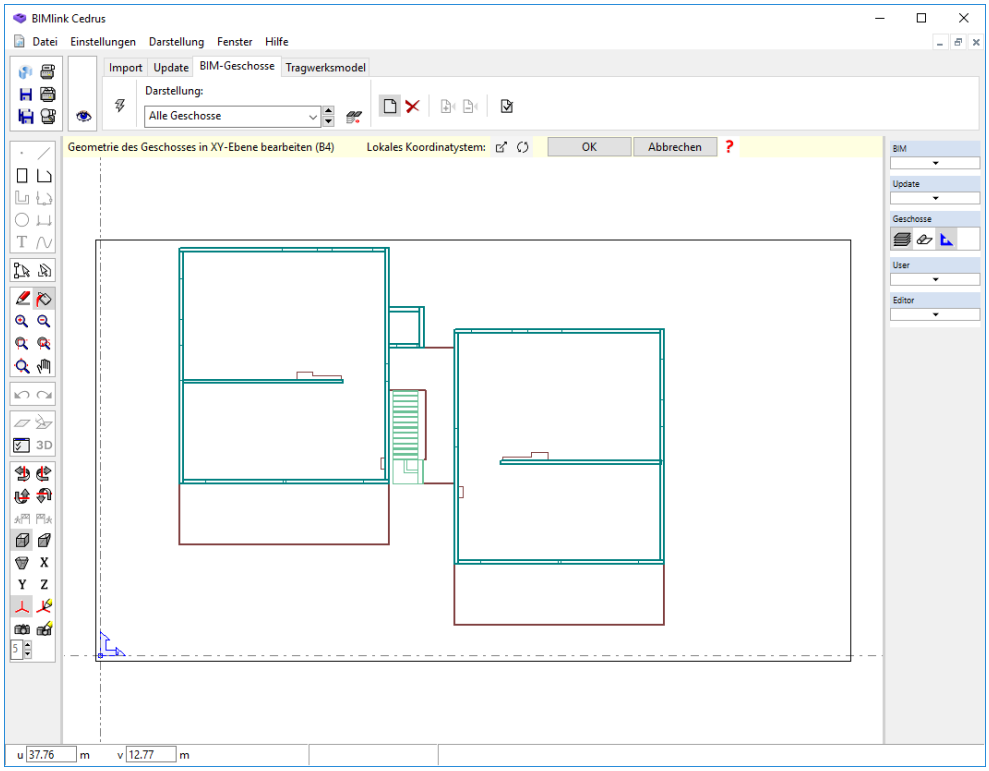
Geometrie im Berechnungsmodell

Mit der Oberkante Z wird die Geschossebene definiert, welche im Berechnungsmodell der Oberkante der CEDRUS-Geschossplatte entspricht. Die ID des Geschosses entspricht der Rangordnung aller Geschosse nach Z sortiert (höchste Zahl = zuoberst). Das lokale Koordinatensystem des Geschosses weist standardmässig die gleiche Orientierung wie das (globale) Modell des BIM-Gebäudes auf. Soll ein anderes Koordinatensystem verwendet werden, so kann diese hier eingestellt werden.

Zugehörige BIM-Elemente

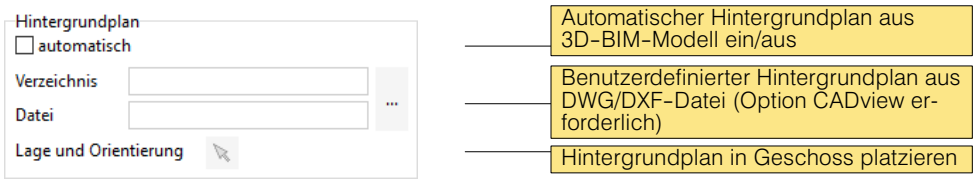
Zu jedem CEDRUS-Geschoss gehört eine Platte mit Ober- und Unterzügen und die untenstehenden Wände und Stützen. Diese Bauteile werden aufgrund der Angabe des Suchbereichs Dz oben und unten aus den vorhandenen BIM-Elementen extrahiert. Dabei sucht BIMlink zuerst alle Platten im angegebenen Bereich zusammen und findet anschliessend alle diese berührenden Stützen und Wände.

Sind mehrere Geschosse mit gleicher Z-Kote vorhanden, so müssen auch mehrere BIM-Geschosse eingegeben werden: Der polygonale Geschossbereiches jedes der entsprechenden Geschosse kann über die Definition eines Polygons eingegrenzt werden. Dieses erfolgt durch die Deaktivierung von 'automatisch' für 'Polygon' und Drücken der Schaltfläche . Die Ansicht wechselt dann in die 2D-Geschoss-Ebene, wo das Polygon mit den Mitteln des Grafikeditors eingegeben wird.



Hintergrundplan

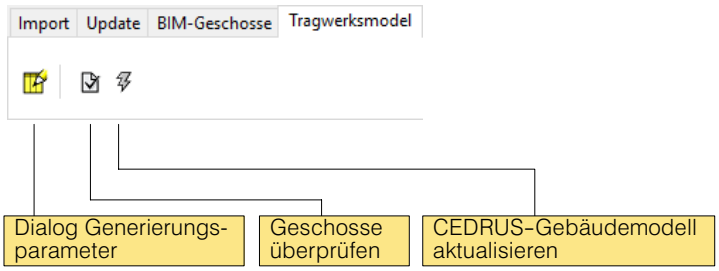
Ist ein BIM-Modell vorhanden, so wird mittels eines horizontalen Schnitts durch diese 3D-Daten für jedes BIM-Geschoss automatisch ein Hintergrundplan generiert. Dieser kann durch den Layer aus der Layergruppe 'Geschosse' in BIMlink dargestellt werden und steht später auch in der CEDRUS-Platteneingabe zur Verfügung.



Ist kein 3D-BIM-Modell vorhanden, so kann als Hintergrundplan eine DXWG/DXF-Datei eingelesen werden. Dies erfolgt im BIM-Geschossdialog durch Deaktivieren der Schaltfläche 'automatisch' bei 'Hintergrundplan' und Drücken der Schaltfläche ..., womit sich der Dialog 'CADview' öffnet (s. C 2.3).

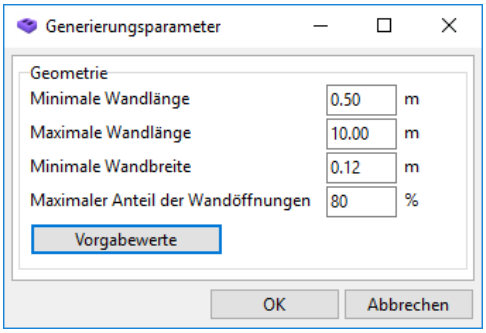
D 6.6 Das Register 'Tragwerksmodell'

Im Register 'Tragwerksmodell' kann spezifiziert werden, welche BIM-Geschosse ins Tragwerksmodell von CEDRUS einfliessen und welche Daten pro Geschoss für CEDRUS generiert werden sollen.



D 6.6.1 Dialog Generierungsparameter

Um ein sinnvolles Berechnungsmodell aus den BIM-Daten zu generieren, werden ein paar Parameter benötigt. Diese Generierungsparameter können im gleichnamigen Dialog eingestellt werden, welcher über die Schaltfläche geöffnet werden kann.



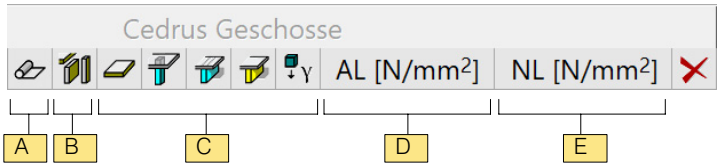
Minimale Wandlänge:
Kürzere Wände werden nicht ins Berechnungsmodell aufgenommen oder, sofern freistehend, als Stütze modelliert.

Maximale Wandlänge:
Längere Wände werden automatisch unterteilt.

Minimale Wandbreite:
Dünnere Wände werden nicht ins Berechnungsmodell aufgenommen.

Maximaler Anteil der Wandöffnungen:
Wandabschnitte mit höherem Anteil an Öffnungen werden als Last modelliert.

D 6.6.2 Tabelle Geschosse



A: 2D-Hintergrundplan der BIM-Daten (horizontaler Schnitt durch 3D-BIM-Modell)

B: Alle Elemente als CEDRUS-Struktureingaben

C: Element individuell als CEDRUS-Struktureingaben

D: Auflast für ganze Geschossplatte

E: Nutzlast für ganze Geschossplatte

D 6.7 Das Register 'Update'

In diesem Register kann eine neue Version des vorgängig importierten 3D-BIM-Modelles aus einer Datei im IFC- oder ALLPLAN-XML-Format eingelesen werden. Dabei liest BIMlink die neuen Daten ein, vergleicht diese mit dem ursprünglichen Modell und stellt die Änderungen geschossweise grafisch dar.

Mit der Schaltfläche 'Import' kann eine neue Version der BIM-Datei eingelesen werden. Bereits vorhandene Update-Daten werden dabei gelöscht. Nach dem Import werden die Änderungen an den Daten grafisch dargestellt.

Mit der Schaltfläche 'Updateverlauf' öffnet sich der gleichnamige Dialog, wo alle bisherigen Updates mit Datum aufgelistet sind.

Mit der Schaltfläche 'Update ablehnen' werden die aktuellen Updatedaten gelöscht.

Mit der Schaltfläche 'Updatecheck' wird überprüft, ob die Änderungen für alle BIM-Elemente akzeptiert wurden. Falls nicht, werden diejenigen Elemente markiert, bei denen dies noch nicht gemacht wurde.

Mit der Schaltfläche 'Update akzeptieren' werden die Änderungen aus den Update-Daten ins aktuelle BIM-Modell übernommen. Dies erfolgt entsprechend den Einstellungen für jedes einzelne BIM-Element (s. D 6.7.2). **HINWEIS:** Dieser Schritt kann nicht mehr rückgängig gemacht werden.

☞ *Der Ablauf eines Updates wird in Kapitel A 2.5.3 anhand eines Beispiels näher erläutert.*

D 6.7.1 Layergruppe Update

Neu

Gelöscht

Verschoben (bei unveränderten Dimensionen)

Veränderte Geometrie (z.B. Dimensionen, Öffnungen)

Übrige Eigenschaften

D 6.7.2 Dialog BIM-Element: Update

Im Register 'Update' des Dialogs BIM-Element (s. unten) kann pro BIM-Element eingestellt werden, was beim Akzeptieren des Updates geschehen soll.

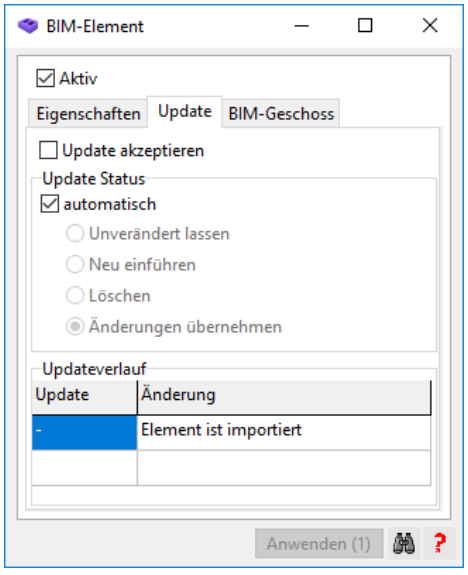
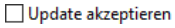


Fig. D–1 *Dialog „BIM–Element“ mit Register 'Update'*

Das Akzeptieren des Updates erfolgt durch die Betätigung der Schaltfläche im Update-Register von BIMLink. Was dabei mit einem BIM-Element passiert kann durch den 'Update Status' vom Benutzer definiert werden (s. unten).

Als Vorgabe wird der Update-Status auf 'automatisch' gesetzt. Will man dies nicht, so kann man ‘automatisch’ deaktivieren und den Status manuell einstellen.

Zum Abschluss muss noch die untenstehende Schaltfläche aktiviert werden, damit das Update gelingt.



D 6.8 Einsatz von BIMlink bei 2D-Geschossplänen

Falls kein 3D-BIM-Modell zur Erstellung eines Berechnungsmodelles zur Verfügung steht, kann BIMlink eingesetzt werden, um das CEDRUS-Gebäudemodell aus den digitalen 2D-Geschossplänen im DWG/DXF-Format zu erstellen. Dazu wird die CADview-Funktionalität verwendet (s. C 2), was eine gültige Lizenz für die Option 'CADview' (CV) erfordert.

☞ *Die Erstellung eines Gebäudes aus 2D-Plänen ist im Einführungsbeispiel von Kapitel A 2.6 detailliert erläutert.*

Die Eingabe eines CEDRUS-Gebäudemodelles aufgrund von 2D-Grundrissplänen im DWG/DXF-Format geschieht in zwei Schritten, die in den folgenden Kapiteln beschrieben werden.

D 6.8.1 Schritt 1: Geschossstruktur mit 2D-Plänen erstellen

Zuerst wird für jedes Geschoss, welches im Berechnungsmodell berücksichtigt werden soll, ein sogenanntes BIM-Geschoss-Objekt erstellt. Die Eingabe eines BIM-Geschosses erfolgt im Register 'BIM-Geschosse' von BIMlink.

Dort wird mit der Schaltfläche der Dialog BIM-Geschoss geöffnet, wo für jeden 2D-Plan eine Z-Koordinate definiert und die DWG/DXF-Datei als Hintergrundplan eingelesen wird. Die Z-Koordinate ist die Oberkante der entsprechenden Geschossdecke.

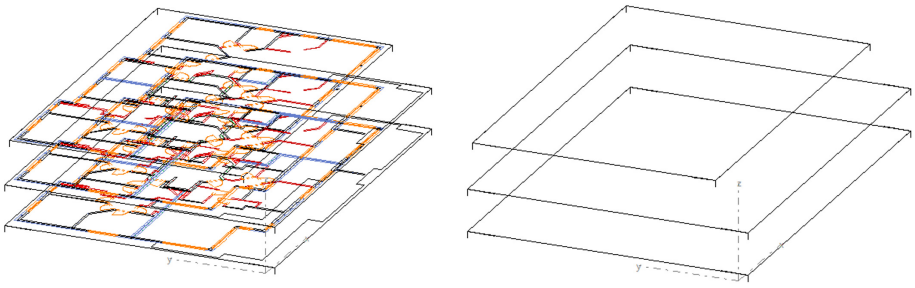


Fig. D–2 *Geschossstruktur mit und ohne Darstellung des 2D–Planes*

Damit ist die 3D-Geschossstruktur definiert und im Register 'Tragwerksmodell' kann das entsprechende CEDRUS-Gebäudemodell generiert werden.

D 6.8.2 Schritt 2: CEDRUS-Geschosse konstruieren

Zurück im CEDRUS-Hauptfenster können nun die einzelnen Geschosse als normale Platten konstruiert werden. Dabei stehen die im ersten Schritt eingelesenen 2D-Pläne als Hintergrundpläne zur Verfügung und müssen nicht mehr für jedes Geschoss eingelesen werden (s. C 2.2).

D 6.9 Export in IFC-Datei

Das Gebäudemodell von CEDRUS kann in eine IFC-Datei exportiert werden. Der Export kann über das Hauptmenu „Datei“ > „Export“ > „IFC“ gestartet werden.

Dabei werden alle Platten, Stützen, Pfähle und Wände des Geometriemodelles als „Coordination View“-Modell in eine IFC-Datei der Version IFC2x3 gespeichert.

Ein Export ist nur möglich, falls das Geometriemodell erzeugt werden konnte. Die exportierten Daten entsprechen der Geometrie, wie sie in der Gebäudeansicht mit eingeschaltetem Layer 'Baustoffe' dargestellt wird.

D 7 BIM-Interface to Nemetschek Allplan

D 7.1 Introduction

Building Information Modeling (BIM) is the exchange of three-dimensional, digital building data between the involved planning partners. CEDRUS has a BIM interface, which allows a three-dimensional, component-oriented model from a CAD system (for example, an architectural model) to be imported and a analysis model generated from it. With the current version of CEDRUS 3D building models can be imported, which were created with the CAD software Allplan.

☞ *The BIM interface requires a valid building license (option 'G') as well as a valid AP license (service option)*

CAD model ≠ analysis model

An imported CAD model can not be used directly for static and dynamic analysis. This has several reasons:

- CAD models are volumetric models that describe the three-dimensional geometry of the components. A analysis model, on the other hand, consists only of surfaces and axes, which represent the static system.
- In the CAD model, all components are described as geometrically as possible, whereas only the statically active structural elements are permitted in the analysis model. These should also be defined geometrically as simple as possible in order to ease result verification.
- Characteristic for the majority of the CAD-3D models that occur in practice is that they are optimized for architectual visualisation. Most CAD systems do not guarantee consistent, three-dimensional data, i. They allow substantial drawing inaccuracies (e.g., overlaps). Consistent data, however, is precisely the basic prerequisite for a useful analysis model.

In order to obtain a reasonable analysis model, the geometry of the components must be postprocessed from the CAD model.

Creation of the analysis model

To ensure that the effort for the required postprocessing does not negate the advantage of the 3D data import, CEDRUS has sophisticated algorithms that allow the automatic creation of the analysis model from the CAD model.

The following adjustments are made:

- Overlapping or completely covering components are merged
- Slabs on top of each other are combined
- Wall top edges are adapted to slab surfaces
- The surfaces of the walls touching each other are framed
- Column axes are adapted to the surfaces of the slabs
- Walls and columns, which do not stand on a slab, are provided with a foundation at their base
- Non-bearing structural elements are modeled as loads

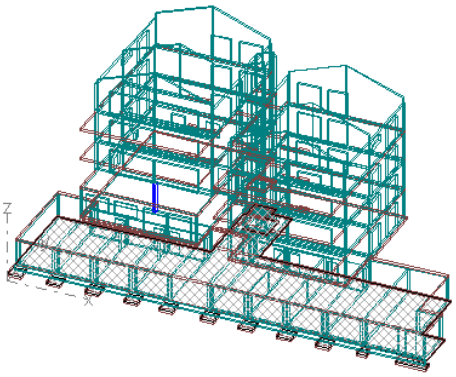
As a result of these automatic adjustments, the post-processing time can generally be considerably reduced or even completely eliminated. Nevertheless, it is the responsibility of the engineer to check the generated static model and, if necessary, adjust it.

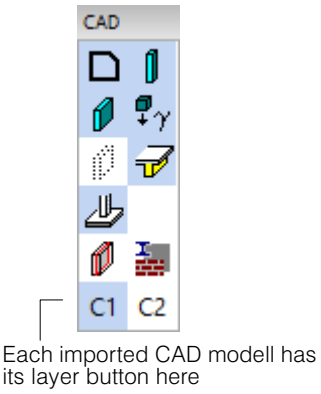
D 7.2 Workflow of a 3D CAD import

Use Allplan to create an XML file that contains all drawing elements required for the structure analysis.
Start CEDRUS and create a new building calculation. Change to the CAD register. From importing the external CAD file to the complete calculation model, the following steps are now necessary:

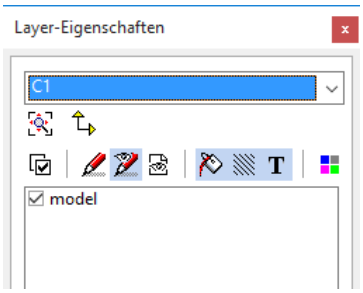
1. Importing the CAD model from an Allplan XML file

Press the 'Import CAD-Modell' button to open a dialog. Select the previously created XML file and press 'Open'. The CAD model is now read into a new layer in the CAD layer group and displayed in the graphics area.





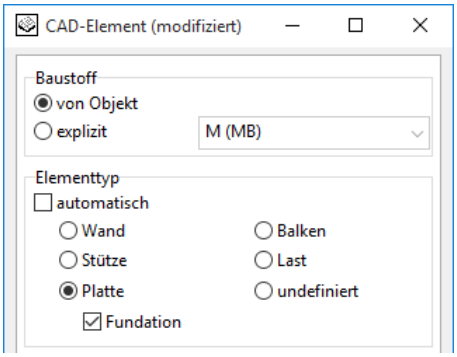
A CAD model can have multiple sublayers whose visibility is controlled in the Layer Properties dialog. The dialog is opened via the context menu of the CAD model layer button.

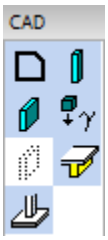


A CAD model that is no longer required can be deleted with the 'Delete' function in the context menu of its layer button.

2. Selection and adjustment of the CAD elements to be modeled

When importing, each recognized 3D object is inserted individually as a CAD element in CEDRUS. CAD elements are normal graphic objects with attributes and can be deleted or edited. The values of the attributes are taken from the CAD model data during import and can then be changed in the properties dialog.

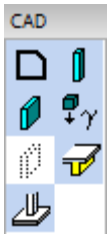




The most important attribute of a CAD element is the so-called element type. It determines the modeling in the calculation model. The following element types are available:

Walls:	Modeled as supporting, vertical wall
Columns:	Modeled as vertical pillar
Slabs:	Modeled as horizontal slab (possibly with offset)
Beams:	Modeled as downstand or upstand beams of a slab
Foundation:	Modeled as base slab (with area support zone)
Loads:	Modeled as additional dead load on the underlying slab
undefined:	Is not included in the analysis model

With the default 'automatic', the element type is determined directly from the geometric CAD data, whereby unassignable objects receive the element type 'undefined'.

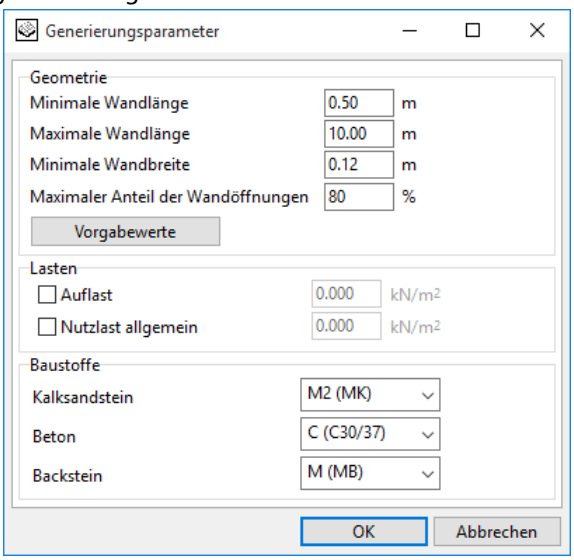


In order to view and control the imported data, each element type is drawn in a different color. The same color values are used as in the building model. In addition, the visibility of the CAD elements per element type can be individually controlled by turning the corresponding layer button on or off.

3. Definition of building materials and generation parameters

For the generation of the analysis model the building materials still have to be defined. These can not always be assigned correctly automatically during import and must be adapted manually in these cases. This can be done jointly for all CAD elements by assigning a building material from the building material list to the material names identified during the import.

To do this, press the 'Generation Parameters' button , which opens the dialog below. Assign a building material to each of the listed names.



In addition to the names of the building materials, individual generation parameters can be set here:

Minimal length of wall:
Shorter walls are not included in the calculation model or, if detached, modeled as a column

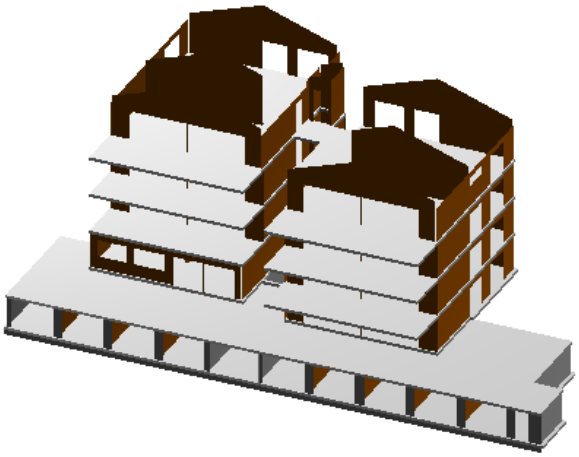
Maximal length of wall:
Longer walls are automatically divided

Minimal width of wall:
Thinner walls are not included in the analysis model

Maximal ratio of wall openings:
Wall sections with a higher ratio of openings are modeled as load

After closing the dialog with 'OK' the new building materials are assigned correctly. To check these settings, you can display the CAD elements in the color of their building material by activating the corresponding layer button 'materials'.

☞ *The material can also be changed individually in the property dialog for each CAD element*

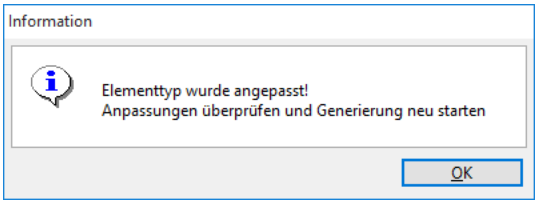


4. Generation of the calculation model

Now press the 'Generate geometry Model' button , to run the automatic generation.

Some CAD element can not be included in the analysis model for geometrical or functional reasons according to the specified element type. For example, the top walls of a building with gable roof can not be modeled as load-bearing walls since they are not under any plate. Such walls can only be accommodated as a load in the calculation model.

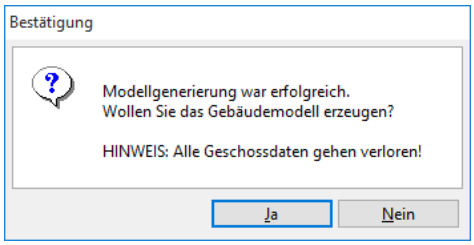
If possible, CEDRUS automatically adjusts the element type in these cases and displays the following message:



The affected CAD elements are highlighted by their flashing display.

☞ *The adjustments made by CEDRUS can always be undone by 'Undo'.*

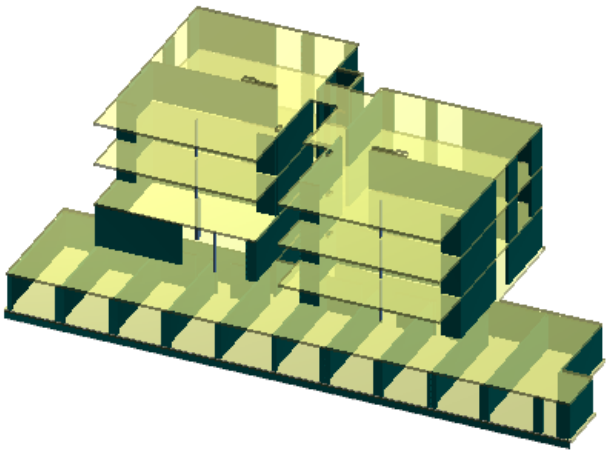
If all CAD elements are converted without errors, the dialog below appears:



By clicking 'Yes', the project data of the analysis model are now generated from the CAD elements, with all existing floors being deleted beforehand.

5. Check and refine the generated model

Now change to the 'Geometry' tab and press the button to display the geometry of the analysis model . In the process, the data of all floors are now checked and converted into a consistent building model.



☞ *Sometimes error messages occur during the verification of a floor, since the generated geometric data does violate the tolerances set in CEDRUS. In this case, you must adapt the corresponding geometries using the known means of graphical input.*

After successful, critical checking of the generated model, the liveloads in the individual floors have to be specified since they can not be automatically imported from the CAD model. Then the analysis model is ready for the necessary static and dynamic calculations.

Vol. E Building Option H

E 1 Introduction

Option H is an extension of the Building Module of Option G. It represents, on the one hand, an extension of the stacked slab model for the top-down analysis of horizontal loads. These aspects are described in detail in E 2. The following list gives some additional keywords with chapter and page information, where information may be found:

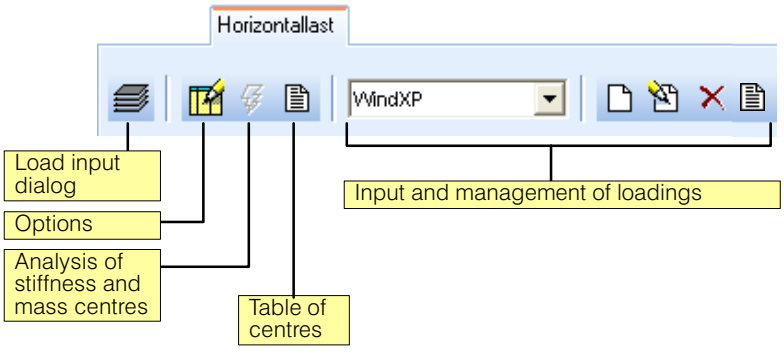
- Input of loadings — E 2.1
- Stacked slab model — D 5.2.1,E 2.3.1
- Generation of seismic loads for the equivalent method — E 2, E 4
- Definition of design spectra — E 2.2.2
- Mass and stiffness centres — E 2.1.5
- Combination loadings — E 2.2.1
- Results for walls and columns — E 2.3.3
- Consideration of horizontal loads in the dimensioning of slabs — E 2, E 4 E 2.3.4, E 2.3.5
- Masonry verification (MURUS program required) — E 2.2.3, E 2.4
- Dimensioning of reinforced concrete walls (FAGUS required) — in preparation

On the other hand, Option H allows the creation of frame models for earthquake analyses using STATIK, which is explained in E 3.

E 2 Analysis with the CEDRUS Building Model

After a detailed description of the input possibilities in the three Tab sheets 'Horizontal Loads', 'Analysis' and 'Stacked Slab Model' follows a E 2.4 with a description of the procedure for masonry verification (“Murus Analysis”)

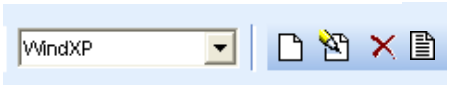
E 2.1 Tab sheet 'Horizontal Loads'



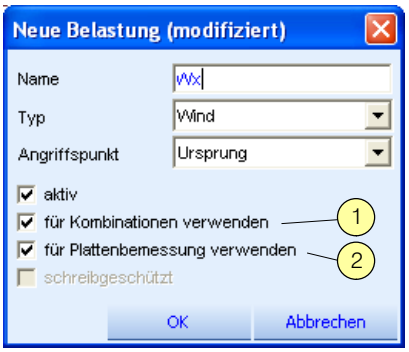
In this Tab sheet the centres of stiffness and mass for each story are determined and the loadings together with the horizontal loads are input.

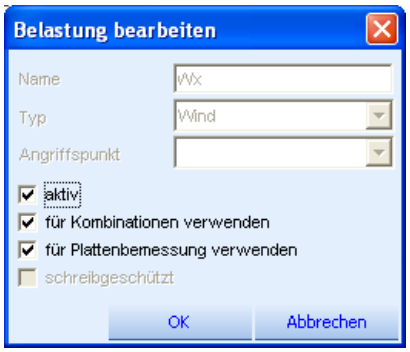
E 2.1.1 Management of Loadings

An arbitrary number of loadings consisting of horizontal loads can be input and managed. For this purpose the following self-explanatory tools are available:



The following dialog with the loading properties appears when a new loading is introduced or when modifying an existing loading:





Under Type the selections 'Wind', 'Earthquake', 'Earth Pressure' and 'others' are available. They have the following meaning:

Loading type	Loading considered in slab analysis	Used for auto-combinations in wall analyses	Available for manualcombi-nations and wall analyses	Exported to STATIK model
Wind	positive and negative if (2) on	if (1) on	yes	positive and negative, if active
Earthquake	positive and negative if (2) on	if (1) on	yes	---
Earth Pressure	if (2) on	if (1) on	yes	if active
others	---	---	yes	---

For a new loading the default value for the points of load application are selected. There are the following three possibilities:

- Origin: This is the origin of coordinates of the building.
- Centre of mass: The centres of mass are determined for each story using $G + \psi_2 * Q$. where G stands for the permanent loads including weights of walls (half of wall below and above the floor slab) and Q the variable loads. ψ_2 is specified in the Building Definition dialog.
- Centres of stiffness: On their determination see E 2.1.5.

Automatically generated loadings

If, in the Options dialog of the Tab sheet 'Horizontal Loads', the generation of earthquake loads is required, the following four loadings (of type Earthquake) are automatically created (see Page E-4); note, here equivalent signifies “static-equivalent”:

!EXTP: equivalent force in X direction with positive torsional loading
!EXTN: equivalent force in X direction with negative torsional loading
!EYTP: equivalent force in Y direction with positive torsional loading
!EYTN: equivalent force in Y direction with negative torsional loading

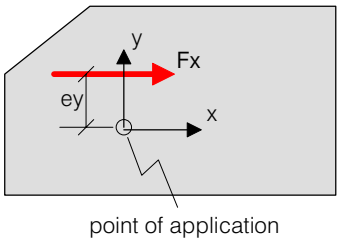
E 2.1.2 Load Input Dialog

The Building Definition dialog is replaced by the Load Input dialog in the Tab sheet 'Horizontal Loads':

ID	z	Beschreibung	Angriffspunkt	ex	ey	Fx	Fy	T
L5	11.40	DG	Ursprung	0.00	5	80.0	0.0	0.0
L4	8.60	OG2	Ursprung	0.00	5	70.0	0.0	0.0
L3	5.80	OG1	Ursprung	0.00	5	60.0	0.0	0.0
L2	3.00	EG	Ursprung	0.00	5	50.0	0.0	0.0
L1	0.00	UG	Ursprung	0.00	0	0.0	0.0	0.0
L0	-3.00		Ursprung	0.00	0.00	0.0	0.0	0.0

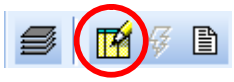
Its visibility can be controlled using the button shown.

For each story a force can be input for each of the directions x, y and a torsional moment about the z axis. The coordinates x,y,z refer to the global building coordinate system.



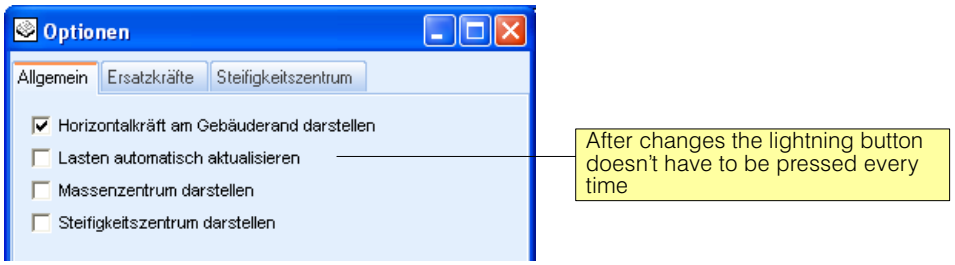
Since the loads act on the more or less rigid story membrane, the point of application of the torsional moment is not important. The loads Fx and Fy act at the chosen point of application, in some cases with an eccentricity normal to the load direction. The eccentricity in the load direction is unimportant.

E 2.1.3 Options Dialog

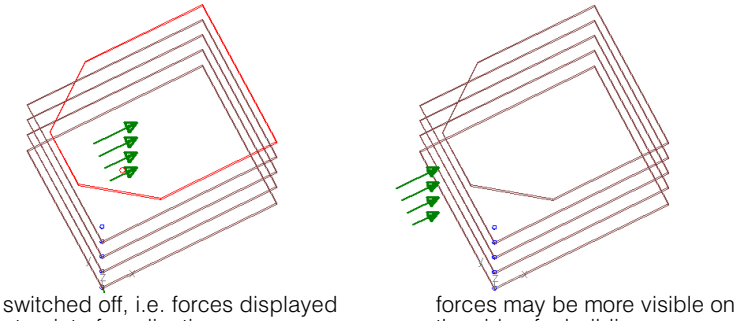


This button opens the dialog with the following options:

Tab sheet 'General'



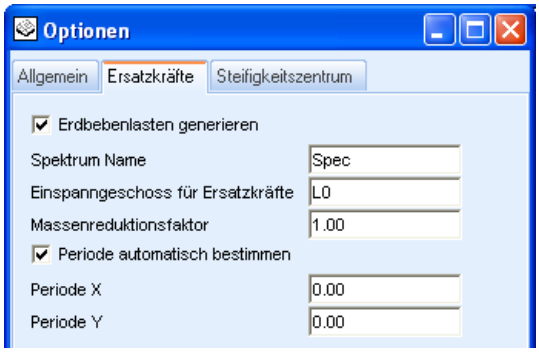
Switch to 'Show horizontal forces acting at the side of the building':



switched off, i.e. forces displayed at point of application

forces may be more visible on the side of a building

Options for generating the equivalent earthquake forces



In this way, the loadings with the statically equivalent forces for the earthquake analysis can be automatically generated. In addition, the following are required

- The **mass distribution**, which is known for the model for each story with $G + \psi_2 * Q$. G represents the permanent loads including the weights of walls (half of wall below and above the floor slab) and Q the variable loads; ψ_2 is specified in the Building Definition dialog.
- Some codes allow a reduction of mass in the determination of the statically equivalent loads (e.g. a factor 0.85 in EC). If necessary, the default value of 1.0 for the **Mass Reduction Factor** can be changed in the dialog.
- The **periods** of the fundamental mode of vibration in the X and Y directions.
  With the setting 'automatic' they are determined according to the formula proposed in the SIA261 code, $T = 0.5h^{0.75}$, and otherwise they are input explicitly.
- The **design spectrum**: It is specified in the Tab sheet 'Analysis' and in the above dialog Tab sheet the corresponding 'Spectrum Name' must be entered.
- The **story widths** for any eccentricity of the centres of mass, which are necessary for the generation of the torsional loadings. They are automatically imported by the slabs.

In the dialog one can also select the floor to have fixed supports, if one wants to exclude rigid basement floors from the seismic model.

By activating the lightning button the following equivalent forces are created:

!EXTP: equivalent force in X direction with positive torsional loading
!EXTN: equivalent force in X direction with negative torsional loading
!EYTP: equivalent force in Y direction with positive torsional loading
!EYTN: equivalent force in Y direction with negative torsional loading

Remarks on torsional loadings:

The torsional loadings should comply with the formulas (42) and (43) of the code SIA261. The formula (42) is

$$e_{d,sup} = e + 0.5e + 0.05b \qquad \text{(i.e. 1.5e is split into } e + 0.5e\text{)}$$

$e_{d,sup}$ is the maximum eccentricity of the centres of mass and stiffness to be considered. The formula (43) for the minimum eccentricities is included in following considerations.

The formula is specifically for the equivalent beam method (cantilever embedded in the ground) and has to be adapted to the stacked slab model. The three terms of the above formula have the following meanings and are treated accordingly:

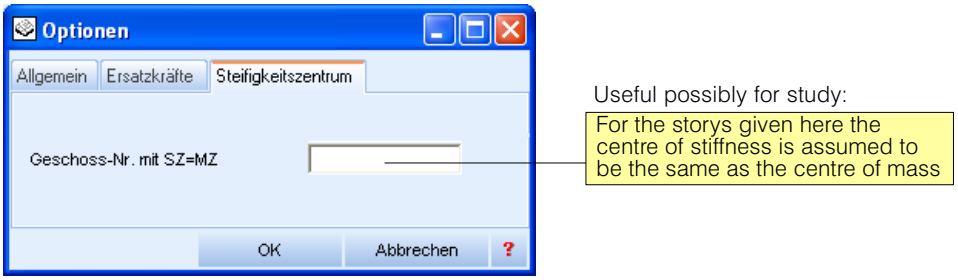
Term 1: e Actual or rather the planned eccentricity of the centre of mass with respect to the centre of stiffness: The statically equivalent earthquake force F_i for story i is an inertia force and has to act therefore at the centre of mass of the floor. Since the equivalent member has its axis in the centre of stiffness, if the force acts in the axis of the member as a compensation a torsional moment Fi * e must be introduced.

Different to the equivalent member, the equivalent force in the case of the stacked slab model can be applied directly at the centre of mass. The walls underlying the slab transmit the equivalent force according to their stiffnesses and the resulting wall forces are in equilibrium with the equivalent force. Thus Term 1 is implicitly included in the stacked slab model and does not have to be included in the torsional loading that has to be introduced.

Term 2: 0.5e With this term a dynamic amplification (resonance effects during the earthquake) is taken into account, which results from the actual eccentricity of the centre of mass. This term is considered in the torsional loadings.

Term 3: 0.05b The planned eccentricity e of the centre of mass with respect to the centre of stiffness is subject to various arbitrary (random) uncertainties, e.g. the method of fixture, how the live loads are included in the determination of the masses. These random eccentricities are covered in the code by 0.05b (b=width of building normal to the direction of excitation) and considered accordingly in the torsional loadings.

Options regarding centre of stiffness



E 2.1.4 Updating the Loads

After changes in the loads or the loading options the loadings and their display are updated.

E 2.1.5 Evaluation of the Mass and Stiffness Centres

Using this button a table with the mass and stiffness centres of all storys is output. How these are determined is described below.

Centre of mass

The centre of mass of a story is the centre of gravity of the weight of the story, which consists of the following parts:

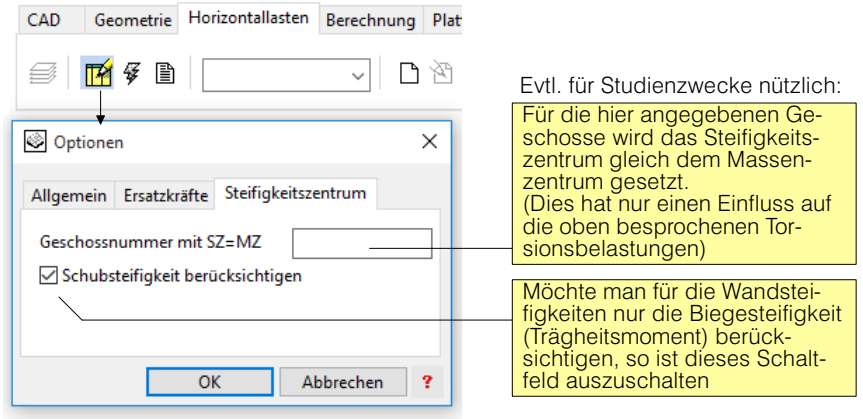
- Weight of the floor slab, which is given by the sum of all permanent loadings on the floor slab. All loadings of the type 'permanent' are taken into account that are active and intended for automatic export. These properties are specified in the dialog with the loadings properties and by default are set as follows:

- ψ_2 * the variable loads acting on the floor slab. ψ_2 is a property of an action assigned to a loading. All loadings of the type 'variable' are taken into account that are active and intended for automatic export (analogous to the permanent loads).
- Weight of half of each wall above and below the foor slab is considered.
- Weight of half of each column above and below the foor slab is considered.

Centre of stiffness

The centre of stiffness of a slab resting on walls and columns is by definition the place where, for an assumed rigid slab membrane, horizontal forces in the x and y directions can be applied, without causing a rotation of the slab membrane. Thus the slab is considered to be flexible in bending and the vertical elements to be fixed at the bottom.

For the analysis model the displacement stiffnesses of the tops of the walls and columns are needed. Normally, the pure bending stiffnesses are assumed and thus the current CEDRUS version does this too.

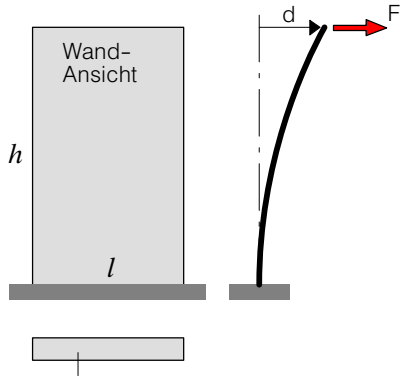


Remarks:

- Long walls greatly influence (i.e. attract) the centre of stiffness, because the bending stiffness increases with the third power of the length of the wall. In the case of buildings with just a few storys it is often not realistic and leads to overestimated torsional loadings. In some cases it is better to subdivide such a long wall.
- The idea of the centre of stiffness comes from the equivalent frame model, in which one takes into account the torsional effects by considering the eccentricities of the centres of mass with respect to the centres of stiffness. Whereas by definition the centre of mass in an individual story has a clear physical meaning, in a building one can only speak approximately of centres of stiffness, since the deformation behaviour of a story depends greatly on how it is integrated into the whole building and also on the distribution of the loads.
- In the case of three-dimensional building models or of the stacked slab model the identified centres of stiffness have no direct influence in the analysis model. They are only required for the torsional loadings when using the equivalent force method (see 'Remarks on Torsional Loadings' on page E-5).

Displacement stiffness s of a wall

This is generally determined as follows:



Wandquerschnitt im Grundriss
mit dem Trägheitsmoment J
um die starke Achse

$$F = s \cdot d$$

$$Stiffness\ \ s = \frac{1}{\frac{1}{s_B} + \frac{1}{s_V}}$$

$$s_B = bendingstiffness \cong \frac{(3...12) \cdot EJ}{h^3}$$

3 corresponds above free, 12 clamped; selected in the program 6 as mean

$$s_V = shearstiffness = \frac{GA^*}{h}$$

A = 5/6 A (shear area of the cross section)*
E and G Correspond to the building material of the wall
*assumption for h: 0.75 * building height*

$h/l \gg 1$ *Bending stiffness decisive*
$h/l \approx 1$ *Both shares*
$h/l \ll 1$ *Shear stiffness decisive*

Remarks:

- Several wall segments in a single plane are treated like a single wall, if they are grouped into a pillar.
- General, grouped pillars are treated accordingly.

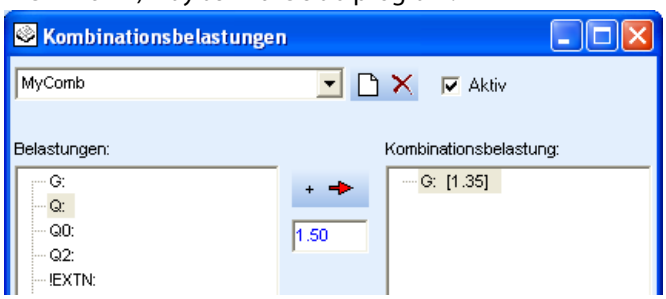
E 2.2 Tab sheet 'Analysis'

E 2.2.1 Combination Loadings

Analogous to the slab program also for the stacked slab model combination loadings (here often just referred to as combinations) are defined. Some combinations are also automatically generated.

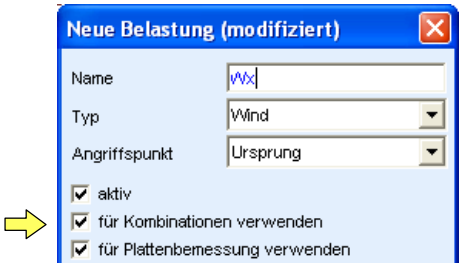
Defining combination loadings

The input of combinations is carried out in the dialog Combination Loadings in an analogous (and well known) way as in the slab program:



Automatically generated combinations

Loadings of the types Wind, Earthquake and Earth Pressure are input in the automatically created combinations, if they are specified accordingly:



Combinations consisting of loadings of type Earthquake

From a loading with the name 'BEL' four combinations result:

!GQ2_BEL_P
!GQ2_BEL_N

These two combinations contain

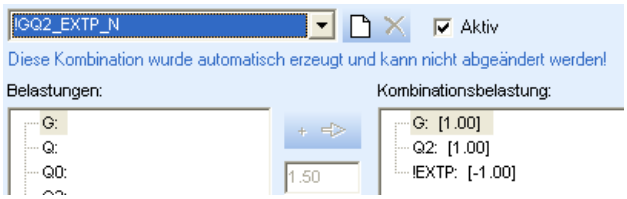
- G * 1.0:
  the permanent loads defined in the floor slabs
- Q2 * 1.0:
  the variable loads defined in the floor slabs, multipled by the corresponding load factors ψ_2.
- the input loading with positive and negative sign (_P, _N)

Based on the 4 automatically generated earthquake loadings

!EXTP: equivalent force in X direction with positive torsional loading
!EXTN: equivalent force in X direction with negative torsional loading
!EYTP: equivalent force in Y direction with positive torsional loading
!EYTN: equivalent force in Y direction with negative torsional loading

the following 8 combinations are created:

!GQ2_EXTP_P
!GQ2_EXTP_N
!GQ2_EXTN_P
!GQ2_EXTN_N
!GQ2_EYTP_P
!GQ2_EYTP_N
!GQ2_EYTN_P
!GQ2_EYTN_N



Combinations of loadings of type Wind and Earth Pressure

From a loading of type Wind called 'WX' the following 8 combinations are obtained (G and Q are the permanent and variable loads input in the floor slabs):

!GiQ_WX0_P: $\gamma_{G,inf}$*G + γ_Q*Q + ψ_0 * WXx
!GiQ_WX0_N: $\gamma_{G,inf}$*G + γ_Q*Q - ψ_0 * WX
!GiQ0_WX_P: $\gamma_{G,inf}$*G + ψ_0*Q + γ_Q * WX
!GiQ0_WX_N: $\gamma_{G,inf}$*G + ψ_0*Q - γ_Q * WX

!GsQ_WX0_P: $\gamma_{G,sup}$*G + γ_Q*Q + ψ_0 * WX
!GsQ_WX0_N: $\gamma_{G,sup}$*G + γ_Q*Q - ψ_0 * WX
!GsQ0_WX_P: $\gamma_{G,sup}$*G + ψ_0*Q + γ_Q * WX
!GsQ0_WX_N: $\gamma_{G,sup}$*G + ψ_0*Q - γ_Q * WX

Normally, there is also a wind load in the Y direction, which generates a further 8 combinations.

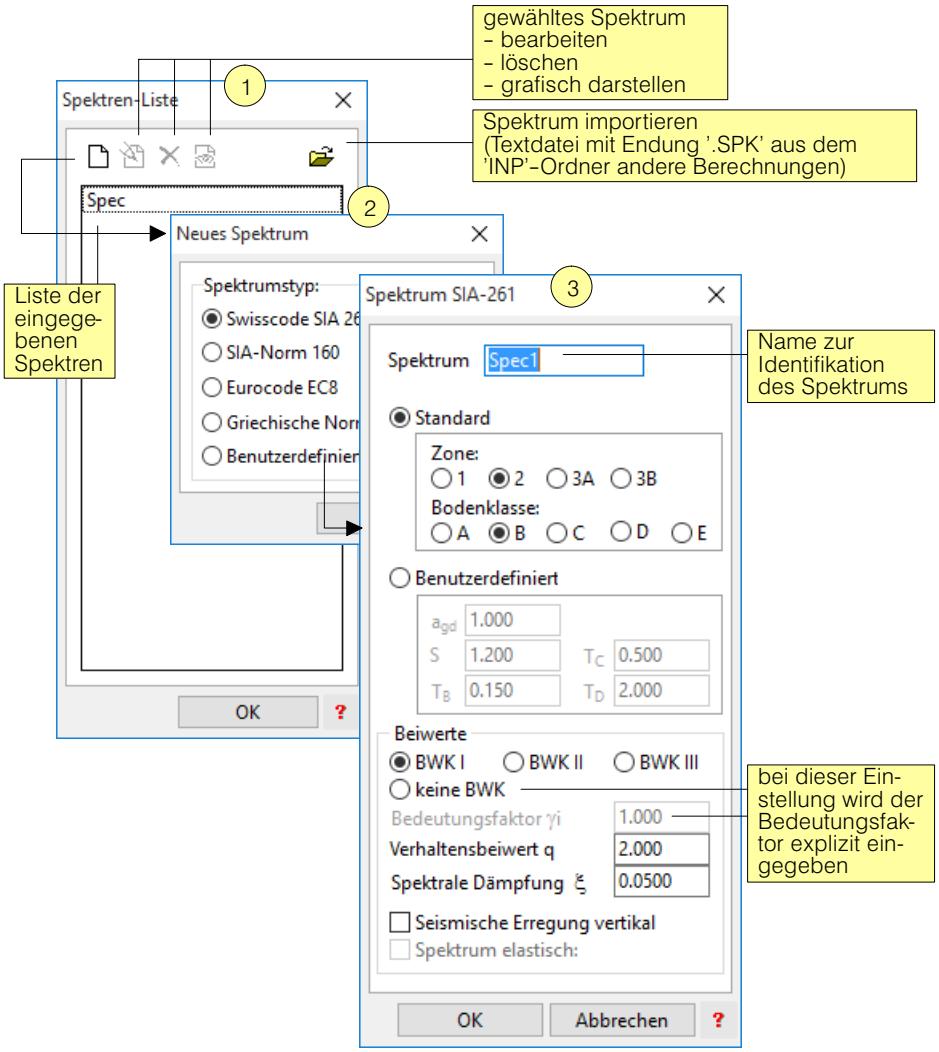
Any additional input loadings of the type Earth Pressure are assumed to be permanent loads. They are also included in the above combinations with their corresponding $\gamma_{G,inf}$- and $\gamma_{G,sup}$ values. It is assumed that the permanent loads from the floor slabs and those from earth pressure act in combination, both favourably or unfavourably.

If there are only earth pressure loadings without wind then the following combinations are created:

!GiQ: $\gamma_{G,inf}$*G + $\gamma_{G,inf}$*(EDL1+EDL2+...)+ γ_Q*Q ; 'EDLi'=Name of Earth Pressure loadings
!GsQ: $\gamma_{G,sup}$*G + $\gamma_{G,sup}$*(EDL1+EDL2+...)+ γ_Q*Q ;

E 2.2.2 Definition of Design Spectra

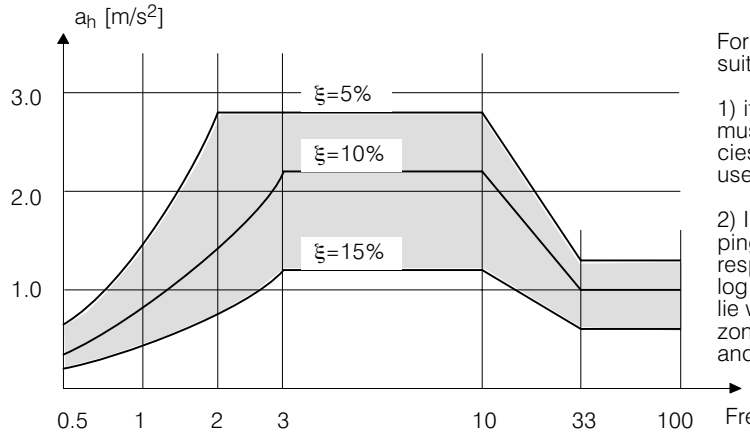
For the generation of earthquake loads it is necessary to have a design spectrum. With the button shown, the Design Spectra Management dialog is opened (1), with which design spectra can be defined and managed:



Before equivalent force loadings can be generated, a spectrum with the name defined in the Options dialog (page E-4) must be given. The input of standard spectra (3) according to the codes listed in (2) is self-explanatory. CEDRUS also permits the input of individual spectra that are based on data from microzonation, but which however are more likely to be used in the response spectrum method with the STATIK model.

User-defined spectra

As an example the following spectrum in the form *Frequency vs. Acceleration* is given. It is based on the importance factor $\gamma = 1$ and a behaviour factor q of 1.0.

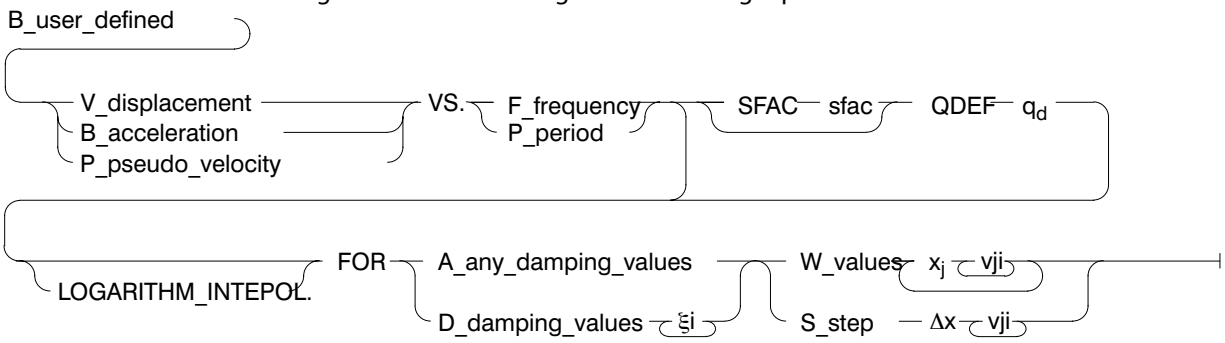


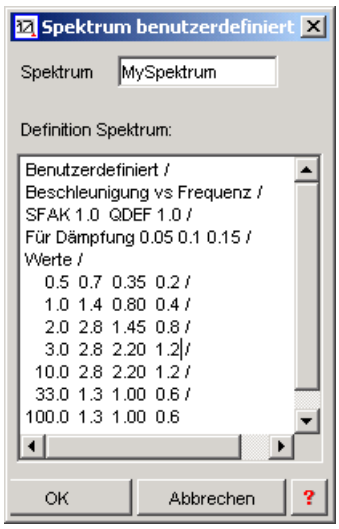
For the spectrum to be suitable

1) its range of frequency must cover the frequencies of all eigenmodes used in the analysis.

2) In addition, the damping values given in the response spectrum dialog must, for each mode, lie within the hatched zone, i.e. here between 5 and 15%.

The input of the above spectrum in the dialog 'User-defined Spectrum' is performed in the dialog's text field according to the following input schema:





The most important rules for the interpretation of the input scheme are illustrated in the example.

Remarks:

- The whole input text consists of a single line, whereby at every place with a line continuation symbol '/' it can be interrupted.

- For the words written in capital letters, at least the initial letter has to be used.

- All numerical values must have a decimal point, otherwise the input is format-free. Between the individual elements (words, numbers) there must be at least one blank character. If these rules are ignored, the program issues a corresponding error message.

sfac is a factor with which all input spectral values v_{ji} are multiplied. If, for example, the input spectral values are based on an importance factor γ=1 and a behaviour factor q=1, then with a corresponding factor sfac=γ/q the spectrum can be immediately adjusted to changes in the values of these two factors.

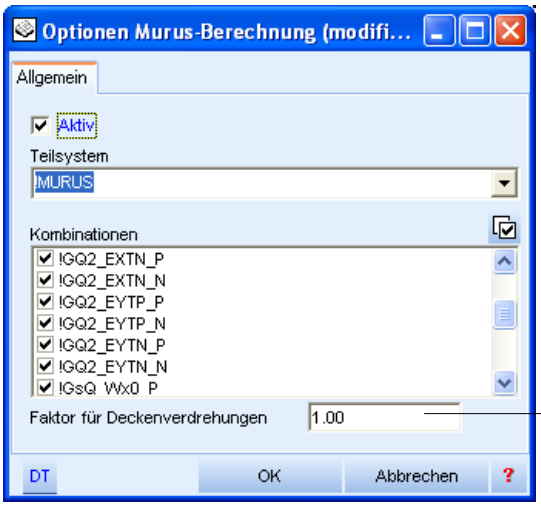
q_d is the behaviour coefficient, with which the displacement results determined on the basis of a design spectrum have to be multiplied as specified in the code.
(SIA 261:16.5.5.1: q_d=q, EC-8: q_d=q, 'if nothing else is specified')

☞ *If one wants to redefine a spectrum that has already been input, e.g. for a behaviour factor of 2.0, then for sfac the value 0.5 instead of 1.0 and for q_d the value 2.0 instead of 1.0 can be input.*

E 2.2.3 Setting-up a Masonry Analysis

In the case of a masonry analysis every masonry wall is analysed with the Masonry program Murus for all requested load cases.

In the Options dialog for a MURUS analyis it is specified for which walls and loading combinations the verification has to be carried out:



The Murus analysis is carried out in the Tab sheet 'Stacked Slab Model'. There it is only available for analysis purposes, if here in the Options dialog it is set to **active**.

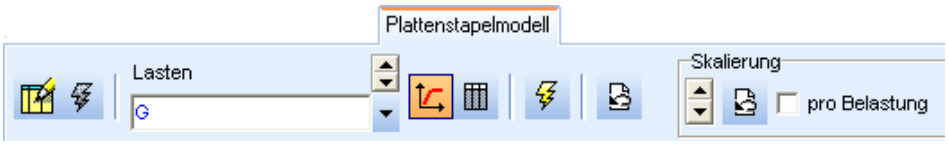
The **partial system 'MURUS'** selected in the selection field 'Partial System' contains by default all walls with masonry material (regarding Partial Systems see Chapt. D 4.1). The verification is performed for all walls included in the selected partial system. The partial system can if necessary be modified or a different one can be created and selected.

In the field 'Combinations' those **combination loadings** are marked (ticked), for which the Murus analysis has to be performed.

The rotations at the top of a wall imposed by the floor slab have to be taken into account in the Murus analyses. For this purpose the elastically determined floor slab rotations due to the quasi-permanent loads are used, which in order to take into account the long-term influences and cracking have to be multiplied by an estimated **Factor for Floor Slab Rotations**, which is input here.

☞ *In Chapter E 2.4 'Murus Analyses from A to Z' all the steps of a Murus analysis and special points to be considered in the modelling are summarized.*

E 2.3 Tab sheet 'Stacked Slab Model'



E 2.3.1 Building the Model and Selecting the Options

While the stacked slab model in the option G of CEDRUS is used for the top-down analysis of vertical loads and is described there in Ch. D 5; with the option H it is extended to the treatment of horizontal loads.

The setting 'Constant forces at bottom for vertical loads' only concerns vertical loads (see D 5.3.1).

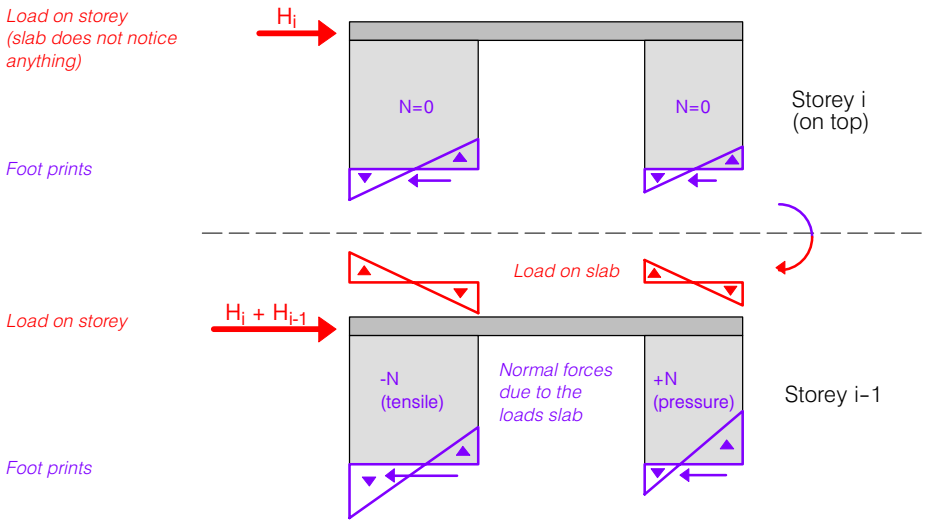
With this button the loading-independent model is built. It is not essential for the user to carry out this step, as it is automatically done when the program starts the analysis. More on this in Chapt. D 5.3.

Analysis model for horizontal loads

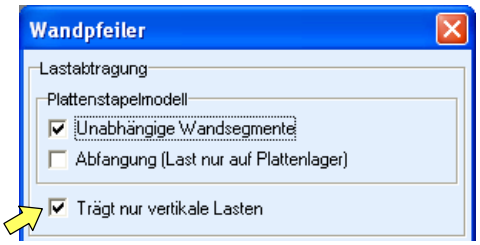
Here too the analysis is based on the stacked slab model, i.e. the loads are transmitted from top to bottom for each story from the slab to the wall pillars and columns and transmitted from them to the next story.

The loads are transmitted through the basically rigid membrane of each story to the vertical elements according to their stiffness in such a way that the resulting shear forces in these elements are in equilibrium with the external loads.

The forces so obtained at the bottoms of the wall pillars and columns are transmitted as a loading to the next floor slab.



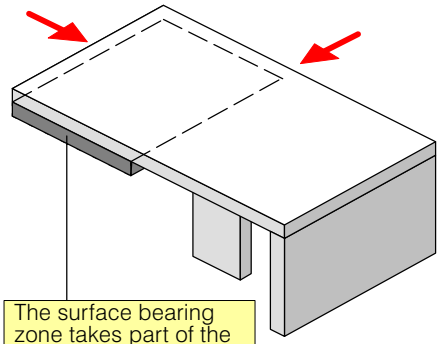
⚠ Wall pillars with the setting 'carries only vertical loads' do not participate in the transfer of horizontal loads:



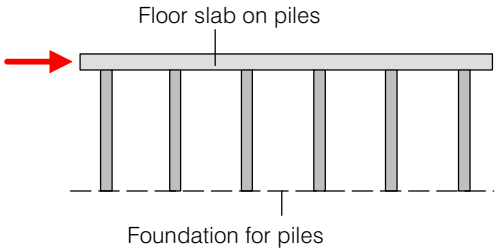
⚠ Walls with point transfer (defined on page D-28) cannot transmit horizontal forces. They have to be set by the user to 'transmits only vertical loads', otherwise the program gives an error message.

Treatment of floor slabs

In this case, the term "floor slabs" should be understood to be slabs which rest completely or partially on the ground and are provided with a corresponding surface bearing. These surface bearing zones, of course, also take on horizontal forces, which is important when it goes downwards (see the figure below). It is not easy to determine the stiffness of a surface bearing zone for horizontal loads. The program assumes rough assumptions, whereby the stiffness depends on the bedding number and can therefore be influenced by it.



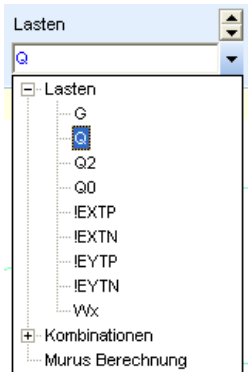
The surface bearing zone takes part of the horizontal forces



If the piles are designed as pendulum supports, they can not absorb horizontal forces. In this case, the floor slab must have a bearing for stabilization. The bedding number should be so small that unintentionally loads are not removed from the piles.

E 2.3.2 Results Selection Field and Starting the Analysis

The results selection field makes available all loadings and combination loadings as well as the 'Murus Analysis' :



The Murus analysis appears in the list if it was activated in the Options dialog (Chapt. E 2.2.3).

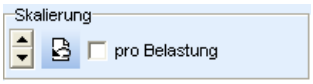
The analysis of the selected result is started using this button.

E 2.3.3 Results in Walls and Columns

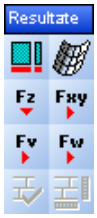
For results of Murus analyses see Chapt. E 2.4. The tasks described here concern the loadings and combinations selected in the selection field.

Graphical

The exaggerated scale of the diagrams and the load arrows in the graphics can be controlled using the following self-explanatory tools:



To display the forces at the bottoms of the walls and columns the following layer is available in the Layer Group 'Results':



Display of footprints of the stacked slab model (see D 5.3).

This Result Layer shows the bending deflections of the slab in the 'Building' view. Normally, however, one would consider the bending deflections in the individual slabs.

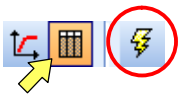
Bottom vertical forces in wall segments;
Labelling: Nu (=total vertical force)
Pointer on the cursor: Nu, Mw(= moment about the major axis of the wall segment)

Horizontal force (sum of both components) at the bottom of the vertical elements in vectorial form.

Wall segments: bottom shear force along wall segments
Columns: bottom shear force in X direction of slab coordinate system

Wall segments: bottom shear force normal to the wall segments
Columns: bottom shear force in Y direction of slab coordinate system

These layers contain the results of a Murus analysis.



Tabular

The output is created fot the specified loading (in the figure = G) and consists of the three listed tables:

The table headings are as follows:

Fusskräfte Z

Geschoss	Bauteil	Wandsegmente				Wandpfeiler			Stützen	Fundation	Summe
		Fuss [%]	fz_{min} [kN/m]	fz_{max} [kN/m]	Fz [kN]	fz_{min} [kN/m]	fz_{max} [kN/m]	Fz [kN]	Fz [kN]	Fz [kN]	Fz [kN]
L5	C1								173.8		
L5	C2								32.7		
L5	C3								32.7		

The Fz correspond to the quantities described above in the graphical output. Besides the total forces for each wall segment, in the tabular output the forces per unit length fz_{min} and fz_{max} are also output (different values in the case of a trapezoidal distribution). In addition, also the values for wall pillars are output, which in the case of grouped wall pillars differ from the segment values.

Fusskräfte XY

Geschoss	Bauteil	Wandsegmente		Wandpfeiler		Stützen		Fundation		Summe	
		Fx [kN]	Fy [kN]	Fx [kN]	Fy [kN]	Fx [kN]	Fy [kN]	Fx [kN]	Fy [kN]	Fx [kN]	Fy [kN]
L5	C1					0	0				
L5	C2					0	0				
L5	C3					0	0				

Here the components of the horizontal bottom forces are output in the global coordinate system of the building.

Fussschnittkräfte

Geschoss	Bauteil	Schnittkräfte						
		Nu [kN]	Vv [kN]	Vw [kN]	Mu [kNm]	Mv [kNm]	Mw [kNm]	
L5	C1	173.77	0	0	0	0	0	
L5	C2	32.72	0	0	0	0	0	
L5	C3	32.73	0	0	0	0	0	

The directions u,v,w are defined here as follows:

u: vertical

v: **wall segments**: in the direction of the wall segment
wall pillars (grouped): direction X of the slab's coordinate system
columns: direction X of the slab's coordinate system

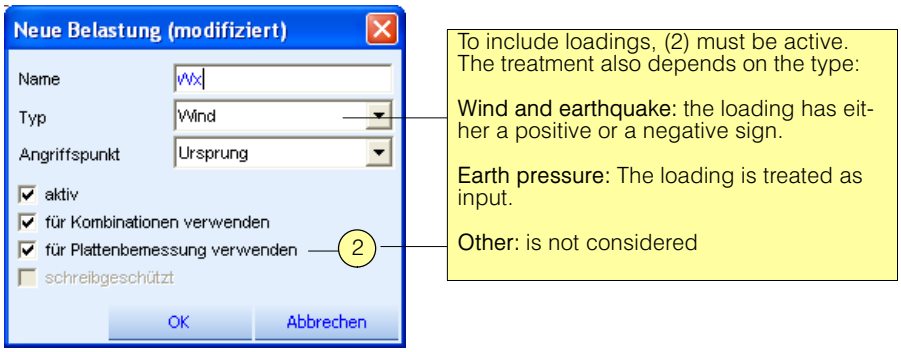
w: **wall segments**: normal to direction of wall segment
wall pillars (grouped): direction Y of the slab's coordinate system
columns: direction Y of the slab's coordinate system

E 2.3.4 Results in Floor Slabs

As mentioned in Chapt. D 5.3.3, the results of the stacked slab analysis come in the form of imported loadings in the floor slabs of a building. Without horizontal loadings the imported loadings !Imp-G for the permanent loads, !Imp-Q for the variable loads and, if present, !Imp-P for prestressing loads appear. These imported loadings contain the bottom forces for the walls and columns that act on the corresponding floor slab.

If, in addition, we also have horizontal loads acting on the building, then under certain circumstances the resulting bottom forces from these loadings appear as imported loadings in the floor slabs.

Which loadings appear in the slab analyses depends, as mentioned in E 2.1.1, on the loading type and, for manually input loadings, on the setting in the Loading dialog :



Loadings of type Earthquake

For each input loading two corresponding import loadings are obtained with different signs. For example, if a loading called 'EXTP' is input, the following two import loadings are obtained:

!Imp_E0 : EXTP+ : input loading EXTP
!Imp_E1 : EXTP- : loading generated from it

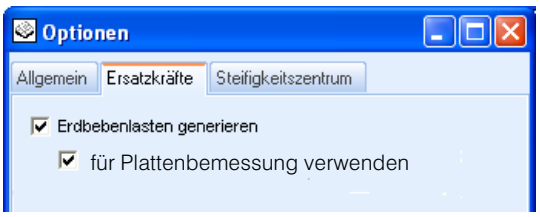
Since with the equivalent force method torsional loadings that act together with the horizontal equivalent forces always have to be considered, one has to include them in the input loadings. Thus normally four loadings have to be input, in order to cover all possible combinations (with respect to signs) of the equivalent and associated torsional loadings, e.g.:

!Imp_E0 : EXTP+ : input loading EXTP with EkX positive, TEkX positive
!Imp_E1 : EXTP- : loading generated from it with EkX negative,TEkX negative
!Imp_E2 : EXTN+ : input EXTN with EkX positive, TEkX negative
!Imp_E3 : EXTN- : loading generated from it with EkX negative,TEkX positive

!Imp_E4 : EY TP+ : input loading EVTP with EkY positive, TEkY positive
!Imp_E5 : EY TP- : loading generated from it with EkY negative TEkY negative
!Imp_E6 : EY TN+ : input loading EVTN with EkY positive, TEkY negative
!Imp_E7 : EY TN- : loading generated from it with EkY negative TEkY positive

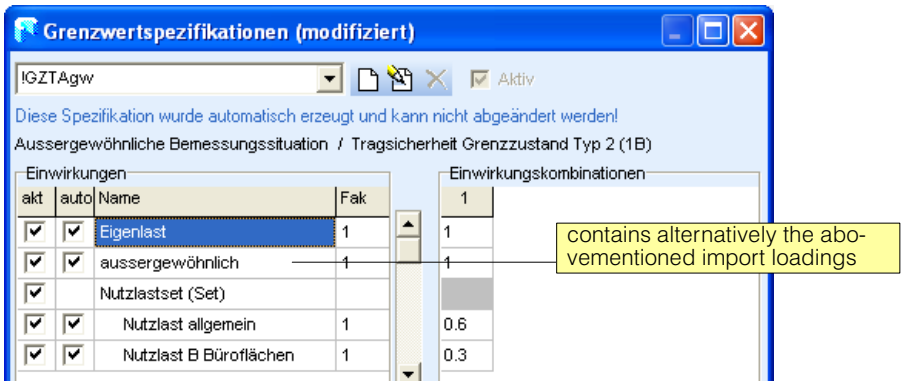
(EkX = equivalent force in X direction, TEkX = torsional loading associated with EkX)

If one allows the equivalent force loadings to be automatically generated by means of the corresponding settings in the Options dialog for horizontal loads, the four



above mentioned loadings, which have to be manually input, are automatically generated and identified by !EXTP, !EXTN, !EYTP and !EYTN. In the case of the setting 'Use to dimension slab' the eight import loadings are created as described above.

A limit state specification 'GZTAgw' for the accidental dimensioning situation is automatically produced :

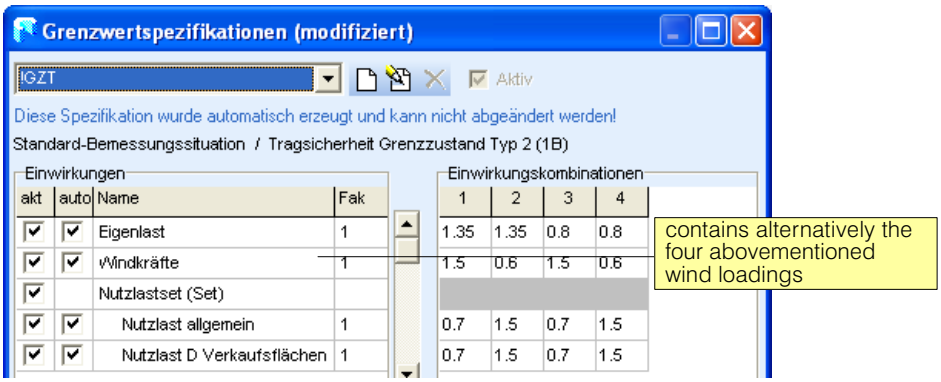


Loadings of type Wind

If e.g. two loadings 'Wx' and 'Wy' of type Wind are input, this leads to the following import loadings in the floor slabs:

!Imp_W0 : Wx+
!Imp_W1 : Wx-
!Imp_W2 : Wy+
!Imp_W3 : Wy-

The automatically generated limit state specification for the ultimate load state may look as follows:



Live and wind loads appear alternatively as the leading action and the accompanying action, respectively.

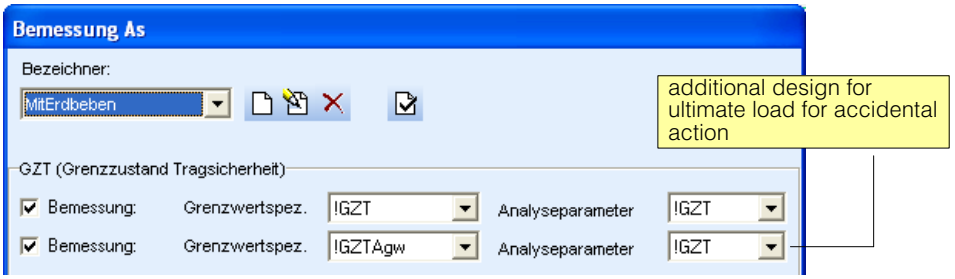
Loadings of type Earth Pressure

These are treated in an analogous way to the type Wind, whereby, however, for each loading only one import loading is created (no additional ones with a negative sign)

E 2.3.5 Design Specifications for Slabs with Seismic Actions

The design of the reinforcement is usually carried out for the ultimate load state and then for *persistent and temporary design situations*. Thus the generated design situations *!Standard* and *!Standard+* are set.

If, in the Options dialog for horizontal loads, consideration of the earthquake loadings in the design of the slab is required, the limit state specification GZTAgw for the *accidental design situation* Earthquake is automatically created. If one also wants to take this into account in the design of the reinforcement, a corresponding design specification has to be defined:



☞ *Slabs, on which walls are supported wholly or partially, must of course be defined for the seismic actions. If in a particular story all walls can transfer their forces directly and completely to the underlying walls and columns, then the floor slab only has to be sufficiently ductile and it is not really necessary to design it for seismic actions.*

E 2.4 Murus Analysis from A - Z

In the case of a Murus analysis the serviceability and ultimate load verifications for all masonry walls is carried out according to the code SIA266.

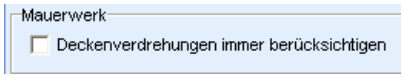
A Murus analysis comprises the following steps:

On creating the building model we have to consider:

1) Masonry walls are recognized by the program by the assigned material type (Masonry). CEDRUS creates a partial system called !Murus, which contains all plane shear walls with the material type *Masonry.*
2) The Murus analysis only considers plane wall pillars. These may consist of one or more segments.
3) Free rotation of the wall supports are handled in the Murus model by having a hinge at the top without eccentricity and being fixed at the bottom. The hinge support condition at the top, however, can be annulled in the wall pillar properties (see Point 4b).
4) Wall pillar properties have to be set correctly:

   a)

   If in this way one deactivates the ability to resist horizontal forces, then this has to be ensured by constructional means, or it must be verified that these walls can also resist the vertical loads under the expected deformations.

   b) 

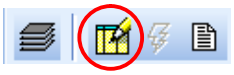
   In the case of a fixed support, the rotations of the floor slab at the top and bottom of the wall determined for the quasi-permanent loads have to be taken into account in the Murus verification. With this check box one can specify that the top of the wall, in the case of walls with freely rotatable supports, is loaded by the floor slab rotations and thus may be considered as fixed.

Loadings

What one has to do in the case of earthquake and wind loads is separately described here.

In the case of earthquake analyses with the equivalent force method

5) Input of a spectrum in the Tab sheet 'Analysis' (see E 2.2.2).



6) In the Tab sheet 'Horizontal Loads' Options dialog open and make the settings according to the description given in E 2.1.3. In particular, the Tab sheet 'Equivalent Forces' in the dialog needs to be considered.

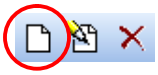
7) Calculate equivalent force loadings and check them. The following loadings should be present:
   !EXTP: equivalent force in X direction mit positive torsional loading
   !EXTN: equivalent force in X direction mit negative torsional loading
   !EYTP: equivalent force in Y direction mit positive torsional loading
   !EYTN: equivalent force in Y direction mit negative torsional loading

   In the process, the following combination loadings are also generated automatically:
   !GQ2_EXTP_P !GQ2_EXTP_N
   !GQ2_EXTN_P !GQ2_EXTN_N
   !GQ2_EYTP_P !GQ2_EYTP_N
   !GQ2_EYTN_P !GQ2_EYTN_N

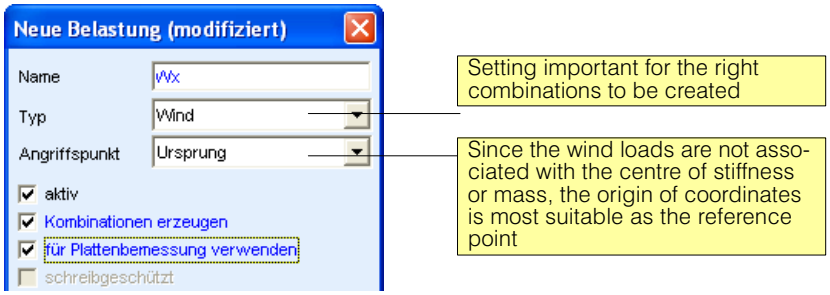
   One can check how these are defined in the Tab sheet 'Analysis'.

Further with Point 8 under 'Continuation for all loadings' .

In the case of wind and/or permanent earth pressure loads



5) In the Tab sheet 'Horizontal Loads' input the **wind loadings** (*normally this is one loading in each of the x and y directions*):.



The input of the load values and eccentricities is carried out in the Load Input dialog (see E 2.1.2).

For the verification, these loadings are entered automatically with both algebraic signs in the generated combinations. If, for example, the two loadings 'Wx' and Wy' are input, the program automatically creates the following 2 x 8 combinations:

for the loading Wx:

!GiQ_Wx0_P: $\gamma_{G,inf}$*G + γ_Q*Q + ψ_0 * Wx
!GiQ_Wx0_N: $\gamma_{G,inf}$*G + γ_Q*Q - ψ_0 * Wx
!GiQ0_Wx_P: $\gamma_{G,inf}$*G + ψ_0*Q + γ_Q * Wx
!GiQ0_Wx_N: $\gamma_{G,inf}$*G + ψ_0*Q - γ_Q * Wx

!GsQ_Wx0_P: $\gamma_{G,sup}$*G + γ_Q*Q + ψ_0 * Wx
!GsQ_Wx0_N: $\gamma_{G,sup}$*G + γ_Q*Q - ψ_0 * Wx
!GsQ0_Wx_P: $\gamma_{G,sup}$*G + ψ_0*Q + γ_Q * Wx
!GsQ0_Wx_N: $\gamma_{G,sup}$*G + ψ_0*Q - γ_Q * Wx

for the loading Wy: the corresponding 8 combinations

Loadings of the type **earth pressure** are assumed to be permanent loads. In the above combinations for the permanent loads G they are taken into account with the corresponding $\gamma_{G,Q,inf}$ and $\gamma_{G,Q,sup}$. Self-weight and earth pressure are entered together with γ_{inf} or γ_{sup}. If there are no wind loads we only have the combinations !GiQ and !GsQ.

Further with Point 8 under 'Continuation for all loadings' .

Continuation for all loadings

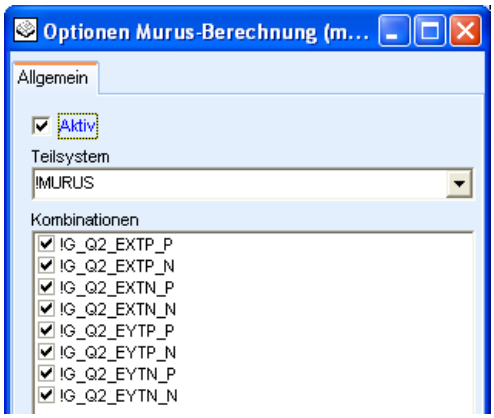
8) In the Tab sheet 'Analysis' open the Options dialog for Murus analyses and activate the following settings:

- Set Murus analysis to active

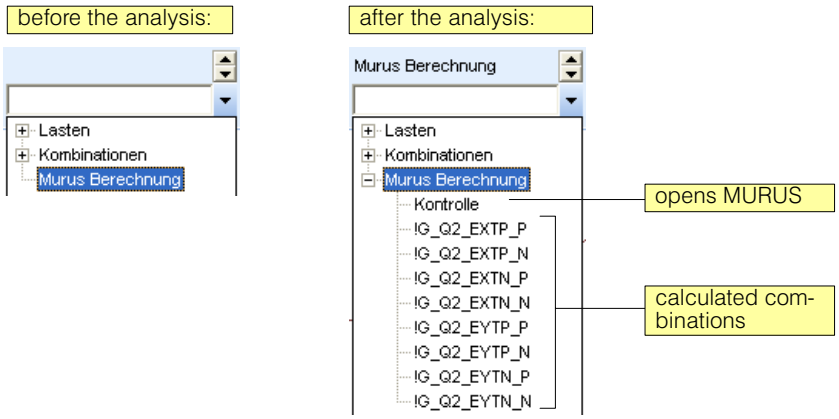
- Specify for which partial system (which walls) the analysis has to be carried out. The default partial system !MURUS includes all plane wall pillars with the material masonry.

- Select combinations to be considered.

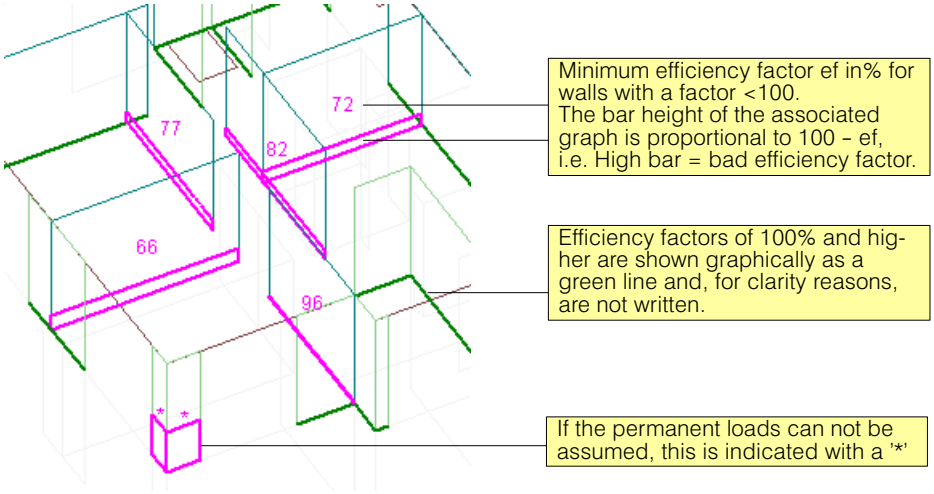
For earthquake analyses the dialog setting may look as follows:



9) In the Tab sheet 'Stacked Slab Model' set the Results selection field to Murus analysis (see figure) and start the analysis.



10) After the calculation, 'Control' is selected in the result tree, and when the adjacent layer switch is switched on, a summary result graphic appears with the following information:



Efficiency factors **ef** are used for loads, e.g. !GQ2_EXTP_P, which are divided into two parts by a '_' sign. The first part is regarded as a permanent load SL and the second part as a variable load VL. The efficiency factor is then defined as follows:

ULS = SL + **ef** * VL

(ULS = Ultimate limit state)

i.e. ULS = !GQ2 + ef * EXTP_P

ef is the factor by which the earthquake loads can be increased until the ultimate limit state safety is reached.

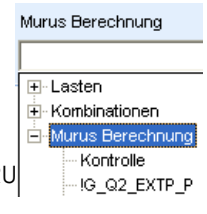
After the analysis the Results selection field is extended to include the following:

'Check': If one chooses this entry and clicks on the MURUS program is started. Imported are all the walls considered with their properties and the section forces from all combinations considered, so that all analyses can be reconstructed in detail. On the use of MURUS the user should consult the documentation.

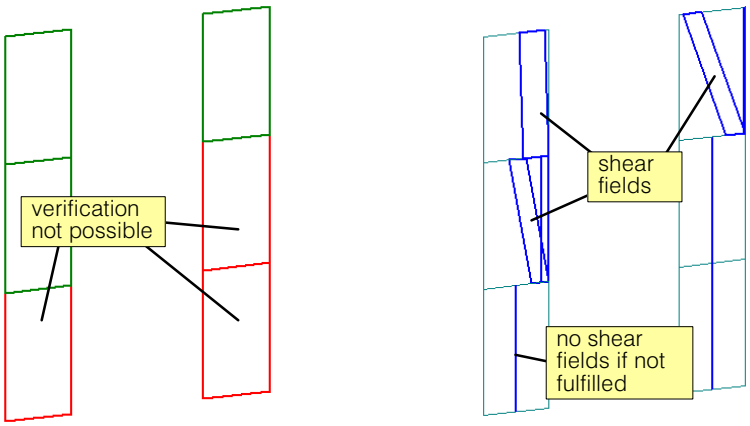
Calculated loading combinations: If a loading combination is selected, then in the Layer group there are two Layer buttons, which show the following results:

: All walls, for which the verification of the selected combination is not fulfilled, are marked red.

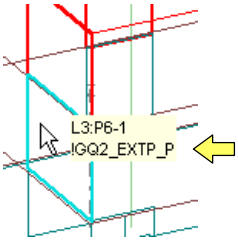
: For all walls for which the verification is positive the calculated shear fields are presented.



If the main branch of the 'Murus Analysis' is selected, then all those walls are marked red that fail in the case of at least one of the selected combinations. If one moves the cursor



over such a wall, then the unfulfilled combinations are shown in the Preselect Hint (if the layer is active):



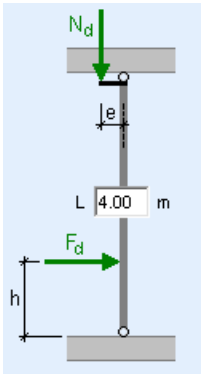
☞ *For a better overview, the program automatically generates the following subsystems when evaluating the results:*
- !MURUS-FAILED: all unfulfilled walls
- !MURUS_V: all MURUS-capable walls, which only carry vertical loads
- !MURUS_H: all MURUS-capable walls, which can also absorb horizontal forces

E 2.5 PYRUS calculation for the dimensioning of columns

E 2.5.1 Overview

With a PYRUS calculation the reinforced concrete columns of a building can be designed respectively verficated with the program PYRUS. This requires an PYRUS license. Observe the following points and conditions:

- Es wird mit dem Hochbau-Modus von PYRUS gearbeitet (siehe PYRUS-Handbuch), d.h. es gelten die dort genannten Voraussetzungen:



- prismatic column
- general cross section
- mounted as a pendular column
- vertical load N_d on top with optional eccentricity
- percentage of N_d is permanent and can creep
- optional impact load

  Frame columns, ie columns which take up larger moments, can not be treated directly. They must be analyzed in PYRUS as general columns.

- As normal force loads are carried at the stacked model normal force results are at the foot of the columns, usually !GSQ for the load combination. The normal forces from the combination !GQ2 (quasi-permanent combination) are assumed to be permanently acting, if this has been activated for the calculation.
- If impact forces are also to be tested, this can be specified accordingly. In such cases, a second design situation with the quasi-permanent normal force and the impact load is investigated as an extraordinary effect during the analyzes.
- Any support cross sections of reinforced concrete are supported. Concrete rolled profiles are also possible, whereby connection-specific aspects are not taken into account.
- An automatic grouping same supports for selectable normal force ranges reduces the computing and work effort.
- The analyzes carried out by the PYRUS procedures take into account second-order effects as well as material-induced non-linearities (cracked state), and the imperfections required by the standards are also taken into account. Details on the analyzes can be found in the PYRUS manual.
- Screen resolution must be at least 1024x768. A higher resolution is helpful.

⚠ Whether the stacked slab model is suitable for determining the normal forces in the columns must be assessed by the engineer in any case. The program offers a special safety factor for uncertainty in the determination of the normal forces, which can be selected individually for each column.

A PYRUS calculation is set up in the 'Calculation' tab (see chap. E 2.5.3) and the actual analyzes are carried out in the 'PYRUS Calculation' dialog in the 'stack of slabs model' tab (see chap. E 2.5.4). The steps of a PYRUS calculation are described by means of an example in chap. E 2.5.2.

E 2.5.2 Steps in a PYRUS-calculation

To get to know the way a PYRUS calculation works, we want to carry out a column design in a step-by-step example. The detailed description of all functions and buttons of the PYRUS calculation dialog follows in the following chapters.

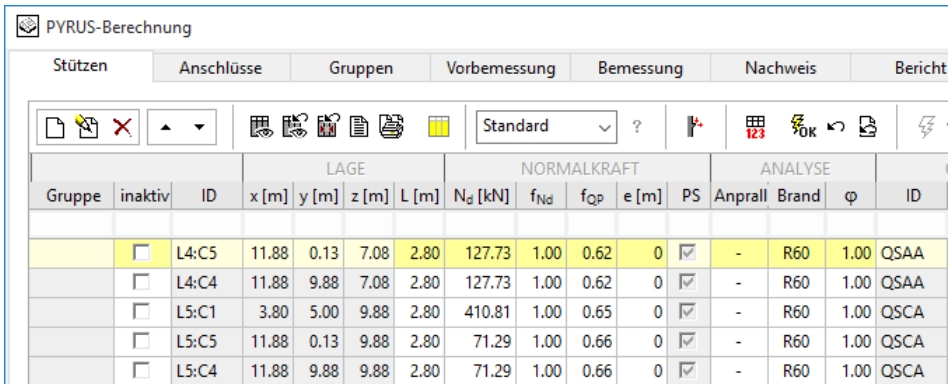
We load the example in CEDRUS via the menu 'Help > Load Examples > Building > Example SIA D0191'.

1. Step: set up a PYRUS-calculation

This is done in the tab 'Calculation' as described in chap. E 2.5.3 .

2. Step: open a PYRUS-calculation

Open the 'PYRUS Calculation' dialog in the 'Stack of slabs model' tab (see page E-29). When opening, the 'Columns' tab is displayed, in which all the columns found in the building are listed in tabular form (columns table). The dialog is hereinafter abbreviated as the PYRUS dialog. Which of the numerous properties are displayed can be set per register in a special dialog (see page E-32).



3. Step: Check / modify the column properties

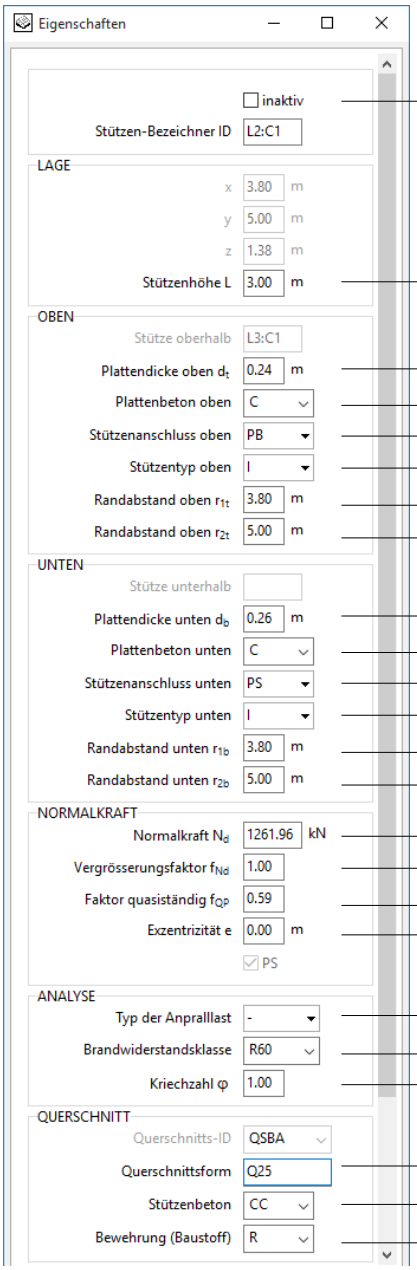
The column properties in the PYRUS dialog are distributed over the two tabs 'Columns' and 'Connections'. They are, if possible, taken over from the building model and can be overwritten for analysis purposes. The following points are to be mentioned:

- The PYRUS analyzes are based on the current data in the PYRUS calculation dialog. Since these can be overwritten, they do not have to correspond to the current building model.
- Overwriting is performed for individual cells by double-clicking on the corresponding cell or for a series of highlighted column rows by selecting 'Properties' from the context menu.
- The properties can be overwritten in the white columns of the tabs 'Columns' and 'Connections'.
- Overwritten properties are shown in bold.
- Overwritten properties can be reset to the original values from the building model for a marked range of columns.
- There is no backflow of overwritten properties into the building model.

☞ *If the table contains many columns, it is helpful to sort these by column contents (see page E-32).*

List of the columns properties:

The easiest way to display the list of columns properties is by means of the 'Properties' dialog, which is opened via the context menu of the support rows:



(M)=taken from building modell
(V)=Default value, not derivable from the building model

(V) for columns which are not to be analyzed

(M) Upper edge lower slab to lower edge upper slab

(M)

(M)

(M) 'PB' if it has a column above, otherwise 'P'

(M) Internal or border column, only used for verification of pressure in slab

(M) Distance columnaxis to edge in the direction of short column length

(M) Distance columnaxis to edge in the direction of long column length

(M)

(M)

(M) 'PA' if it has a column above, otherwise 'PS'

(M) Internal or border column, only used for verification of pressure in slab

(M) Distance columnaxis to edge in the direction of short column length

(M) Distance columnaxis to edge in the direction of long column length

(M)

(M) Ratio of N from combination '!GQ2' to N_d

(V)

(V) CEDRUS internal safety factor for inaccuracies in the building model *)

(V) Selectable are '–' for no 'F' or 'G' for the corresponding traffic area categories

(V) R60 od R90 acc. to SIA262

(V)

(M) Columns cross section determined by the program, regardless of orientation

(M) Selectable concretes from the building material table of the calculation

(M) Selectable reinforcements from the building material table of the calculation

The properties which can not be derived from the building model (marked with (V) in the figure above) must always be carefully considered.

For our example, we only want to make the following adjustments:

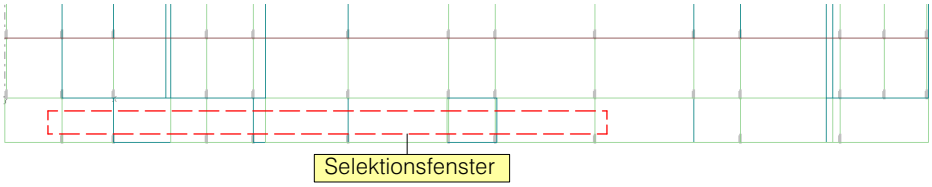
- The impact loads for the supports in the storey L1 in the area of the grid lines E to L (Type F) and
- Increase the enlargement factor f_{Nd} for all columns to 1.1.

*) ***To the enlargement factor f_{Nd}:***
As in determining the normal forces in the columns even with complex computer models always prevails (just think of the difficult to detect stiffness ratios of columns and slabs) a degree of uncertainty, the user, a safety factor is offered to increase the determined column loads for this reason. It is a program-internal factor that is not required by any standard.

Tip for the procedure:

(a) Change the projection direction of the building view to [Y] (the PYRUS dialog remains open)

(b) Select the columns in the required area by pulling a window from right to left:



(c) The column list in the PYRUS calculation dialog now shows only the selected columns. There should be 14, which are already selected (yellow background). Open the context menu (right-click in the PYRUS calculation dialog) and select 'Properties'. Set the impact load type to 'F', and then press [Apply].

(d) Close the Properties dialog, switch back to the previous, general projection direction and click on the graphic area to deselect the selection. All the columns in the PYRUS calculation dialog are now displayed again.

(e) Select all column rows in the PYRUS calculation dialog (button). Open the Properties dialog again in the context menu of the PYRUS calculation dialog and change the value for f_{Nd} to 1.1, then [Apply].

☞ *All changed properties should now be displayed in bold.*

☞ *You can reset the default values for selected column rows by clicking the button* ↶ .

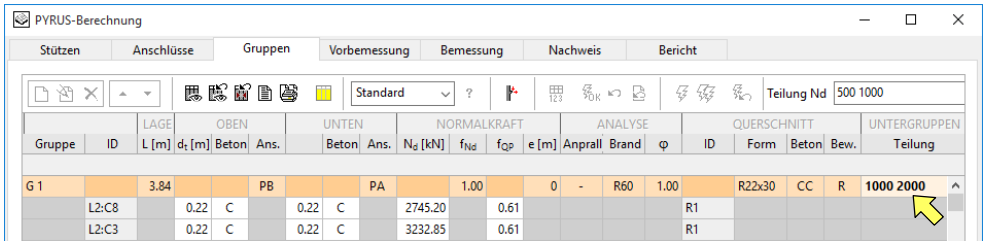
Also check the **settings in the tab sheet 'connections'** and leave it that way.

4. Step: Groupings

The concept of the grouping of columns for the dimensioning process is, above all, very important for larger projects. The design process becomes more efficient and clear, and the number of different column designs can be kept within reasonable limits.

Change to the tab sheet 'Groups'. Here, columns with the same properties are grouped together. Within a group, the supports differ only in respect of the normal force and their quasi-permanent part as well as the connections.

With respect to N_d, subgroups can be formed. This is done in the input field Teilung Nd 100 500 kN by entering an ascending list of limits for N_d. The limits entered here are the default for all groups. However, they can still be overwritten group by group (last column of the group rows):



Per group resp. subgroup assumes that all columns are executed the same, that is, only the support with the largest Nd is dimensioned and the result is for the entire (sub-)group.

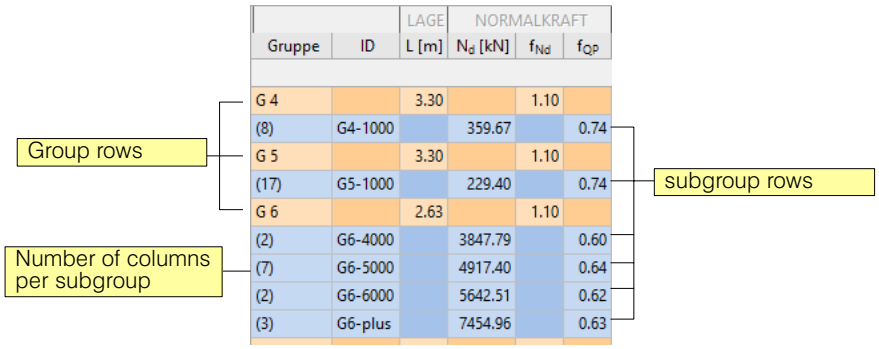
We set the following limits for our example, valid for all groups:
1000, 2000, 3000, 4000, 5000, 6000

With this button in the upper right of the dialog, the view is limited to the group lines.

5. Step: Design

Change to the tab sheet 'Design'.

Here, as in 'Preliminary design' and 'Verification', all columns are no longer displayed individually, but only the headlines of the individual groups are listed with their subgroup rows. Each group has at least one subgroup line, regardless of whether subgroups have actually been formed with normal force boundaries. The analyzes (preliminary design, design, verification) are always carried out for sub-group lines:



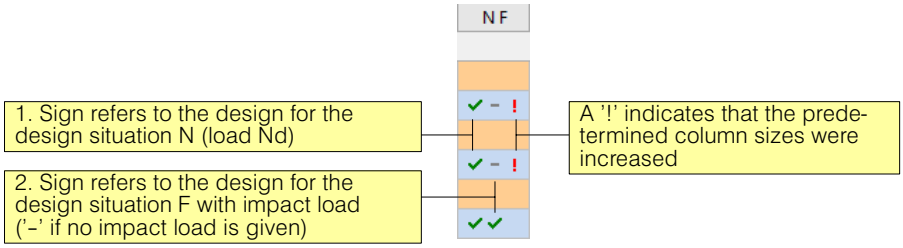
		LAGE	NORMALKRAFT		
Gruppe	ID	L [m]	N_d [kN]	f_{Nd}	f_{QP}
G 4		3.30		1.10	
(8)	G4-1000		359.67		0.74
G 5		3.30		1.10	
(17)	G5-1000		229.40		0.74
G 6		2.63		1.10	
(2)	G6-4000		3847.79		0.60
(7)	G6-5000		4917.40		0.64
(2)	G6-6000		5642.51		0.62
(3)	G6-plus		7454.96		0.63

Design all subgroups by clicking the adjacent button. The elaborate iterative design algorithms require some time, which is why they may only be pre-designed (neighbor register), or for only a few selected subgroups. The results are as follows:

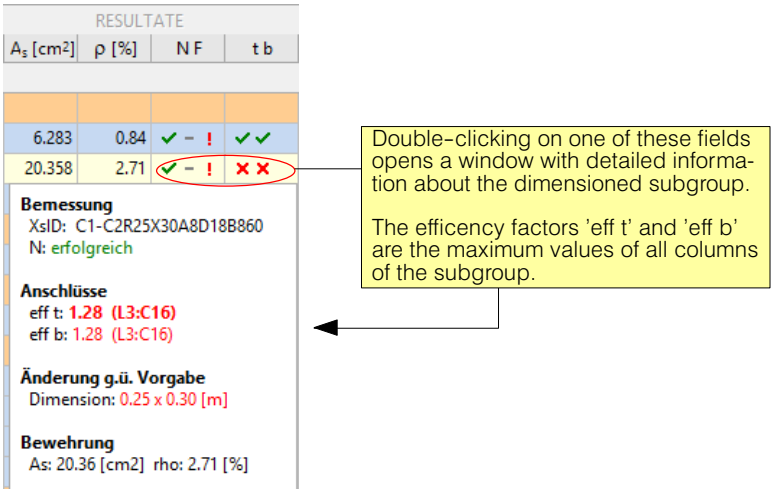
		LAGE	NORMALKRAFT				ANALYSE	RESULTATE					
Gruppe	ID	L [m]	N_d [kN]	f_{Nd}	f_{QP}	Form	XsID	f_D	A_s [cm²]	ρ [%]	N F	t b	Ana.
G 4		3.30		1.10		R22x30							
(8)	G4-1000		359.67		0.74		C1-C2R25X30A		6.283	0.84	✓✓ !	✓✓	B
G 5		3.30		1.10		R22x30							
(17)	G5-1000		229.40		0.74		C1-C2R25X30A		6.283	0.84	✓✓ !	✓✓	B
G 6		2.63		1.10		R25x35							
(2)	G6-4000		3847.79		0.60		C1-C2R25X35A		72.634	8.30	✓✓	✓ –	B
(7)	G6-5000		4917.40		0.64		C1-C2R25X45A		108.950	9.68	✓✓ !	✓ –	B
(2)	G6-6000		5642.51		0.62		C1-C2R25X45A		150.796	13.40	✓✓ !	✓ –	B
(3)	G6-plus		7454.96		0.63		C1-C2R25X60A		150.796	10.05	✓✓ !	✓ –	B

1. view

The first view is the column 'NF', which indicates whether the column dimensioning of all subgroups was successful:



If the designs were successful ('✓'), the results obtained must be checked. In the normal case, the subgroups include a plurality of supports (as seen from the bracketed number in the first column), i.e. the results shown here are the effective value of a subgroup. If you want to see the results of one or more subgroups individually, you can select the subgroup(s) and select the command 'Column in the tab <Columns> / <Connections> show exclusive' in the context menu. Detailed results for the main column of a subgroup can also be obtained by double-clicking on one of the fields in the columns 'NF' or 'tb':



Cross-sectional dimensions: (Changes only possible for cross-sections of the catalog)
If much larger cross-sectional dimensions are obtained than is foreseen, this will be investigated further. The change in the cross-sectional dimensions can be seen on the sign '!', which appears in column 'NF'.

Reinforcement (Column 'ρ[%]'):
In the case of very high reinforcement values, it should be checked whether this is executable (if the catalog cross sections are used) and also economical. In order to find the critical reinforcement contents immediately, the reinforcement contents can be made visible in the Columns register and sorted according to these.

Connections:
In the table above it is noticeable that the connections are not all ok ('x' in the column 'tb'). To see how the efficency factor of the individual columns of certain subgroups look, you can mark them and then select them in the context menu 'Exclusive display columns in tabsheet 'Connections'.

To adjust a connection, you have the following options:

- The normal force is transferred directly to the next column by means of structural measures
- A steel plate, which protrudes by a few centimeters
- Increase the concrete quality of the slab
- Increase the cross-section of the column
- A refined verification method

If you would like to know more about the analysis of a subgroup, you can call up the PYRUS program at any time, which is initialized with the data of the main support of the subgroup.

6. Step: Documentation

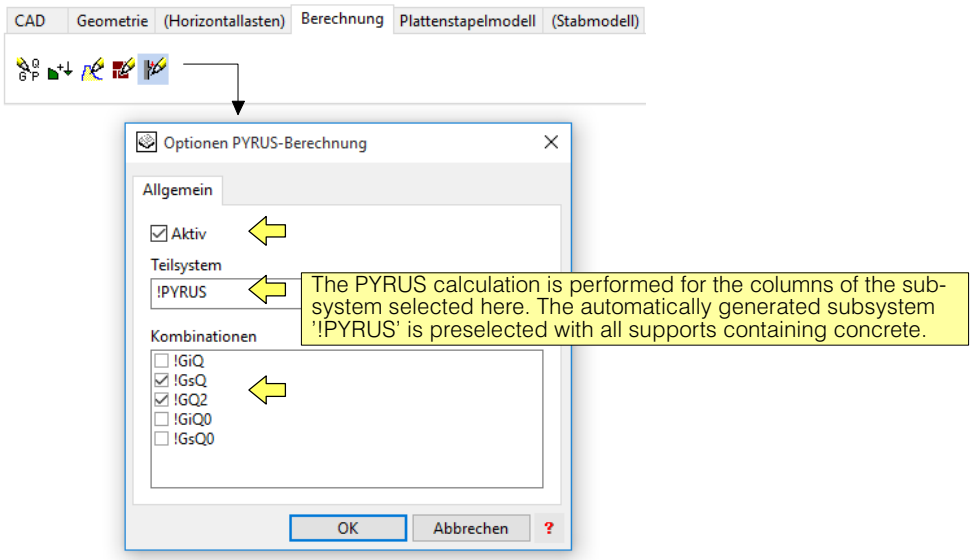
Once the dimensioning process has been completed, the results should be documented. This is advantageously done in the tabsheet 'Report' (see page E-38).

E 2.5.3 Setting up a PYRUS calculation

The dialog 'Options PYRUS-analysis'

The dialog is called via the button shown in the calculation register. It is used to activate the PYRUS calculation and to select the load combinations to be considered (see chap. E 2.5.1 for an overview of the PYRUS calculation).

Before creating a PYRUS calculation, the geometry model must always be created first!



If only vertical loads are considered, then the combinations are normally

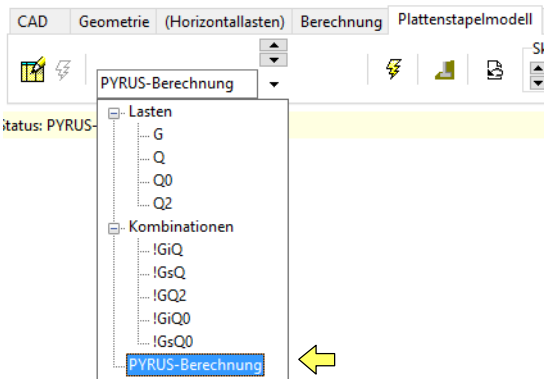
- !GsQ (γ_G * permanent loads + γ_Q * changing loads) and
- !GQ2 (1.0 * permanent loads + ψ_2 * changing loads)

selected. If horizontal loads such as wind or earthquakes are also specified, the combinations generated for this can be switched on additionally. Information on such loads can be found in chapter E 2.4 on the MURUS calculations.

E 2.5.4 The dialog 'PYRUS Calculation'

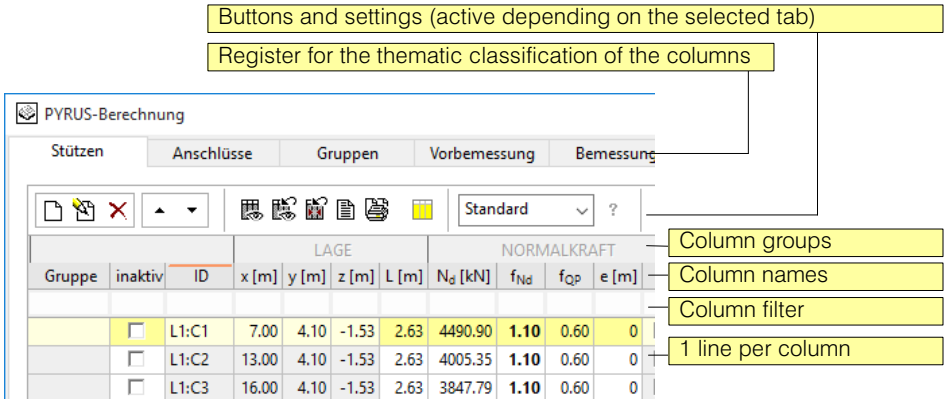
Open dialog

If the 'Active' field is activated in the options dialog described above, the entry 'PYRUS Calculation' appears in the 'Combinations' list box in the 'stack of slabs model' tab:



If this entry is selected and the flash button is pressed, the 'PYRUS-Calculation' dialog of all the columns of the building is listed in tabular form, and all PYRUS analyzes are performed.

Overview of the dialog



			LAGE				NORMALKRAFT			
Gruppe	inaktiv	ID	x [m]	y [m]	z [m]	L [m]	N_d [kN]	f_{Nd}	f_{QP}	e [m]
	☐	L1:C1	7.00	4.10	-1.53	2.63	4490.90	**1.10**	0.60	0
	☐	L1:C2	13.00	4.10	-1.53	2.63	4005.35	**1.10**	0.60	0
	☐	L1:C3	16.00	4.10	-1.53	2.63	3847.79	**1.10**	0.60	0

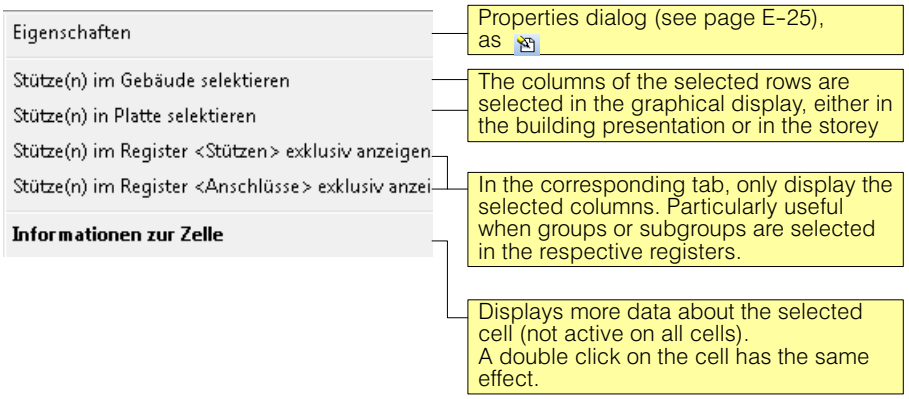
The work sequence corresponds in principle to the arrangement of the register sheets. For reasons of clarity and space, the visibility of the columns is adapted to the respective tab sheet, ie the working step.

The individual tabs is shown below each one chapter dedicated.

General functionalities of the dialogue

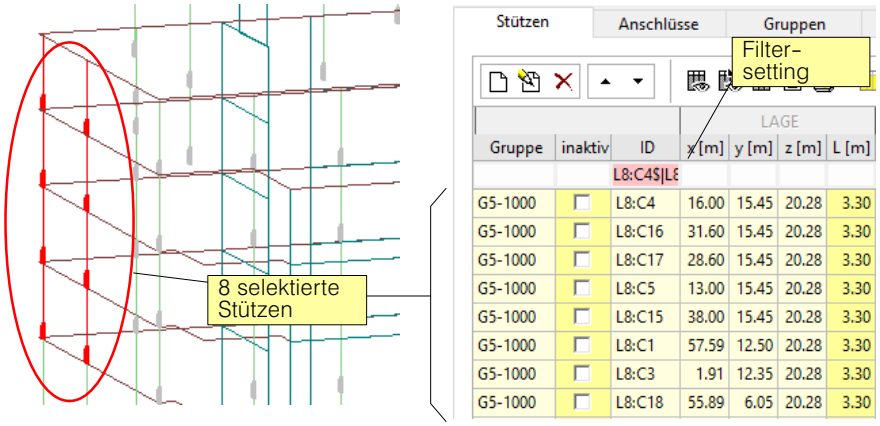
The following functions are only available in certain tab pages.

- The context menu for selected column rows:



- White fields in the support table can be changed, either individually by double clicking on an input field or via the property dialog (then simultaneously for several selected columns).
- The dialog is not completely re-generated every time it is opened, but it is attempted to transfer its contents from earlier workings as far as possible and to make changes that result from modifications to the building model visible. You can also see which values you have overwritten as a user yourself. This is achieved by various display styles of the table fields (see page E-33).
- The column list can be sorted according to the values in a column by clicking on its head (up / down by clicking again). A red marking in the column header indicates an appropriate sort.
- The display of the column lines can be limited by filter criteria (see page E-32)
- If columns are selected in the building graphics resp. if an existing selection is changed, the dialog is adapted as follows:
  - If a column is selected, the entire table is displayed, and the selected table is displayed Support is marked (highlighted in yellow)
  - If several columns are selected, only these are displayed in the dialog

- If no column is selected, the entire table is displayed.
☞ A partial view of the table is implemented via the filter function (see page E-32). By deleting the filter settings, the entire table can be made visible again without having to deselect the selection in the graphic.



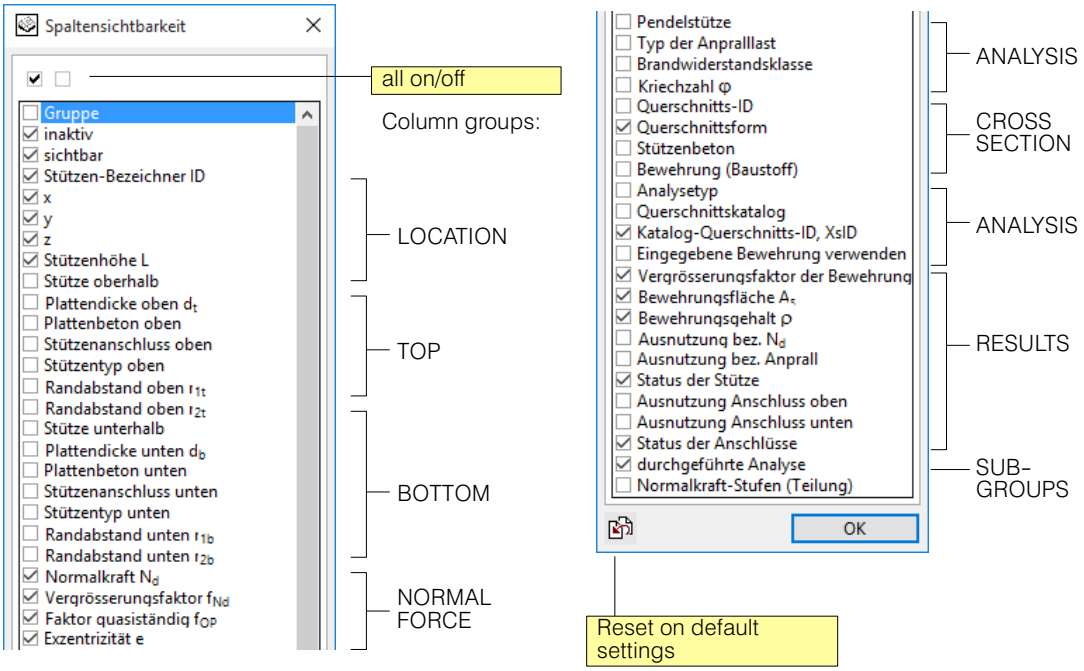
- This can be used to add and remove new column rows, which do not originate from the building model.
- Shows the properties dialog for the selected rows (see page E-25).
- The selected lines are moved one position up / down
- The visibility of the columns can be individually configured for each tab page (see page E-32).
- The visibility of the columns can be individually
- Reset column visibility to default values
- Creates a printable table of the column list with columns visible in the tab (filter settings are taken into account).
- Print entries of the analysis results for the selected columns resp. subgroups in the analysis registers (see page E-39).
- Selects all rows of the column list. This can also be achieved with the key combination <Ctrl + A>, if the column list has the focus.
- Choice of columns-catalog for columns with catalog of cross-sections. The [?] Button provides information about the selected catalog (Internet link).
The catalog selection is primarily relevant in the design registers and is not available in the register of verification. In the first three registers, its only important when calling PYRUS in the case of columns which have not yet been dimensioned with cross-sections.
- Calls the PYRUS program and initializes it with the data of the selected column row.
- Only changed rows are displayed.
- The changes coming from the building model (shown in red) are confirmed (again black) for the selected lines.
- Reset to default values for the selected rows. *Caution: all properties of a column are reset, not only those that are visible in the current tab.*
- Resetting the entire PYRUS dialog. The dialog is then closed. When reopening, it behaves as if it were opened for the first time.

Only in the analysis registers:

- Perform analysis for selected resp. all subgroups.
- Reset the analysis state of the selected subgroups to *not analyzed.*

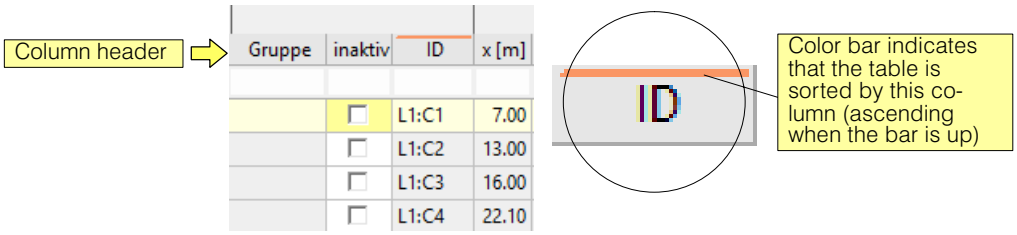
Set column visibility

This is done with the button , which displays the following dialog with all available columns:



Sort the column list

Clicking on a field in the column header will move the entire table to ascending or descending values (changes each time you click) in the corresponding column.



Sort by several columns

If you hold down the <Shift> key when clicking on a column header, an existing sorting criterion remains as a primary sorting criterion. For example, you might want to sort a table sorted by the 'Form' column by ID's. This makes sense only if the primary sort is not unique. This means for the above example that there must be several lines with the same 'form'. If the sorting is still not clear, additional columns can be sorted.

Column list filter

If a text is entered into an input field of the filter line, only those lines of the column table which contain this text are displayed.

☞ *Strictly speaking, the filter text is interpreted as a regular expression (see, for example, at Wikipedia), but this is of little importance to the user here.*

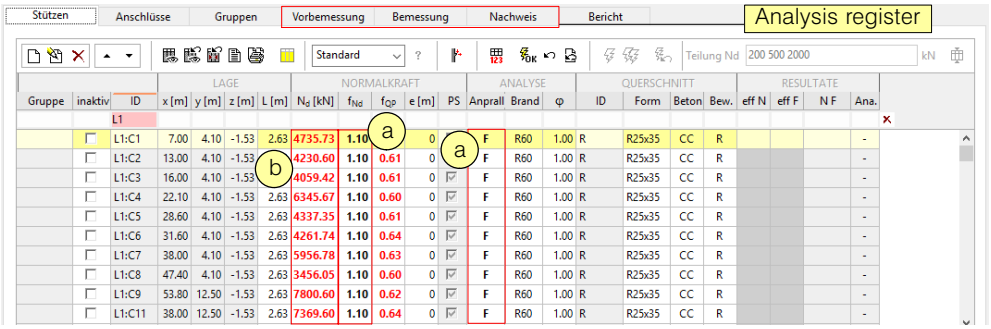
View by changes

Changes to the contents of the column list can have the following reasons:

- The user modifies input fields in the column list.
- The data changes as a result of modifications on the building model.
- Result fields in the column list change as a result of an analysis in the dialog (design or verification).

As you work through the dialogue step by step, this is not necessary to be generated completely every time you open it, but you keep your entries as much as possible and you should be able to see what you have changed. Which resulted as a result of modifications to the building module. In order to achieve this, the following presentation concept is provided:

Values modified by the user



They are displayed in black and bold (a) and are retained until they are reset with this button for the selected rows.

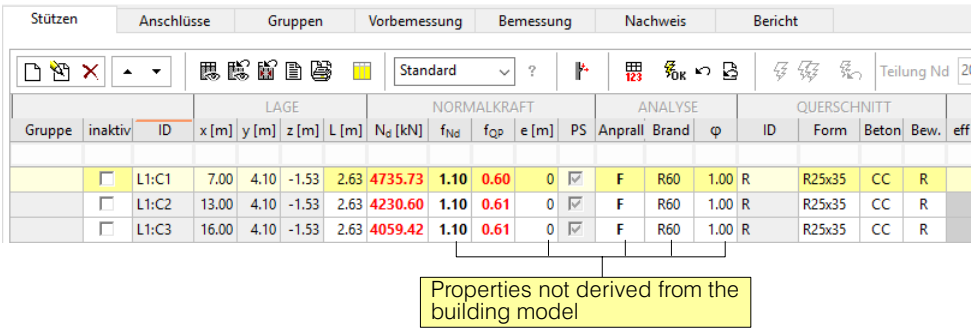
User-modified values (e.g. N_D) are also retained if new values result from a modified building model. A reset would then switch to these new values.

Through modifications to the building model changed values

They are shown in red and bold (b).

The representation is retained until it is transferred to the normal representation for the selected rows with this button. Modified table values after modifications to the building model would then be red again.

Tab sheet 'Columns'



The most important column data are presented here. The list must be checked and supplemented, in particular the characteristics which can not be derived directly from the building model are to be considered.

The selection field Standard is only relevant here if the PYRUS program is called directly with and the column is not yet dimensioned.

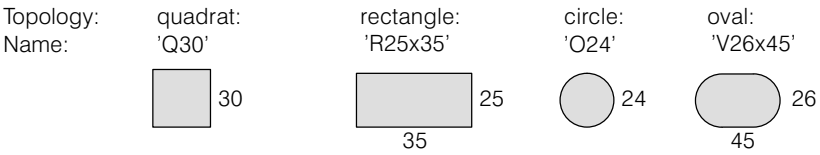
Here, results (column 'NF') of the most recently performed analysis (column 'Ana.') Can also be viewed individually for each column, which is only possible for entire subgroups in the analysis registers.

To the column 'Topology':

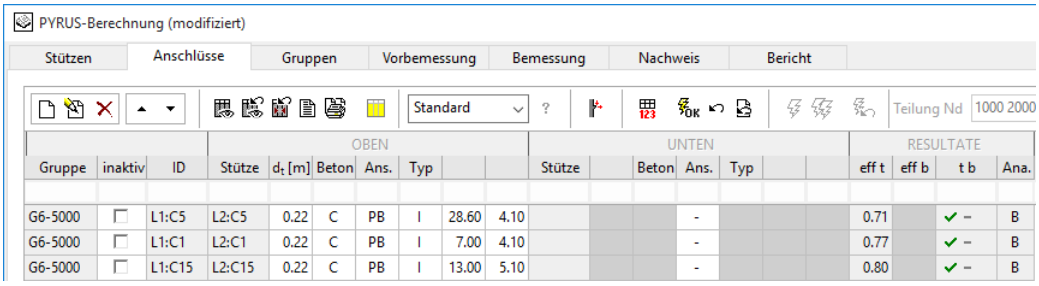
When dimensioning, a distinction is made between cross-sections taken from a catalog and general cross-sections (see below for the tab 'Design'). When the column list is created, column cross-sections, which are recorded in catalogs, are introduced as catalog forms. In the current program version, this is a square, rectangular and circular column cross-section, and the entry in the Column column is then called e.g. 'R25x35' for a rectangular cross-section. Cross-sections, whose type is not included in the catalog, are of the 'general' type and their name in the column Form begins with 'G:'.

The fields of the 'Topology' column are modifiable. The entry can either be overwritten with a catalog-compatible name (for example, an 'R22x30' with 'R25x35'), or a cross-section can be selected from the opening selection list. This list contains all column cross sections occurring in the building calculation.

Examples of catalog-compatible names (all dimensions in cm):



Tab sheet 'Connections'

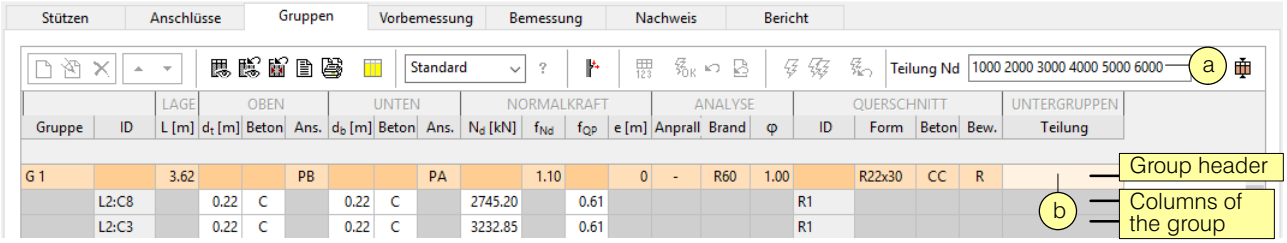


The data required for the verification of the concrete pressures in the concrete slabs at the top and at the bottom are taken from the building model, in particular the connection type has to be checked. This tab sheet is separated from the sheet 'Columns' only for reasons of space. The general functionalities are the same.

Here, results (columns 'eff t', 'eff b' and 'tb') of the most recent connection analysis (column 'Ana.') can be viewed individually for each column, which is only possible for entire subgroups in the analysis registers. 'eff t/b' are the utilization values at the upper and lower connection.

For detailed information, in particular also to columns $r_{1T/B}$ and $r_{2T/B}$, see the PYRUS manual.

Tab sheet 'Groups'



As a preparation for the analysis, columns with the same characteristics (except N_d and f_{QP}) are automatically grouped together. In addition, a subgroup can be made on the basis of the normal forces.

The limits for the standard subgroups can be defined in two places:

(a) The entries in this field are the default values for all columns.

(b) The default values for each group can be overwritten in this field of the group rows.

Example: The entry '1000 2000' leads to the three subgroups with the standard force ranges 0 ... 1000, >1000 ... 2000, >2000.

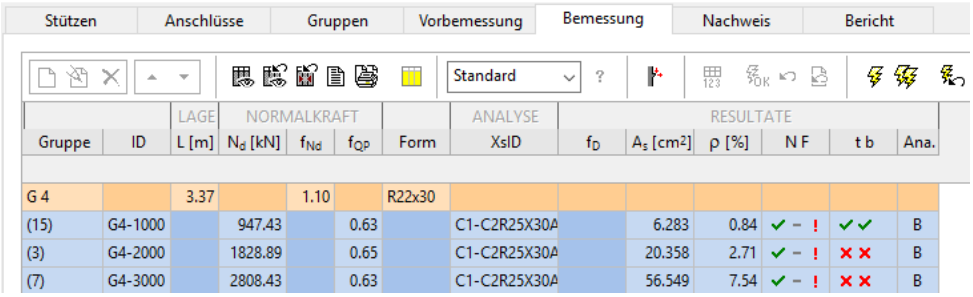
If no normal force limits are specified, a subgroup with the normal force range >0 is generated automatically, since the analyses (see below) are always made for subgroups.

It is assumed that all columns of a subgroup are executed the same, and the analyzes are made only for the column with the maximum normal force. This corresponds, on the one hand, to the requirements of practice and, on the other hand, reduces the calculation effort for the analyzes.

The tabular display lists all associated columns for each group. If this button is pressed (at the top right of the 'Groups' tab), the column rows are dimmed.

☞ *If you want to undo the default subgroup for a group, you can enter a correspondingly large standard value in the 'Distribution' field of the group line.*

Tab sheets 'Preliminary design' and 'Design'



Since the design is per subgroup (see above), only the groups with their subgroup rows are listed here. In both tabs looks the same can be designed per subgroup of the column cross-section. The difference is the following:

Design: The program determines the optimal cross sections of the cross-section catalog used (dimensioning strategy and optimization criterion described in the PYRUS manual) for cross-sections of the catalog and the necessary cross-sections for general cross-sections. Since this process is computer-intensive, it can take a considerable amount of time on many supports. For this reason, there is the possibility of a faster preliminary design described below.

☞ *Column cross-sections are either catalog cross-sections or of the 'general' type. This can be seen from the description in the column 'Topology' of the support list (see page E-34).*

Preliminary design: In order to be able to save computing time in an early stage of the project, there is the preliminary design. The iterative design process is abbreviated by checking, in the case of general cross-sections, whether successful verification is possible with the maximum possible reinforcement content. For catalogs only the cross sections with the maximum reinforcement are examined.

Functions in the tab sheet

In addition to a part of the functions already described, the following tools are available in these two register pages:

In this selection field, the cross-section catalog to be used for catalog cross-sections is set.

The selected subgroups are designed.

All subgroups are designed.

The analysis state of the selected subgroups is reset to *not analyzed*.

A print entry with the detailed design results for the selected subgroups is generated (for details see tab sheet 'Report').

Results

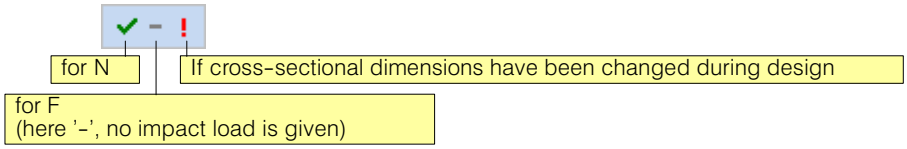
XsID — Shows the name of the found catalog cross-section. XsID is a unique identification that completely describes a cross-section (i.e., geometry, reinforcement and, depending on the catalog, also building materials).

f_D — Shows the factor with which the starting values of the reinforcements to be dimensioned must be multiplied (for details see FAGUS manual).

A_s [cm²] — Total cross-sectional area of the reinforcement

ρ [%] — Geometrical reinforcement content

N F — Status of the column analysis for the most recently executed analysis (the analysis type is shown in the column on the right).

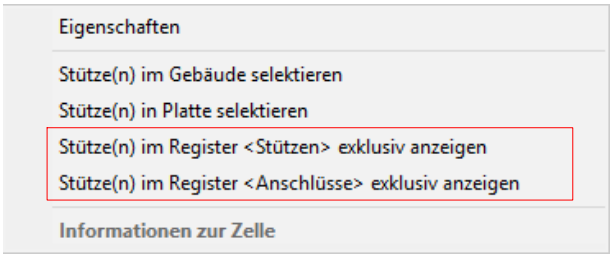


('✓' = successful, 'x' = not fulfilled, '-' = outdated, empty = not carried out the tab corresponding analysis).

t b — Status of the verifications for the connections of the column/slab (analog to the NF status)

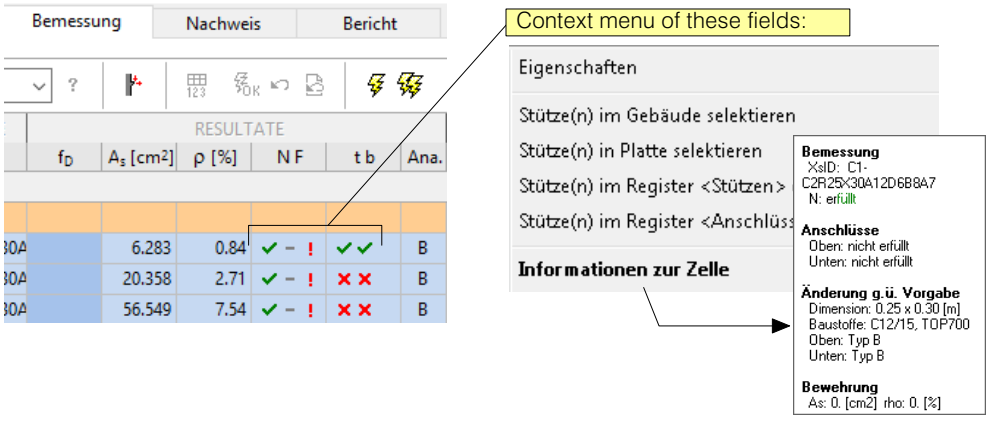
Ana. — Indicates which type of analysis was last carried out, and thus on which analysis the results are based (<V> Preliminary design, <B> design, <N> verification).

☞ *If you want to have information from a unfulfilled subgroup about each participating column, so the subgroup lines can be selected and one of the following commands can be selected in the context menu,*

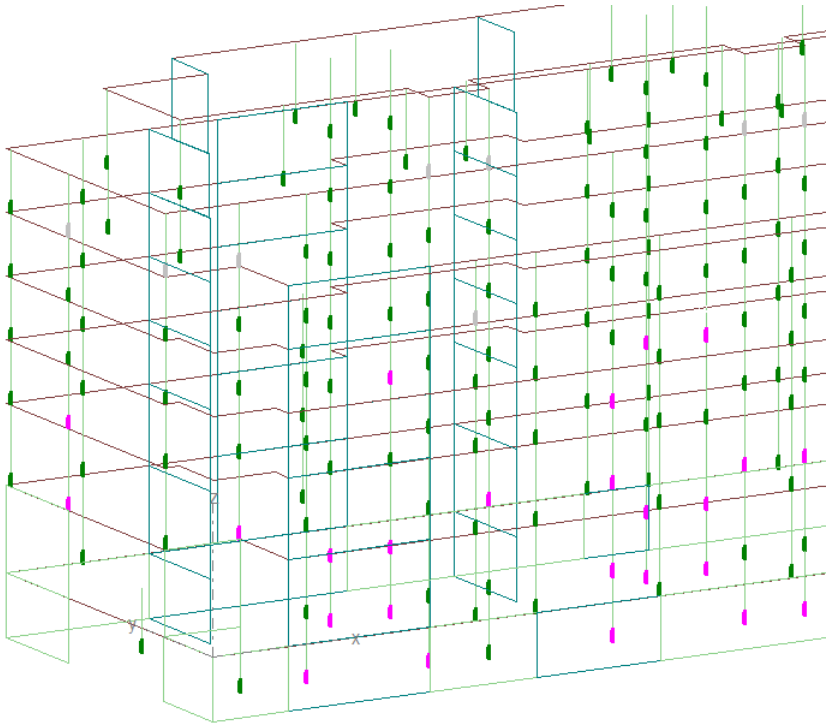


just exactly display the columns of the selected subgroup in the relevant register. It is also possible to treat several subgroups together in this way. The display of the subgroups in the building graphic can also be accessed via the context menu.

A summary of the most important results can be obtained by double-clicking on the fields in the columns 'NF' and 'tb' resp. with 'information about the cell' from its context menu:

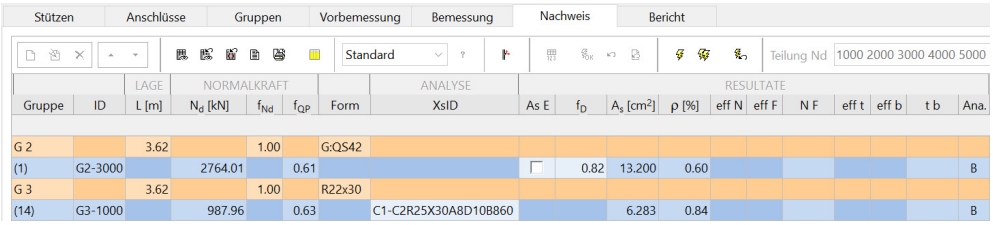


In the graphics area, the analyzed columns are colored: green for successful and red for unsuccessful analyzes:



☞ *Remember that when you select a column in the graphic area, the corresponding row in the column table is marked and immediately visible.*

Tab sheet 'Verification'



		LAGE	NORMALKRAFT				ANALYSE		RESULTATE								
Gruppe	ID	L [m]	N_d [kN]	f_{Nd}	f_{QP}	Form	XsID	As E	f_D	A_s [cm²]	ρ [%]	eff N	eff F	N F	eff t	eff b	t b	Ana.
G 2		3.62		1.00		G:QS42												
(1)	G2-3000		2764.01		0.61			☐	0.82	13.200	0.60							B
G 3		3.62		1.00		R22x30												
(14)	G3-1000		987.96		0.63		C1-C2R25X30A8D10B860			6.283	0.84							B

The utilization for the supports and their connections (ok, if ≤ 1.0) is verificated for each subgroup line.

In columns with **catalog cross sections**, the reinforcement is clearly defined. Concerning materials, there are catalogs in which these are fixed and those with freely selectable materials, i.e. it can then be used those which are set in the column table.

In the case of supports with **general cross sections**, the reinforcement can be fixed in two ways. If the selection field is activated in the 'As E' column, exactly the entered reinforcement is used without any magnification factors. If the selection field is switched off, all fixed reinforcements are used as well as the starting values of the reinforcements to be dimensioned multiplied by the factor fD (neighboring column) (see FAGUS manual).

Functions in the tab sheet

In addition to a part of the functions already described, the following tools are available in these two register:

The selected subgroups are verificated.

All subgroups are verificated.

The analysis state of the selected subgroups is reset to *not analyzed*.

A print entry with the detailed results for the selected subgroups is generated (for the range, see below for the tab 'Report').

Results

eff N Efficency factor for the design situation normal force. The result value indicates how much the variable fraction of N_d (= N_d * (1-f_{QP})) can be increased until the column fails.

eff F Efficency factor for the design situation, Which also contains the impact load as an extraordinary action. The output value indicates how much the impact load can be increased until the column fails.

N F Status of the column verification ('✓' = successful, 'x' = not fulfilled), if the last analysis was a verification.

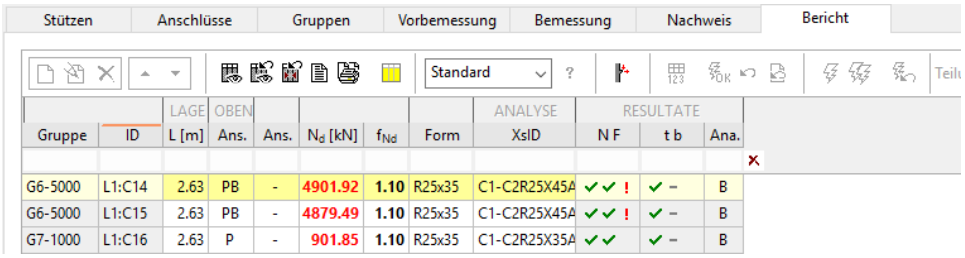
eff t | **eff b** Efficency factor of the upper and lower connection.

t b Status of the verification for the connection column-slab. The view corresponds to that of the NF-status.

Ana. Indicates which type of analysis was last performed and thus on which analysis the results shown are based.

'Informationen about the cell' from the context menu is available for the values in the columns 'NF' and 'tb' (as in the case of the design).

Tab sheet 'Report'



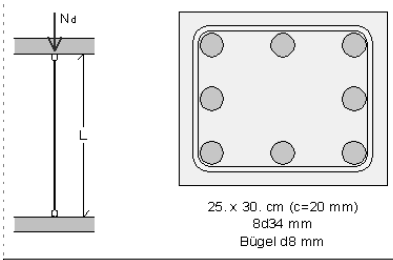
		LAGE	OBEN					ANALYSE	RESULTATE		
Gruppe	ID	L [m]	Ans.	Ans.	N_d [kN]	f_{Nd}	Form	XsID	N F	t b	Ana.
G6-5000	L1:C14	2.63	PB	-	4901.92	1.10	R25x35	C1-C2R25X45A	✓✓ !	✓ –	B
G6-5000	L1:C15	2.63	PB	-	4879.49	1.10	R25x35	C1-C2R25X45A	✓✓ !	✓ –	B
G7-1000	L1:C16	2.63	P	-	901.85	1.10	R25x35	C1-C2R25X35A	✓✓	✓ –	B

The documentation of the column analyzes is advantageously carried out in the tab 'Report'. There are two options for printable output:

Comprehensive data per column

Print entries with comprehensive data and results per column for the last performed analysis. They are made with the button shown left for the selected columns and look something like this:

Stütze L3:C15



25. x 30. cm (c=20 mm)
8d34 mm
Bügel d8 mm

Geometrie:	L	= 3.37 m
Imperfektion:	e_{0z}	= 9.0 mm
	e_{0y}	= 8.4 mm
Belastung:	N_d	= -2664.7 kN
	f_{QP}	= 64.8 %

Anschlüsse:	Typ	Platte	Position
Oben:	PB	d_{f1} = 0.22 m Beton C(C30/37)	Innenstütze
Unten:	PA	d_{f1b} = 0.22 m Beton C(C30/37)	Innenstütze

Bemessung erfolgreich

Norm	SIA (2003)
Analyseparameter	IGZN
Kriechparameter	1.
Grenzdehnungen	-2.0, -3.0, 20.0 [‰]
Teilsicherheitsbeiwerte	γ_c = 1.50, γ_s = 1.15
Brand	R60

Querschnitt:	
Katalog	Standard
XsID	C1-C2R25X30A8D34B8A7
Baustoffe	CC , R
As , p	72.63 cm² , 9.68 %

Bemessung N	erfüllt

Anschlüsse:	Ausnutzung	Details
Oben:	eff_t = 0.98 (<=1 erfüllt)	Ac1 = 47.x52. cm k_c = 1.81 R_u = 2707.77 kN
Unten:	eff_b = 0.98 (<=1 erfüllt)	Ac1 = 47.x52. cm k_c = 1.81 R_u = 2707.77 kN

Output of the support table

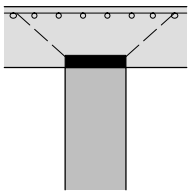
The printable table of the column list with the columns visible in the tab page provides a concentrated overview of the results of the analysis of all columns. (Filter settings are taken into account).

An output table is also available in other tab pages. In 'Report', however, you can configure it specifically for the results documentation by selecting the appropriate column separations without having to re-define the appropriate settings for other tab pages. Also think about the other ways to design the table, such as:

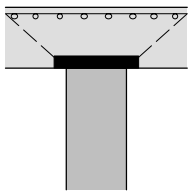
- Sort by one or more columns (see page E-32)
- Filter settings (see page E-32)

E 2.5.5 Columns and punching shear

As the danger of punching in buildings mainly depends on columns, the punching and column verifications are connected together. The decisive parameters, however, are for the most part different, which is why it is not useful to include the punching analyzes (see chap. B 9), which are treated in the storeys, in the support table. However, the following figure shows the extent to which the design of the column connection can have an influence on the punching verification:



Weakening of the slab by an inner connection plate

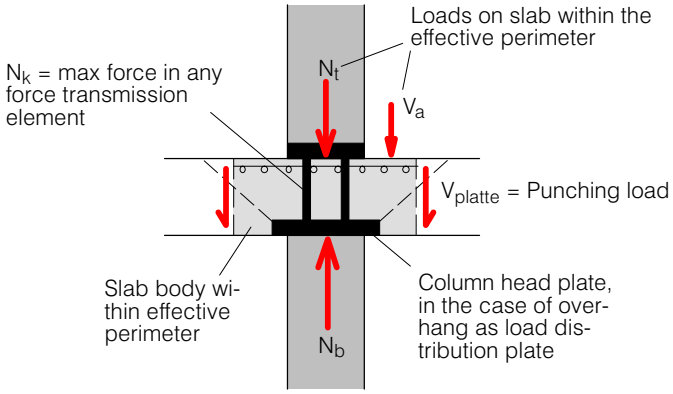


Weakening of the slab, enlargement of the effective perimeter

For the corresponding inputs for the punching shear objects, see chap. B 9.2.

Force flow through slab with continuous columns

In the case of continuous columns, the flow of the force is as follows:



N_t	Corresponds to the force determined in the stacked slab model for the upper column
N_b	Corresponds to the force determined in the stacked slab model for the lower column, minus the dead weight of the column
N_b-N_t	This difference corresponds to the load that comes from the slab.
N_b-N_t-V_a	Corresponds to the punched load.

The verification of permissible concrete pressure over the head plate shall be provided for N_b-N_t.

E 2.6 Berechnung von Stahlbetonwänden

Das Gebäudemodell von CEDRUS ermöglicht die Bemessung der Bewehrung aller Geschosswände aus Stahlbeton. Um eine wirtschaftliche und praxistaugliche Wandbewehrung mit wenig Benutzeraufwand ermitteln zu können, sind in CEDRUS verschiedene Berechnungsverfahren implementiert. Diese Verfahren und ihre Anwendung werden nachfolgend erläutert.

Zur Bemessung der Bewehrung von Flächentragwerken wie Platten und Scheiben bietet sich die Finite-Elemente-Methode (FEM) an, wo die konkreten Bewehrungslagen aus den ermittelten, elastischen Bewehrungsgehalten konstruiert werden können. Die FE-Methode ist in CEDRUS sowohl für Platten als auch für Scheiben (Option 'S') implementiert.

CEDRUS (S)-Berechnung von Wandscheiben mittels FEM

Wie in E 2.8 beschrieben, kann mit der sog. CEDRUS (S)-Wandscheibenberechnung die erforderliche Bewehrung eine Gebäudewand mit beliebigen Öffnungen ermittelt werden. Im Gegensatz zu Stahlbetonplatten, wo sich die FEM zur Ermittlung des erforderlichen Bewehrungsgehaltes im Praxiseinsatz durchgesetzt hat, führt die FE-Bemessung der Wandbewehrung bei der einfachen Rechteckwand oft nicht zu befriedigenden Bewehrungsanordnungen. Dies liegt daran, dass der in der FE-Berechnung vorausgesetzte elastische Spannungszustand sich sehr stark vom plastischen Spannungszustand einer realen Stahlbetonscheibe unterscheidet (Rissbildung, Umlagerungen im Grenzzustand der Tragsicherheit). Dies gilt insbesondere für höher beanspruchte Erdbebenwände.

Alternative Verfahren für Geschosswände

Die Geschosswände in einem Gebäude sind spezielle, ebene Flächentragwerke, welche nur Kräfte in ihrer Ebene aufnehmen (d.h. ebener Spannungszustand). Die meisten Geschosswände sind rechteckig und oben und unten fest mit Stahlbetonplatten verbunden. Eine praxistaugliche Bewehrung einer solchen, einfachen Geschosswand weist i.d.R. eine konstante, vertikale und horizontale Bewehrung auf. Einzig für hochbelastete Erdbebenwände muss am Wandende jeweils eine zusätzliche, konzentrierte (Knoten-)Bewehrung eingelegt werden.

In einem Gebäude weisen viele Wände Öffnungen auf, was insbesondere für Aussenwände gilt. Sind die Öffnungen gross, so werden die vertikalen Lasten im Grenzzustand der Tragsicherheit nur von den Wandbereichen zwischen den Öffnungen abgetragen. Damit müssen für die Wandbemessung nur diese Bereiche als Geschosswände im Gebäudemodell modelliert werden (s. Bild unten).

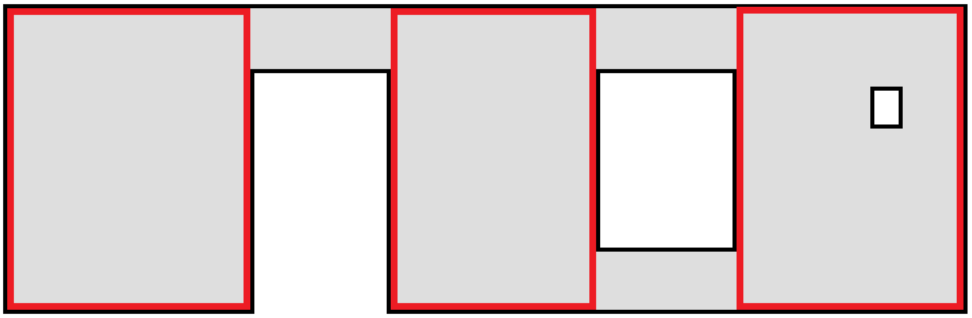
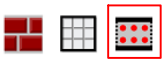


Fig. E–1 *Wand mit Öffnungen und entsprechende Geschosswände für Berechnungsmodell*

Für die Bemessung dieser einfachen Geschosswände bietet CEDRUS alternative Verfahren an, welche mit weniger Benutzeraufwand zu wirtschaftlicheren Ergebnissen und direkt zur Bewehrungsanordnung führen.

FAGUS (A)-Berechnung als Biegestab oder Spannungsfeld



Alternativ zur FE-Berechnung als ebenes Flächentragwerk kann eine Geschosswand ohne grössere Öffnungen als vertikaler Stab mit rechteckigem Querschnitt betrachtet werden (s. mittleres Bild unten). Die Beanspruchung der Wand besteht dabei aus Moment, Normal- und Schubkräften. Die plastische Bemessung der Bewehrung erfolgt mit dem Querschnittsprogramm FAGUS, womit direkt eine verteilte horizontale und vertikale Wandbewehrung berechnet wird. Für Erdbebenwände kann eine konzentrierte, vertikale Knotenbewehrung bestimmt werden (s. E 2.7).

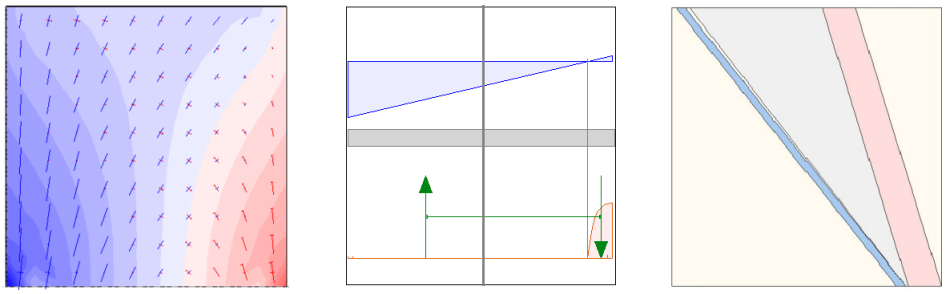


Fig. E–2 *Modelle:*
CEDRUS (S): Finite Elemente; FAGUS (A): Stab mit Querschnitt; FAGUS (A): ebenes Spannungsfeld

Die Geschosswand kann als weitere Alternative auch mittels Spannungsfeldern bemessen werden (s. links Bild oben). Dieses plastische Verfahren ist ebenfalls in FAGUS implementiert und ermittelt direkt die horizontalen und vertikalen Bewehrungslagen, wobei auch die Knotenbewehrungen für Erdbebenwände bestimmt werden können. Beide Verfahren sind im FAGUS-Handbuch ausführlich beschrieben.

Für die automatische Berechnung aller Wandbewehrungen in CEDRUS (s. E 2.7) werden die Geschosswände standardmässig als Stab modelliert. Einzelne Wänd können aber auch als Spannungsfelder berechnet werden (s. FAGUS-Handbuch).

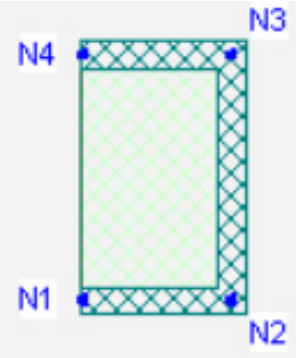
E 2.7 FAGUS (A)-Berechnung der Wandbewehrung

E 2.7.1 Einleitung

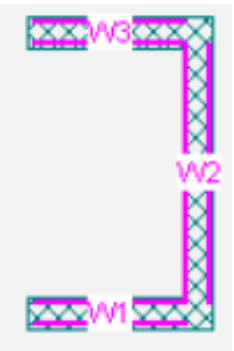
Das Gebäudemodell von CEDRUS erlaubt die detaillierte Bemessung der Bewehrung von Stahlbetonwänden mit den Funktionen des Programm FAGUS (Querschnittsanalyse). Dabei können die Durchmesser und Abstände der horizontalen und vertikalen Bewehrungsstäbe von Einzelwänden, aber auch von Wandpfeilern berechnet werden. Neben der Bemessung ist auch der Nachweis einer bestehenden Bewehrung möglich.

Definition der Wandpfeiler

Im Register 'Gebäudeelemente' jeder CEDRUS-Geschossplatte kann definiert werden, welche Wandsegmente als Einzelwände und welche gruppiert als zusammengesetzter Wandpfeiler berechnet werden sollen. Dies wird in Abschnitt D 3.4.3 ausführlich beschrieben. Jedes Wandsegment und jeder Knoten eines Wandpfeilers erhält dort einen eindeutigen Bezeichner, wobei die Segmente von W1 bis Wm und die Knoten von N1 bis Nn durchnummeriert werden (s. Bild unten).



Wandpfeiler mit Knotenbezeichner



Wandpfeiler mit Wandbezeichner

Neben der Gruppierung der Wandpfeiler werden auch die Baustoffe für den Beton und der Betonstahl aus den CEDRUS-Geschosseingaben entnommen. Dabei werden die Werte entsprechenden der Wandobjekte übernommen.

Spezifikation der Wandbewehrung

Die Eingabe aller Parameter der Wandbewehrung erfolgt im Dialog 'Wandbewehrung' (s. E 2.7.2). Dies sind folgende Daten:

- **Bewehrungskorb eines Wandknotens**
  Pro Wandpfeiler kann der Benutzer definieren, ob in den Knoten eine spezielle, vertikale Bewehrung eingelegt werden soll. Dieser Bewehrungskorb im Wandknoten Nx wird insbesondere für Erdbebenwände benötigt. Die Knotenlänge KL (s. Bild) kann vom Benutzer als Faktor f_L zur Wandbreite b spezifiziert werden (s. Bild unten).

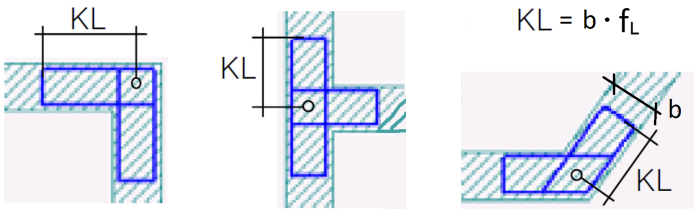


Fig. E–3 *Bewehrungskorb mit Knotenlänge KL*

- **Bewehrung eines Wandsegmentes**
  Die Bewehrung eines Wandsegmentes Wx besteht aus vertikaler und horizontaler Bewehrung, welche pro Wandsegment konstant ist. Die Bewehrung eines Wandsegmentes ist für beide Richtungen zweilagig. Sind Knotenbewehrungen vorhanden, so verteilt sich die Bewehrung zwischen den Bewehrungskörben. Ohne Knotenbewehrungen erstreckt sie sich über die gesamte Wandsegmentlänge (s. W1 im Bild unten).

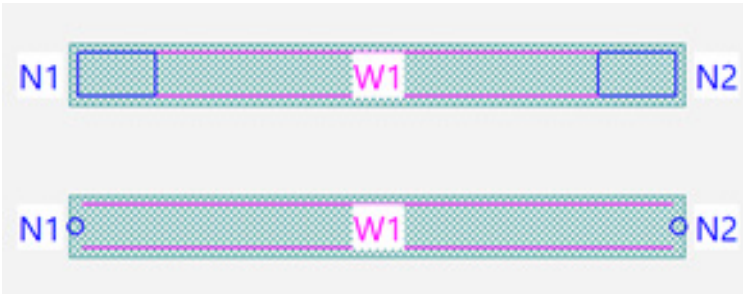


Fig. E–4 *Einzelwand mit (oben) und ohne Knotenbewehrung (unten)*

- **Bewehrungsparameter**
  Für die Bewehrungsberechnung können folgende Daten spezifiziert werden, welche für alle Wandpfeiler gelten:

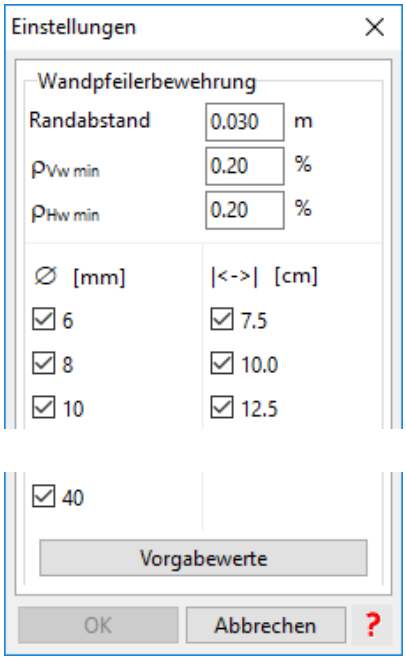
  - Randabstand der Bewehrung

  - minimaler Bewehrungsgehalt horizontale Wandbewehrung

  - minimaler Bewehrungsgehalt vertikale Wandbewehrung

  - Auswahl der Stabdurchmesser und -abstände für Bemessung

  Diese Daten werden im Dialog 'Bewehrungsparameter' verwaltet (s. Bild):



Bemessung und Nachweis der Bewehrung

Im Dialog 'Wandbewehrung' kann für jeden Wandpfeiler separat angegeben werden, ob die Bewehrung bemessen werden soll oder ob es sich um eine bestehende Bewehrung handelt. Damit ist es möglich, sowohl ein neues Gebäude zu bemessen, ein Bestehendes zu überprüfen, als auch nur einzelne Wände eines Bestehenden zu bemessen (z.B. Verstärkung). Je nach Berechnung und dem Vorhandensein einer Knotenbewehrung sind die Knoten- und Wandbewehrungen vorzugegeben oder werden gesucht (s. Tabelle unten).

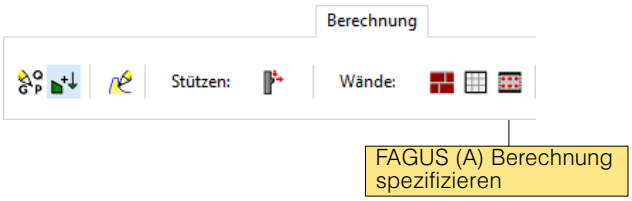
Berechnungsart	Knotenbewehrung		Wandbewehrung	
	vorhanden	vertikal	vertikal	horizontal
Bemessung	Ja	gesucht	gegeben	gesucht
Bemessung	Nein	-	gesucht	gesucht
Nachweis	Ja / Nein	gegeben / -	gegeben	gegeben

Beim Nachweis kann die Bewehrung als Bewehrungsfläche a_s oder als Stabdurchmesser mit Abstand (Wand) bzw. Anzahl (Knoten) angegeben werden. Ohne Benutzereingabe wird standardmässig eine Minimalbewehrung gemäss Dialog 'Bewehrungsparameter' verwendet. Als Minimalbewehrung wird die gewichtsmässig geringste Durchmesser/Abstand-Kombination gewählt, welche den minimalen Bewehrungsgehalt nicht unterschreitet. Dabei werden alle Stabdurchmesser und -abstände aus den Bewehrungsparametern verwendet. Als Resultat wird pro Wandpfeiler der Ausnutzungsfaktor für die vertikale und horizontale Bewehrung ausgegeben.

Bei der Bemessung resultiert eine konkrete Bewehrungsanordnung (Durchmesser/Abstand). Für die gesuchten Bewehrungslagen kann bei Bedarf ein fixer Stabdurchmesser oder -abstand vorgegeben werden. Ohne fixe Vorgabe wird die gewichtsmässig geringste Durchmesser/Abstand-Kombination gewählt, welche nicht unter der minimalen und nicht unter der statisch erforderlichen Bewehrung liegt. Dabei werden nur diejenigen Durchmesser und Stababstände verwendet, welche in den Bewehrungsparametern für die Bemessung aktiviert sind. Alle Resultate (Ausnutzungsfaktor etc.) beziehen sich auf die resultierende Bewehrungsanordnung und nicht auf die statisch erforderlichen Bewehrungsflächen.

FAGUS (A)-Berechnung mit dem Plattenstapelmodell

Bei dieser Berechnung werden die Wände aufgrund der Beanspruchungen des Plattenstapelmodelles berechnet. Damit können Windkräfte und Erdbeben-Ersatzkräfte berücksichtigt werden. Diese Berechnung erfolgt innerhalb von CEDRUS und eignet sich i.d.R. nur für die Gebäude, bei denen der Erdbebennachweis gemäss Norm mit Ersatzkräften ausreichend ist. Die exklusiv vertikal tragenden Wände können aber bei jedem Gebäude direkt mit diesem Verfahren bemessen werden.

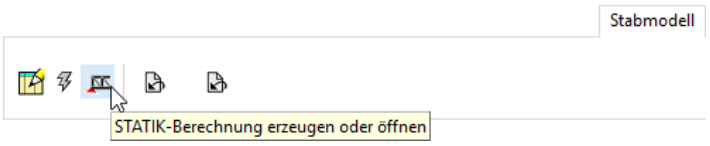


Diese Berechnung wird zuerst im Register 'Berechnung' des Gebäudes aufgesetzt (s. oben), was im Abschnitt E 2.6 näher erläutert wird. Die Analyse wird anschliessend im Register 'Plattenstapelmodell' unter „FAGUS (A)-Berechnung“ > „Wandbewehrung“ durchgeführt (s. unten).

Nach Betätigen der Blitz-Schaltfläche öffnet sich der Dialog 'Wandbewehrung', wo die detaillierte Bewehrungsanordnung bemessen bzw. nachgewiesen werden kann. Eine ausführliche Beschreibung dieses Dialogs finden Sie im nächsten Abschnitt.

FAGUS (A)-Berechnung mit dem Stabmodell

Bei dieser Berechnung werden die Wände aufgrund der Beanspruchungen des Stabmodelles berechnet. Diese Berechnung erfolgt innerhalb von STATIK und eignet sich zum Erdbebennachweis der Stahlbetonwände aller Gebäude, da STATIK auch Antwortspektren-Analysen unterstützt.

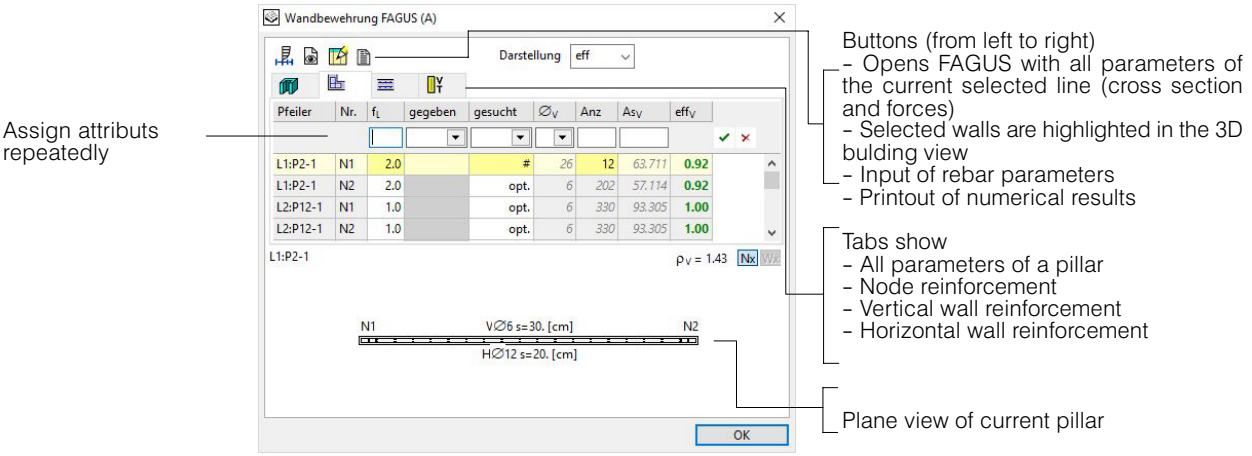


Die Erzeugung des Stabmodelles und der Wechsel ins Programm STATIK erfolgt im Register 'Stabmodell' von CEDRUS (s. oben). Die Berechnung erfolgt über das Register 'Resultate' von STATIK. Eine detaillierte Beschreibung zum Stabmodell findet sich in Kapitel E 3. Die interaktive Berechnung in STATIK erfolgt analog zur FAGUS (A)-Berechnung von CEDRUS im Dialog ‘Wandbewehrung’ (s. E 2.7.2).

☞ *Sowohl für die Wandberechnung in CEDRUS wie in STATIK benötigen Sie eine gültige Lizenz der FAGUS Option ‘Analyse’.*

E 2.7.2 Wall reinforcement dialog

The same wall reinforcement dialog can be used in CEDRUS and STATIK to design and analyse concrete walls. When this dialog is open the building is also shown in 3D view in the standard drawing area of the main GUI.



In this dialog input and results are displayed as numerical tables. In the first tab sheet each line represents a wall or a pillar. If the wall/pillar has nodes, the nodal parameters can be seen in the second tab sheet. The next two tab sheets represent the vertical and horizontal reinforcements.

The buttons allow following:

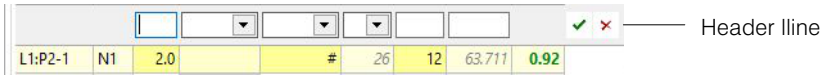
Check with FAGUS
This button opens FAGUS with the current selected line (cross section and forces) and allows to verify the calculation in detail.

Show selected wall in the 3D building view
The selected pillar in the table is shown and also selected in the 3D building view. This can help to locate certain elements faster.

Export wall reinforcement
The reinforcement of all the walls can be exported as a 2D drawing containing the actual bar diameter and spacing and saved into a PDF- or DXF-file. The drawing can be used as a background layer in the CAD-software for modelling the wall reinforcement.

Print out
The calculation results are shown as tables.

Assign attributs repeatedly
By means of the header line shown below several attributs can be assigned to all previously selected lines (yellow coloured) at same time.



Needed steps: First set up all desired attributs in the header line and then apply with ✓.

With the symbol ✗ all default setting of the header line are deleted.

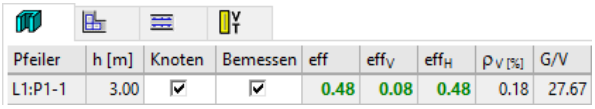
Synchronisation von 3D-Grafik und Tabellenansicht

Die Wandbewehrung wird für alle Wände berechnet, welche sich im in der Berechnungsspezifikation angegebenem Teilsystem befinden. In der Tabelle werden jedoch immer nur diejenigen Wandpfeiler aufgelistet, welche gerade in der 3D-Grafikfläche dargestellt werden. Ist die Darstellung in der 3D-Grafikfläche reduziert (z.B. durch Namensfilter oder Layerschalter), so ist auch die Anzahl der Zeilen in der Tabelle reduziert, womit sich viel übersichtlicher arbeiten lässt.

Zudem kann man einen in der Grafikfläche dargestellten Wandpfeiler in der Tabelle auswählen und anzeigen lassen, indem man ihn in der Grafik selektiert. Wie im obigen Abschnitt beschrieben geht auch das Umgekehrte durch die Betätigung der Schaltfläche.

Tabelle Wandpfeiler

Im ersten Register des Wandbewehrungsdialogs wird pro Wandpfeiler eine Zeile angezeigt. Hier kann spezifiziert werden, ob der Pfeiler an den Knoten einen Bewehrungskorb aufweist oder nicht und ob es sich um eine Bemessung oder einen Nachweis handelt (s. Bild unten).



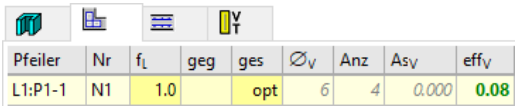
Pfeiler	h [m]	Knoten	Bemessen	eff	eff_V	eff_H	$\rho_{V\,[\%]}$	G/V
L1:P1-1	3.00	☑	☑	0.48	0.08	0.48	0.18	27.67

Neben der Höhe h des Wandpfeilers werden zusammenfassend folgende Berechnungsresultate dargestellt, wobei sich die Werte immer auf die konkrete Bewehrungsanordnung (Stäbe/Abstände) bezieht. Mit den tabellierten Werten für die Ausnutzung wird sofort ersichtlich, ob die Berechnung erfolgreich war (grün) oder nicht (rot). Der Wert G/V ist ein Mass für die Wirtschaftlichkeit der ermittelten Bewehrung:

- eff_V Ausnutzung der vertikalen Bewehrung
- eff_H Ausnutzung der horizontalen Bewehrung
- eff grösserer Wert von eff_V und eff_H
- ρ_V Bewehrungsgehalt der vertikalen Bewehrung
- G/V spezifisches Bewehrungsgewicht (kg Stahl / m^3 Beton)

Tabelle Knotenbewehrung

In dieser Tabelle sind nur diejenigen Pfeiler aufgelistet, wofür der Benutzer die Knotenbewehrung aktiviert hat (s. Tabelle Wandpfeiler). Jede Zeile entspricht einem Knoten eines Wandpfeilers und wird durch den Bezeichner des Pfeilers und die Nr. des Knotens identifiziert.



Pfeiler	Nr	f_L	geg	ges	$\varnothing_V$	Anz	As_V	eff_V
L1:P1-1	N1	1.0		opt	6	4	0.000	0.08

Die Kolonnen der Tabelle sind:

- f_L Grösse des Bewehrungskorbes als Faktor zur Wandbreite (s. E 2.7.1)
- geg Art der **vorgegebenen** Bewehrung (nur Nachweis):
  '-' minimale Bewehrung (= 4 Stabe mit kleinstem Durchmesser)
  'as' Bewehrungsfläche
  'd+#' Stabdurchmesser und -anzahl
- ges Typ der **gesuchten** Bewehrung (nur Bemessung):
  'opt' optimale (= minimales Gewicht) Kombination Stabdurchmesser/ Anzahl
  'd' vorgegebener Stabdurchmesser
  '#' vorgegebene Anzahl Stäbe
- $\varnothing_V$ Stabdurchmesser
- Anz Anzahl Stäbe im Knoten
- As_V Bewehrungsfläche des Knotens
- eff_V Ausnutzung der vertikalen Bewehrung (Wände und Knoten)

Tabelle Wandbewehrung (vertikal und horizontal)

In diesen beiden Tabellen wird die Wandbewehrung berechnet. Die Tabelle zeigt die **vertikale** Bewehrung, welche Biegung und Normalkraft aufnimmt. Die Tabelle zeigt die **horizontale** Bewehrung, welche Schub und (bei zusammengesetzten Pfeilern) auch Torsion aufnimmt. Jede Zeile entspricht einem Wandsegment und wird durch den Bezeichner des Pfeilers und des Segmentes (Nr) identifiziert.

Pfeiler	Nr	b [m]	geg	ges	$\varnothing_V$	s_V	as_V	eff_V	ρ_{Vw} [%]
L1:P1-1	W1	0.20	-		8	25.0	402	0.08	0.20

Die Kolonnen der beiden Tabellen sind:

- b — Breite der Wand
- geg — Art der **vorgegebenen** Bewehrung (nur Nachweis):

  '-' minimale Bewehrung (= 4 Stabe mit kleinstem Durchmesser)

  'as' Bewehrungsfläche

  'd+#' Stabdurchmesser und -anzahl
- ges — Typ der **gesuchten** Bewehrung (nur Bemessung):

  'opt' optimale (= minimales Gewicht) Kombination Stabdurchmesser/ Anzahl

  'd' vorgegebener Stabdurchmesser

  '#' vorgegebene Anzahl Stäbe
- $\varnothing_V$ — Stabdurchmesser
- s_V — Stababstand
- as_V — **vertikale** Bewehrungsfläche (pro Wandlänge), Summe beide Seiten
- as_H — **horizontale** Bewehrungsfläche (pro Wandlänge), Summe beide Seiten
- eff_V — Ausnutzung der **vertikalen** Bewehrung (Wände und Knoten)
- eff_H — Ausnutzung der **horizontalen** Bewehrung (Wände und Knoten)
- ρ_V — Bewehrungsgehalt der **vertikalen** Bewehrung (Wände und Knoten)

Druckausgabe

Mit der links dargestellten Schaltfläche können die Resultate der Bewehrungsberechnung in druckfähiger Tabellenform ausgegeben werden. Dabei öffnet sich ein Dialog mit der Druckvorschau, welcher die zwei untenstehenden Tabellen zeigt. Neben den Werten aus den Tabellen des Wandbewehrungsdialogs wird zusätzlich ein Schätzwert G für das totale Bewehrungsgewicht der Wände ausgegeben.

Wandbewehrung - Zusammenfassung

"FAGUS (A) : Bemessung"

Pfeiler	h [m]	N	B	eff	eff_V	eff_H	ρ_V [%]	G/V [kg/m³]	G/V_V [kg/m³]	G/V_H [kg/m³]	G [kg]
L5:P8	2.80		X	0.06	0.06	0.01	0.18	28.54	14.27	14.27	63.9
L4:P4-1	2.80		X	0.07	0.07	0.02	0.19	30.11	15.06	15.06	33.7
L4:P5-1	2.80		X	0.07	0.07	0.02	0.19	30.11	15.06	15.06	33.7
[illegible]											
L1:P14	3.00	X	X	0.21	0.18	0.21	0.14	21.48	10.74	10.74	51.5
Summe											**2164.4**

Wandbewehrung

"FAGUS (A) : Bemessung"

Wand			Vertikale Bewehrung					Horizontale Bewehrung		
Pfeiler	Nr	b [m]	Anz	$\varnothing_V$ [mm]	s_V [cm]	as_V [mm²/m]	As_V [cm²]	$\varnothing_H$ [mm]	s_H [cm]	as_H [mm²/m]
L5:P8	W1	0.20		8	25.0	402		8	25.0	402
L5:P8	W2	0.20		8	25.0	402		8	25.0	402
L5:P8	W3	0.20		8	25.0	402		8	25.0	402

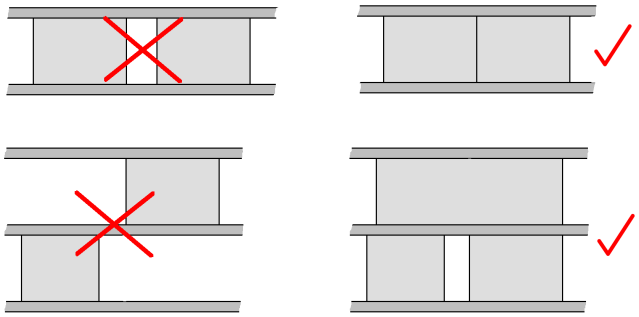
E 2.8 CEDRUS (S)-Berechnung von Wandscheiben

E 2.8.1 Einleitung

Wie in E 2.6 erläutert, bietet CEDRUS unterschiedliche Berechnungsmethoden für spezifische Stahlbetonwände an. Bei einer Wandscheibenberechnung kann die erforderliche Bewehrung einer ebene Scheibe, welche aus einer oder mehreren zusammenhängenden Wänden gebildet ist, bestimmt werden. Diese Wandscheibe kann beliebige Öffnungen aufweisen und sich über mehrere Geschosse des Gebäudes erstrecken (z.B. Abfangscheibe).

Wandscheibe aus Teilsystem

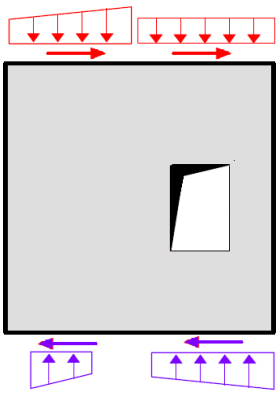
Welche der Geschosswände zusammen eine Wandscheibe bilden, kann der Benutzer über die Definition von Teilsystemen angeben (s. D 4). Dabei bilden alle Geschosswände im Teilsystem eine einzige Wandscheibe (s. Bild unten). Es können mehrere Wandscheiben berechnet werden. Vor dem Aufsetzen der Berechnung müssen die Teilsysteme bereits vorhanden sein.



Nur diejenigen Wände dürfen im selben Teilsystem sein, welche sich berühren und in der gleichen Ebene liegen. Unterschiedliche Baustoffe und Wandbreiten sind zulässig.

Berechnung mit Gleichgewichtskräften

Die Grundidee der CEDRUS (S) Wandscheiben-Berechnung besteht darin, dass innerhalb der untersuchten Scheibe ein statisch zulässiger Spannungszustand bestimmt wird, welcher mit den statischen Randbedinungen des Plattenstapelmodelles im Gleichgewicht ist. Dies gelingt, imdem man die betroffene Wand gedanklich von den Platten trennt und an den Trennflächen die im Modell wirkenden, vertikalen Kräfte als externe Last auf die Scheibe wirken lässt. Da die Wände keine eigene Belastung haben, ist die Summe der angreifenden Kräfte Null. Es handelt sich also um ein Gleichgewichtssystem, welches keine kinematischen, äusseren Randbedinigungen hat. Diese Idee fuktioniert auch für eine Gruppe von zusammengesetzten Wänden.



$\Sigma V = 0$

$\Sigma H = 0$

$\Sigma M = 0$

Die Kräfte innerhalb der Scheibe und die dadurch erforderlichen Bewehrungen werden dann in einer FE-Berechnung mit dem Scheibenmodul von CEDRUS bestimmt, welcher eine entspechende Lizenz voraussetzt. Damit eine FE-Berechnung möglich ist, wird die Scheibe in einem Punkt gehalten, wobei die Reaktionen immer Null ergeben. Dies führt zu einer Lösung, welche den Tragsicherheitsnachweis erfüllt.

E 2.8.2 Ablauf einer Wandscheiben-Berechnung

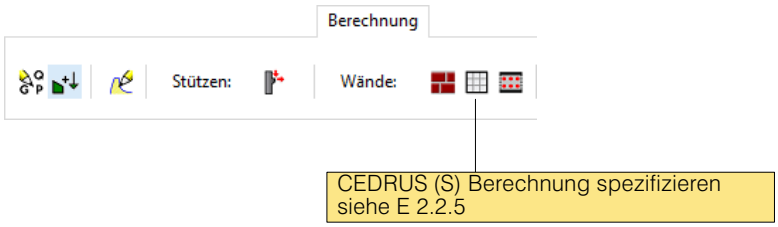
Eine Wandscheibenberechnung läuft wie folgt ab:

1. Teilsysteme bilden

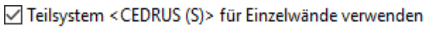
Um einzelne Wände als Wandscheibe zu berechnen, fügen Sie diese dem vordefinierten Teilsystem 'CEDRUS (S)' hinzu (s. LEERER MERKER). Für jede aus mehreren Wänden bestehende Wandscheibe erzeugen Sie ein Teilsystem und addieren die beteiligten Wände.

2. CEDRUS(S) Berechnung aufsetzten

Wechseln Sie im Gebäudemodul ins Register 'Berechnung' und betätigen Sie bei 'Wände' die Schaltfläche 'CEDUS (S) Wandscheiben'.



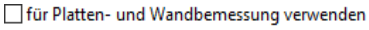
Wechseln Sie im Gebäudemodul ins Register 'Berechnung' und betätigen Sie bei 'Wände': die Schaltfläche 'CEDUS (S) Wandscheiben'. Setzten Sie die Wandscheibenberechnung gemäss LEERER MERKER auf.



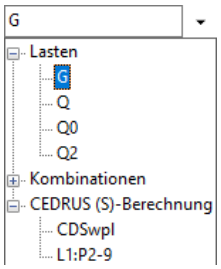
Wählen Sie dabei alle im vorherigen Schritt erstellten Teilsysteme an. Haben Sie Einzelwände in 'CEDRUS (S)' hinzugefügt, so aktivieren Sie die obenstehende Schaltfläche.

3. Horizontallasten aktivieren (falls erwünscht)

Falls Sie Horizontallasten (wie Ersatzkräfte oder Wind) bei der Berechnung berücksichtigen möchten, so müssen Sie dies angeben. Welchseln Sie dazu ins Register 'Horizontallasten' und aktivieren die untenstehende Schaltfläche bei 'Optionen für Horizontallasten' und bei 'Belastung bearbeiten' für jeden Lastfall.



4. FE-Berechnung einer Wandscheibe öffnen



Wechslen Sie nun ins Register 'Plattenstapelmodell'. Bei der Resultatauswahl sind unter 'CEDRUS (S) Berechnung' nun alle betroffenen Teilsysteme und jede im ersten Schritt angegebene Einzelwand aufgelistet. Hinter jedem Eintrag in der Liste steckt eine FE-Scheibenberechnung. Um eine Wandscheibe zu berechnen, wählen Sie nun den entsprechenden Listeneintrag und drücken die Schaltfläche . Damit wird zuerst das Plattenstapelmodell berechnet und anschliessend öffnet sich ein neues Fenster mit der CEDRUS-Scheibenberechnung für die ausgewählte Wandscheibe.

5. FE-Scheibenberechnung ausführen

Das CEDRUS-Fenster der Wandscheibe ist eine normale Scheibenberechnung, wobei die Struktur- und Lastdaten nicht verändert werden können. Nur das standardmässig generierte FE-Netz kann vom Benutzer verändert werden.

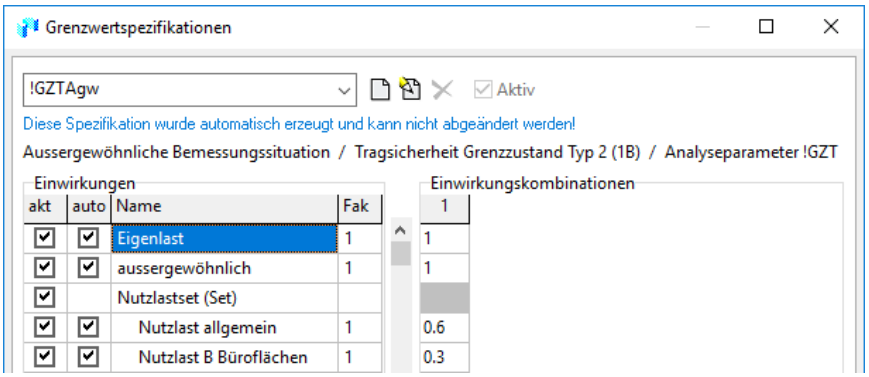
Strukturdaten

Die Strukturdaten der erzeugte Scheibenberechnung entsprechen der Geometrie und dem Baustoff der beteiligten Wände und beinhalten auch alle Wandöffnungen.

Lastendaten

Als Belastungen werden die vertikalen Lasten des Gebäudes analog zur Plattenberechnung übernommen ('Imp-G', 'Imp-Q' etc.). Falls Sie im 3. Schritt auch horizontale Belastungen mitberücksichtigt haben, so wurden auch diese aus dem Plattenstapelmodell importiert. Je nach Typ der Horizontallast sind nun folgende Daten in der Scheibenberechnung zu finden.

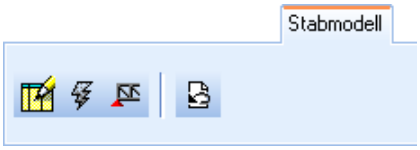
Die aus dem Plattenstapelmodell importierten Ersatzkräfte werden mit '!Imp-Ex' bezeichnet. Für die Bemessung wird zudem die Grenzwertspezifikation 'GZTAgw' für die aussergewöhnliche Bemessungssituation der einzelnen Geschosse generiert (s. Bild).



Die aus dem Plattenstapelmodell importierten Windlasten werden mit '!Imp-Wx', die Erdrücke mit '!Imp-Dx' bezeichnet.

E 3 Model for STATIK

E 3.1 General



With the frame model, the Building Module makes a further analysis model available besides that of the stacked slab model. This is mainly the case for earthquake analyses on buildings, for which the equivalent force method is not admissible. The automatically generated frame model is a genuine 3D model, which is handled by the structural analysis program STATIK. STATIK with its Dynamics Module possesses the required analysis tools for earthquake analyses.

With a frame model, normally the basic behaviour of a building structure can be simulated very well. It provides interpretable results, which are also suitable for dimensioning purposes. Of course, special buildings and situations will be encountered, in which one has to use more complicated models. Here, the Building Module of Cubus is open-ended. For example, a model with surface structures is currently being developed.

In the following chapter the essential properties and parameters of the frame model are dealt with. A prerequisite for a successful application of the frame model is a thorough understanding of what is written here.

⚠ The progress from the first geometric model of the building (usually CEDRUS slabs stacked one on top of the other) to a useful frame model is not always simple and presents a challenge to the engineer with regard to structural modelling. In the final section E 3.4 of this chapter there are instructions for performing a stepwise solution of this task.

E 3.2 Properties and Parameters of the Frame Model

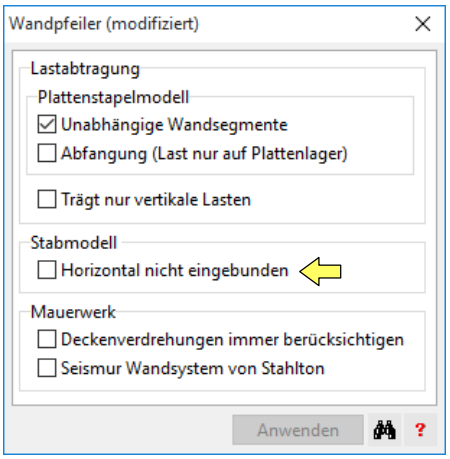
The following information concerns exclusively the frame model generated by the Building Module. In STATIK the model can be modified as much as one likes according to the possibilities of the program.

The frame model is characterized by the following properties:

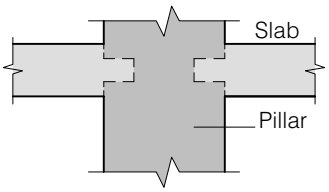
E 3.2.1 Modelling the Slabs

The slabs are modelled in respect to displacements in their planes as **rigid membranes.** This is achieved using nodal constraints. One node in each story plane is a master node and all other nodes in the plane are slave nodes with respect to DX, DY, RZ connected rigidly to the master nodes.

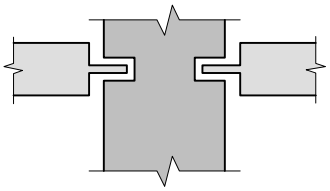
Exception: There is the possibility to detach single pillars from this membrane connection (only permitted with embedded storeys, see page E-64):



horizontal embedded (standard):

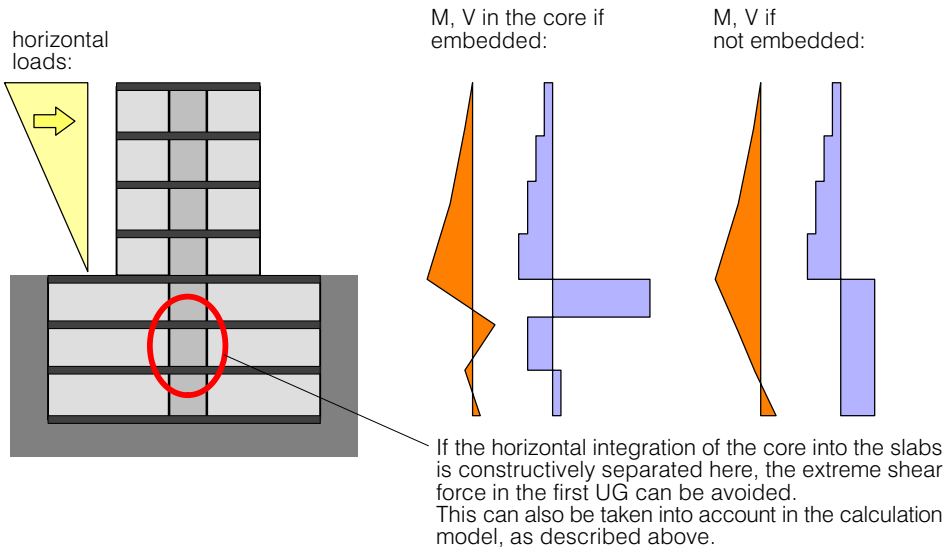


horizontal not embedded:



⚠ This property applies to the node of the frame model generated for the pillar head. It is only accepted by the program if exactly two members are connected to this node in the frame model, one downwards and one upwards.

Application example:



The core is fully clamped down in the base level as all pillars of the program by default. This can be corrected by the user in the STATIK model if necessary.

With regard to bending there are the three following models:

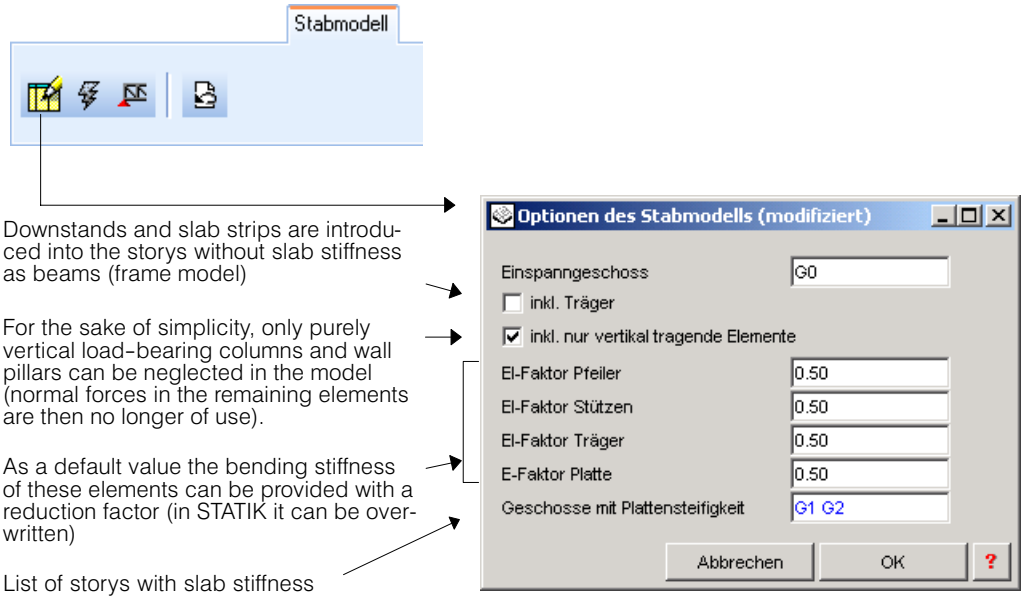
a) Fully flexible: The bending stiffness of the slabs is neglected. The slabs serve solely as rigid 'distance spacers' for the nodes in a story plane. This amounts in effect to a pure **cantilever wall model**.

b) Frame model: Besides the vertical frame members also horizontal frame members (beams) are introduced. This leads therefore to a frame model. All downstands and any slab strips present are modelled as beams (see below).

c) Model with slab stiffness The bending stiffness determined in the slab analysis program is introduced into the STATIK model as a super element for all nodes in a story plane. This corresponds to the modelling of a slab with an FE program. This model is then suitable if for the dynamic analyses and to determine the section forces in the wall pillars the best possible consideration of the slab stiffness in bending is required and a reasonable model with slab strips is difficult to achieve. Therefore, the influence of the slab stiffness can be estimated very quickly. Details of the slab stiffness model are given later in this chapter.

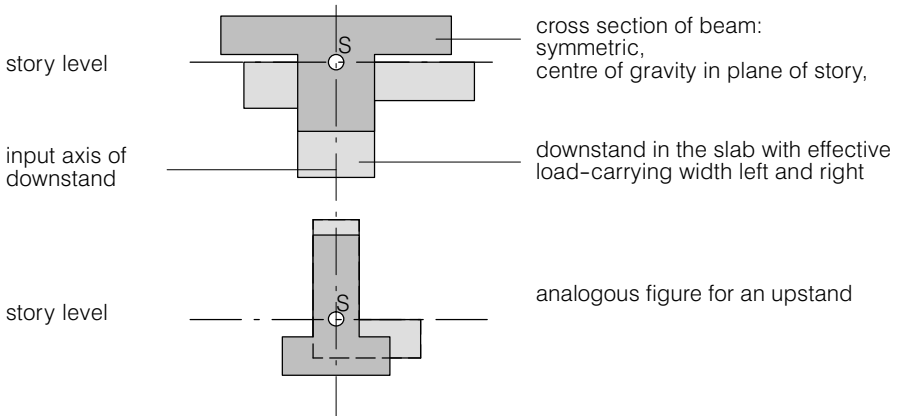
⚠ In the current version of the program, this model has the disadvantage that the moments that are resisted by the slab are not shown. One has to remember this in the dimensioning of the slab.

Mixed forms of the abovementioned models may be present in a building. In the dialog with the 'Options for the Frame Model' the model parameters can be specified as follows:

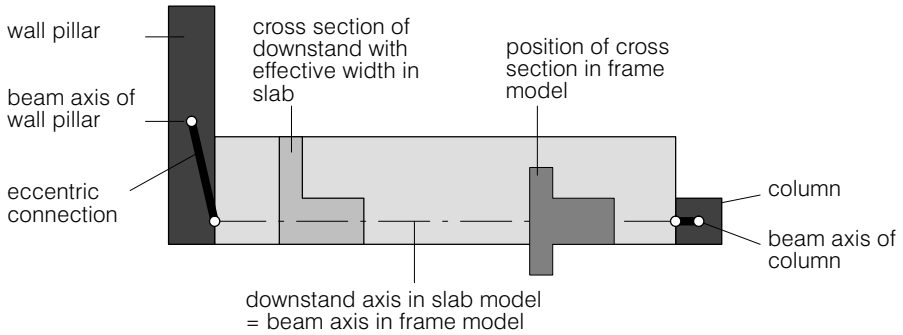


How are downstands made into beams?

The cross section of the generated downstand is formed from the web of the downstand and a symmetrically arranged 'flange'. The width of the flange corresponds to the effective load-carrying width specified for the downstand and the flange thickness is the thickness of the neighbouring slab. The cross section is chosen to be symmetric, since the portion of the slab is also in reality not subjected to an out-of-plane bending action. The following figure shows the modelling of both a downstand and an upstand.



The following figure shows the modelling of a downstand in the plan view, which connects on the left side to a wall pillar and on the right side to a column:

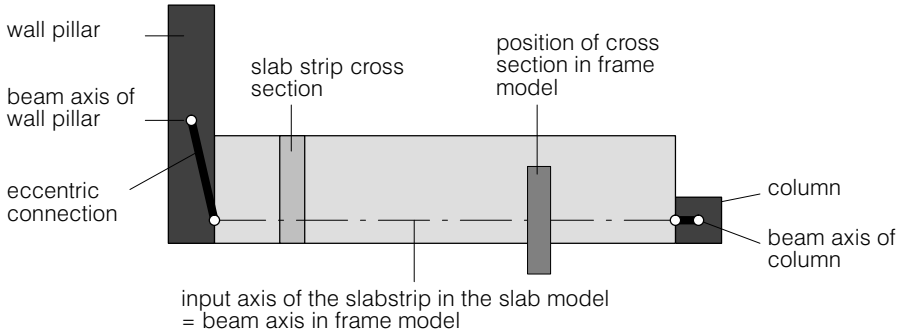


What are slab strips and how are they made into beams?

The frame effect of the slab can be taken into account by means of slab strips. They are defined by the user and consist of an axis line and the width properties 'width left' and 'width right. The input is made in the slab in the 'Building' Tab sheet (see Chapt. D 3.4.2). In the case of very irregular wall and column patterns it is scarcely possible to define a suitable slab strip model. In such cases one has to use Model c) described above, if the slab stiffness has to be considered.

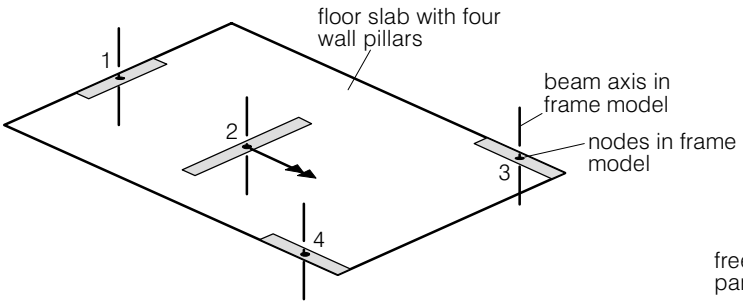
From the slab strips the frame model (beam of rectangular cross section) is formed. The width of the cross section corresponds to the width of the slab strip and the height is the same as that of the slab thickness along the axis of the slab strip.

The following figure shows the modelling of a downstand in plan view, which connects on the left side to a wall pillar and on the right side to a column:

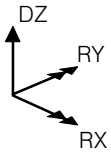


On the 'Slab with Condensed Stiffness' model

With this model the slab effect is considered in relation to its stiffness completely in the frame model. This is illustrated by a simple example.

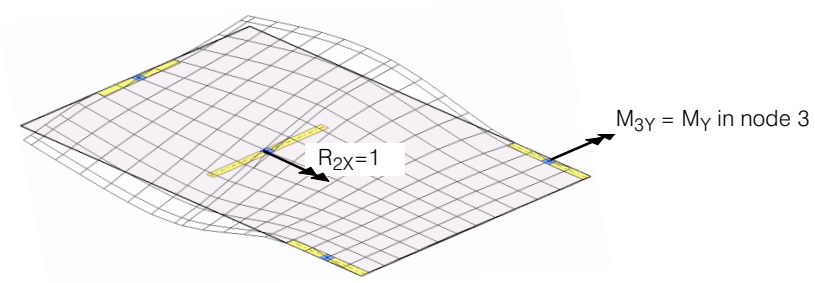


free displacement parameters per node



The slab stiffness determined with an FE slab model is condensed to the 4 story nodes (each with the 3 degrees of freedom DZ,RX,RY) of the frame model. A 12x12 stiffness matrix is obtained, which is input into a STATIK model.

The following example illustrates how the term $k_{9,5}$ is determined. It represents the fixed moment M_{3Y} (parameter 9) due to a unit rotation R_{2X} (parameter 5).
Note that the stiffness of the walls is considered (stiffening beam along the stretch of wall)



			1 D_{1Z}	2 R_{1X}	3 R_{1Y}	4 D_{2Z}	5 R_{2X}	6 R_{2Y}	7 D_{3Z}	8 R_{3X}	9 R_{3Y}	10 D_{4Z}	11 R_{4X}	12 R_{4Y}
1	F_{1Z}	=												
2	M_{1X}	=												
3	M_{1Y}	=												
4	F_{2Z}	=												
5	M_{2X}	=												
6	M_{2Y}	=												
7	F_{3Z}	=												
8	M_{3X}	=												
9	M_{3Y}	=					$k_{9,5}$							
10	F_{4Z}	=												
11	M_{4X}	=												
12	M_{4Y}	=												

Remarks on the Slab Model:

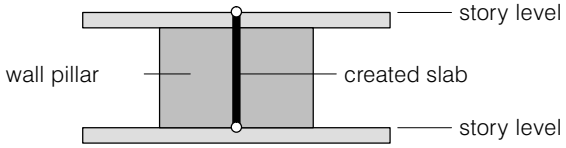
- In the determination of the slab stiffness for a story, all walls and columns that connect to the top and bottom of the slab are taken into account.
- In the case of storys, whose slabs are treated with the slab stiffness model, the setting 'Include beam' in the Options dialog for the frame model has no effect, i.e. no beams are generated.
- If the slab stiffness, as described, is input into the STATIK model, then this behaves as if the corresponding slab were integrated as a complete FE model.
- The slab stiffnesses can be introduced into a building one story at a time as desired.

E 3.2.2 Modelling the Wall Pillars

Wall pillars, together with the columns, form the vertical support elements of a building. They consist of a single wall segment (wall polygon side in the slab input) or of a group of wall segments. The grouping of wall segments to wall pillars is carried out in the slab input in the 'Building' Tab sheet (see chapt. D 3.4.3).

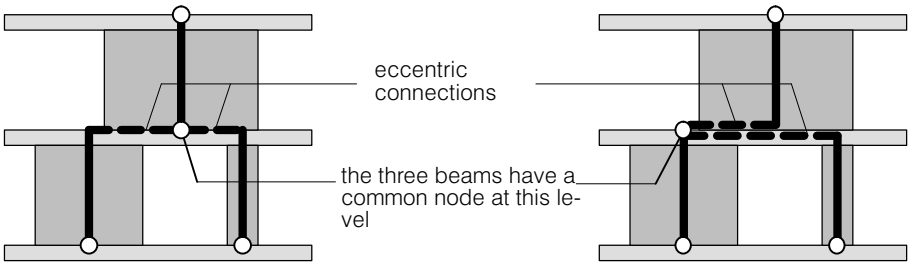
Wall pillars are modelled in the frame model as follows:

- Out of every wall pillar a vertical beam (which for the analysis is subdivided into 3 beam elements) is formed. The beam axis lies in the centre of gravity of the wall pillar cross section. The upper end of the beam has the level of the slab it supports. The lower end is at the level where the wall pillar (or a part of it) on being extended first meets a floor slab or a foundation zone.



The following points apply to the wall pillar of a story that does not stand on a floor slab with the model *Slab Stiffness*! Otherwise the wall pillar is connected directly to the slab.

- If wall pillars are supported by other wall pillars or columns, then the elements that stand one on top of the other are in general rigidly connected to each other at the corresponding story level. This is done in such a way that only one node is introduced at the end of one of the beams and the other beam ends are connected eccentrically to this node. The two models shown in the following figure are equivalent statically but the program does not necessarily find the 'best' solution on the left:

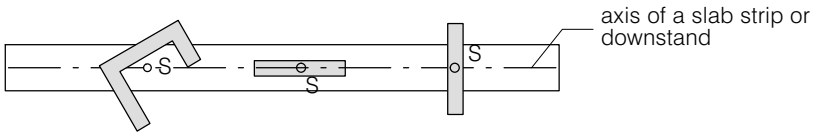


  The conditions (tolerances, etc.), under which the program assumes that two vertical elements stand one on top of another, are described in Section E 3.2.4.

- If there are unsupported wall pillars (see E 3.2.4), the program issues a warning and introduces at the bottom of such beams a vertical support.

  The mentioned support obviously leads to an incorrect normal force in the neighbouring vertical elements, but often this hardly has any influence on the dynamic behaviour. If in the analysis one is not interested in the support details of such wall pillars, then by introducing the support in a simple way one can obtain a simple makeshift model for a quick preliminary dynamic analysis.

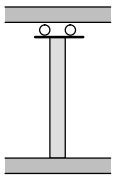
- A wall pillar can also be supported by a beam (downstand or plate strip). This support is possible if the centre of gravity of the wall pillar lies on the axis of the beam (see Page E-54) (only small tolerance permissible) and if in the options for the frame model 'incl. beam' is activated.



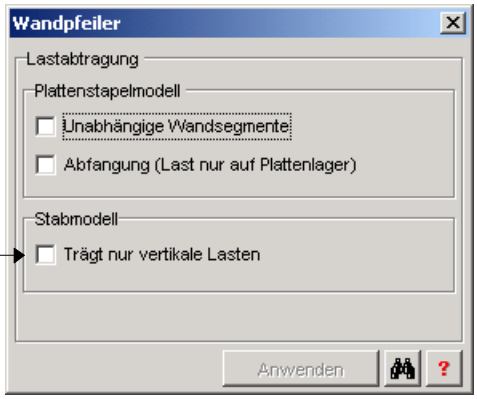
The three wall pillars are supported, since their centre of gravity lies on a beam axis (in the frame model with beams)

With and without resistance to horizontal forces

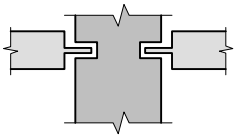
For every wall pillar it can be specified whether or not it should withstand horizontal forces (e.g. due to earthquakes). Specification is made in the wall pillar dialog (see Chapt. D 3.4.3).



Wall pillars that do not have to resist horizontal forces are modelled with a shear hinge at the top. Thus they cannot transfer any shear forces to the floor slab, only resist moments due to eccentric loading.

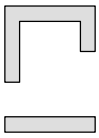


Embedded in membrane slab



In special cases, the Embedding of a wall pillar be lifted into the horizontal rigid basement membrane (panel 'not horizontally embedded'). This setting is only involved storeys allowed (see page E-64). Details are on page E-52 beschrieben.

Plane or general wall pillars



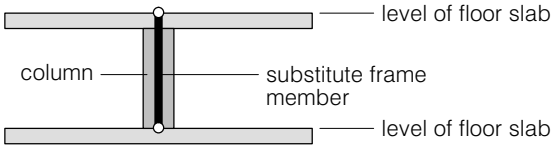
The above statement only applies to **general wall pillars** in the two main axes of the cross section.

In the case of **plane wall pillars** the minor axis is treated differently. If the wall at the connection to the floor slab is defined to be freely rotatable, the bottom of members of such wall pillars are provided with a hinge to allow rotation about the minor axis. No shear hinges are introduced in this direction.

E 3.2.3 Modelling the Columns

Together with the wall pillars, columns form a building's vertical structural elements. They are modelled in the frame model as follows:

- Each column becomes a vertical member of the frame (note, for the analysis they are subdivided into 3 elements). The upper end of the member has the level of the slab that it supports. The lower end is at the level where the member's axis when extended first meets another floor slab or foundation zone.



- If columns rest on other columns or wall pillars, the the corresponding elements are rigidly connected to the corresponding floor slabs (with the exception of any bending hinge that has be introduced, see below). This is accomplished by simply introducing a node and the ends of the frame member are eccentrically connected to these nodes (see above in the case of the wall pillars).
- In the case of the **Cantilever Model** (see E 3.2.1) the columns only carry gravity loads. To prevent a frame action, at the lower end of each column member a rotational hinge is introduced.
- Whether in the case of a **Frame Model** (see E 3.2.1) moments have to be resisted, depends on the support conditions specified in the slab input. If the column is free with respect to rotation, it only resists gravity loads and the column member is provided with a hinge support. Otherwise it has no hinges and can act as a frame member.
- The same holds for the bottom support of columns as for wall pillars (see E 3.2.2).

E 3.2.4 When are vertical elements placed one on top of the other?

If one is not using the Slab Stiffness model, the program has to know if and how the vertical elements (wall pillars and columns) are placed on top of one other and how, as described in E 3.2.2 and E 3.2.3, they are connected with one another.

Prerequisite for the possibility of vertical elements standing one on top of another are point and/or segment supports, as described below.

If the geometric model of a building is too complex, i.e. several walls are placed criss-cross on top of one another, or if an element does not rest on another one, the program doesn't allow a rigid connection and thus there is no force transfer between these elements. In such cases a warning is given, the corresponding elements are marked at the bottom of their axes and an external vertical support is introduced (☞ E 3.2.2).

With such external supports, it is nevertheless, as mentioned above, often possible to carry out a dynamic analysis. However, if one also wants more information on the transmission of vertical forces, this is obviously not a feasible model. The following correctives are possible (for a detailed explanation see Chapt. E 3.4):

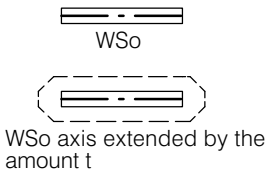
- Choose the model *Slab Stiffness* for the story with the difficult support conditions (Options dialog of the frame model).
- In the case of columns and short sections of wall, which are within the field of a slab: Introduce slab strips, whose axes lie below the axes of the vertical elements.
- In the case of shear walls, which are in fact supported at two points, but the program does not recognize them as being correspondingly connected:
  The shear wall is modelled in the supporting floor slab as an upstanding beam, on which the wall axis can then be supported.
- If, for example, several shear walls lie skew across a longer wall pillar section, this may be a reason why the program refuses to place elements on top of each other. In such cases a solution is to subdivide the longer wall section into several segments.

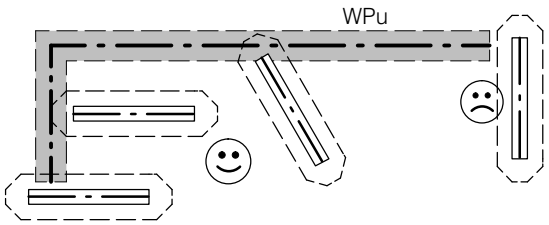
If the program detects elements, unexpectedly, as not lying one on top of the other, this may be due to the specified tolerances for point and segment supports, which therefore are discussed below. Whether vertical elements are actually connected with one another, however, may also be due to other conditions, which among others are based on stability considerations and are difficult to explain. A missing connection can be recognized in each case and treated as explained in chapt. E 3.4.

Point support of a wall pillar

A point support of a **wall pillar WPo on a wall pillar WPu** arises if the axes of two segments WSo and WSu of the wall pillars intersect. This is subject to a tolerance $t \approx (t_o + t_u)/2$, by which the axis of the upper segment WSo is lengthened and widened.

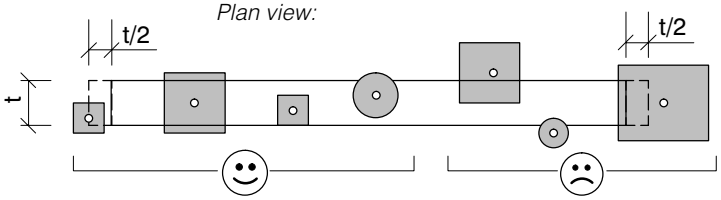
Plan view:





A point support of a **wall pillar on a column** arises if a wall pillar segment lengthened by its thickness t lies over a column axis.

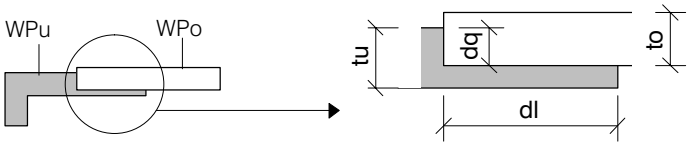
Plan view:



Segment support of a wall pillar

A segment support of a wall pillar WPo on a wall pillar WPu results if two wall pillar segments WSo and WSu lie over one another to a certain extent. Here certain tolerances apply, which are described in the following figure.

Plan view:

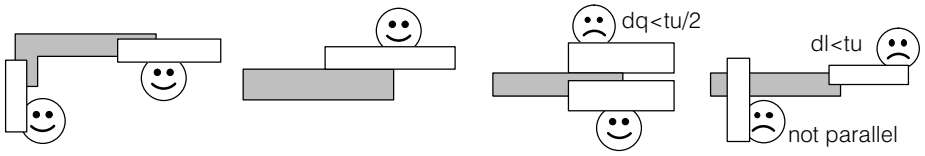


The axes of the wall pillar segments lying one on top of the other must be parallel!

Conditions placed on the overlapping tolerances dl and dq:

dl ≥ tu/2

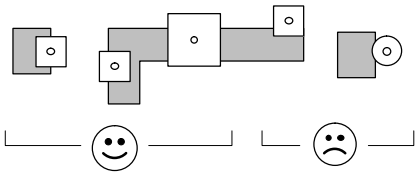
dq: if to ≤ tu: dq ≥ 0
otherwise dq ≥ tu/2



Point support of a column

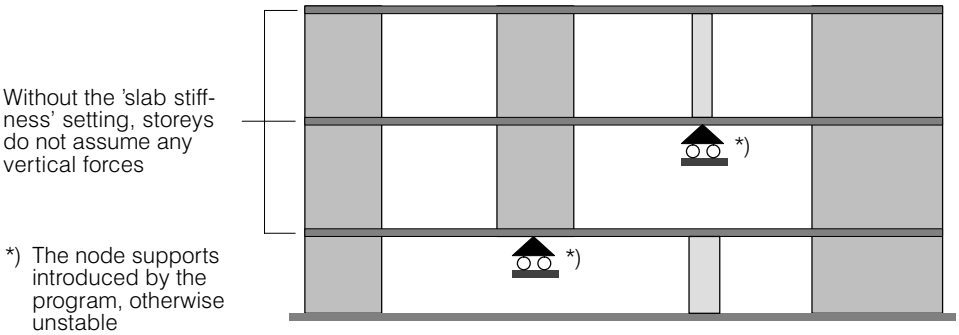
A point support of a column lying on another column or on a wall pillar arises if the column axis lies within the area of the underlying element:

Plan view:

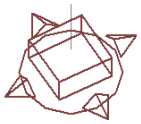


E 3.2.5 Node support

If vertical elements which are not intercepted are generated when the space frame model is created, a corresponding warning is issued and the affected members are marked. During the generation of the STATIK calculation, so-called node supports are introduced at these locations in order to prevent a system instability of the STATIK calculation. In the following figure, such a situation is displayed.



The introduced node supports are always of the following type:

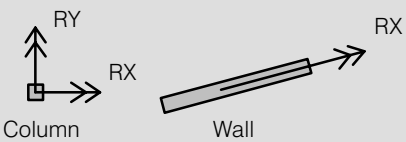
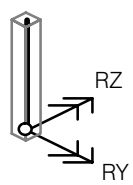


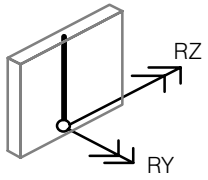
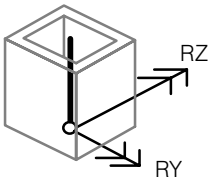
Support -signature in STATIK

The membrane degrees of freedom DX, DY, RZ, which are rigidly connected to each other in a slab plane by node connections, are free.

The displacement degrees of freedom RX, RY, DZ associated with the slab bending are blocked.

While the vertical displacement DZ is always blocked in such node supports, the rotational stresses RX, RY can be canceled out again by means of member end joints. The following table shows when which joints are inserted at the lower end of a member resting on an node support.

	Support conditions in storeys (CEDRUS) 		STATIK model:	Joint components inserted at the foot of a member in node supports
	RX	RY		
Columns	free	free		**RY, RZ**
	blocked	free		**RZ**
	free	blocked		**RY**
	blocked	blocked		-

Walls	free			**RYF*), RZ**
	blocked			**RYF*)**
Pillars				**RYF*), RZF*)**

*) Spring with a estimated stiffness

Discussion of the node supports

In reality, there are no walls and supports that are not intercepted, because they always stand on storeys or foundations, which can carry off the loads in principle. If the cantilever wall model is used, which is often the case in the initial phase of an analysis, the slab rigidity is not taken into account, and it can then have non-captured vertical elements. If such elements are displayed when the member model is created, one should check whether the model is basically in order. Some typical cases are discussed below:

- **Interception slabs:** They are often found above underground garages and, as the name suggests, serve to support walls and columns that do not reach the foundation. With regard to earthquake resistance, they are not ideal and must be thoroughly constructed. For the calculation model, consideration is probably given to the slab stiffness (see page E-56). This eliminates the node supports in the slab.
- **Attic storey:** They often differ greatly from the storeys below, and often they come to not intercepted walls and columns. The interception slab is also to be carefully analyzed. It is not so delicate, however, since it only needs to transfer loads from one storey. The slab stiffness model is also recommended for these "interception slabs".
- **Not interceptioned columns:** Since they normally do not absorb earthquake forces, their node supports do not pose a problem in the normal case. The same applies to pillars, which are designed for the absorption of exclusively vertical loads.
- **Walls with horizontal loads:** An earthquake wall, which simply stops on a non-specific interception slab, is probably not very useful. If such walls occur, a special warning is issued. It is therefore necessary to clarify whether this is actually the case, or whether the generated model has to be refined by hand. Since a full joint around the strong axis of pillars would be very real, a spring joint with a program stiffness is introduced here. If other continuous pillars are present, no decisive moments are likely to occur in such spring joints introduced for stabilization purposes.

Node supports, of course, falsify the static model. The extent to which such node supports can nevertheless be used is explained separately for vertical and horizontal loads. The insignificant spring joints for pillars are not discussed here.

Node supports under vertical loads

As described below in E 3.2.8, there are two models for transferring vertical loads to the STATIK model.

In the **load export model**, the loads are transmitted as surface loads and the loads which are then absorbed by the node supports are, of course, missing in the adjacent walls and supports. It is often possible to live with this, since this does not necessarily have any influence on the load transfer of the horizontal forces and on the natural vibrations - but it can also, as explained in the section on horizontal loads!

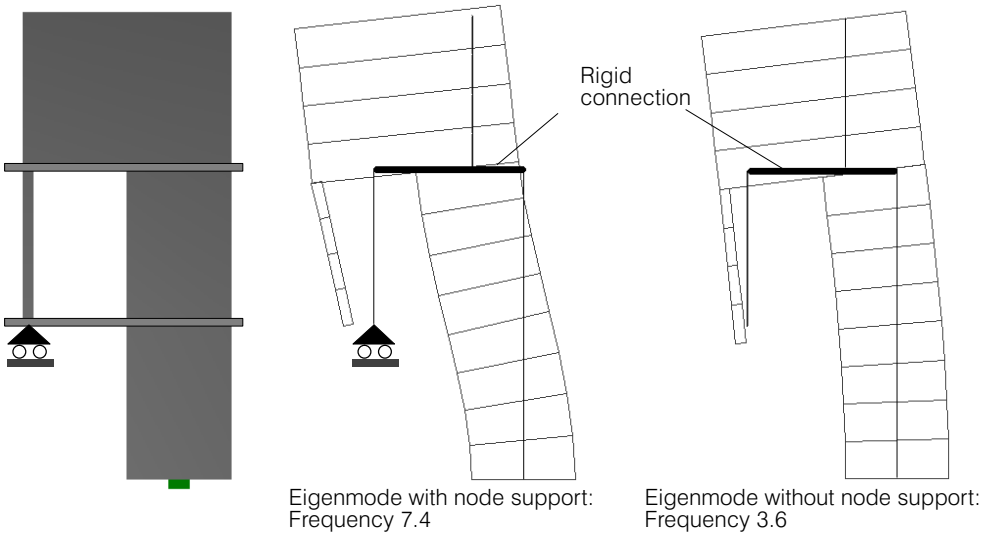
In the **sectional forces export model**, the sectional forces determined by the stacked slab model are transferred directly to STATIK, ie, the vertical loads. In STATIK, the vertical loads are not counted. Thus the influence of the node supports for the vertical loads is switched off and no vertical forces are lost. However, only sectional forces and, for example, no displacements are available for these loads.

Node supports under horizontal loads

In cases, as illustrated in the first illustration in this chapter, where the node supports are necessary to avoid instability, horizontal forces and displacements can not cause reactions in the emergency camps, which do not adversely affect the static model.

However, the following figure shows a case where the system would also be stable without an node support and horizontal loads would lead to non-negligible reactions in the node support.

⚠ In such cases, e.g. can be detected by checking the reactions for lateral forces, the node supports must be removed in STATIK.

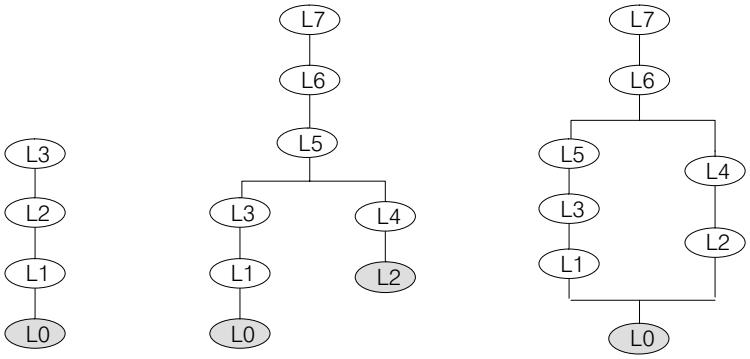


E 3.2.6 Supports of the STATIK model

Storey topology

Vertical elements (i.e., pilars and columns) of a storey are extended downwards until they hit a different storey. If there is no intercepting storey, this is an input error that is reported by the program. If elements of the slab Lj meet on the slab Lk, Lj and Lk are connected topologically with one another. The topography of a building can be displayed graphically.

Here are some examples of building topologies:



Base level

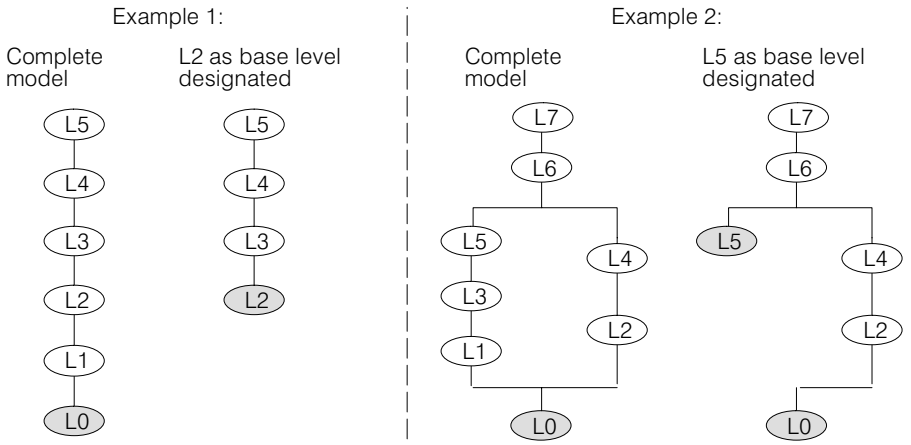
All columns and pilars, which hit on base levels, are generated as bars that are fully clamped at the bottom.

Automatic base level

Storeys of the above defined types 2, 3, 4 function automatically as so-called base levels. Such storeys must always be at the lower ends of free branches of a topological space (gray-backed storey symbols in the figure above).

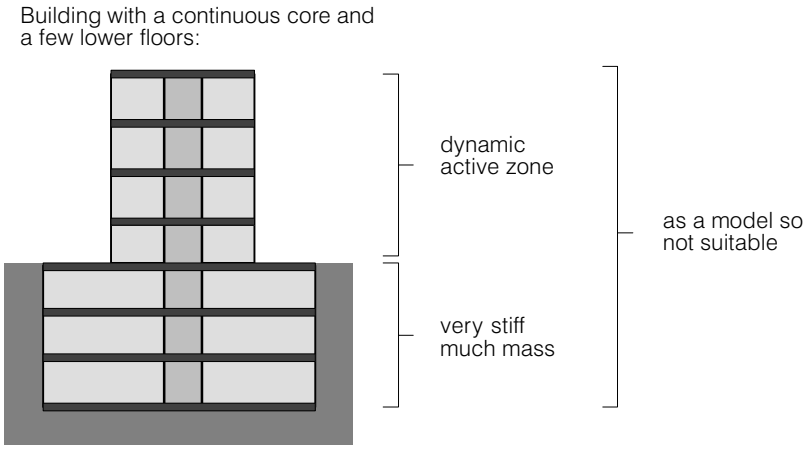
User defined base level

If you do not want to take stiff lower floors into the space frame model, you can cut a building down by setting fixtures downwards (see option dialog for the bar model page E-68). If a storey is referred to as a base level, the topological space is cut off at the relevant location. The base floor itself acts only as a limited area support zone.

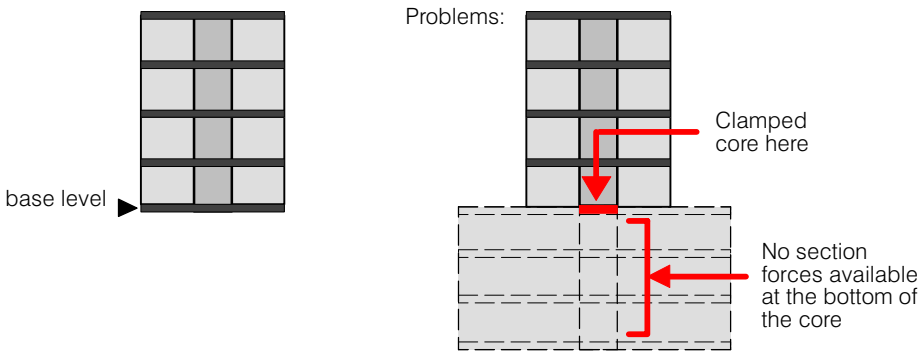


Embedment levels

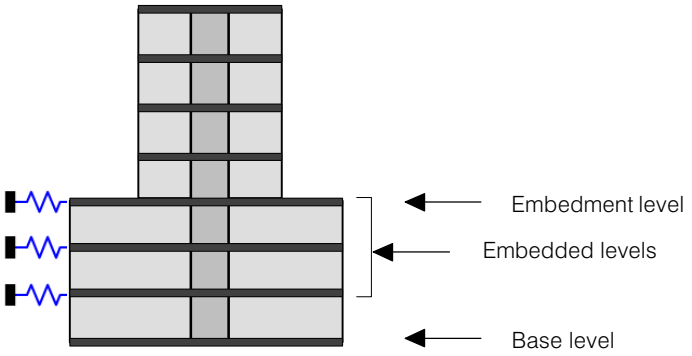
The following figure shows a situation that requires special treatment in space frame model generation:



Model 1 (less suitable)



Better model with embedment levels (see also page E-52)



In addition to **base levels,** it is also possible to specify **embedment levels** in the option dialog for the space frame model (see page E-68). They have the following effects on the generated space frame model:

- Floors below a specified embedment level in the topological space are also considered integrated
- Embedded levels have no masses
- Embedded levels are provided with a support in the master node without stiffness for components DX and DY and clamping for rotation RZ (user-adaptable!).
- The specified embedment level is the zero level for the generation of earthquake lateral forces. If several embedment levels are defined, it is the deepest among them.

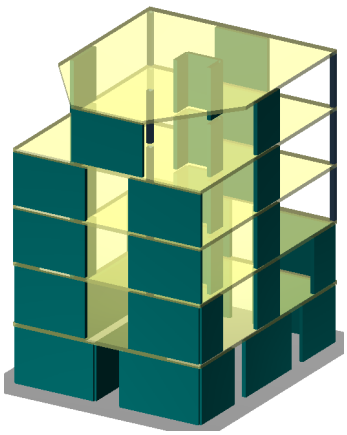
E 3.2.7 Automatically-Generated Partial Systems

For all systems of members connected with one other in a frame model a partial system is automatically created to simplify inspecting and checking these systems (on partial systems see D 4.1).

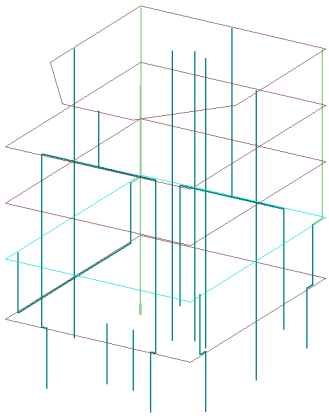
E 3.2.8 Generated Loadings and Mass Distributions

The building module also generates for the STATIK model

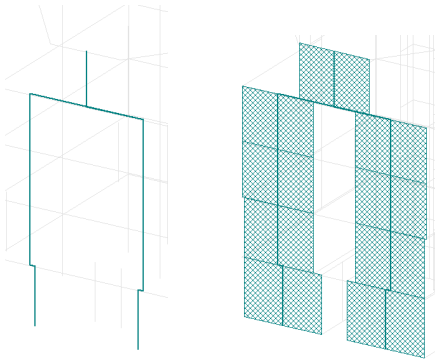
- the loads with the constant and variable vertical loads of the building
- the loads with the storey masses resp. weights for dynamic analysis
- the input horizontal loads of the type 'wind' and 'earth pressure' (see Chap. E 2.1.1)



Building model



Complete frame model



As partial system, generated members of a contiguous subsystem

Generation of storey weights for dynamic analyzes

For the frame model the following loadings are generated for each floor:

G: Combination (without load factors) of all permanent loadings for the floor slab that have been activated in the loading properties of the check box 'Export automatically', together with half the weights of the columns and walls connecting from above and below. The loading is assigned the action 'Self-weight' .

Q: Combination (without load factors) of all variable loadings for the floor slab that have been activated in the loading properties of the check box 'Export automatically. The loading is assigned the action 'Q-Psixxx', where instead of 'xxx' stands the ψ_2-factor given for the story in the Building Definition dialog (e.g. 'Q-Psi0.3').

M: The required sum G + Q*ψ_2 required for the mass distribution.
ψ_2 is specified in the Building Definition dialog for each story.

For STATIK, the loadings are created in the form of area loads at story level. The slab loadings may contain point, line and area loads.

Vertical loads

The following loads are generated for the vertical loads:

G: Loadings with the permanent loads *) on the whole building; any export factors <> 1.0 (load characteristics) are not considered

Q: Loading with the variable loads *) in the whole building; any export factors <> 1.0 (load characteristics) are taken into account (also in the following combinations)

Q0: Loadings with the variable loads *) multiplied by in the whole building. The ψ_0 are taken from the influences which are assigned to the loadings entered.

Q2: Loadings with the variable loads *) multiplied by ψ_2 n the whole building. The ψ_2 are taken from the influences which are assigned to the loadings entered.

GQ2: Combination loading from G and Q2.

*) Depending on the setting in the option dialog for the space frame model, the sectual-force determination in STATIK and the load export from the building are performed according to one of the following two models:

Section force export model

In this model, not the loads, but the section forces determined in the slab stack model of the building are transferred to STATIK. Formally, it is also loads which, however, do not contain load elements but rather the section forces determined in the building model. The section forces for the vertical loads are thus not determined in the space frame model but taken over from the building model.

Loading export model

In this model, loads in the form of surface loads are transferred to STATIK, and the section forces under these loads are determined in STATIK on the space frame model.

Model comparison

The section force export model is set as default. With respect to the load export model, the following advantages and disadvantages have to be weighed against each other in the specific case:

\+ ***No loss of normal force due to non-intercepted walls and columns:*** At the foot of non-intercepted walls and columns, supports for the vertical displacement are automatically inserted to stabilize the space frame model. These supports receive reaction forces that should be transferred to adjacent walls and columns by the slab. When dimensioning these walls and columns, these standard components are missing. When using the section force model, the section forces of all walls and columns in the stacked slab model are determined and no normal forces are lost.

\+ ***Consideration of slab properties in the transfer of storey loads on walls and columns:*** In the transfer of surface loads in STATIK working with a loaded area of constant thickness. In the stacked slab model, the actual slab properties, such as different thicknesses and joint lines, are taken into account, and the stiffness ratio of walls and plate can be better controlled.

o ***More realistic section forces:*** The stacked slab model, particularly for the permanent loads, is a quite good model for the transfer of vertical loads. The application of the dead weight to the entire, elastic, spatial model often leads to questionable results in higher buildings. In special cases, however, the 3D model can also be interesting.

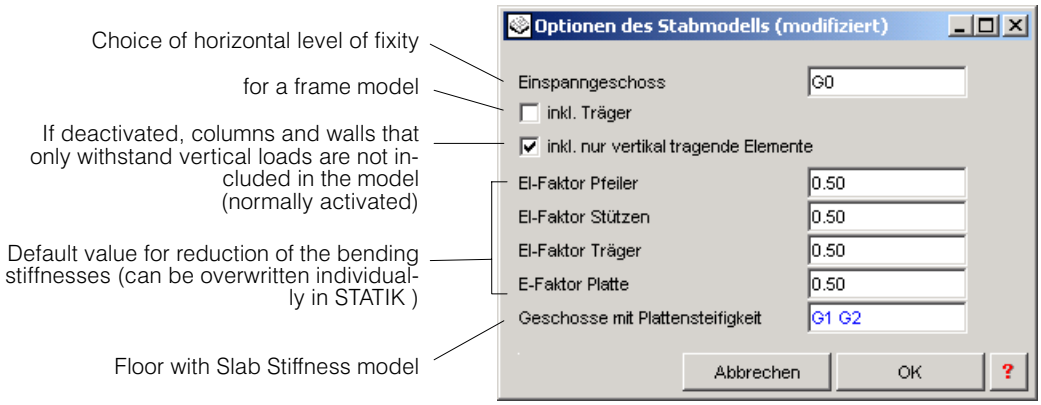
\- ***Only section force results for vertical loads:*** Since only the section forces are transmitted for the vertical loads in the section force model export, only those section force results can be obtained in STATIK for these loads and, for example, no shifts.

E 3.3 Generation of the Frame Model

For the generation of the frame model the geometric building model has to be analysed and in particular its topology determined. Under topology one understands the way in which the various structural elements are connected with each other, whether walls rest directly on other walls and if they do, in which areas, etc.

Optionendialog für das Stabmodell

The button on the left calls the following dialog, in which the options for the generation of the frame model can be specified:



Reduction factors for bending stiffnesses

Since during an earthquake in the components subjected to bending stress considerable cracking is to be expected, it makes no sense to carry out the natural vibration calculation on which both the lateral force and the response spectra method is based on a model with stiffnesses of the homogeneous state.

The program refers to SIA261, especially to 16.5.2.2 and 16.5.5.2.

The following information, taken from the script for the lecture “Erdbebensicherung von Bauwerken I”, ETH Zürich, FS 09; Alessandro Dazio und Thomas Wenk

Die folgende Tabelle gibt Richtwerte für die Abminderung des Trägheitsmoments infolge Rissebildung [NZS95]:

Riegel	Stützen	Wände
Rechteckq. $\rightarrow 0.40 I_g$	$\frac{N}{f_c' A_c} > 0.5 \rightarrow 0.80 I_g$	$\frac{N}{f_c' A_c} = 0.2 \rightarrow 0.45 I_g$
T- oder L-Q. $\rightarrow 0.35 I_g$	$\frac{N}{f_c' A_c} = 0.2 \rightarrow 0.60 I_g$	$\frac{N}{f_c' A_c} = 0.0 \rightarrow 0.25 I_g$
	$\frac{N}{f_c' A_c} = -0.05 \rightarrow 0.40 I_g$	$\frac{N}{f_c' A_c} = -0.1 \rightarrow 0.15 I_g$

wobei:

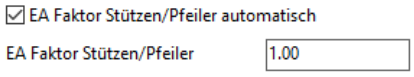
N: Normalkraft (Druck positiv)
f_c': Zylinderdruckfestigkeit
A_c: Fläche des ungerissenen Querschnitts
I_g: Trägheitsmoment des ungerissenen Querschnitts

[NZS95] New Zealand Standard: “Concrete Structures. Part 1: The Design of Concrete Structures, Part 2: Commentary”. Standards New Zealand, Wellington 1995.

☞ *The preset stiffness reduction factors can be individually overwritten in STATIK.*

EA-factors for columns and pillars

In the case of the natural vibration calculation, pure axial vibrations of vertical elements often occur early, which are not of interest in the normal case. To avoid having to work with unnecessarily many natural vibrations (long tables, more resource consumption), these axial oscillations can be "pushed backwards" by increasing the axial stiffness with an EA factor. The option dialog offers two settings:



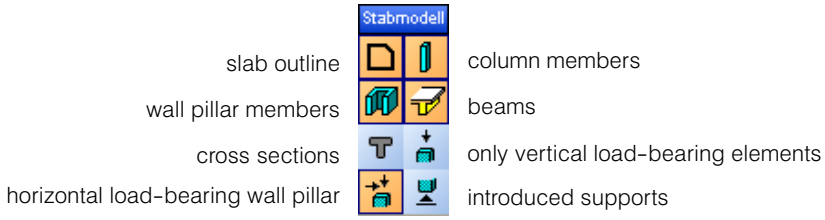
- **Control panel 'EA-factor columns / pillars automatic':** For vertical members whose stiffness has little or no influence on the oscillation behavior in the horizontal direction, an EA factor of 100.0 is automatically set. For the remaining members, the factor is set to 1.0.
  If you want to analyze vertical excitations, you have to switch off this control panel!
- **Value for EA-factor:** If the above control panel is switched off, the EA-factor set here applies to all columns and pillars. Default setting is 1.0

⚠ Care must be taken that the static model is not impermissibly corrupted by the axial stiffening of the vertical elements.

Generation of the frame model

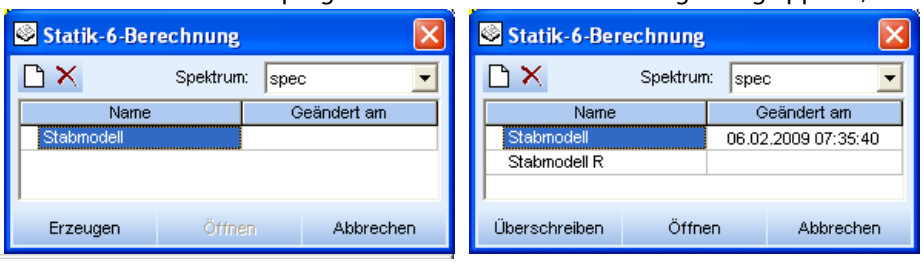
The generation of the frame model is started with the button shown. If necessary, this can be done with the neighbouring button to the right of it, which starts the program STATIK. It is recommended, however, to check the generated frame model already in the Building Module, so that one can carry out any necessary corrections before changing to STATIK.

The generated frame model is presented graphically and can be checked visually. To control the view of the frame model, besides the methods already described it has its own group in the Layer menu:



Generating the input and call STATIK

With this button the STATIK program is started and the following dialog appears,



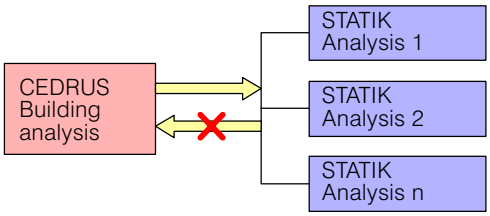
which provides the following possibilities,

a) with a new row is introduced, which can be chosen to create a new STATIK model Erzeugen ,

b) after choosing the corresponding row, overwrite an existing STATIK model Überschreiben or

c) after choosing the corresponding row, open an existing STATIK model Öffnen .

In the cases a) and b) new STATIK input files corresponding to the actual frame model are created and a new STATIK analysis with this input is performed. With c) an existing STATIK analysis is started. Thus several STATIK model variants can be managed.



☞ *There is no data flow from STATIK back to the Building Module of CEDRUS. After modifications of the building model a new STATIK model must be created. One can overwrite an existing model or generate a new one*

To work with the STATIK program one should consult the manual, especially the appendix 'On the Use of the Response Spectrum Method with Buildings'.

Frame model reset (is carried out automatically by the program as needed and is therefore only in special cases of importance).

E 3.4 Steps to Obtain a Useful Frame Model

In this chapter we describe how one obtains a suitable frame model for earthquake analyses from an existing geometric model. Not everything that one inputs as a CEDRUS slab and stacks to form a building can be transformed by the program into a suitable frame model by just pressing a button without beforehand investing some thought. Thus it is a question of tidying up the input data for the program to have a chance of generating a frame model and for this model to be able to handle seismic forces.

Before we proceed we want to acquaint the reader with an important principle:

The model has to be purged of unnecessary details. The formwork plan is not a model that can be used for calculation purposes. Earthquake analyses are so complex and of necessity involve strong simplifying assumptions.

E 3.4.1 Preparatory Work

Before generating the frame model the following preparatory work must be carried out:

❑ The walls of all **slabs** have to be checked for the following points (in the Walls dialog of CEDRUS slabs):

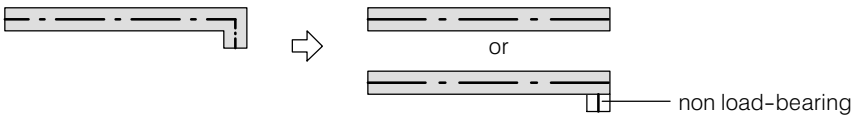
- **Support is load-bearing or non load-bearing:**

  Non load-bearing walls only transfer their own weight and are not structural elements as such

- **Support fully rotatable or general**:

  Walls with freely rotatable supports are provided with hinges about the minor axis in the frame model, as long as we are dealing with a plane wall pillar.

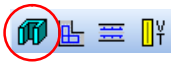
- Whenever possible **short T and L-shaped pieces at wall ends** have to be removed or input as non load-bearing.



❑ The wall pillars of all **slabs** must be checked for the following points (in the Tab sheet 'Building Elements' for the **slabs**):

- **Groups of wall segments** (see D 3.4.3):

  By default each wall segment forms an independent wall pillar, which goes into the model as a frame member. If one wants to group several segments into a single wall pillar, select them, then **<RMB>Group**. Groups can be ungrouped by selecting them and **<RMB>Ungroup**.

  ☞ *With the button*  *groups can be made visible.*

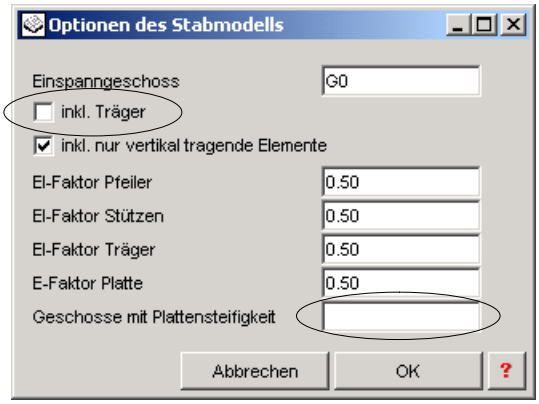
- **Wall pillar properties** (Wall Pillar dialog):

  If the check box is activated, the wall pillar does not resist horizontal loads.

E 3.4.2 The Cantilever Wall Model

We want to begin with a simple model, the Cantilever Wall model. Here the horizontal forces are withstood solely by the wall pillar and column systems fixed to the foundation. Any frame action due to the beams or the slab is not taken into account. The model is described in detail in Ch. D 5.

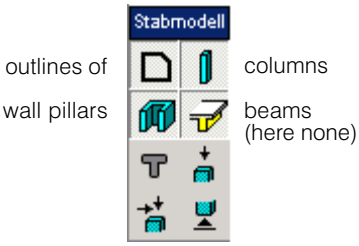
A pure Cantilever Wall model is generated, if in the Options dialog for the frame model the check box 'incl. beams' is deactivated and no story is entered for the Slab Stiffness model.

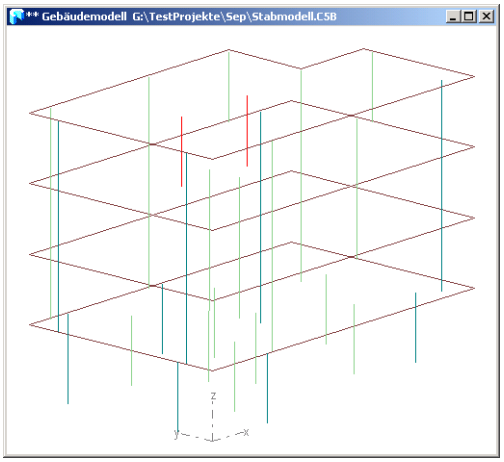


As already mentioned (see E 3.2.4), it can happen that for some vertical elements at first no clear-cut support is found. The program gives a corresponding warning and shows the correspondingly marked elements. Before describing how one can master this situation we want to draw your attention of the possibilities of visualisation control, without which an overview of the model for large buildings is easily lost.

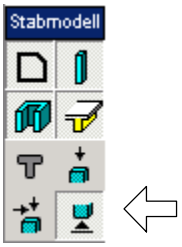
Visibilty Control

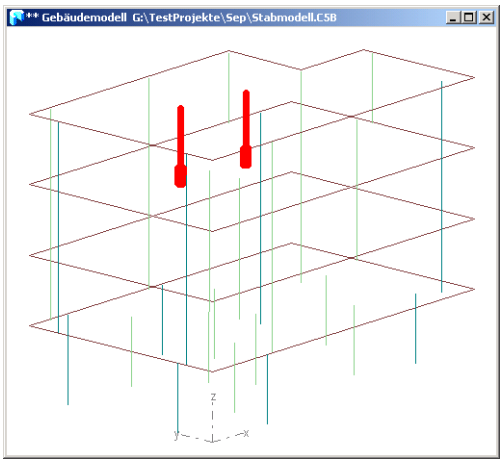
View after generating frame model:



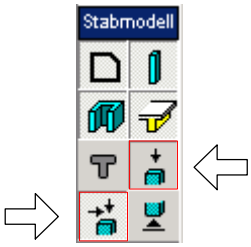


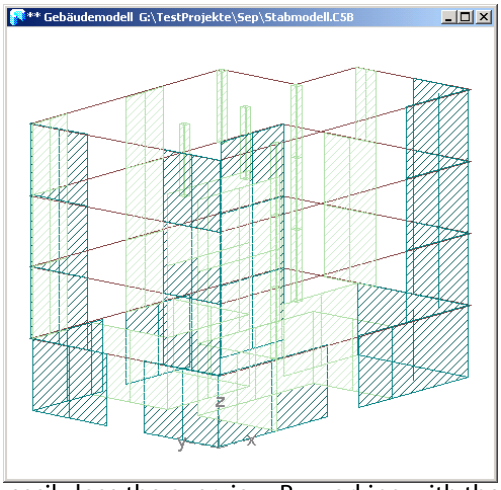
Display of additional vertical supports:



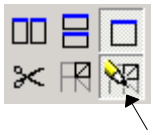


Extending vertical elements, separated into those that only resist vertical or also earthquake forces

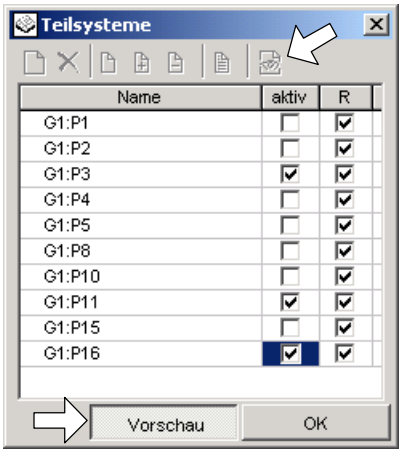


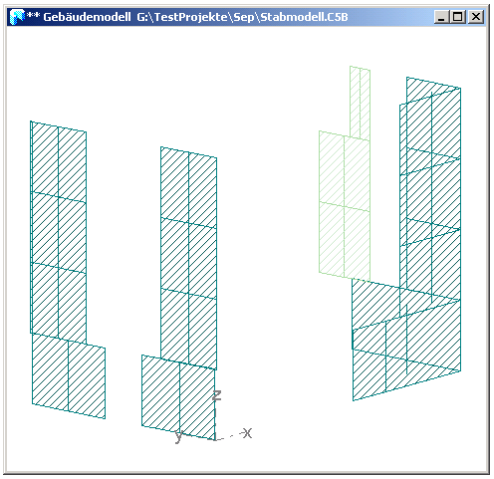


Here, even with this simple building, one can easily lose the overview. By working with the **automatically generated partial systems** for stacked elements the picture as a whole is disentangled and becomes much clearer.

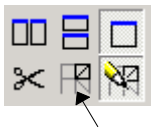


Open the Partial Systems dialog (top left in program window):

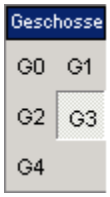


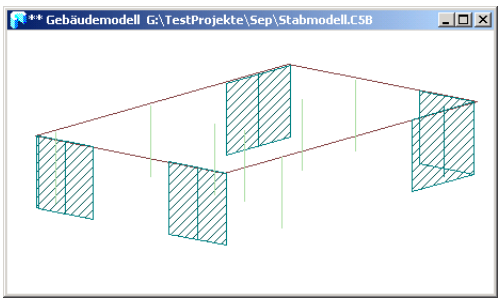


As shown, you can activate the desired partial system and the button [Preview]. Running through all partial systems is also no problem, if with the arrow you select the button at the top of the dialog (only possible if Preview is deactivated) and then select (highlight) any desired row. With the arrow key the selection can be moved down or up and so one can change from one partial system to another. If you close the Partial System dialog, the settings in the column 'active' remain active and with the button shown on the left you can change quickly from the whole system to partial system display and vice versa.

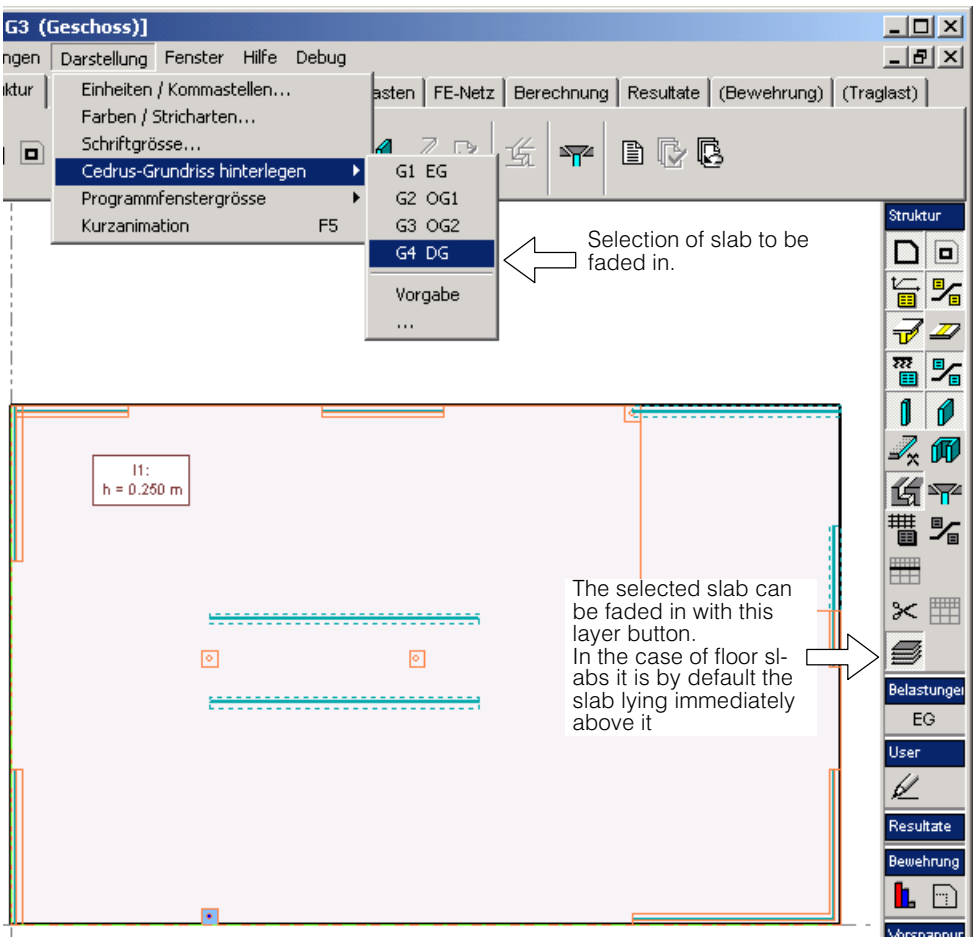


Limiting viewing to selected storys:





To tidy up the geometry it is often necessary in viewing a slab to fade in another slab in the correct position above it, in order to see why, e.g., the vertical elements do not lie above one another. What happens is shown in the following figure. The elements in plan view of the faded in slab are displayed in red and can be grabbed.



Tidying up the generated cantilever wall model

After inspecting the cantilever wall model generated by the program using the presentation method illustrated above you can decide whether you want to accept it or make changes.

Changes have to be given proper consideration, above all when the program has introduced artificial supports because vertical elements were not supported. Such cases can be dealt with as follows:

Solution 1: Accept inaccuracy of the normal forces

Keep the supports, above all if they were only introduced under elements that only support vertical loads. Thus you save yourself the bother of entering the input data again, at the cost of dispensing with more accurate results for the normal forces, which in any case you probably have or do not require in the early phase of an earthquake investigation. The moments and shear forces in the walls, both for the equivalent force or the response spectrum method, are hardly affected and the most important results are obtained quickly.

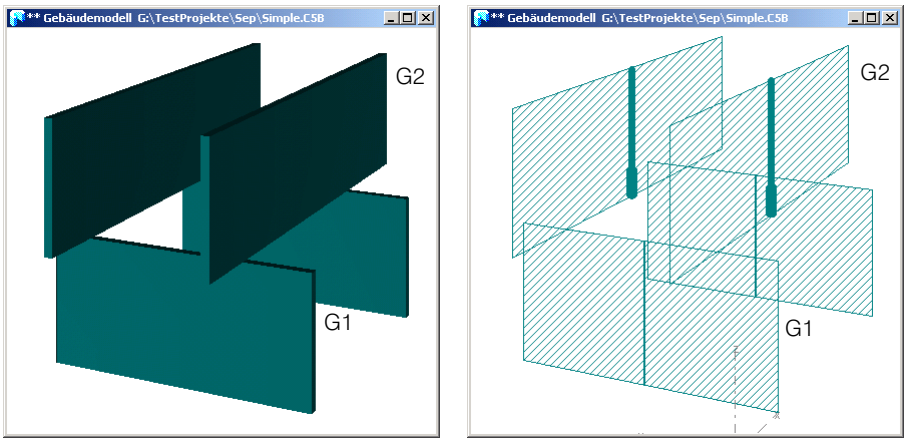
Solution 2: Take into account slab stiffness

For the story with the extraneous supports select the **Slab Stiffness Model** (Options dialog of the frame model). Thus the problem of the normal forces of Solution 1 is overcome.

Solution 3: Introduce beams as supports

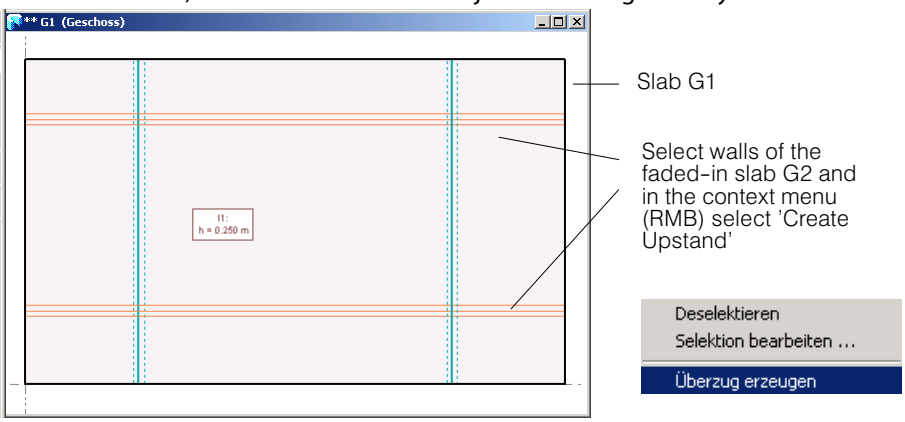
- For shear walls that rest on other vertical elements, a **support in the form of an upstand** is possible.

Example:



In the frame model (figure on right) it can be seen that the two upper walls of story G2 are not supported. We now have to introduce the upstands in the floor slab G1 under the walls of G2.

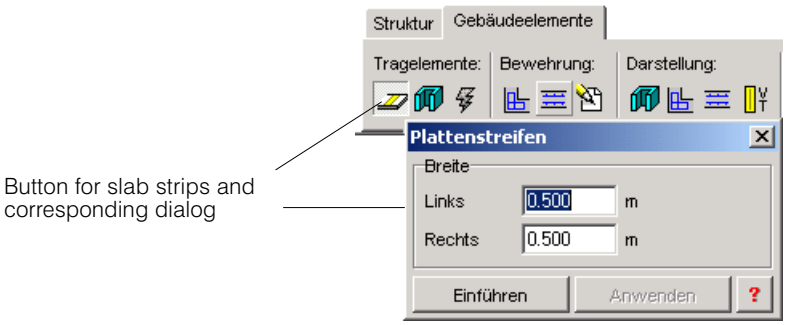
The program helps you in the introduction of such upstands. Fade in the overlying floor slab, so that the walls under which the upstands have to be introduced are visible. Select the desired wall and in the context menu select 'Create Upstand'. The upstand created is introduced with half the height of the story and weightless (weight already taken into account in the wall) and in some cases it is adjusted to the geometry of the actual slab.

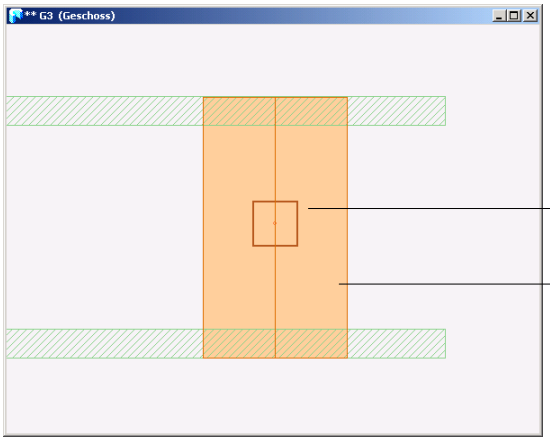


☞ *With this solution we no longer have a pure Cantilever Wall model.*

- Vertical elements, which in reality simply rest on the slab, **can be supported by means of a beam**. It could be a downstand that already exists in the slab and is already activated in the frame model, i.e. if in the Options dialog for the frame model 'incl. beams' is activated.

  If there is no available downstand, then a slab strip (Tab sheet 'Building Elements') can be introduced (☞ D 3.4.2). The axes of the vertical elements to be supported have, within a small tolerance, to lie exactly on the axes the downstands or slab strips. Slab strips also have to be activated in the Options dialog for the frame model by activating 'incl. beams'.





To allow consideration of the beams in the new frame model, before regeneration in the Options dialog for the frame model 'incl. beams' must be activated.

☞ *With solutions 2 und 3 we no longer have a pure Cantilever Wall model.*

E 3.4.3 The Frame Model

The Frame Model is selected by activating 'incl. beams' in the Options dialog. This setting has an effect in the case of storys whose slabs incorporate downstands or slab strips and and that are not input for the Slab Stiffness model.

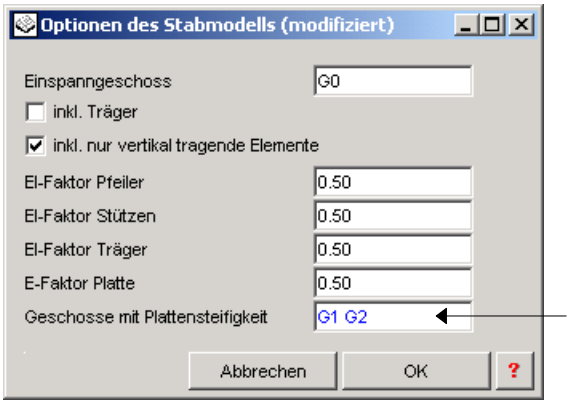
One selects the Frame Model in the cases mentioned under E 3.4.2 or if one really wants the earthquake forces to be resisted by a frame structure. Then, of course, you have to model all downstands or slab strips in the slab.

E 3.4.4 The Slab Stiffness Model

In the case of a Slab Stiffness model the frame effect of the slab has to be taken into account using an FE model of the slab.

The model can be activated for each story in the Options dialog of the frame model.

In storys with slab stiffness the setting 'inc. beams' has no effect.

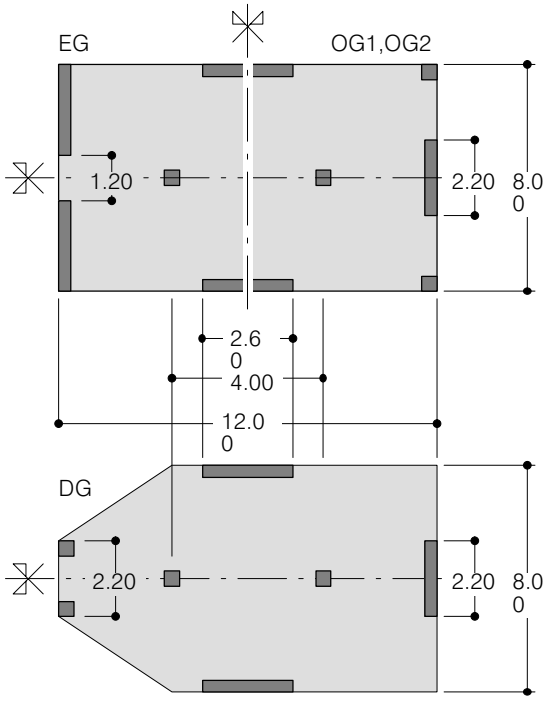


E 4 Example with Earthquake Analysis

E 4.1 Problem Description

For a four story building with a ground floor EG, two identical upper storys OG1 and OG2 and a top story DG, the following have to be dimensioned

1) with the stacked slab model analyse the loads top-down and
2) with a frame model alternatively with the equivalent method and the response spectrum method determine the sectional forces due to seismic action and dimension the load-bearing walls in the classical way (non-ductile).



slab thickness:	0.24
wall thickness:	0.20
columns:	0.25x0.25
material:	concrete C25/30
nodes	EG: 3.0
	OG1: 5.8
	OG2: 8.6
	DG: 11.40
perm. loads	self-weight
EG,OG1,OG2	+ 1 kN/m2
DG	+ 5 kN/m2
variable loads (living area):	
EG,OG1,OG2	3 kN/m2
DG	2 kN/m2
seismic zone	3B
soil class	C
structure class	I
behaviour factor	2.0

For the seismic analysis the stiffness of the walls must be multiplied by a factor 0.4.

E 4.2 Solution Part 1 with CEDRUS

In the following it is assumed that you have experience with the use of the basic version of CEDRUS.

E 4.2.1 Starting the Program

▶ Start CEDRUS and open a new Building Analysis using the menu 'File > New... > Building Analysis'.

Now you can see the Building Window with the Building Definition dialog.

▶ In the menu 'Settings > Materials' set the desired material class 'C25/30' for the given material 'Concrete'.

☞ *The building materials can only be changed in the Building Window and apply for all slabs.*

E 4.2.2 Input of the Ground Floor Slab

Since there is only one ground floor slab in the building, we want to input it directly as a floor slab.

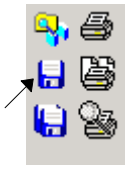
▶ Click on the button for a new, empty story (in the story section of the Building Definition dialog). Write 'EG' in the field of the column 'Description'.

A new floor is input in this way, its floor slab (still empty) was opened and its document window was placed over the Building Window. The shown level of 3.0 corresponds to the specified value.

If the ground floor slab already exists as an independent CEDRUS analysis, with the menu 'File > Import > CEDRUS-?' we can import it. This is not the case here, so the slab is input in the familiar way in the CEDRUS Base Module.

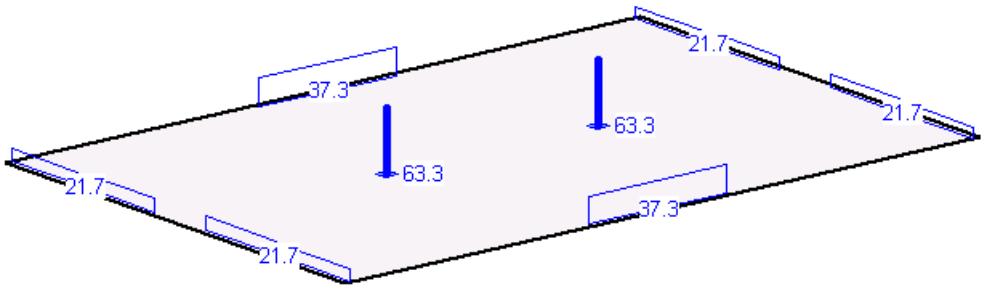
▶ Input as usual the ground floor slab including loads.

Note:
- area-supported columns, vertically rigid, are rotationally free
- walls supported freely rotatable, vertically rigid



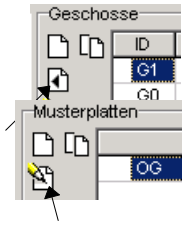
▶ Save the slab data.

▶ For a plausibility check, analyse, e.g., the bending deflections and the reactions for the live load.



E 4.2.3 Input of the Master Story for the Upper Storys

Since the two upper storys are identical, it is useful to input first a master story. But since this master story is similar to the ground floor slab we can make use of the possibility of creating a master story as a copy of an independent story.



▶ Select story 'G1' and then the button shown.

In this way a master story is created as a copy of the existing story 'G1'.

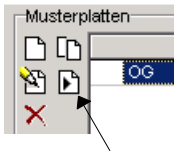
▶ Rename the master story 'OG' and then use the button to work on the slab.

▶ Modify the slab according to the input description, i.e. introduce
1)the walls in the Y direction
2) and the 4 corner columns

▶ Save the slab data.

E 4.2.4 Introducing the storys OG1, OG2 and DG

The two upper storys OG1 and OG2 as well as the top story DG can now be generated from the master story 'OG' .

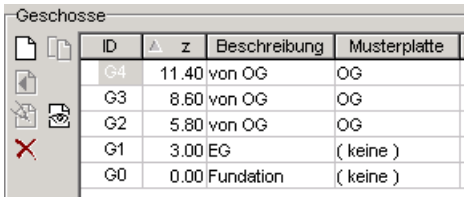


▶ Select the master story 'OG' in the Building Definition dialog and then activate the shown button once.

Thereby a new independent story 'G2' is created from 'OG'.

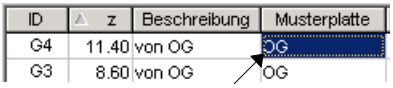
▶ Change the level of G2 to 5.8.

▶ Repeat twice the last but one point to generate the storys G3 and G4. For the new nodes each time the story height of the underlying story is automatically given (by default), which in the present example leads directly to the desired nodes:



In 'G4' we now have to modify the loads according to the problem description. Thus in this way the story slab is differentiated from the master story and, before we can make changes to 'G4', we have to make this story independent of the master story.

▶ Click on the field G4-'Master story'.



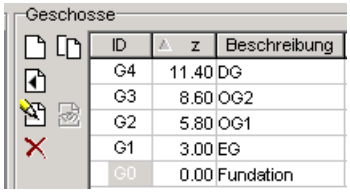
Use the arrow button that appears to open the selection list and select '(none)'.

Thereby the story slab 'G4' is uncoupled from the master story and is now independent. Now changes to the master story only affect 'G2' and 'G3'.

▶ Next, in the live load loading, change the load value to -2 kN/m^2 and the additional permanent load to -5.

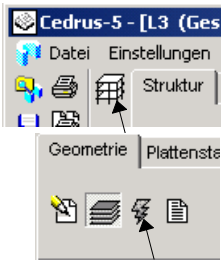
▶ Save the slab data.

▶ Change the descriptions of the storys as follows:



E 4.2.5 Generation of the Building Model

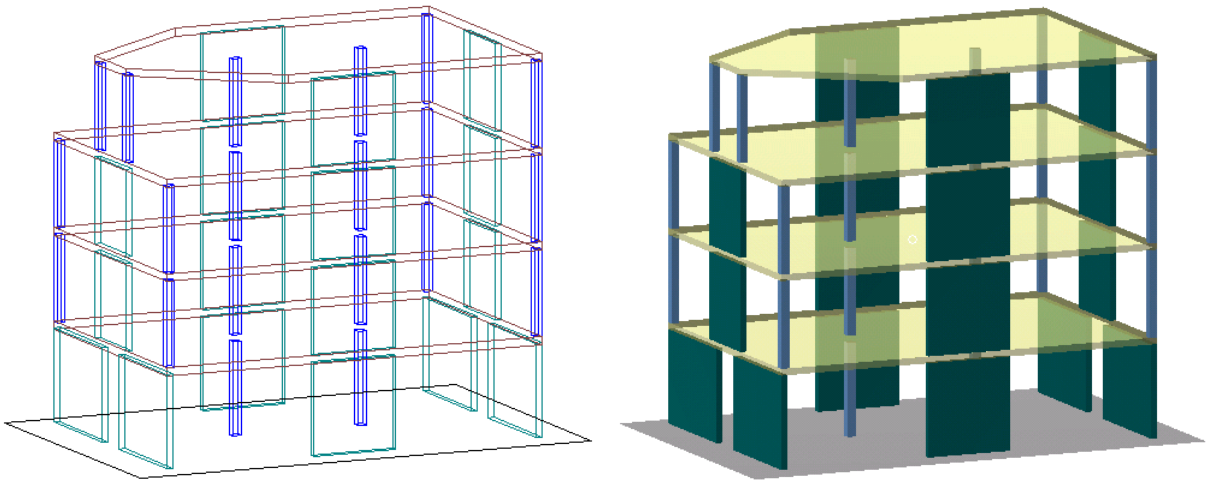
We must now go back to the Building Window. This is best done as follows:



▶ Use the shown button above left in the Program Window

and the Building Window appears again.

▶ Use the lightning button to generate the geometric building model. Answer any questions posed by the program in the affirmative.



Note the layer button for controlling the visibility. With the button in the group 'Storys' the visibility of the individual storys can be set individually and with those in the group 'Geometry' visibility can be selected for individual types of structural element.

☞ *In the rendered presentation (Fig. above on right) many functions, especially the change of Tab sheet, are blocked.*

E 4.2.6 Stacked Slab Model

Reminder: The Stacked Slab model is used to analyse a building for vertical loads. The program investigates how the loads are transmitted via the slabs, walls and columns to the foundation and the floor slabs are dimensioned individually, as in the basic CEDRUS module.

▶ Change to the Tab sheet 'Stacked Slab Model' and using the lightning button create the Stacked Slab model.

▶ Activate the layer button for 'Load Transfer'. This produces a figure that allows one to check how walls and columns transfer their loads (here trivial).

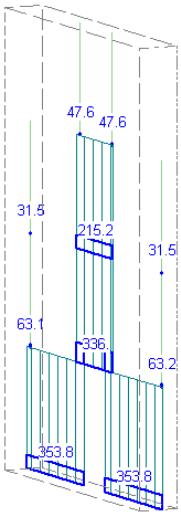
▶ This starts the analysis of the Stacked Slab model.

Results in the vertical elements

The results, in each case at the bottom of vertical elements, can be output graphically (using the layer button shown here) and in tabular form using the button .

Switching on and off of the labelling of the graphics results is done in the context menu (<RMB>) of the corresponding layer buttons.

☞ *Consider the possibilities of the Clipping Box (Fig. right) or the partial systems, to restrict the visibility as required*



Results in the floor slabs

The results in the individual floor slabs are also available. Open the floor slab using the button shown below and request the results, as in the basic CEDRUS module.

dependent on master story:

independent of master story:

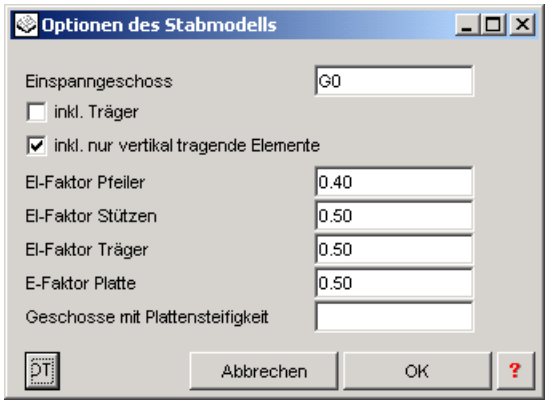
E 4.2.7 Frame Model

Reminder:

- The frame model is used to analyse a building for horizontal loads.
- Wall pillars which resist and transmit horizontal loads have to be specified accordingly (see E 3.2.2). By default this is the standard setting for all wall pillars.

▶ Change to the Tab sheet 'Frame Model' and activate the button for the Options of the frame model:

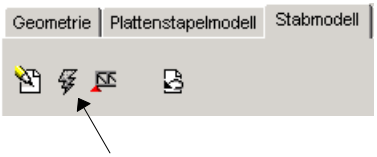
▶ The dialog that appears should be set as follows:



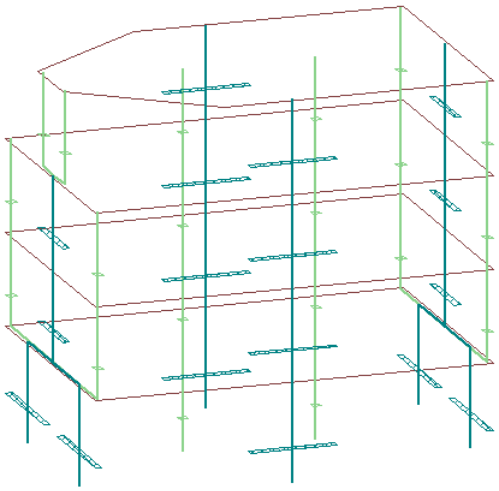
Remarks on the selected setting:

- No beams are input (pure cantilever wall model)
- Without 'Only vertical load-bearing elements' the columns would not be modelled, which would give just a very rudimentary model.
- The bending stiffnesses (here only relevant for the wall pillars) are multiplied by a factor 0.5 according to the problem specifications (takes into account the cracked state of the concrete)

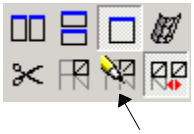
▶ Close the dialog and activate the button to create the frame model:



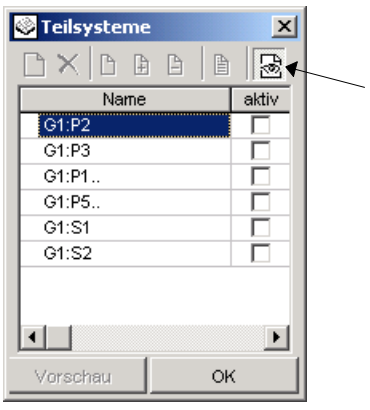
▶ With the layer button on the left, the generated connections of the individual members can be made visible.



The program automatically creates a partial system for each wall tower (stacked contiguous wall/column systems). We will look at them more closely.

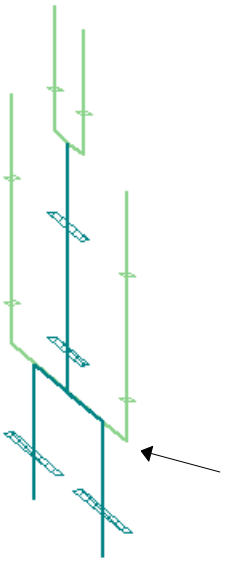


▶ Open the Partial Systems dialog and click on the button in the shown dialog.



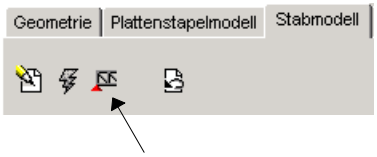
▶ Use the arrow key (down/up arrow) , to move from one partial system to another and view all.
Example:

Thus this completes the frame model as far as CEDRUS is concerned, and it should now be prepared for the dynamic analysis with STATIK.



On this story all member end nodes are rigidly connected (eccentric connections to a common node)

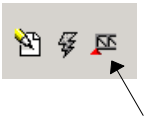
▶ Use the button to **call STATIK**:



▶ In the dialog that appears activate the button for a new STATIK model and then click on [Create].

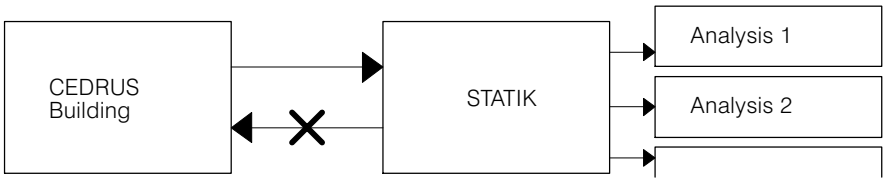
E 4.3 Solution Part 2 with STATIK

It is assumed that you have experience in using the Base Module of STATIK. In addition you should consult the chapter on Dynamics in the STATIK manual and the supplementary documentation 'On the Use of the Response Spectrum Method for Buildings'.



Using this button you can start STATIK with the input data created in the Building Module. STATIK now runs in a seccond program window completely independent of CEDRUS.

☞ *Starting STATIK from the Building Module is always a one-way process. You can start STATIK over and over again from the Building Module and create thereby a new STATIK analysis or work with an existing STATIK model.*



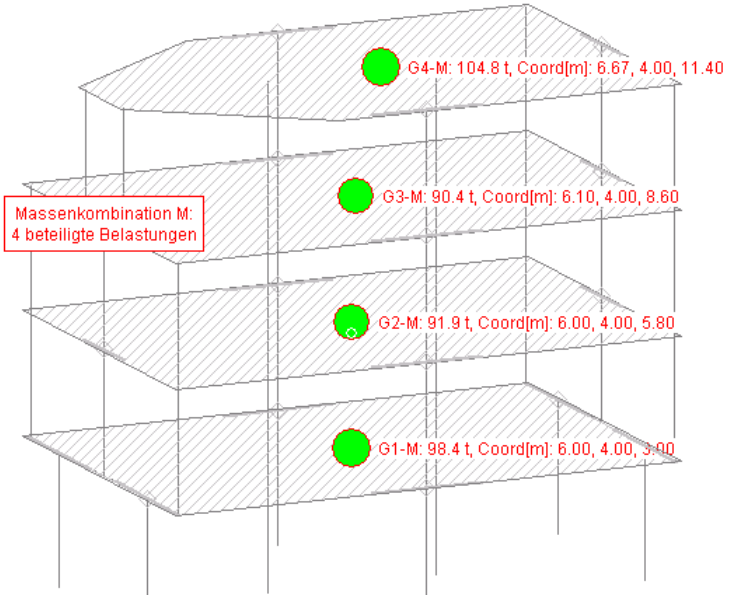
E 4.3.1 Checking the Generated Model

Look at the automatically generated STATIK model (e.g. the structure legend...).

▶ Change to the Tab sheet 'Loads' and click on the button to check the loadings (if active).

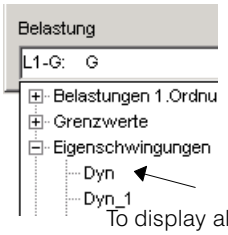
▶ Now select from the loadings list the mass combination M.

You can now see the masses summed for each story. The position of the circle representing the masses corresponds to the centre of gravity.



You can also check that for each story three loadings were generated:
Li.G: dead loads
Li.Q: live loads
Li.M: loads for mass distribution

E 4.3.2 Free Vibration Analysis



▶ Change to the Tab sheet 'Results' and select the results of the free vibration analysis.

The specification of the free vibration analysis was created automatically (for 10 eigenvalues). Changes to the specification can, as usual, be carried out in the Tab sheet 'Analysis'.

Discussion of the results:

To display all 3 tables at the same time

Eigenfreq. Dyn \Result\Dynamic\Frequencies\Dyn

EIGENFREQUENZEN

'für Berechnung: Dyn; Massenbelastung=M"

EW-Nr.	ω^2 [(rad/s)²]	ω [rad/s]	Periode [s]	Frequenz [s⁻¹]
1	97.82	9.890	0.6353	1.57
2	152.71	12.358	0.5085	1.97
3	553.24	23.521	0.2671	3.74
4	3146.84	56.097	0.1120	8.93
5	3835.26	61.929	0.1015	9.86
6	14629.58	120.953	0.0519	19.25
7	25749.25	160.466	0.0392	25.54
8	25773.03	160.540	0.0391	25.55
9	70396.31	265.323	0.0237	42.23
10	75877.65	275.459	0.0228	43.84

MODALE PARTIZIPATIONSFAKTOREN

'für Berechnung: Dyn; Massenbelastung=M (Massennormiert)"

EW-Nr.	X	Y	Z
1	16.49505668	-0.00000697	-0.00172803
2	-0.00001079	-14.25074397	0.00001487
3	-0.00015714	0.18486891	-0.00043034
4	0.00007210	-7.72701439	-0.00029733
5	9.16007547	0.00006844	0.00636254
6	0.00004529	-4.42588785	0.00302276
7	0.00649529	6.26084947	0.00294149
8	4.94427945	-0.00820160	0.01304809
9	-2.25664256	0.00005723	0.00686274
10	-0.00004676	-5.20085013	0.00854545
Σp_i	385.53225480	348.66026021	0.00035191

relevant 1st X mode
relevant 1st Y mode
1st torsional mode
2. Y mode
2. X mode
3. Y mode

MODALE MASSEN (sortiert nach Grösse pro Richtung)

'für Berechnung: Dyn; Massenbelastung=M"

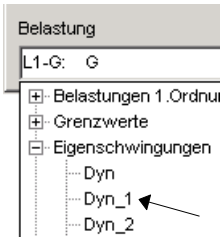
EW-Pos.	X-Richtung EW-Nr.	me_X [t]	Σme_X [%]	Y-Richtung EW-Nr.	me_Y [t]	Σme_Y [%]	Z-Richtung EW-Nr.	me_Z [t]	Σme_Z [%]
1	1	272.1	70.6%	2	203.1	52.7%	8	0.0	0.0%
2	5	83.9	92.3%	4	59.7	68.2%	10	0.0	0.0%
3	8	24.4	98.7%	7	39.2	78.3%	9	0.0	0.0%
4	9	5.1	100.0%	10	27.0	85.3%	5	0.0	0.0%
5	7	0.0	100.0%	6	19.6	90.4%	6	0.0	0.0%
6	3	0.0	100.0%	3	0.0	90.4%	7	0.0	0.0%
7	4	0.0	100.0%	8	0.0	90.4%	1	0.0	0.0%
8	10	0.0	100.0%	5	0.0	90.4%	3	0.0	0.0%
9	6	0.0	100.0%	9	0.0	90.4%	4	0.0	0.0%
10	2	0.0	100.0%	1	0.0	90.4%	2	0.0	0.0%

Gesamtmasse = 385.5 t

With modes 1 and 5 92.3% of the masses are excited for an excitation in the X direction

With modes 1 and 5 90.4% of the masses are excited for an excitation in the Y direction

An excitation in the Z direction is only of interest in special cases. In order not to eliiminate Z modes that are of interest at low frequencies, the normal force stiffness of the vertcal elements was increased!



▶ You can also see the mode shapes graphically. You will be able to see the properties in the numerical tables confirmed in the visually appealing graphics.

Often the buildings and the load-bearing walls are not completely symmetric and (esp. for frame models) the analysis of a larger number of eigenvalues may be necessary before the decisive modes are included in order to be able to excite 90% of the masses as prescribed in the code.

E 4.3.3 Equivalent Forces

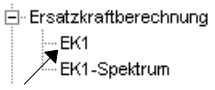
An equivalent force analysis is automatically generated by the Building Module. Changes to the specification can, as usual, be carried out in the Tab sheet 'Analysis'. This also applies to the default spectrum, which has to be modified for the actual case investigated.

▶ Change to the Tab sheet 'Analysis' and click on the button to modify spectra.

▶ Select the only available spectrum 'Spec' in the list that appears and then click on the button to carry out modifications.

▶ Make the necessary changes and exit the open dialogs.

seismic zone	3B
soil class	C
building class	I
behaviour factor	2.0



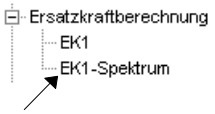
▶ In the Tab sheet 'Results', in the Results Tree choose the branch Equivalent Force Analysis and in it choose 'Equiv' and request the results table.

Ersatzkräfte F_{ri}

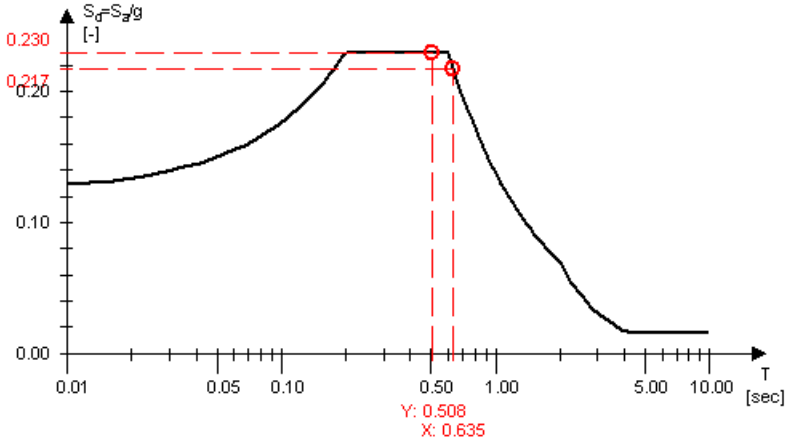
"für Berechnung: Erk; Massenkombination=M"

Kote [m]	Gewicht [kN]	Σ Gewicht*λ [kN]	X-Richtung GFak	X-Richtung F_{di} [kN]	X-Richtung $V_x = \Sigma F_{di}$ [kN]	Y-Richtung GFak	Y-Richtung F_{di} [kN]	Y-Richtung $V_y = \Sigma F_{di}$ [kN]
11.40	1048.17	1048.17	11.400	**357.34**	357.34	11.400	**378.35**	378.35
8.60	903.65	1951.82	8.600	**232.40**	589.74	8.600	**246.07**	624.42
5.80	919.05	2870.87	5.800	**159.41**	749.14	5.800	**168.78**	793.20
3.00	984.41	3855.28	3.000	**88.32**	837.46	3.000	**93.51**	886.71

I	:	1.00 (Massenreduktionsfaktor)
Verteilung	:	Gewichtsfaktoren = Kote (linear)
Spektrum	:	Spec ,Zone=3b,BK=C,BWK=I,g=1.00,q=2.00,x=0.05
Perioden	:	aus Eigenwertberechnung: Dyn
T_x	:	0.635 [s-1]
M_{eff}/M_{tot}	:	0.71
Sa_x	:	0.2172
T_y	:	0.508 , von Eigenmode 2
M_{eff}/M_{tot}	:	0.53
Sa_y	:	0.2300



▶ Select the result 'Equiv-Spectrum' and display the spectrum.



For both directions of excitation it shows the calculated periods and the associated acceleration values used for the equivalent force analysis.

Note on the equivalent force method of analysis:

- The requirements for the equivalent force method are laid down in the codes.
- To determine the equivalent forces, the fundamental modes of a building in the X and Y directions together with the periods are required.

  In the specification Equivalent Force Analysis it can be specified that the fundamental modes can be determined automatically using a free vibration analysis. To find these fundamental modes the program first searches in each direction for the fundamental mode with the largest modal mass. If this modal mass is smaller than 50% (can be specified) of the total mass, this search is treated as unsuccessful.

  If fundamental modes were found, these still have to fulfil the following condition: The ratio of the modal mass in the considered direction to that in the direction normal to it must have a minimum value of 10.0 (can be specified). This ensures that these are unique and independent fundamental modes in the directions X and Y. An assessment of the fundamental modes of vibration used has to be checked by the user in every case.

  If the above conditions are not fulfilled, the program uses the maximum spectral values to determine the equivalent force. In the Specification dialog, however, one can also input manually the periods or switch to 'Maximum Spectral Values' (in this case no periods are required). The usefulness of the results thus generated has to be assessed by the user.
- **When carrying out an equivalent force analysis the corresponding loadings are automatically created**

  These are

  1) the **loadings with the substitute forces** in the X and Y directions with the identifiers **xxx%X** and **xxx%Y** (xxx=identifier for the equivalent force analysis) in the centres of mass. *Since, in general, there are no nodes in the centres of mass the forces, together with the corresponding moments, are introduced in the neighbouring nodes.*

  2) the **torsional loadings** to take into account inaccurately determined and incidental eccentricities of the centres of mass in relation to the centres of stiffness (for the identifier see below).

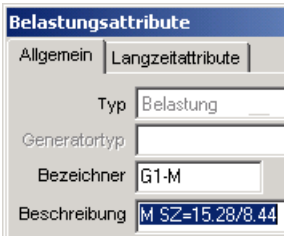
  According to SIA261 16.5.2.7 the masses must be considered by applying a factor 0.5e (e=eccentricities of the centres of mass with respect to the centres of stiffness) or 0.05b. b corresponds to the width of the building normal to the direction of loading and is determined as the difference of the maximum and minimum coordinates of the respective floor slab (load area) (can be corrected, see below). Thus the total corrected eccentricity is

  $e_{korr} = \pm (0.5e + 0.05b)$ which can vary from story to story.

  ☞ *In the code it states that $e_{d,sup} = 1.5e + 0.05b$. The 1.5 instead of the 0.5 in the above formula in the code assumes that the equivalent forces are introduced at the centre of stiffness and that for torsion therefore the total mass eccentricities $e_{d,sup}$ and $e_{d,inf}$ have to be used. In our case the equivalent forces are introduced in the centres of mass, which is why for the torsional loadings only the correection values e_{korr} have to be considered.*

  The eccentricities e_i are determined as the differences between the centres of mass and centres of stiffnesss in STATIK.

  The centres of mass can be viewed in the graphics as a mass combination 'M' or in the numerical output as the mass distributions in the storys.



  *The centres of stiffness are determined in the Building Module of CEDRUS and exported in the descriptions of the loadings for the story masses to STATIK. If these descriptions contain text in the form **SZ=15.28/8.44**, this indicates that the centre of stiffness of the correspoinding story is at X=15.28 and Y=8.44 (values in [m]). If necessary, these descriptions can be changed manually by the user.*

  *The widths b can also, if desired, be overwritten. This is done, as in the case of the centres of stiffness, in the description of the loadings for a story mass. After a blank insert the text B=bx/by (Example **B=6.25/9.98**). bx denotes the story dimension in the Y direction, i.e. the width that is needed for a loading in the X direction.*

To check the values used together with the equivalent force table the following table is also output:

Torsionsmomente infolge Zusatzexzentrizitäten

"der Massenzentren für Ersatzkraftberechnung: Erk"

Kote [m]	Richt.	SZ [m]	MZ [m]	e=MZ-SZ [m]	b [m]	F_{di} [kN]	F_{di}(0.5e+0.05b) [kNm]	F_{di}(0.05b) [kNm]
11.40	X	4.00	4.00	0.00	8.00	357.34	142.95	142.93
11.40	Y	11.79	6.67	5.12	12.00	378.35	1195.16	227.01
8.60	X	4.00	4.00	0.00	8.00	232.40	92.96	92.96
8.60	Y	6.00	6.10	0.10	12.00	246.07	160.01	147.64
5.80	X	4.00	4.00	0.00	8.00	159.41	63.76	63.76
5.80	Y	6.00	6.00	0.00	12.00	168.78	101.27	101.27
3.00	X	4.00	4.00	0.00	8.00	88.32	35.33	35.33
3.00	Y	6.00	6.00	0.00	12.00	93.51	56.11	56.11

Richt. : Belastungsrichtung
SZ : Steifigkeitszentrum
MZ : Massenzentrum
F_{di} : Ersatzkraft im Massenzentrum
F_{di}(0.5e+0.05b) : Torsionsmoment für Ersatzkraftberechnug
F_{di}(0.05b) : Torsionsmoment für Antwortspektrenanalysen

Coordinate values normal to direction of loading

The torsional loadings, which have to be taken into account in the case of the equivalent force analysis, are identified by **xxx%TEX** and **xxx%TEY** (xxx = identifier for the equivalent force analysis) and for each story contain a torsional moment about the global Z axis equal to $\mathbf{M_{iZ} = F_{di}\, e_{i,korr}}$.

According to SIA261 16.5.3.4 with the response spectrum method only the second term of e_{korr} has to be considered. With this method the program also creates corresponding torsional loadings obtained from the equivalent force analysis, which are denoted by **xxx%TAX** and **xxx%TAY**.

In performing the equivalent force analysis the following three limit state specifications (accidental design situation) are also created:

- **!EE!xxx:** (xxx = name of the equivalent force analysis)
  for verification and dimensioning according to the equivalent force analysis:

  Besides the permanent loads G and the variable loads Q this limit state specification also contains the equivalent loads with the loadings in the X and Y directions and the corresponding torsion loadings xxx%TEX/Y alternatively in the eight possible algebraic sign combinations.

- **!EAtot!xxx:** (xxx = name of the equivalent force analysis)
  for the output of limit state values of individual quantities using the response spectrum method (without corresponding components):

  Besides the permanent loads G and the variable loads Q this limit state specification also contains the torsional loadings xxx%TAX/Y for random mass eccentricities determined on the basis of the equivalent loads together with the results of the response spectrum analysis Aws_K2.

- **!EApart!xxx:** (xxx = name of the equivalent force analysis)
  for reinforced concrete analyses (dimensioning of reinforcement), in which the response spectrum results are imported separately from the limiting values of section forces from vertical and torsional loading:

  Besides the permanent loads G and the variable loads Q this limit state specification also contains the torsional loadings xxx%TAX/Y for random mass eccentricities determined on the basis of the equivalent loads.

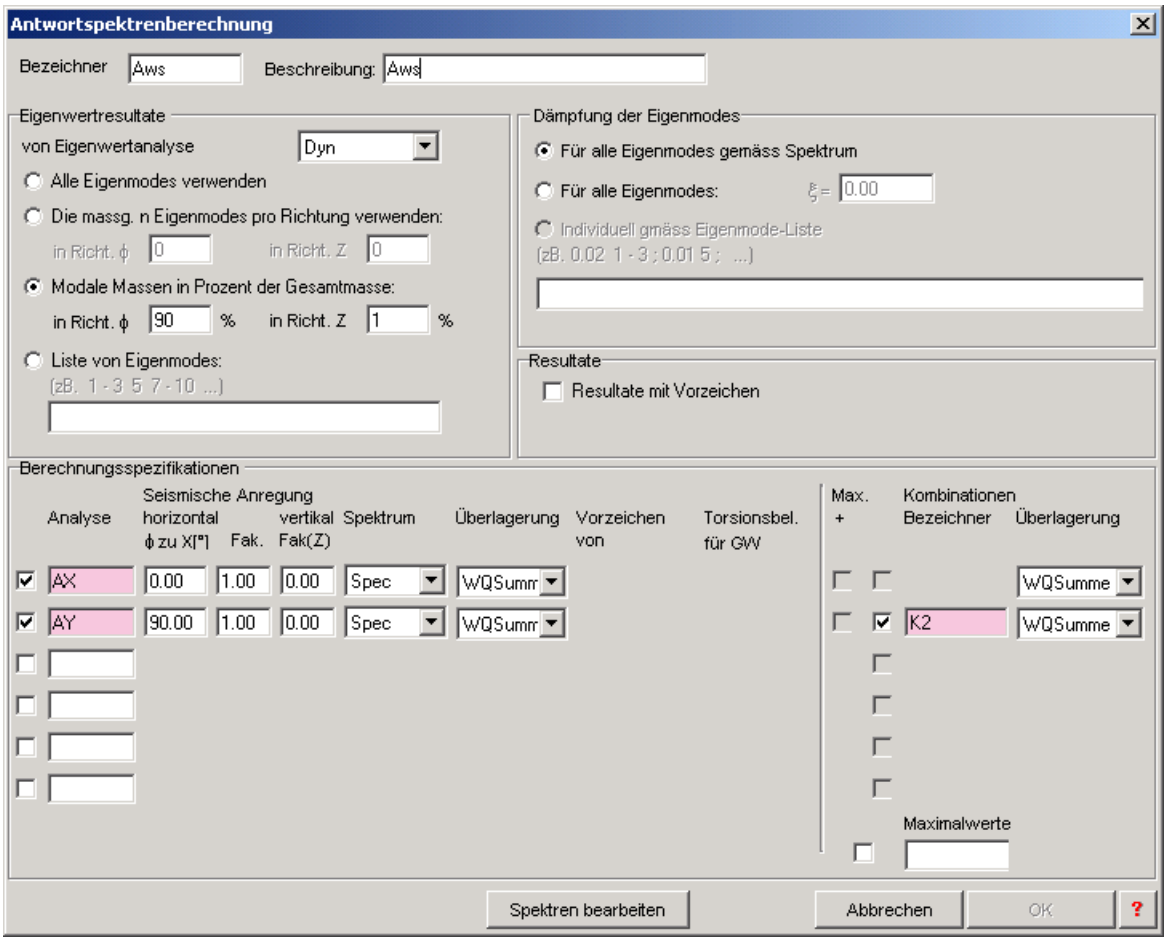
E 4.3.4 Response Spectrum Method

If the requirements for the equivalent force method are not fulfilled, the response spectrum method can be used.

☞ *Even if the code allows the equivalent force method to be used, the stresses with the response spectrum method may often be substantially smaller in imprtant parts of the struc-*

ture. Since, when using the response spectrum method for the dimensioning of the load-bearing walls, it is not necessary to take into account higher frequencies (see e.g. SIA 262, 4.3.9.4.4), normally more economic solutions are possible than with the equivalent force method.

A response spectrum analysis is automatically generated by the Building Module. Changes to the specification can, as usual, be carried out in the Tab sheet 'Analysis'. The following figure shows the dialog for the state as generated



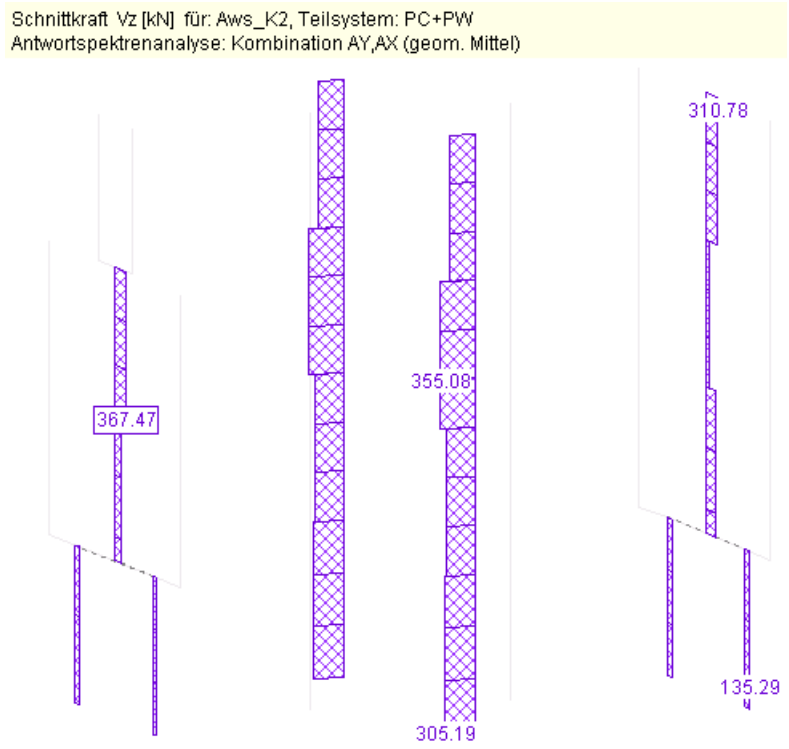
Two analyses 'AX' and 'AY' are specified for an excitation in the X and Y directions and the combination K2 of both results. In this example, the latter is effectively only the summary of the X and Y results, since due to symmetry the two directions depend on each other.

☞ *The spectrum 'Spec' used was also automatically generated and has to be adjusted therefore to actual needs [modify spectrum].*

For detailed information on the response spectrum method see the documentation of STATIK.

The following results are by default:

▶ Request the shear forces and moments in the wall pillars for the combination 'K2'. For this purpose use the automatically defined partial system 'W' for the wall pillars.



Torsion

The torsional effect due to random eccentricities of the centres of mass and stiffness of the individual storys has to be taken into account according to the codes. This may be done in several ways:

- For each direction of excitation considered one creates a loading with M_z=eF in each story plane. (e=eccentricity, F=equivalent force for the story). This loading has to be taken into account with its algebraic sign. The absolute values of these results have to be added to those obtained from the response spectrum analysis.

  This method is recommended in the Eurocode.

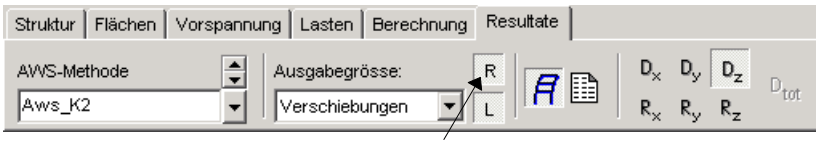
  ☞ *When creating an equivalent force analysis the load directions are created automatically for the torsional loadings. They are called XXX%T**E**X, XXX%T**E**Y (XXX = name of the equivalent analysis)*

- The required eccentricity of the masses can be input directly in the Specifications dialog of the free vibration analysis. Disadvantage: To include all relevant cases a series of independent free vibration and response spectrum analyses has to be carried out.

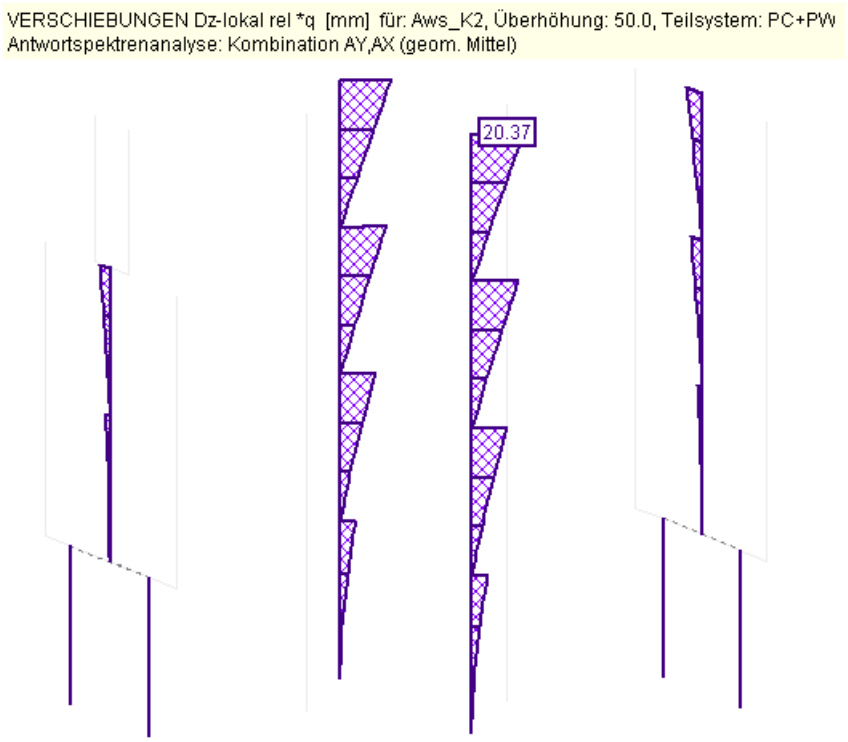
Interstory drift

Sometimes the maximum horizontal interstory drift is required. To determine this correctly, one cannot simply subtract the displacements at the top and bottom of a wall pillar that are given by a response spectrum analysis, since these results represent the maximum values determined for each section. Rather, one has to find the displacement difference for each eigenmode using the superposition method (e.g. square root of the sum of the squares SRSS).

STATIK provides correct interstory drift results for the response spectrum method individually for each vertical member.



Click on the button [R], which appears in the case of displacement results from response spectra analyses. The results can be obtained in local or global directions (button under it). In local directions the components Dy and Dz and in global directions normally the components Dx and Dz are of interest.



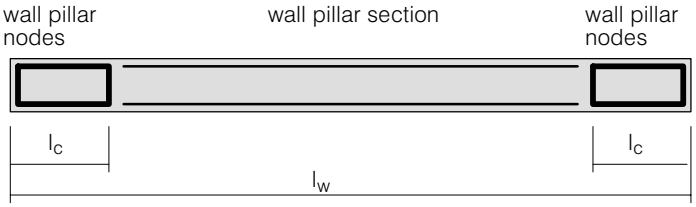
E 4.3.5 Dimensioning (Reinforced Concrete Analyses)

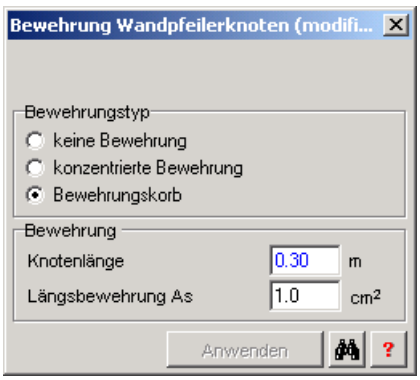
Here the reinforcement is dimensioned conventionally, i.e. for non-ductile behaviour. In general, verifications are possible, provided all reinforcement is input with the actual cross sectional areas.

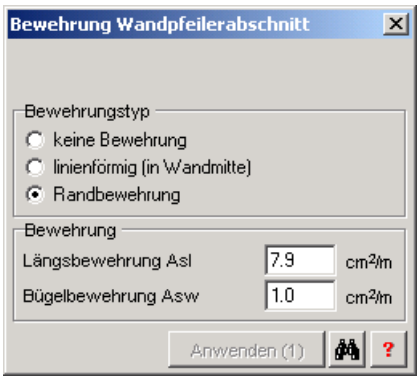
The reinforcement of load-bearing walls

To be able to dimension the reinforcement, the reinforcement layouts in the walls pillars have to be available. Not to make the first part of the example too complicated, we dispensed with the reinforcement specification details in the building's input.

Now we want to make up for this. The reeinforcement is defined in the master story and th independent floor slabs in the CEDRUS Building Module (see Chapt. D 3.4.4). The reinforcement should be as follows in all wall pillars:







The selected reinforcement type 'Reinforcement Cage' denotes a uniformly distributed longitudinal reinforcement along the rectangle shown. The condition for the length of the wall pillar nodes denoted by l_c in SIA 262: 5.7.1.2 are:

- $l_c \geq 300$ mm (applies for all short walls)
- $l_c \geq l_w/10$ (applies to longer walls on the ground floor: l_c=34 cm)

The longitudinal reinforcement is dimensioned in the wall pillar nodes. That in the wall pillar sections remains constant, as input.

The stirrups reinforcement is dimensioned and can be input with a factor 1.0.

▶ Start the Building Analysis and input the reinforcement in the floor slabs 'G1' and 'G4' as well as in the master story 'OG' . Save the slab data.

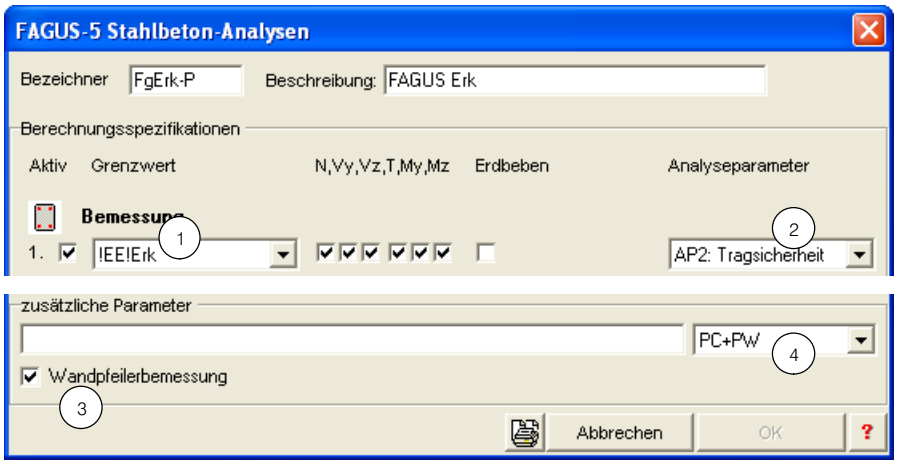
▶ Generate the frame model again by overwriting the old model, i.e. with 'Overwrite' .

The STATIK model is thereby overwritten, with the exception of any existing spectra.

You are now in the STATIK program again.

Dimensioning of the wall pillar reinforcement for equivalent forces

The reinforced concrete analysis 'FgErk-P' was created for this purpose. It can be viewed in the Tab sheet 'Analysis'. The associated dialog looks as follows:



1 The dimensioning is based on the limit state forces for the section forces, which were developed using this limit state specification.

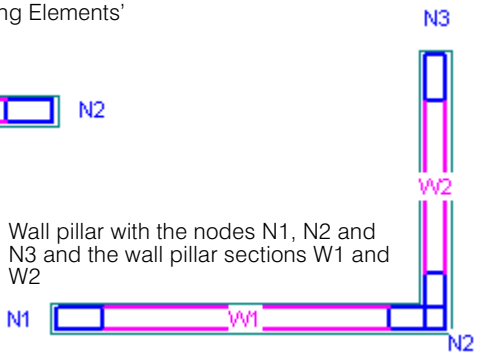
2 For the dimensioning, FAGUS uses the set of analysis parameters under the name shown here. Sets of analysis parameters can be viewed in the menu 'Settings > Analysis Parameters' and if necessary modified.

3 If this control field is active (default for wall pillars in the Building Module), then the output for the reinforced concrete dimensioning is not in the standard form for this module, but is adjusted to the specification of the wall pillars from the Building Module. If one wants the standard output, the control field can be deactivated.

The output takes into account the numbering of the wall pillar nodes and sections in the corresponding floor slab of the Building Module. These numberings can be viewed in the 'Building Element' Tab sheet of a floor slab (Activation of labelling with Layer button for wall pillars):

Representation of the Tab sheet 'Building Elements' of a floor slab (CEDRUS):

Shear wall with the nodes N1 and N2 and the wall pillar section W1

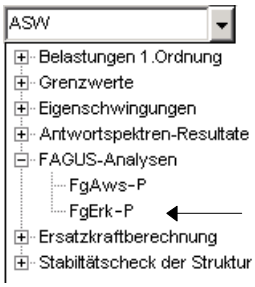


Wall pillar with the nodes N1, N2 and N3 and the wall pillar sections W1 and W2

4 The dimensioning is carried out in the partial system 'PC+PW', which comprises all wall pillars (simple shear walls and groups of wall pillars).

☞ *When working with a frame model, which also contains beams in addition to the wall pillars and columns, the Building Module generates a second reinforced concrete dimensioning called 'FgErk-B', in which the partial system for the beams is specified and the control field 'Wall Pillar Dimensioning' is deactivated. In this case the standard output of the reinforced concrete dimensioning is created.*

▶ In the Tab sheet 'Results' start the FAGUS reinforced concrete analysis 'FgErk-P':



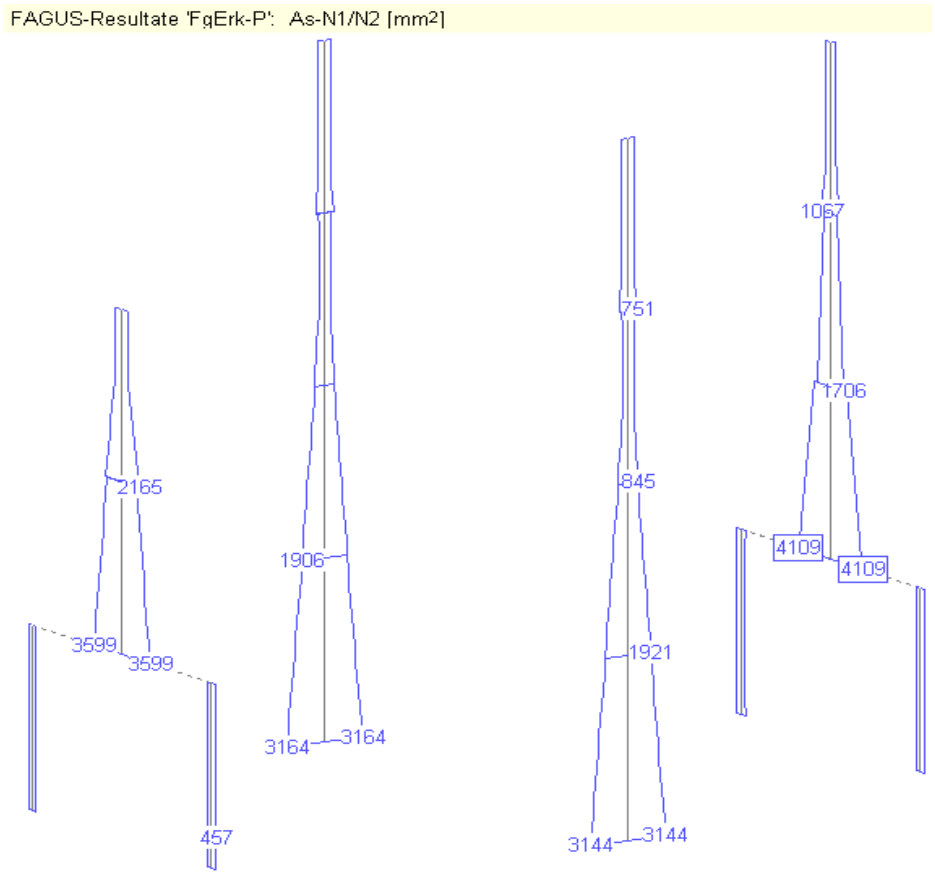
The folllowing four graphical outputs can be created (which contain the individual output quantities) as explained below in the tabular output):

Reinforcement As for the wall pillar nodes N1 and N2

Stirrup reinforcement Asw of the wall pillar section W1

Total longitudinal reinforcement of the whole wall pillar

Geometrical reinforcement of the wall pillar.



Tabellarische Ausgaben

To begin with, the numerical output consists of a table with the abovementioned results, which can also be presented graphically:

Wandscheibenbewehrung (1. Wandabschnitt)

Stab	Elem	Distanz [m]	Längsbewehrung N1 [mm²]	Längsbewehrung N2 [mm²]	Längsbewehrung W1 [mm²]	Bügel A_{sw} W1 [mm²/m]	Längsbewehrung total A_{stot} [mm²]	Längsbewehrung total ρ_{tot} [%]	Text
G1-P1	1	0.00	**457**	**457**	**2086**	**239**	3000	**0.4**	

The individual output quantities have the following meaning:

N1,N2: Longitudinal (axial) reinforcement required in the corresponding wall pillar nodes M,N,V,T. The output value was calculated for the corresponding input reinforcement type, i.e. either for a point or uniformly distributed over the stretches shown in the cross section graphics.

W1: Input (minimum) constructional longitudinal reinforcement in the wall pillar section W1 (was not dimensioned)

A_{sw} W1: Stirrups reinforcement Asw for the wall pillar section W1 (takes into account the shear force in the wall segment and has nothing to do with the required cover for the reinforcement in the wall nodes)

A_{stot}: Total cross sectional area of the axial reinforcement in the wall pillar

ρ_{tot}: Total geometrical reinforcement content of the wall pillar

If there are wall pillars with several sections, then the reinforcement in the sections 2 to max. 5 are presented in another table with the following heading:

Wandscheibenbewehrung (restliche Wandabschnitte)

Stab	Elem	Distanz	Knoten				Wandscheiben verteilt				Bügel A_{sw}			
			N3	N4	N5	N6	W2	W3	W4	W5	W2	W3	W4	W5
		[m]	[mm²]	[mm²]	[mm²]	[mm²]	[mm²]	[mm²]	[mm²]	[mm²]	[mm²/m]	[mm²/m]	[mm²/m]	[mm²/m]
G1-P8	1	0.00	**139**	114			**978**	**568**			20	19		

For wall pillars with even more sections the results can be viewed either in the additionally generated tables 'Result Points and Combinations' or one has to work with the section check possibility (see below).

Resultatpunkte und Kombinationen (min/max): Bemessung

Stab	Elem	Distanz	ASL_MNV_N1	ASL_MNV_N2	ASL_MN_N1	ASL_MN_N2	ASL_V_N1	ASL_V_N2
			[mm²]	[mm²]	[mm²]	[mm²]	[mm²]	[mm²]
G1-P1	1	0.000	103	103	**100**	**100**	3	3

The columns in this table signify:

ASL_MNV_N*: Total required longitudinal reinforcement in node *due to M,N,V,T

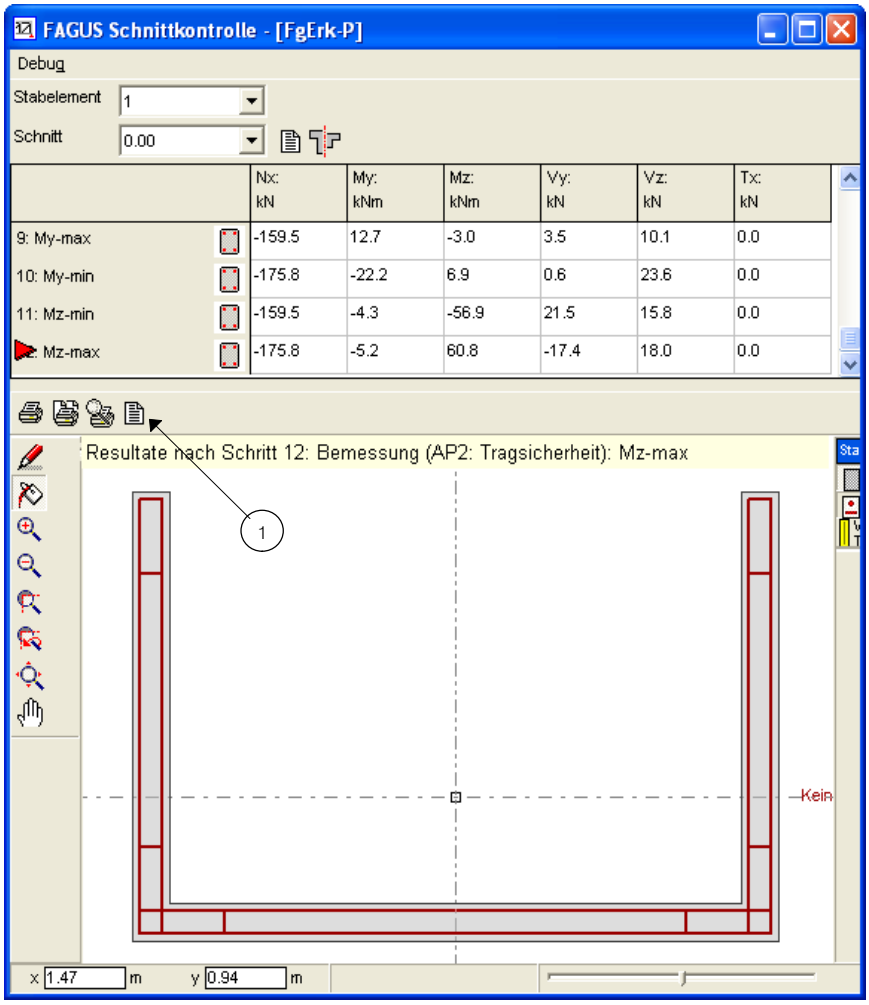
ASL_MN_N*: Total required longitudinal reinforcement in node * due to M,N

ASL_V_N*: Total required longitudinal reinforcement in node * due to V,T

ASW_N*: Stirrups reinforcement for the corresponding wall section

Results using the section check

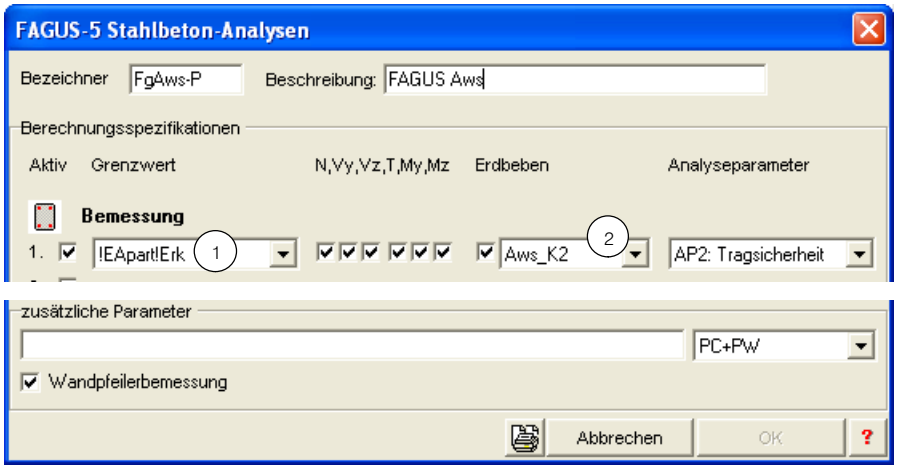
In the selection list for the output quantities the 'Section Check' can be activated. Select it, then select a member and click on the lightning button to start the execution, and then the following section check window appears for the selected member. In it all results can be seen in detail.



(1) Detailed text output for all cross section elements

Dimensioning of wall pillar reinforcement for the response spectrum results

For this purpose the reinforced concrete analysis 'FgAws' is created. It can be viewed in the Tab sheet 'Analysis'. The corresponding dialog looks as follows:

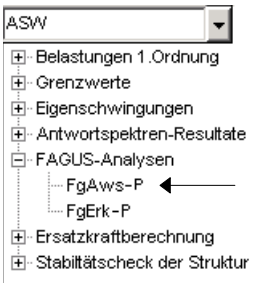


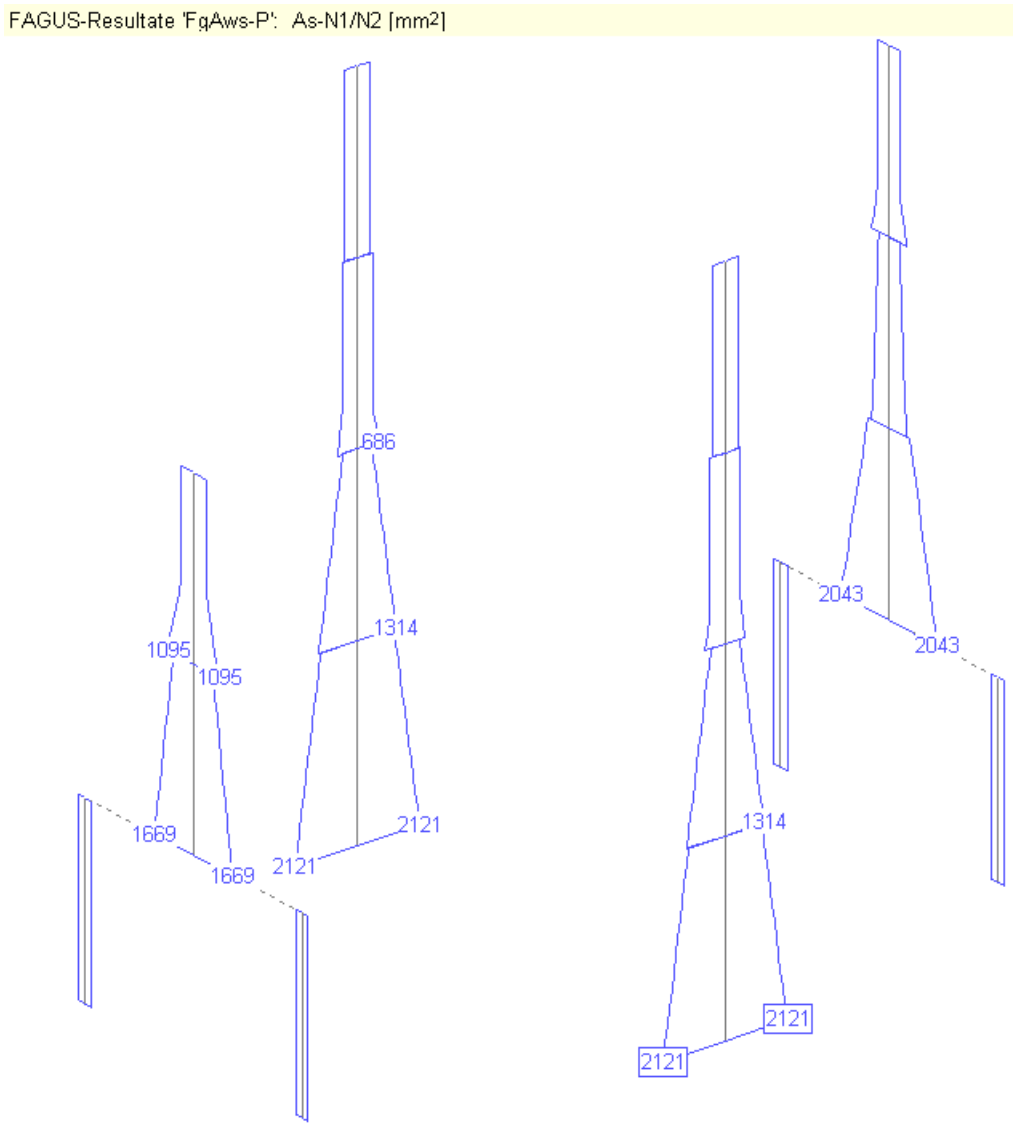
(1) The dimensioning is based on the limit state values of section forces, which in turn are based on the limit state specification. It contains all loadings (incl. torsion due to seismic effects) with the exception of the response spectra results.

(2) Here the response spectra results required for the dimensioning are selected.

For further details of this dialog see the above explanations for dimensioning using the equivalent force method. The explanations given there on the reinforcement and the various forms of output also apply here .

▶ In the Tab sheet 'Results' start the FAGUS reinforced concrete analysis 'FgAws-P':




FAGUS-Resultate 'FqAws-P': As-N1/N2 [mm2]
686
1314
1095
1095
1669
1669
2121
2121
2043
2043
1314
2121
2121

Vol. F Dynamic Analysis

F 1 Natural Vibrations

F 1.1 Basic Theory

The optional module Dynamics of CEDRUS permits the analysis of natural vibrations for slabs with general mass distributions. As results you get circular frequencies, corresponding mode shapes and participation factors.

For this purpose the following eigenvalue problem must be solved:

$$([K] - \omega_i^2[M])\{\Phi_i\} = \{0\}$$

where:

ω_i : eigenvalue (circular frequency) of i-th mode
$\{\phi_i\}$: mode shape of i-th mode
$[M]$: mass matrix
$[K]$: stiffness matrix

The mode shapes (eigenmodes) are normalized as follows:

$$\{\Phi_i\}^T[M]\{\Phi_i\} = 1$$

The modal participation factors are defined as follows:

$$P_i = \left\{\begin{matrix}1\\0\\0\\\vdots\end{matrix}\right\}^T [M]\{\Phi_i\}$$

F 1.2 Mass Distribution

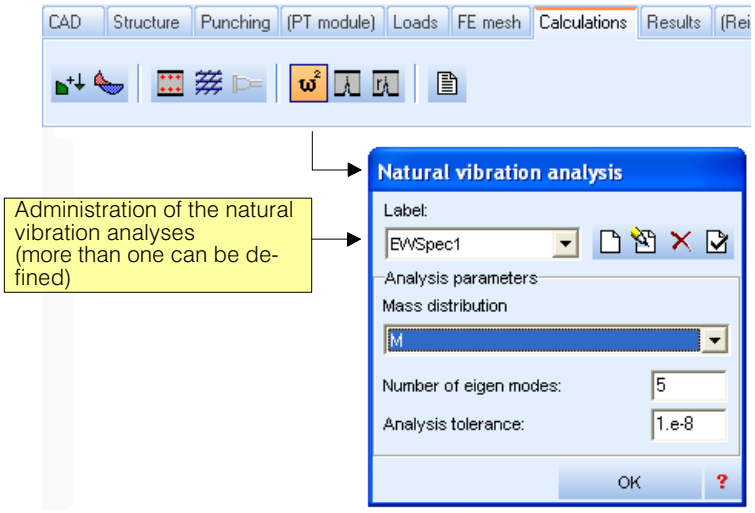
Mass distributions are defined in the 'Loads' tabsheet just like normal loadings but with the type 'mass distribution'. They can consist of specific masses, distributed masses (polygonal or along a line) and concentrated masses.

For the numerical analysis the mass distribution is transformed into translatory (in Z-direction) acting concentrated masses in the nodes of the FE-mesh, i.e. the mass matrix has only diagonal nonzero elements.

☞ *'Exotic' mass distributions can lead to numerical convergence problems (e.g. it is impossible to calculate more eigenvalues than the number of nodes with associated concentrated masses). Generally the mass of the slab should be considered, thus avoiding these problems.*

F 1.3 Setup of a Natural Vibration Analysis

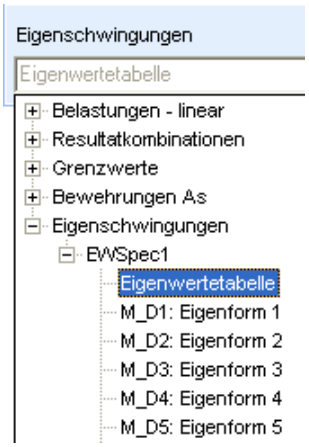
A natural vibration analysis is setup in the tabsheet 'Calculaions' as follows:



The dialogue should be self-explanatory. The default tolerance value 1E-8 for the calculation algorithm should be appropriate in normal cases.

F 1.4 Output of Results

For each natural vibration analysis defined in the 'Calculations' tabsheet you will find a corresponding branch in the result tree of the 'Results' tabsheet offering a table of the frequencies and participation factors of all calculated eigenvalues as well as the modal shapes (eigenmodes) that can be viewed graphically and numerically just like displacements.

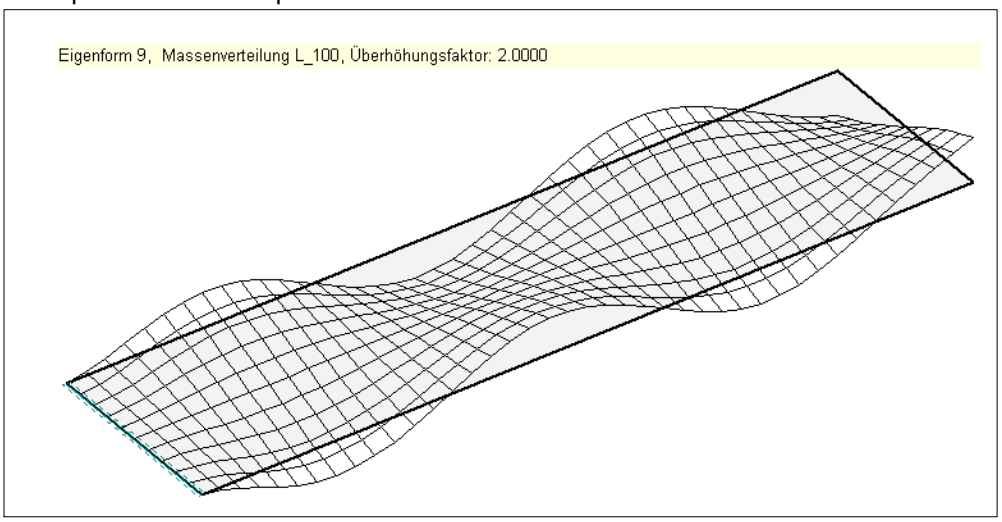


Example of a table of frequencies and participation factors::

Eigenfrequencies for distributed mass M

Eigenmode	ω^2 [(rad/s)^2]	ω [rad/s]	Frequence [s^{-1}]	Period [s]	PF	PF^2 [t]
1	2120.8	46.05	7.33	0.136437	2.818	7.940
2	2727.4	52.22	8.31	0.120311	10.726	115.043
3	4905.7	70.04	11.15	0.089707	4.431	19.635
4	5689.6	75.43	12.01	0.083299	8.335	69.474
5	8505.7	92.23	14.68	0.068128	5.758	33.159
Sum						245.252
Total Mass						330.475

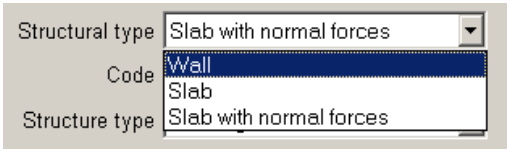
Example of a modal shape:



Vol. G Walls

G 1 Introduction

The optional module „Walls" allows for the linear elastic analysis and dimensioning of plates subjected to pure in-plane or combined bending/in-plane action. If licensed you can choose between the follwing 3 structure types:



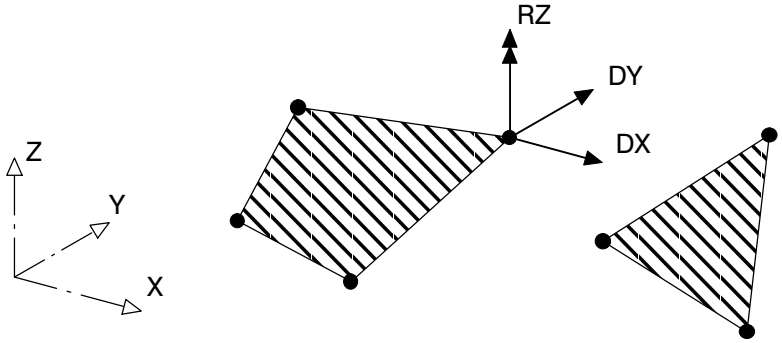
Due to the full integration into the user interface of the slab analysis program, the data input for the walls program is very similar to the slab part. Thus this manual for walls does only handle the aspects that are specific to the analysis of walls. For all the other topics, that both modules have in common, please refer to the part A 2 of this manual. The following sections concentrate on the specialities of a wall analysis, i.e. the model for geometry, loads and the output and outline some specialties realated to combined bending/in-plane action.

The structure type „slab with normal forces" is particularly usefull for prestressed slabs, where the distribution of the normal forces due to the anchorage and deviation forces can be investigated. For details see G 3.1.

G 2 Basic Theory

G 2.1 Element Model and Solution Method

The module walls a Finite Element (FE) program for the linear elastic analysis of plates subjected to in-plane action. The element models used are triangular and quadrilateral elements of arbitrary shape with the three displacement degrees of freedom DX, DY (displacement in directions X and Y) and RZ (rotation about the Z axis) in the corner nodes.



In-plane action, not bending action is considered. The stiffness matrices and the load vectors are built according to the so-called "Free-Formulation" method. This method does provide accurate results for deformations and stresses. The most important properties of the element models are:

- shape functions for deformations as linear combinations of eigenmodes.
- consistent distributed and initial element loads.
- stress results in corners as well as in the center of the element (four-node elements only).

For a detailed study of this element model refer to the following:

- P.G. Bergan, M.K. Nygard: "Finite elements with increased freedom in choosing shape functions", Intl. J. of Num. Meth. in Engineering, 20 (1984), P 643-664.
- M.K. Nygard: "The Free Formulation for Nonlinear Finite Elements with Application to Shells", Thesis, Institutt for Statikk, Universitete Trondheim.

The FE method in CEDRUS essentially involves the following steps:

1. Determination of the element matrices for the hybrid method.
2. Determination of the load vectors (right hand sides of the system of equations).
3. Summation of the element stiffness matrices to form the global stiffness matrix.
4. Solution of the resulting system of equations for the unknown nodal displacement parameters (possibly iteratively in the case of supports that do not take tension).
5. Calculation of the section forces in the elements for the now known nodal displacement parameters.

One should observe that the FE method is an **approximate numerical method**. The numerical solution, however, converges for an ever finer element mesh, within the limits of numerical accuracy, to the exact theoretical solution of Kirchhoff's plate bending theory.

G 2.2 Modelling

G 2.2.1 Geometry

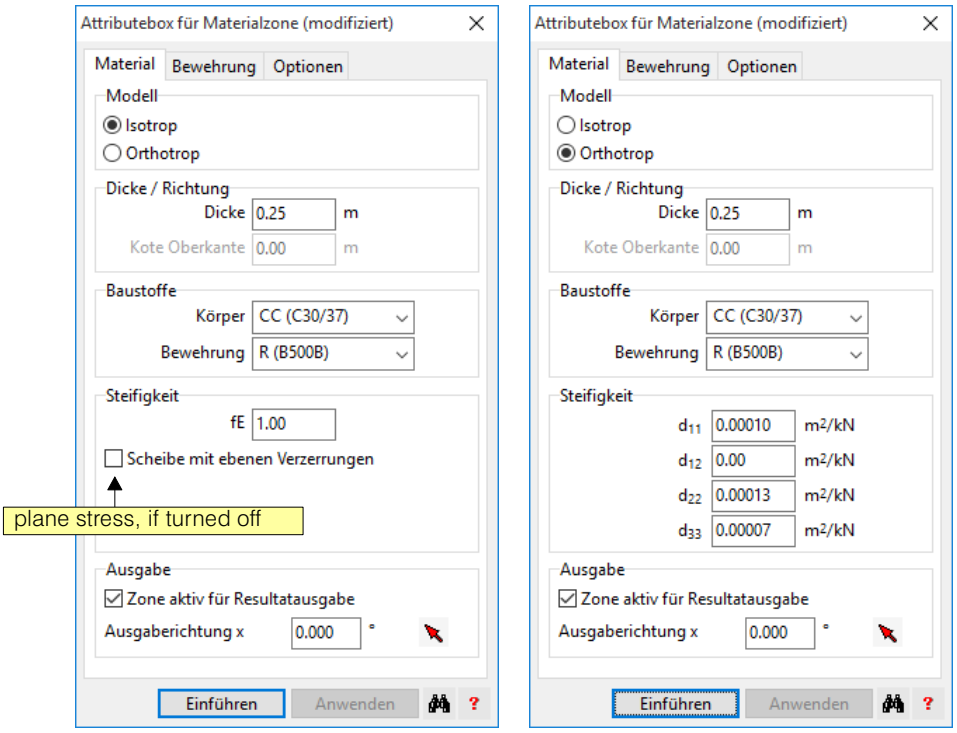
The geometry of the plan outline is basically fixed by the following conditions:

- **Outline:** An arbitrary closed polygon.
- **Recesses and openings:** Arbitrary closed polygons.
- **Point supports:** Points for the definition of support conditions for single nodes. The direction of the node can be chosen at will (i.e. skew supports are possible).
- **Line supports:** Lines or polygons for the definition of support conditions. Internally a series of point supports are generated. The direction of the supported nodes is taken from the orientation of the line is located on.
- **Material separators:** Lines or polygons which divide up the slab into several zones with different material attributes.
- **Lines of symmetry:** Along these lines the slab may bend but normal to them it may not rotate. Thus one has a special type of linear support, which may only lie on the slab boundary

G 2.2.2 Thickness and Material

The structure can be subdivided by separator lines into several zones. For each zone the material model has to be specified.

The input is made, as in slabs, in the dialog 'attribute box for material zone':



In CEDRUS, the two material models **Isotropic** and **Orthotropic** are supported and explained in this section. In the description of the material models the following notation is used:

$\sigma_x,\ \sigma_y,\ \tau_{xy}$: in-plane stresses
$\varepsilon_x,\ \varepsilon_y,\ \gamma_{xy}$: (corresponding) strains
E, ν: elastic modulus and Poisson's ratio

Isotropic Material

Isotropic material is directionally independent and is completely described by two elastic constants, i.e. the modulus of elasticity (or Young's modulus) E and Poisson's ratio ν. The program allows for two different models, the plane stress and the plane stretching model.

Plane Stress

In plane stress it is assumed, that the stress perpendicular to the middle plane of the wall vanish ($\sigma_z = 0$), which is a good model for thin walls. The relationship between in-plane strains and stresses is governed by the following elasticity matrix (i.e. a matrix containing the elastic constants):

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \frac{1}{f_E\, E} \begin{bmatrix} 1 & -\nu & 0 \\ -\nu & 1 & 0 \\ 0 & 0 & 2\,(1+\nu) \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix}$$

$$\varepsilon_z = \frac{-\nu}{E}\,(\sigma_x + \sigma_y)$$

$$\sigma_z = 0$$

f_E = factor of the stiffness (deafult=1.0)

Plane Stretching

In plane stretching it is assumed, that the strain perpendicular to the middle plane of the wall vanish ($\varepsilon_z = 0$), which is a good model for walls of infinite (or very high) thickness. The relationship between in-plane strains and stresses is governed by the following elasticity matrix:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \frac{1+\nu}{f_E\, E} \begin{bmatrix} 1-\nu & -\nu & 0 \\ -\nu & 1-\nu & 0 \\ 0 & 0 & 2 \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix}$$

$$\varepsilon_z = 0$$

$$\sigma_z = \nu\,(\sigma_x + \sigma_y)$$

f_E = factor of the stiffness (default=1.0)

☞ *The user must specify the thickness of the wall even in the plane stretching model. If one wants to analyze a wall of unit width, which is the natural choice for plane stretching analysis, the thickness must be explicitly set to 1.0.*

Orthotropic Material

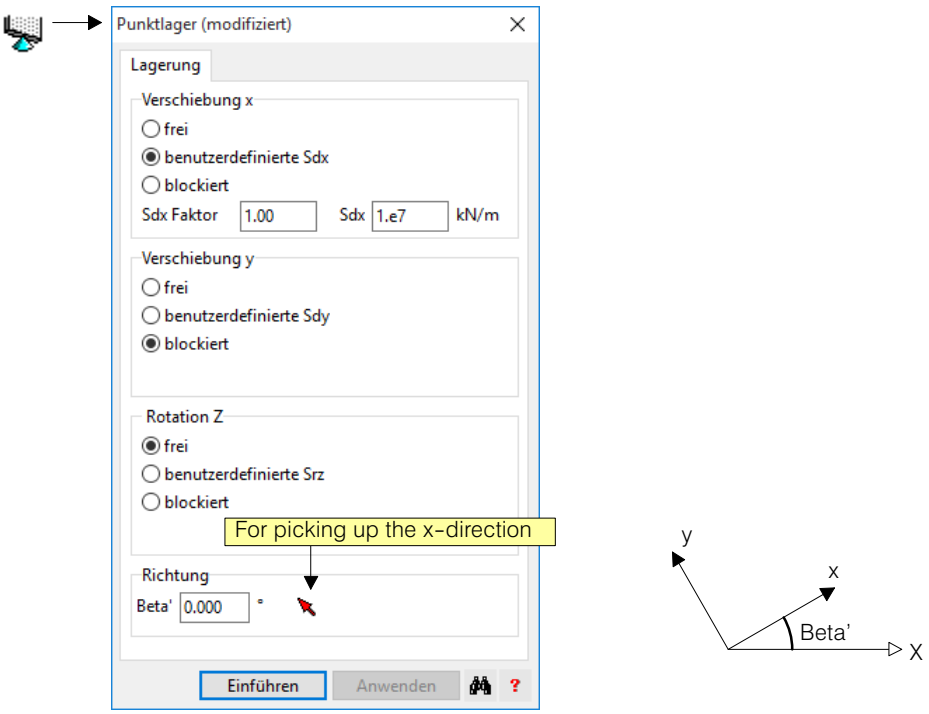
Orthotropic material exhibits different properties in the two directions x and y normal to one another. It is described by the following elasticity matrix:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} d_{11} & d_{12} & 0 \\ d_{21} & d_{22} & 0 \\ 0 & 0 & d_{33} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} \qquad (d_{12} = d_{21} <= 0)$$

$$\sigma_z = 0$$

Here the condition $d_{11} * d_{22} > d_{12} * d_{21}$ must be fulfilled. The x-direction (material direction) is selectable.

G 2.2.3 Point Supports



For every point support a local x-direction can be specified. All the input and output refers to this orientation. The stiffness terms s_{dx} , s_{dy} (for displacements) and s_{rz} (for rotation) can be defined:

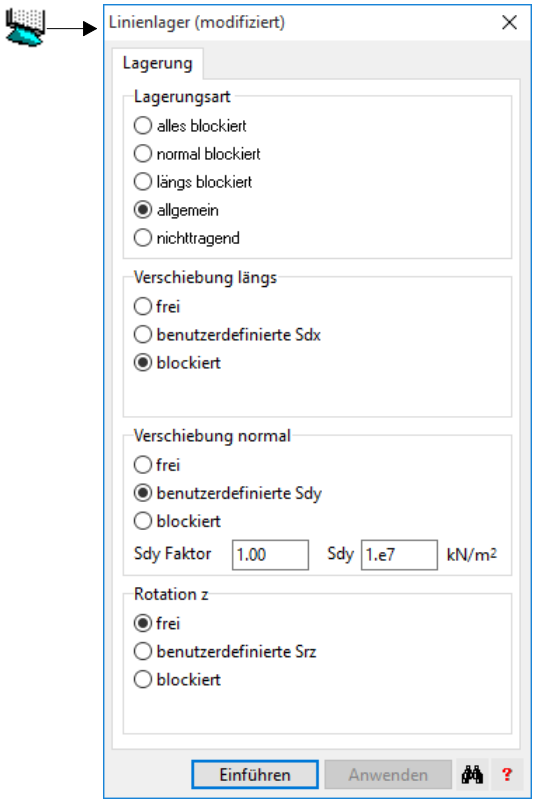
- free (no restriction of displacement / rotation)
- blocked (no shift possible)
- spring (input of the spring stiffness):

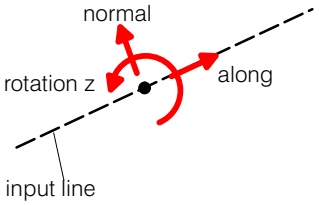
s_{dx}	:	spring stiffness in direction of node
s_{dy}	:	spring stiffness perpendicular to direction of node
s_{rz}	:	rotation stiffness around Z-axis

The angle beta' (see above in the dialog shown) of the X-direction with respect to the global X-direction must be specified. The storage conditions and the output of the reactions refer to this direction.

When spring-mounted, a corresponding spring stiffness (force per unit displacement or moment per unit twist) must be entered.

G 2.2.4 Line Supports



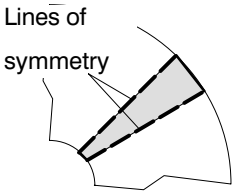


The boundary conditions of a wall can be specified with line supports as well. Internally, a line support is a series of point supports with local orientations according to the direction of the line. The stiffness terms s_{dl} , s_{dn} (for displacements) and s_{rz} (for rotation) can be defined:

s_{dl}	:	longitudinal spring stiffness (= in direction of the line) per unit length
s_{dn}	:	normal spring stiffness (= perpendicular to the line) per unit length
s_{rz}	:	rotation stiffness around Z-axis per unit length

In the case of spring-loaded bearing components, the spring stiffnesses to be input each refer to the unit length of the support line. The conversion to the FE nodes takes place according to the distances between them.

G 2.2.5 Lines of Symmetry



Lines of symmetry are special line supports, along which the wall is free to move while the displacements perpendicular to the line and the rotations are blocked. Symmetry conditions are really only meaningful at wall boundaries, which is why lines of symmetry are only allowed to lie on the plan outline.

Lines of symmetry are used to demarcate parts of the wall in the model, be it a genuine line of symmetry or if by means of a symmetry condition one obtains the most favorable boundary condition. Note, that with symmetry conditions the loads too always act symmetrically.

An example of genuine symmetry is given by the circular plate - even if with the condition of a symmetrically acting load it is rather academic. Here the input of a sector with the corresponding symmetry conditions along the radial boundaries suffices.

Line supports on lines of symmetry are permitted. If a line support should work as a line of symmetry, one has to block the longitudinal displacement and the rotation. Note: An eventually provided spring stiffness must divided by two.

Point supports on lines of symmetry are allowed. The direction of the support must however correspond to the direction of the line. Note: An eventually provided spring stiffness must divided by two.

G 2.2.6 Loads

As in the case of the slabs, the loads are combined in individual loadings. The wall module knows the following load elements:

Area loads (i.e. loads per unit area)

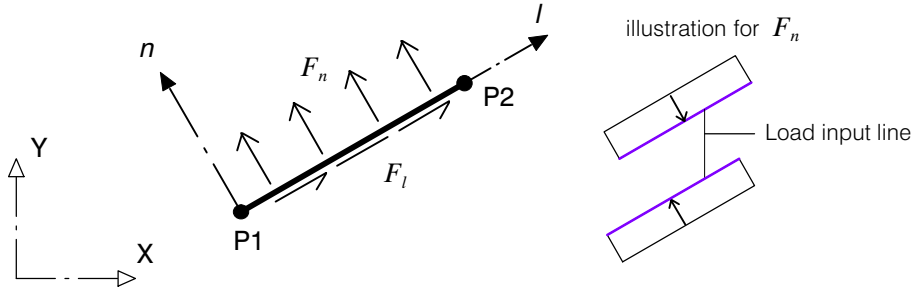
Rectangular or arbitrary polygons for:

- body force (e.g. dead or self-weigh in X and Y)
- uniformly distributed force in X and Y
- differential temperature loading
- initial strain (strain X, Y and XY)

Line loads

- Constant or trapezium distributed
  - longitudinal force F_l : force in direction of the line
  - normal force F_n : force perpendicular to the line (counter clockwise)

  The direction of the line load goes from the first point P1 to P2.



Point loads

- forces in X and Y

Displacements of point supports

- the corresponding displacement parameters of the support nodes have to be blocked
- displacement in X and Y
- rotation around Z

The load types 1) to 3) are independent of the FE mesh, so that they can be arbitrarily arranged geometrically.

Loads are combined to individual load cases, which can be combined or superimposed in any way for the calculation of the results. The load cases can be assigned to particular action types, like dead weight loads, surcharge loads, imposed loads etc., whereby in standard cases a load superposition automatically carried out by the program to determine the design section quantities is possible.

G 2.3 Results

G 2.3.1 Raw Results

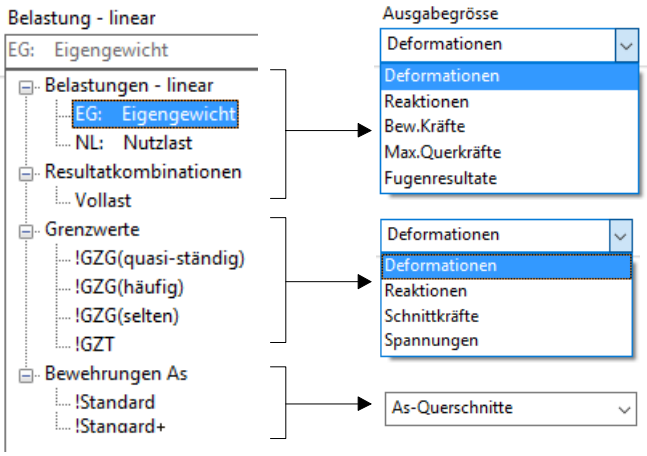
The solution algorithms of CEDRUS produce raw results for each load case input:

- The nodal displacements
- Reactions of the supports
- Section forces at the element centers in global coordinate directions
- Section forces at the element corners in global coordinate directions, which are then immediately and zone-wise converted to averaged nodal values and are not given as separate values

These calculated and binary stored raw results serve as a basis for determining the results types described below in the desired form of presentation.

G 2.3.2 The Structuring of the Output of Results

Output results are not created unless the user specifically demands them. Firstly, it is defined for what the results are required (e.g. load case number), then comes the choice of the quantities (e.g. section forces), a possible component (e.g. m_x) and finally the presentation form (e.g. isolines), which can still be influenced by certain parameters.



From a list one first chooses for what the results are wanted. The list contains

- All input load cases
- Any defined load case combinations
- All automatically or manually produced limit state specifications
- Required reinforcement

Load case combinations are fixed combinations of load cases provided with arbitrary factors, of which the user can define as many as desired. In the output of results they are treated in exactly the same way as individual load cases.

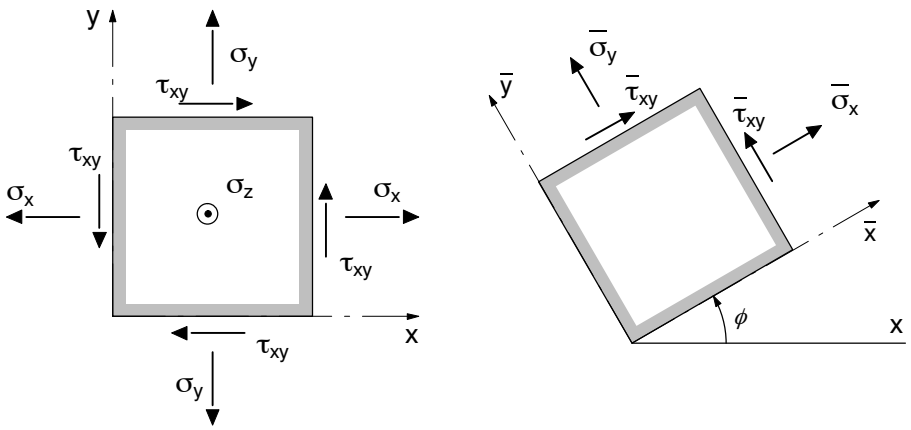
The quantities for load cases and load case combinations

Deformations:

With the deformations it is a question of disablement in x- and y-direction as well as the rotations about the Z-axis, respectively, in each node of the FE mesh. The rotations of nodes not acting as supports are output in the global coordinate system. For point and line supports the x-direction of the input object is adopted and the y-direction normal to it. The x-axis of line supports shows the support direction.

Stresses:

The stresses in a wall consist of the components $\sigma_x, \sigma_y, \tau_{xy}$. The following figure shows the forces acting on an infinitesimal slab element:



The output of the wall section stresses is carried out in zone-wise definable output directions. The transformation formulas for a free direction ϕ are as follows (Mohr's circle for the stresses):

$$\bar{\sigma}_x = \sigma_x \cos^2\phi + \sigma_y \sin^2\phi + 2\tau_{xy} \sin\phi \cos\phi$$
$$\bar{\sigma}_y = \sigma_x \sin^2\phi + \sigma_y \cos^2\phi - 2\tau_{xy} \sin\phi \cos\phi$$
$$\bar{\tau}_{xy} = (\sigma_x - \sigma_y) \sin\phi \cos\phi + \tau_{xy}(\cos^2\phi - \sin^2\phi)$$

In zones with plane stress the stresses in z-direction vanish, while the following relations governs the zones with plane stretching :

$$\sigma_z = \nu (\sigma_x + \sigma_y)$$

Depending of the output format principal stresses σ_1 and σ_2 are calculated.

In addition the output of nominal stresses is possible:

von Mises: $$\sigma_{eqM} = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}}$$

Tresca: $$\sigma_{eqT} = |(\sigma_1 - \sigma_3)| \qquad \sigma_1 > \sigma_2 > \sigma_3$$

Section forces:

The section forces consist of the components n_x, n_y and n_{xy} ,which correspond to the stresses ($\sigma_x, \sigma_y, \tau_{xy}$.) multiplied by the thickness of the wall. Therefore the same transformation formulas can be used..

Reactions:

The reactions, arranged node-wise, are output according to the individual supports. In the graphical output of the line supports the possibility exists, of combining the nodal reactions in sections, provided the section length is given.

Quantities for limit state specifications (envelope values)

Deformations:

One can obtain the envelope values of the displacements.

Reactions:

One can obtain the envelope values of for point and line supports.

Reinforcement forces:

The section forces n_x, n_y and n_{xy} are combined to reinforcement forces according to the combination rules specified in the limit state specifications.

The reinforcement forces at a point are the four forces required to determine the reinforcement in two orthogonal directions. Their calculation is based on the well-known linearised plasticity conditions:

$$na_{x>} = Max\ (\ n_x + n_{xy}\ ,\ n_x - n_{xy}\ )$$
$$na_{x<} = Min\ (\ n_x + n_{xy}\ ,\ n_x - n_{xy}\ )$$
$$na_{y>} = Max\ (\ n_y + n_{xy}\ ,\ n_y - n_{xy}\ )$$
$$na_{y<} = Min\ (\ n_y + n_{xy}\ ,\ n_y - n_{xy}\ )$$

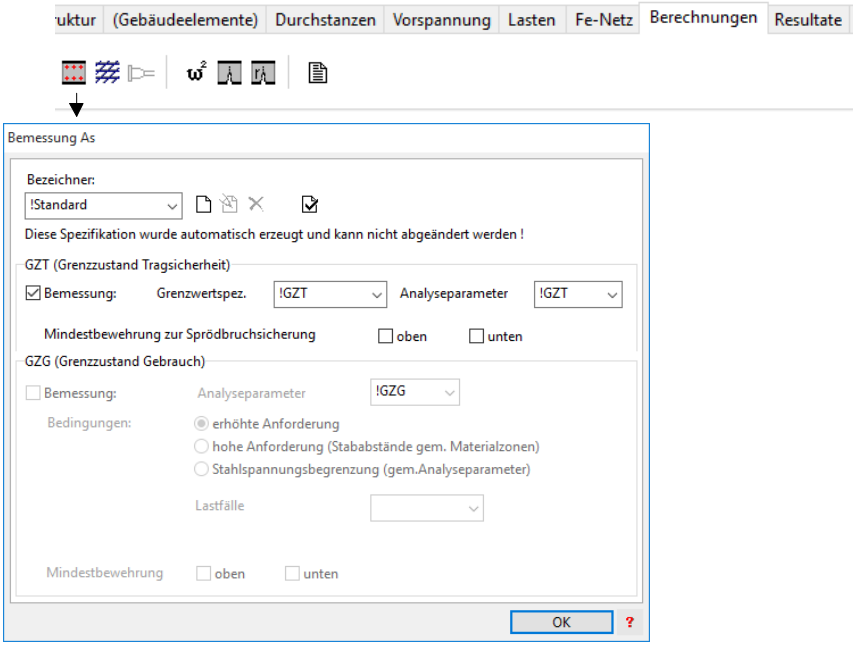
with:

$na_{x>}$	maximum reinf. force in x-direction (=tension -> reinforcement)
$na_{x<}$	minimum reinf. force in x-direction (=compression-> concrete)
$na_{y>}$	maximum reinf. force in y-direction (=tension -> reinforcement)
$na_{y<}$	minimum reinf. force in y-direction (=compression-> concrete)

The output directions are defined for each zone.

Required reinforcement

The required reinforcement of the wall in two orthogonal directions a_x and a_y is determined on the basis of design limit values of the reinforcement moments described above.



It is assumed, that all the tensional forces are taken by the reinforcement and all the compressive forces are taken by the concrete.

The ultimate limit state and / or the service limit state can be selected as the criterion for design specification. In the case of service limit state, a permissible steel tension is to be entered. The rated stress values of the reinforcement forces described above serve as stresses. Appropriate limit values have to be selected.

The reinforcement, which is used exclusively for the absorption of the tensile forces, is measured. The concrete tensile strength is assumed to be zero. The output reinforcement represents the entire required reinforcement and is therefore independent of the number of layers to which it is distributed. Concrete pressure forces are not checked.

Concrete and reinforcing steel material are properties of the material zones. The value for the permissible steel stress for usability is defined in the above dialog and applies to the whole wall.

The orthogonal reinforcement directions correspond to the output directions and can be defined zone-wise (output direction X in the attribute box for material zones).

Forms of Presentation

The following table provides information about the possible forms of presentation for the different derived quantities:

Results and their form of presentation:		Contour plot	Principle values	Numerical/graphical	Tables	Axonometrical	Sections *)	Beam sections **)
Load cases and load case combinations:	displacements	✓		✓	✓	✓		
	point and line reactions				✓	✓		
	stresses	✓	✓	✓	✓		✓	
	section forces	✓	✓	✓	✓		✓	✓
	nominal stresses	✓		✓				
Limit state values:	displacements	✓		✓	✓			
	reactions				✓			
	reinforcement forces	✓		✓	✓		✓	
Reinforcement:	reinforcement sections	✓		✓	✓		✓	✓

*) **Sections:** These are simple sections through the corresponding contour plot. In the case of design limit values and reinforcement these section results can only be determined in zones, whose output direction coincides with that of the section direction or the normal to it, respectively.

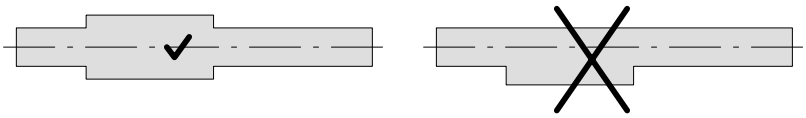
) **Beam sections: These are sections with a selectable width, which cut a "beam" out of the disc. They are available for those results which are defined per unit of length, such as section forces and reinforcement cross sections. The integral of the result variable is output as a result value in a point of the beam over the cross-sectional width.

G 3 Slab with Normal Forces

G 3.1 Model

"Slabs with normal forces" (SwN) are a combination of slobs with pure bending and walls with pure membrane action. Therefore the nodes of a SwN model have 6 degree of freedoms, DZ, RX, RY from the slab aund DX, DY, RZ from the wall.

The SwN model of CEDRUS does cover structures with a continous middle plane only. i.e. zones with different thickness are all interconnected without eccentricity.

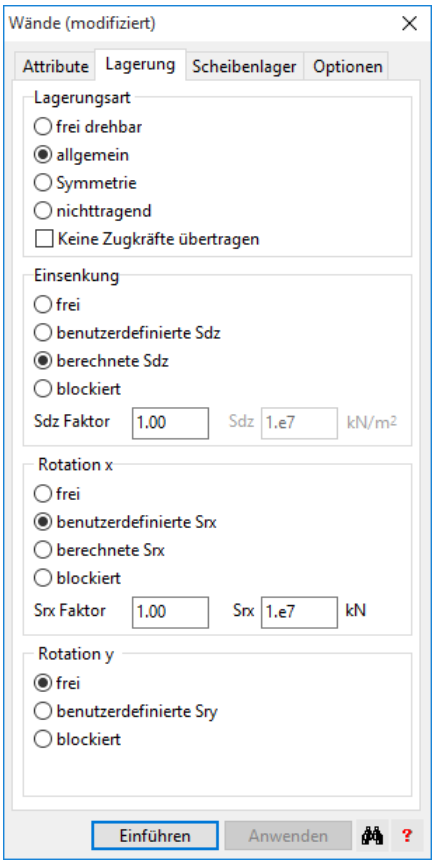


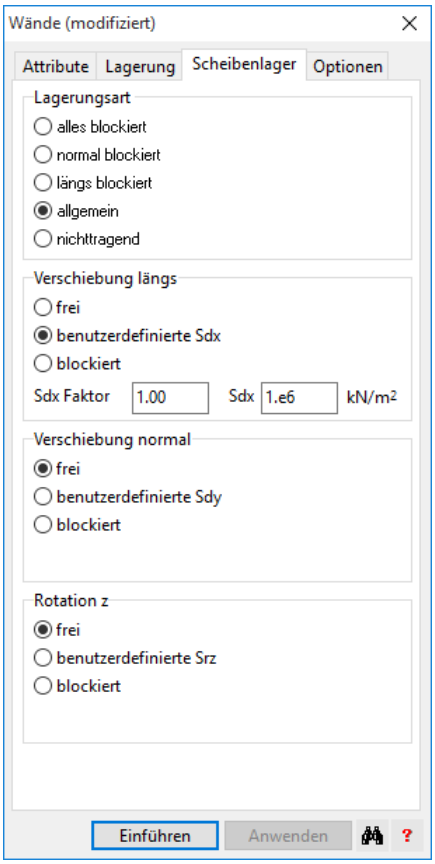
This applies to **downstanding beams** as well. Therefore, if you start a new project as a slab and enter (eccentrically connected) downstanding beams, after a change of the structure type to SwN (see G 3.2) the beam-zones will be centrically connected.

The feature SwN is especially tailored for pre- and posttensioned flat slabs, which are not affected by this restriction.

In order to input all the data needed to properly model a combined slab-wall system, the attribute-dialogs of the supports (e.g. walls and columns) have two tabsheets regarding the support conditions.

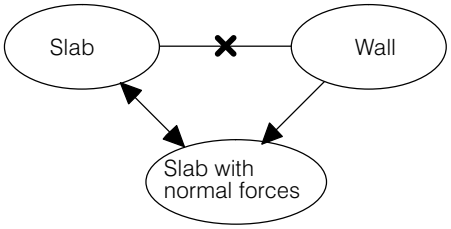
⚠ The structure must be stable as a slab as well as a wall. Particular attention must be paid to the settings of the supporting components of the slab and wall parts, especially in walls and point supportings.





G 3.2 Changing the Structural Type

Having initially choosen the structural type of a new project, there are some restrictions in changing it later. This is shown in the following graph:



G 3.3 Results

Results and their form of presentation:		Contour plot	Principle values	Numerical/graphical	Tables	Axonometrical	Sections *)	Beam sections **)
Load cases and load case combinations:	displacements	✓		✓	✓	✓		
	point and line reactions	✓		✓	✓	✓		
	stresses	✓	✓	✓	✓		✓	
	section forces	✓	✓	✓	✓		✓	✓
Limit state values:	nominal stresses	✓		✓	✓	DZ	✓	
	displacements	✓		✓	✓	✓		
	reactions	✓		✓	✓			
	reinforcement forces	✓		✓	✓		✓	✓
Reinforcement:	reinforcement sections							✓

*) **Sections:** These are simple sections through the corresponding contour plot. In the case of design limit values and reinforcement these section results can only be determined in zones, whose output direction coincides with that of the section direction or the normal to it, respectively.

) **Beam sections: These are sections with a selectable width, which cut a "beam" out of the disc. They are available for those results which are defined per unit of length, such as section forces and reinforcement cross sections. The integral of the result variable is output as a result value in a point of the beam over the cross-sectional width.

Vol. H Prestressing Module

H 1 Introduction

H 1.1 Overview

With the Prestressing Module (PT) of CEDRUS the effect of prestressing can be treated either as an external load or as a resistance within the context of design and dimensioning. The program is intended to aid the search for a suitable prestressing arrangement and provides all calculated data for further planning and design. Thanks to its data exchange capabilities, CEDRUS allows the export of diagrams, tables and graphs into other applications for post-processing, e.g. CAD systems, spreadsheets and word processors.

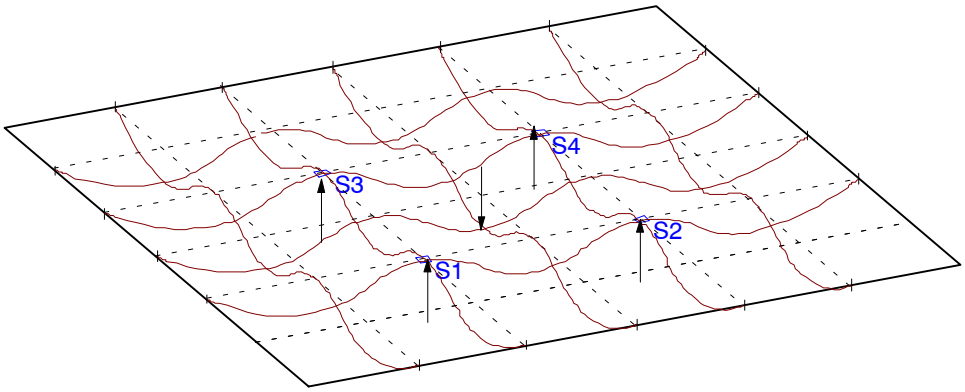
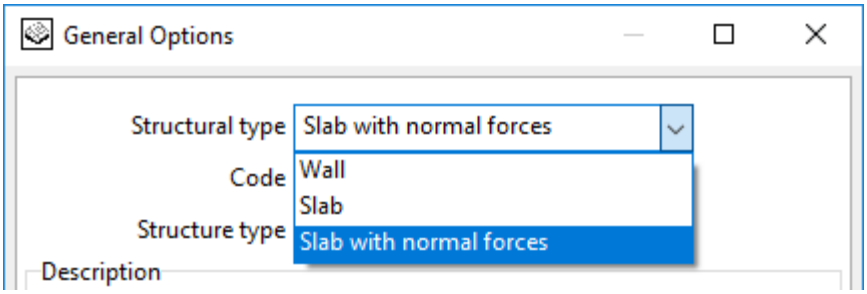


Fig. 1 *Diagrammatic illustration of prestressing*

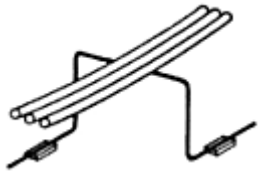
Slab or **Slab with Normal Forces**:
If the option 'Wall' is available, the calculation model 'slab with normal forces' can be selected in the 'General Options' dialog:



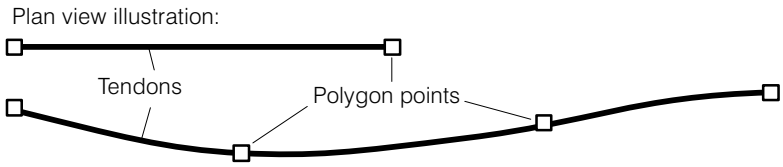
In this model, the normal forces can be found in the board due to prestressing by FE analysis and used in determining the amount of bar sections (more on this under).

H 1.2 Tendons and Supports

H 1.2.1 Tendons



The term 'tendon' is used in the following as a generic term for every kind of prestress element (cables with bundles of wires, strands, bars). In contrast to most other graphical 3D objects the geometrical input of the prestressing is done separately for the horizontal and vertical directions.
With the aid of the Graphics Editor the horizontal direction is drawn in the plan view as a straight line or as a polygon. In the case of curved tendons the individual polygon points serve as discrete points of the continuously curved tendon. By moving these discrete points, the introduction of intermediate points or changes in the tangents, the changing shape of the curve can be influenced by the user.

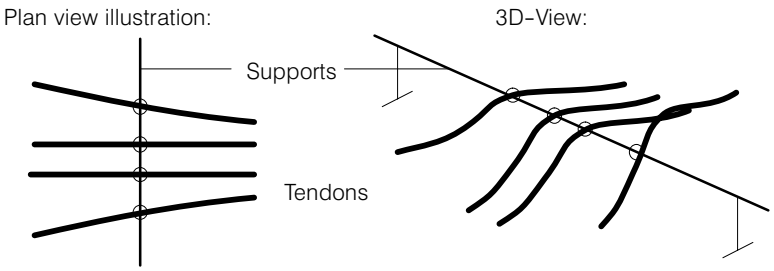


The vertical curve can be set on the one hand in its polygon points. On the other hand assumes the tendon in each intersection of its plan view line with supports (see below) its properties.

H 1.2.2 Supports

The vertical variation of the tendon is also defined in the plan view along the straight line with height attributes at so-called “supports”. Just like on the construction site the supports serve to support the tendons at the points of intersection, whereby the following attributes can be assigned:

- z-coordinate (different input possibilities regarding slab geometry)
- slope of tendon
- rules regarding minimum radius and inflexion points



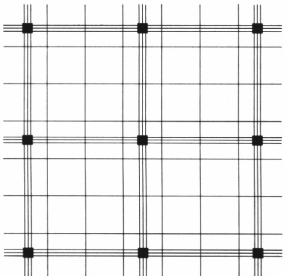
H 1.3 Input Procedure

Normally one attempts to compensate the permanent loads in the slab by means of the deviation forces from the prestressing. In the case of a vertically directed tendon, with second order parabolas, in the ideal case without friction losses straight constant deviation forces are produced.

With the standard cases described in H 2 many useful tendon profiles can be described in a simple manner.

The basic steps of the interactive input comprise:

- Input of the tendons in plan view
- Input of the supports for fixing the vertical profile
- Definition of the stressing program
- Assignment of the tendon groups to CEDRUS load cases



The choice of the arrangement of the tendons is governed both by statical and constructional considerations. Basically, the prestressed elements can be placed in one or two directions. Regarding the subdivision into field and column strips (sometimes referred to as 'support strips') running along the grid over columns there are different recommendations (e.g. 50 % of the tendons in the field and the other 50 % in these concentrated column strips). In addition to compensating the dead load, with regard to punching shear behaviour as well one wants to profit from as many of the favourable effects of prestressing as possible.

To obtain an initial estimate of the required prestressing force for the input of the tendons a small program is available, with which a “pre-dimensioning” sheet with all characteristic values can be printed out.

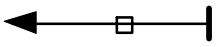
For a better overview of the input several tendons can be grouped together. Each group corresponds to a layer in the Graphics Editor. One can switch between the individual groups. “Inactive” tendon groups can, if necessary, be graphically superimposed. A standard subdivision could look like the following:

- Group 1: Tendons in x-direction, supports in y-direction
- Group 2: Tendons in y-direction, supports in x-direction

For further treatment in CEDRUS the tendon groups are put together to form one or more CEDRUS load cases.

For control purposes, at any time during the input the following diagrams can be included:

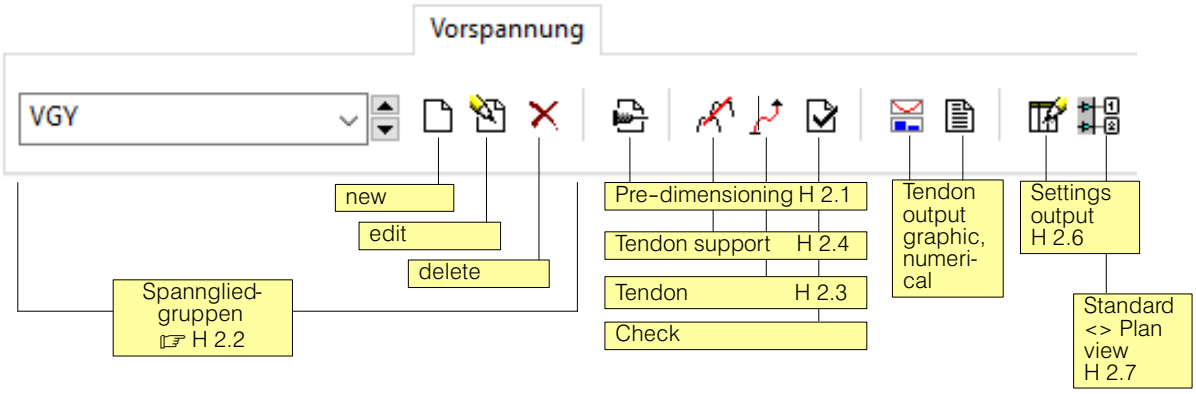
- 3D view with arbitrarily selectable viewing direction
- Longitudinal view of a selected tendon
- Presentation of the force variation along the tendon itself based on the selected stressing procedure
- Presentation of the deviation forces in the section
- Presentation of the deviation forces in plan or in 3D view



The amount of graphical information produced can be controlled using various options (presentation in the prestressing direction, tendon attributes etc.) and the diagrams can also be output on a printer or plotter to any desired scale.

General stressing procedures can be defined by specifying the anchorage forces or the anchorage movements, whereby arbitrary stressing (tensioning, overtensioning) and release of tension (wedge draw-in during lock off, anchorage slip) is possible. The variation of the prestressing force calculated by the program takes into account all friction losses.

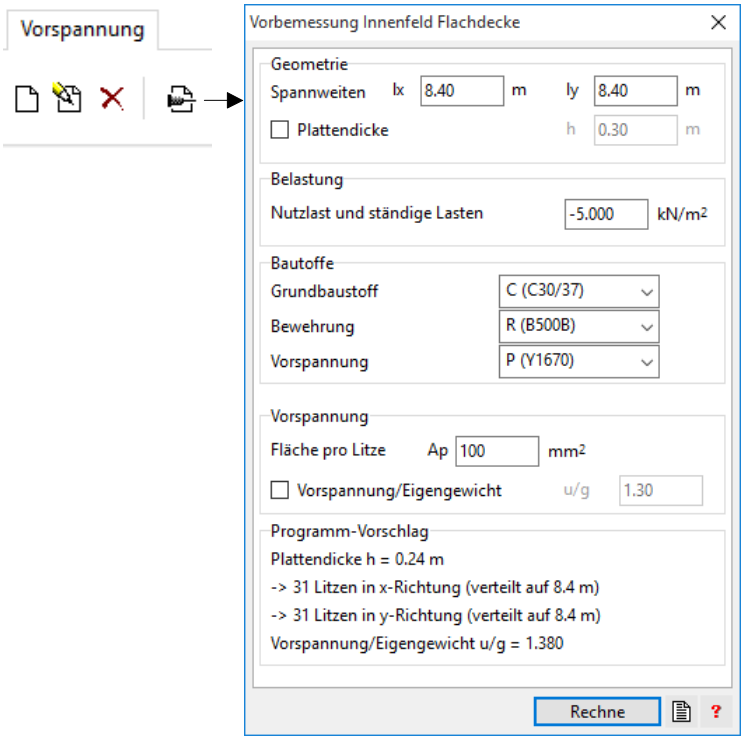
H 2 Basics



Further chapters: H 2.5 Tendon geometry
H 2.8 Pretension as an external effect on the slab
H 2.9 Dimensioning the reinforcement

H 2.1 Preliminary Design

For the preliminary design of an interior field of a flat slab is within the tendon entering a small tool available with which a sheet for preliminary design can be printed with all the characteristic values.



The preliminary design is due tabled values and is not exactly the same as a CEDRUS analysis.

H 2.2 Tendon groups

Tendons and associated holders are grouped into so-called prestressing groups. The criteria for division into several groups are as follows:

- All properties (tendons and holder) a bias section are combined in one layer. About the appropriate Layer button in the Layers group 'prestressing', the visibility on and off.

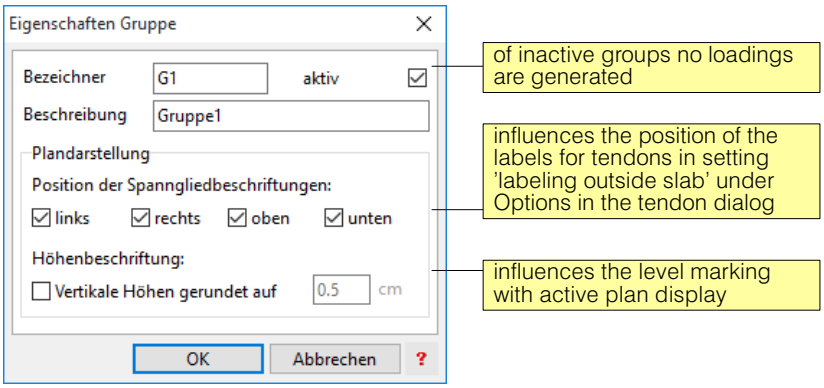
  For reasons of clarity, it is e.g. useful to manage the tendons in two bias directions in respective groups.

- All the votes of the tendons on the plate forces are summarized per prestressing group in a separate load. If you want these loads vary so you must have the corresponding prestressing groups.

The management of the group is carried out via the following instruments, which allows reintroduce bias groups, delete and edit their properties. The selection box is used to select the current group to be worked with which:

Newly input tendons and holders are assigned to the selected in the dialog above prestressing group. That this is possible, first at least one group must be defined. Changing the allocation is not possible. Typed objects can however be moved by 'cut' and 'paste' in a different prestressing group.

The characteristics of a prestressing group are summarized in the following dialog:

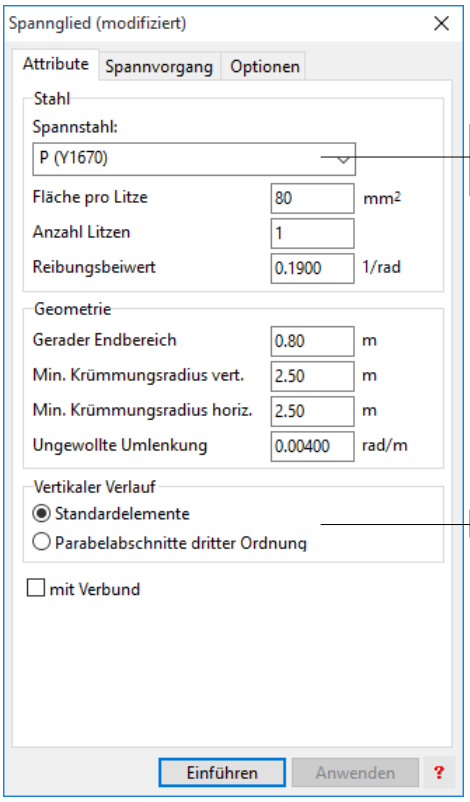


To position the tendon labels: The labels are positioned along the ticked edges, preferably the closest.

H 2.3 Tendon-Dialog

The properties dialog for tendons has three tabs:

Attributes



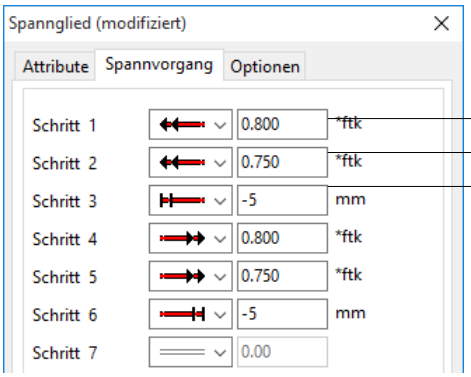
The attributes are self-explanatory

may choose from the menu 'Settings'> 'Materials' imported materials of the type of prestressing steel

refer to the chapter H 2.5.2

Tensioning process

Both ends of a tendon several clamping operations can be defined in terms of forces on the press respectively up ranges (negative values for relaxation):

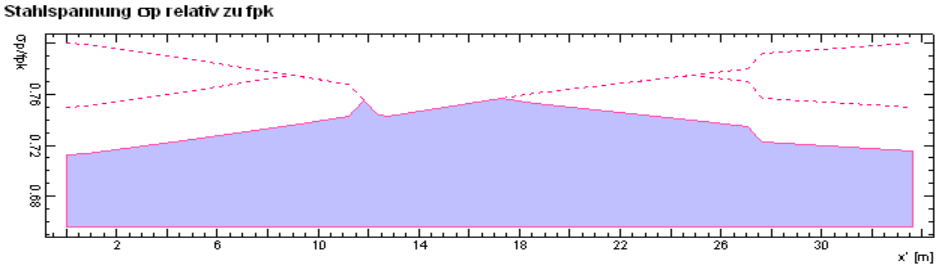


tension on the tendon start

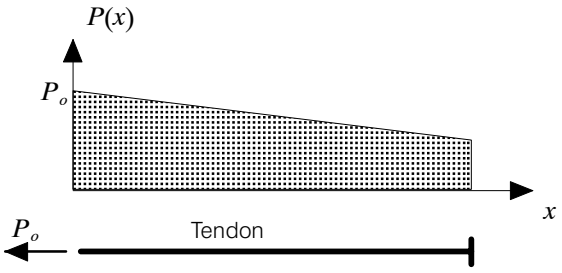
let up on smaller force

Wedge retraction of 5 mm (sign!)

The chip transactions presented could lead to the following clamping force profile:



The force curve in a tendon for a first-time tensioning is calculated using the following formula:



$$P(x) = P_o \cdot e^{-\mu (\alpha + \Delta\alpha\, x)}$$

$P(x)$: Prestressing force at a distance x from the clamping point

x : Distance from the clamping point along the tendon

P_o : Prestressing force at the clamping point

μ : Friction (Typical values in the range: 0.1 .. 0.3)

α : Sum of all deflection angles (Absolute values, radian) at a distance x
The deflection angle is determined from the spatial course of the axis of the tendon.

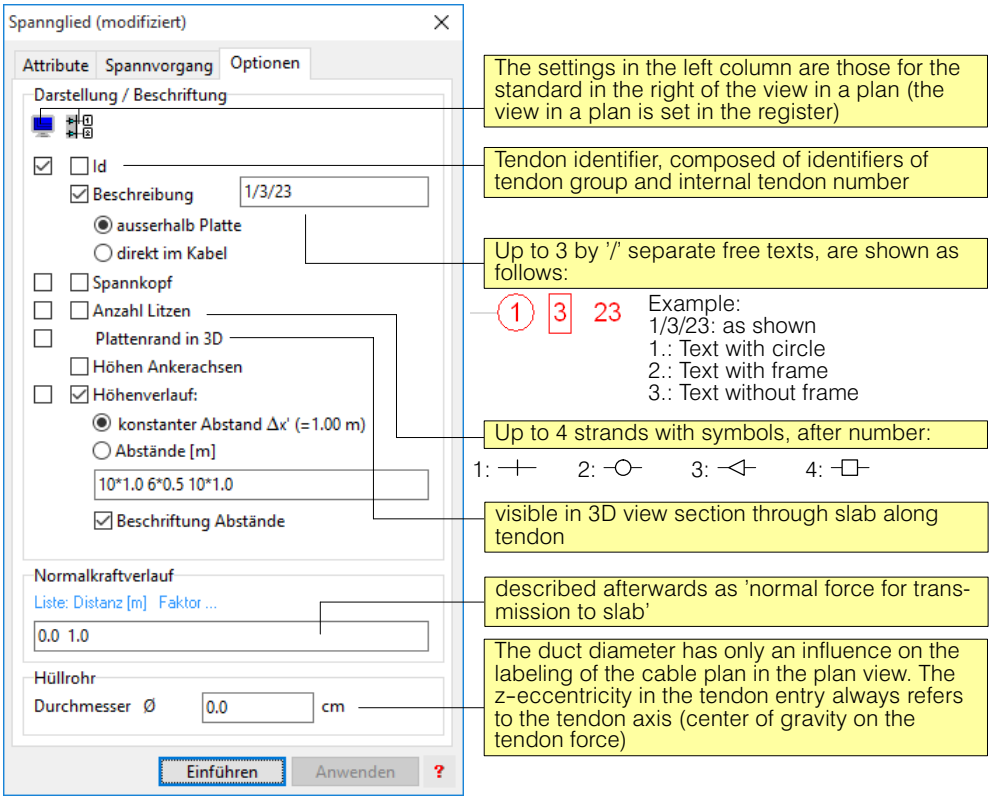
$\Delta\alpha$: unintentional deviation (per unit length)
values in the range: 0.004 .. 0.008 /m)

The associated movement capacity is proportional to the area under the curve and is (without taking into account the concrete shortening):

$$\Delta L = \int_0^L \frac{P(x)}{EA} dx$$

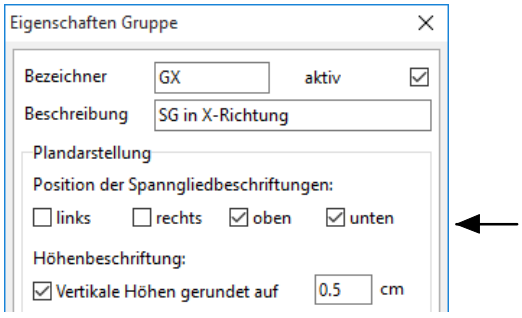
Options

This is about the labeling of the tendons and to the normal force component which in its environment transmits the tendon on the slab:



Description 'outside slab'

In the field 'Description' you can optionally enter a unique identifier for the tendon group. If the input field is empty, is displayed instead of the description of the 'Id'. In the Properties dialog of the tendon group can be set, at which slab edges the labels should be available:



Height profile / heights anchor axes

The caption of the height profile of the tendons is carried out either at constant intervals (acc. Setting in the Preferences dialog Edition ', H 2.6) or according explicitly given distance list. The distances are entered in [m] and along the outline curve, starting with the tendon start measured.
The attached written heights understood as distance from the formwork to bottom cladding tube according to the set duct diameter dialog 'tendon > Options'. The rounding of numerical values can be set in the Properties dialog of the tendon group (see illustration above).

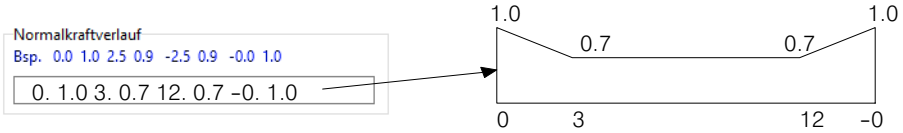
At the height of the lettering anchor axes (only in plan to see) the duct diameter are irrelevant.

Normal Force to transmission to slab

The normal force proportion a tendon transfers to the plate strip directly surrounds it, is required at two locations:

1) In the load generation of slabs with different thicknesses (siehe Chap. H 2.8.4).
2) In determining the prestressed reinforcement in beams sections, provided that the normal force is not taken over in the bar section of a slice calculation (siehe Kap. H 2.9).

This normal force component can be reduced as a result of supports jamming or spreads the introduced at anchor clamping forces along a tendon. The course is entered as follows:



The string entered describes the polygonal normal force along the course of a tendon by number pairs (distance and value).

The **distances** refer to the tendon start resp. on the tendon end, when you start with a minus sign and are measured along the floor plan line of the tendon.

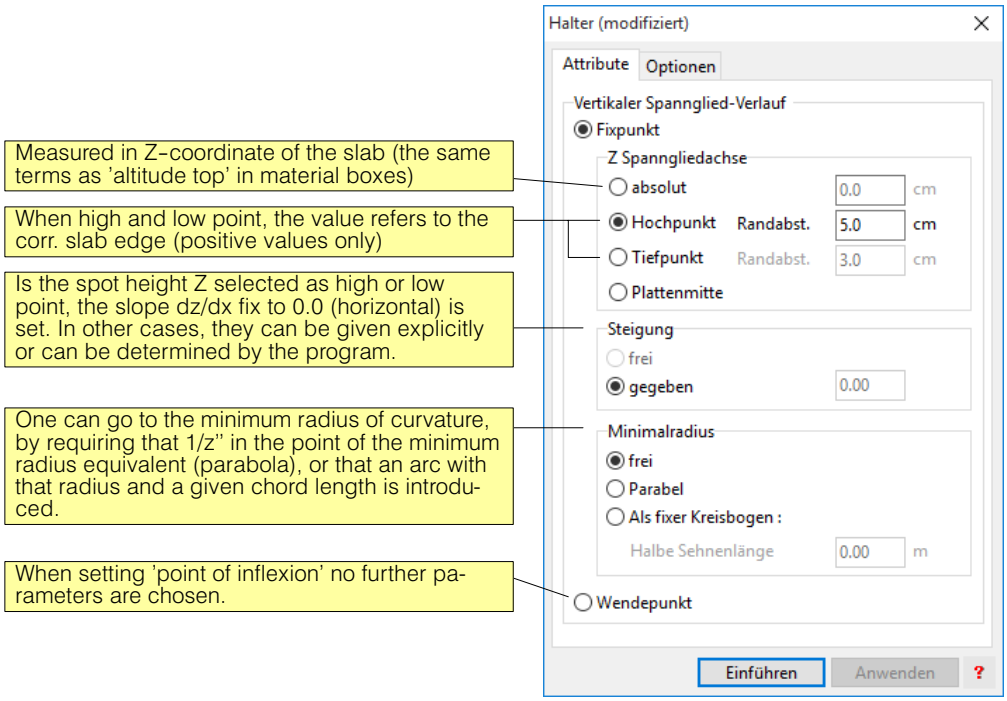
The **values** indicate how much of the tendon force at the relevant place in the accompanying slab strip acts as a normal force. A value 1.0 means that the normal force of the full tendon force equivalent.

If the first distance value greater than 0.0, so its value reverse applies to the tendon start. The same applies analogously to the tendon end. Is the text as '0.0 1.0', the value of 1.0 for the whole tendon applies.

H 2.4 Supports- and Tendon point-Dialog

The properties dialog of the two objects are nearly equal. It is both in order to define the vertical position of a tension member to a particular location (see also Kap. H 2.5).

Attribute



The tendon point dialog has against the above graduated Attributes tab of the tendon support dialog still has two additional properties:

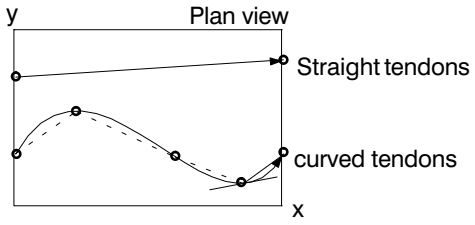
'Free' means that from the point forth no conditions are placed on the vertical course.

The proposed program from the tangent direction of a tendon point in the plan can hereby be overwritten. The inputted angle is measured on the basis of global X in the counterclockwise direction.

H 2.5 Tendon Geometry

As already mentioned in the introduction, the tendon profile is first drawn in plan view with the Graphics Editor and then brought into the desired vertical position using the supports. It is also possible to assign height attributes directly at every input point. For all internal calculations (e.g. of friction losses) the three dimensional profile of the tendon is used.

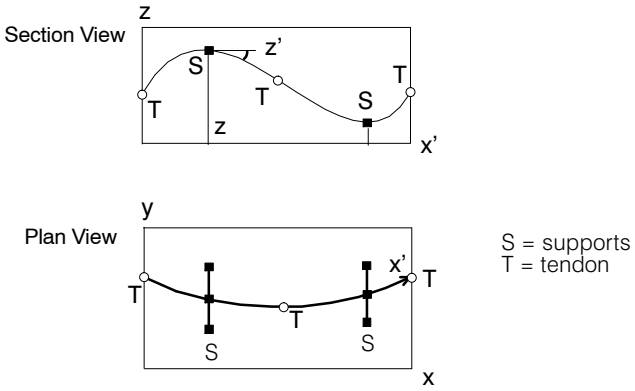
H 2.5.1 Geometrical description in the plan view



If the tendon was defined in plan by just two input points, a straight line profile is assumed. Curved tendons are produced by inputting several points, whereby the curved variation between two points is assumed to be a polynomial of third order. The slope of the tangent at a particular point is given by the slope of the two neighbouring stretches (i.e. the bisector). This default value can be overridden by the user.

H 2.5.2 Geometrical description in the side view

The definition of the tendon's vertical profile is given by the supports or by specifying height attributes at the tendon input points. The so-called folded view in the vertical section, which is permitted by the program, corresponds, as may be seen in the following pages of the manual, to the development of a curved surface in vertical section through the respective tendon. Therefore, the abscissa designated by x' represents the projection of the tendon onto the plan view.



Segments of tendons

In view of the deviation forces one often wants to work with special generating elements (e.g. "straight line-parabola", "parabola-parabola"). By means of the following example it is shown how in most cases met with in practice the support attributes can be controlled.

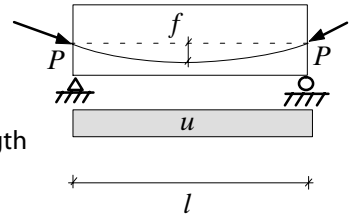
A tendon can be given the attribute "**Standard elements**" or **"Polynomial segment of 3rd order**".

Normally one works with standard elements (described below).

Segments of standard elements

The program tries in this case to describe the profile of the tendons between the support points with (quadratic) parabolic sections and straight lines, based on the attributes specified at the supports or at the intermediate points. The advantage of (quadratic) parabolas lies in the practically constant deviation forces:

$$u \approx P \frac{8f}{l^2}$$



with:
u = deviation force per unit length
P = prestressing force
f = parabolic height (sag)
l = span

In the following some standard cases are put together to show how they are created by the program on the basis of the attributes in the section (support and polygon points, respectively). As mentioned already in the introduction, the tendon profile is first drawn in the plan view using the Graphics Editor and then brought into the desired vertical position by inputting the supports. It is also possible to assign height attributes directly to the support points. In all internal calculations (e.g. of friction losses) the 3 D profile of the tendon is used.

It could happen that some conditions in a certain section are in conflict with each other or that the input values do not allow reasonable results to be obtained (e.g. chord lengths that are too long for circular arcs). In this case the program may have to ignore some conditions to still be able to draw a curve. All input height attributes (height z), however, are always used! In the vicinity of a support one should not assign height attributes to the polygon points, to avoid defining them twice. At the ends of the tendons a height input is always required by the program otherwise the start and end points are specified in a standard way at the middle axis of the slab (reason: avoiding tolerance problems with the calculation of the points of intersection between the tendons and the supports). The required user input is demonstrated by means of the two span beam shown in Fig. 2 .

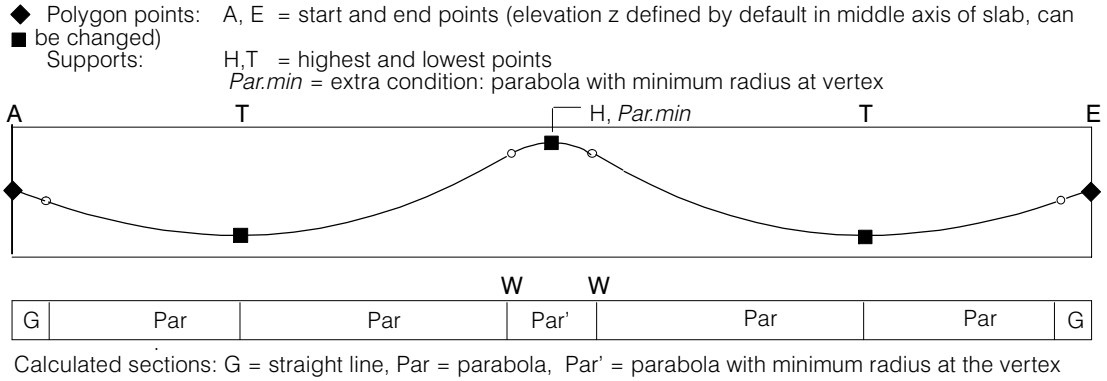
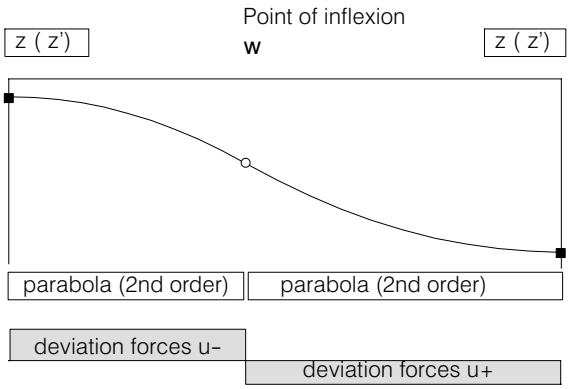
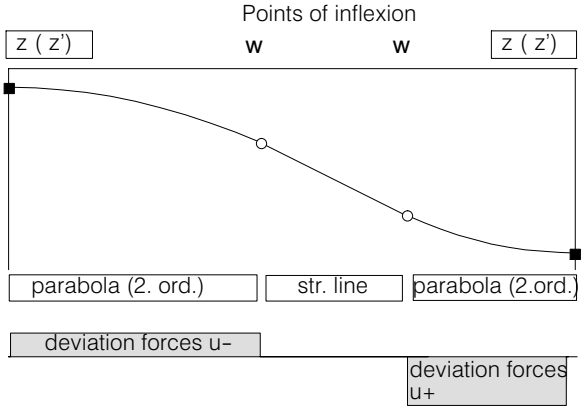


Fig. 2 *User input for a prestressed two span beam*

Further examples of single sections:

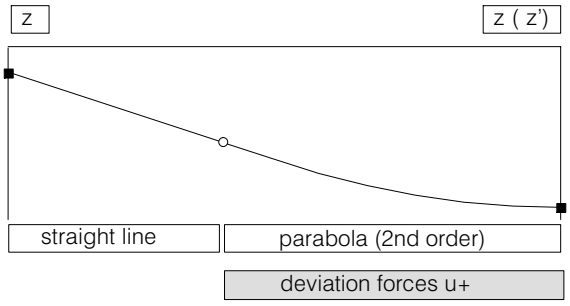


If, in the section under consideration, a point of inflexion is introduced between the highest and lowest points, then this defines the subdivision of the two parabolic sections. If no extra information is supplied the point of inflexion is placed in the middle of the section.

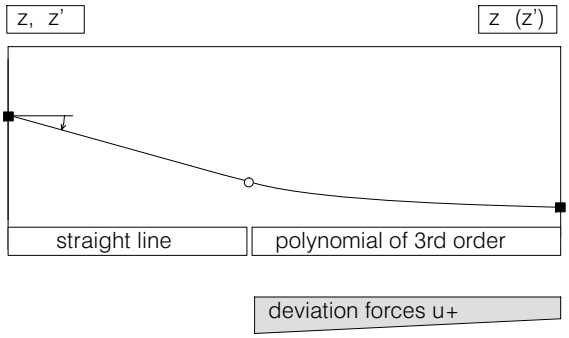


If there are two inflexion points, then the part in between is a straight line. Further inflexion points lying in between do not influence the shape of the curve, i.e. only the two outermost inflexion points are effective.

Fig. 3 *Geometry in a middle section*

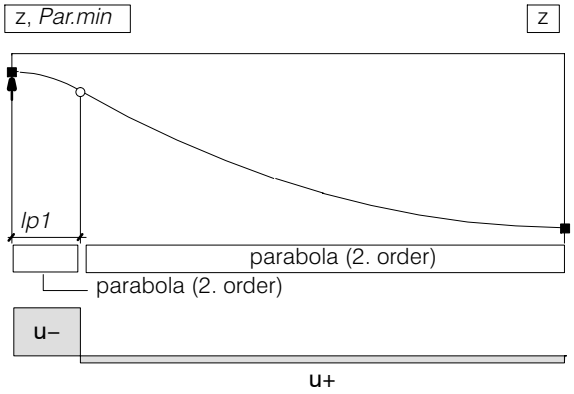


To avoid the start and end sections being geometrically redundant, at the anchorages only the height is specified. Then, optionally, at the end of the first section also the slope of the tangent z' can be specified.

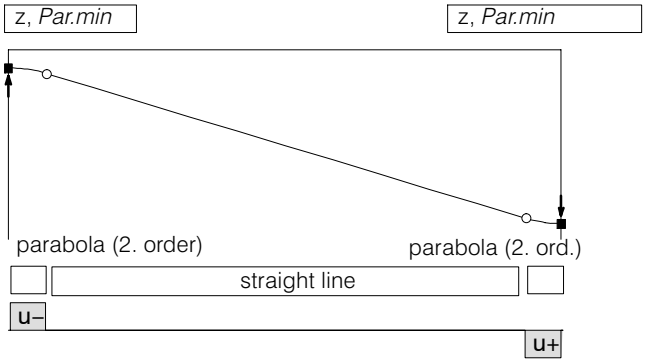


If, in addition to the height at the start, the slope is specified, the program uses at the end of the initial straight line a polynomial of 3rd order.

Fig. 4 *Geometry at start of tendon: straight line – parabola*

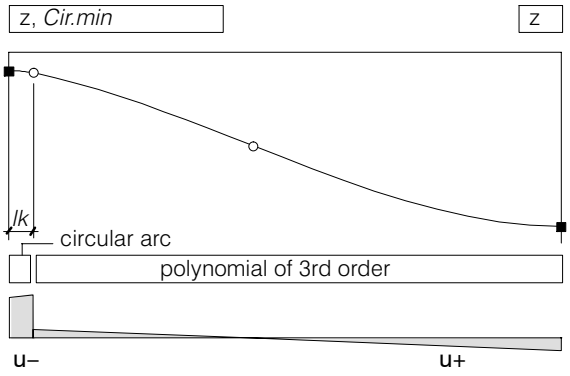


If a point of inflexion is moved closer to an end point, the radius of the shorter parabola section is always smaller. In the extreme, just permissible, case the minimum radius at the vertex of the curve is reached. This corresponds to the condition *Par.min* (parabola with minimum radius at vertex), which can be input as a support attribute, i.e. in this case the corresponding length of the parabola *lp1* is determined by the program.

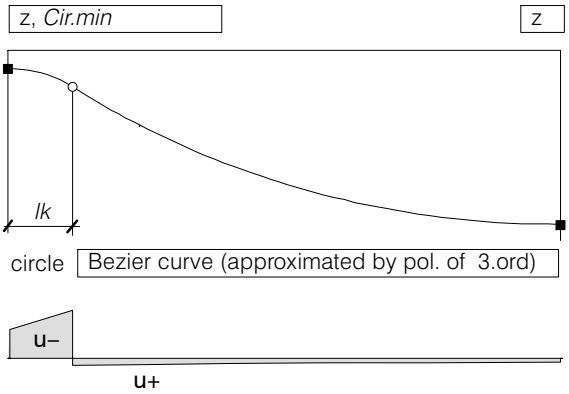


If, in the highest and lowest points, the condition Pmin is input, a trapezium can be produced.

Fig. 5 *Parabolas with minimum radius at vertex*

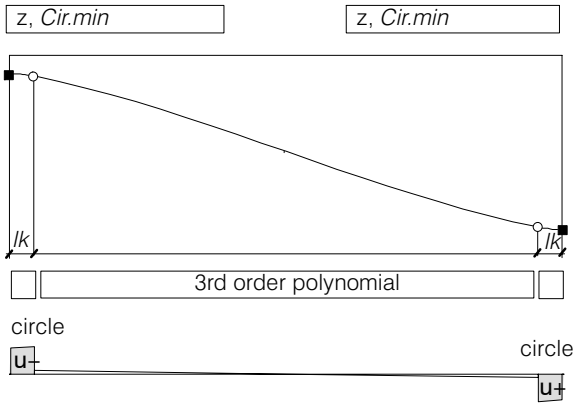


In contrast to the condition *Par.min* (see above) for which the minimum radius is just reached at the vertex, with the condition *Cir.min* a circular arc segment of half chord length l_k is produced. Purely geometrically therefore, at a given distance l_k the steepest variation (max. shear forces due to prestressing !!) is enforced. The program of course takes no account of constructional constraints (stiffness of the duct, etc.). If the length l_k is too short, the point of inflexion of the connecting 3rd order polynomial may not occur at the desired position.

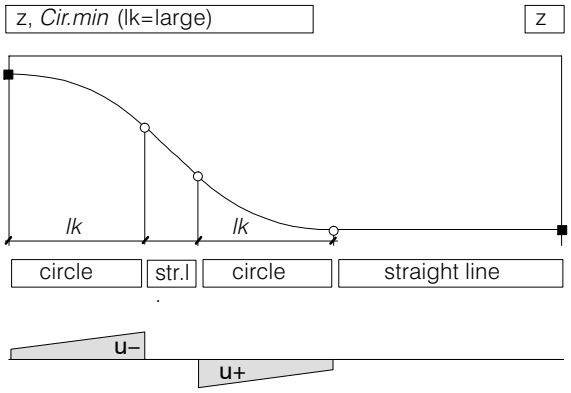


With a higher value of l_k the second section is approximated by several polynomials of 3rd order, in order to produce as smooth a curve as possible (approximation with Bezier curve).

Fig. 6 *Circle instead of parabola at vertex*



Section with circular arc conditions at both ends. For smaller values of l_k the transition is with a polynomial of 3rd order.

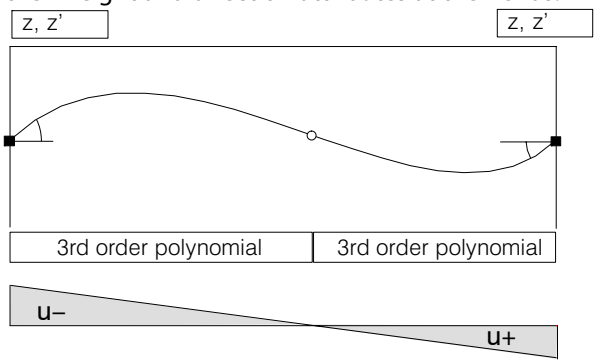


Circular arc section with very large value of l_k. In the extreme case the curve passes directly from one circle to another.

Fig. 7 *Further possibilities with circular arcs at vertex*

Parabola segments of third order

The setting 'parabola segments of third order' is used in special cases, for example when a given tensioning path is to be recorded as precisely as possible. They are clearly defined by their height and direction attributes at their ends.



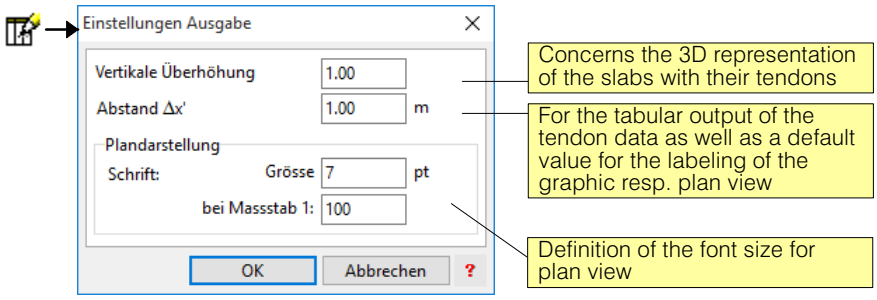
For “Non-Standard” input of tendons the program uses cubic polynomials. If the tangent slope is not explicitly given, it is calculated from the two adjoining polygon sides (bisector).

This input possibility is needed above all to recalculate existing objects, by inputting sequentially the tendon heights measured from the plan.

Fig. 8 *Cubic polynomials*

H 2.6 Dialog 'Output settings'

The following dialog allows certain settings for the tabular and graphical outputs of tendons:

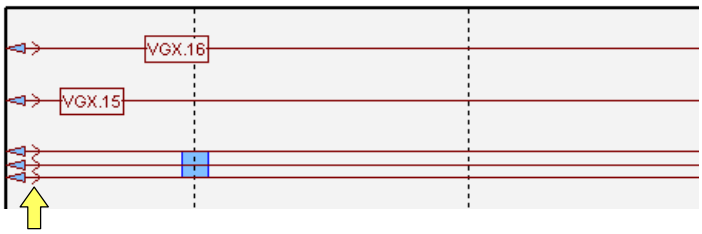


H 2.7 Standard presentation or Plan view

If this button is turned on, so the plan view is on, otherwise the default display applies. Both displays can be configured under 'Options' in the tendon-Dialog (see H 2.3).

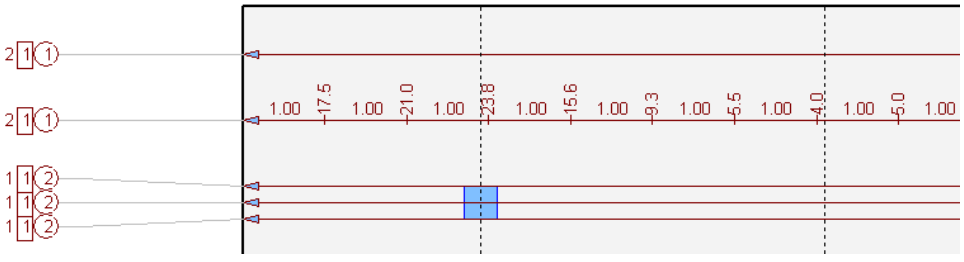
Standard presentation

The standard presentation is more useful while typing and for documentation purposes, and the output is normally used to CubusViewer (printing entries). The font sizes react as usual, that is, they are independent of the selected scale. Exclusively in this display there is a symbol of the tendon start, the one as know when defining the clamping operations must:



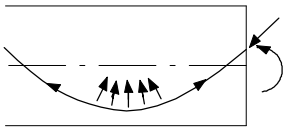
Plan view

The plan view is already closer to the layout plan and the most important addition to the standard presentation offers the possibility of individual labeling of the tendons outside the slab outline. As an example, the label look like and contain the item number, installation and tensioning sequence:



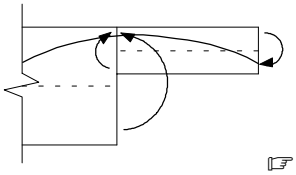
The plan view is especially suitable for the DXF export ('Export' > 'DXF' in the context menu of the graphic area). The final editing is then performed in a CAD program.
The plan view works with a to scale font size that is defined in the dialog 'output settings'.

H 2.8 Deviation Forces



As a result of prestressing the following loading elements are generated:

- Deviation forces
  These continuously distributed forces are proportional to the curvatures of the tendon (determined in the vertical section) and to the prestressing force. By not taking into account the friction losses the deviation forces would be constant for each section (2nd order polygon) or linear (3rd order polygon)
- Friction forces
  In sloping tendon sections, due to friction a small vertical component is produced, which has to be considered for vertical equilibrium to be fulfilled.



- Anchorage forces
  Due to the anchorage forces vertical **point loads** and correspondingly directed **end moments** are introduced. To guarantee consistency such concentrated forces are also introduced at abrupt changes in the slab thickness.

☞ *The loading elements generated by the the Prestressing Module can be checked individually in the Tabsheet 'Loads' as with 'conventional' loads.*

H 2.8.1 Exclusive slab model

If the effect of the prestressing to the exclusive slab model examined, only the vertical component and the Endmoments may be introduced as loads. The normal forces due to prestressing are not simply "forget", but their inclusion will be discussed the following sections:
- H 2.8.4 Load generation for slabs with different thicknesses
- Design of conventional reinforcement: Section "Normal force due to prestressing"

H 2.8.2 Slab with normal forces

If you have licensed the S option (slices), as addition to the regular slab model, the model 'slab with normal force' to choice (see Chap. G 3). For slab with normal force, the normal force can be determined automatically in the slab due to prestressing. The disk model proceeds from respect to a global mean surface centrally arranged slab zones, i.e. there is no interaction between moments and axial forces.

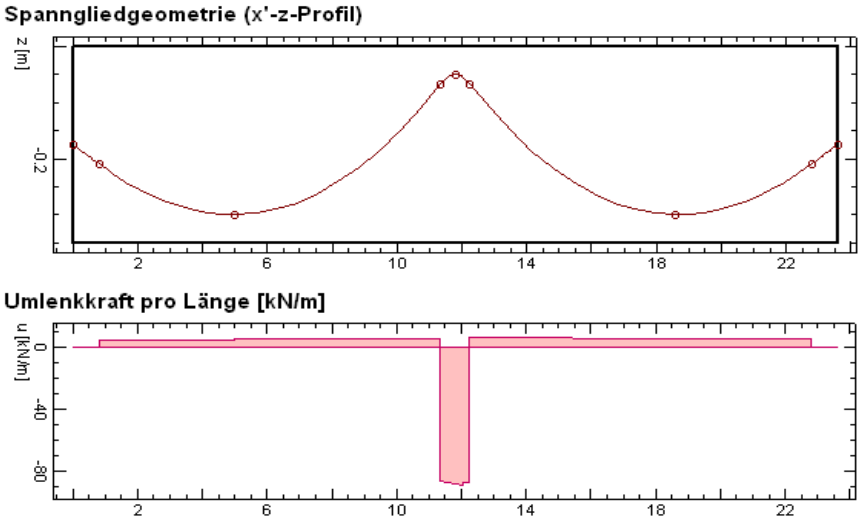
Reality | Calculation model for normal forces

H 2.8.3 Load generation for slab with constant thickness

For slab with constant thickness give the tendons following forces to the slab from:

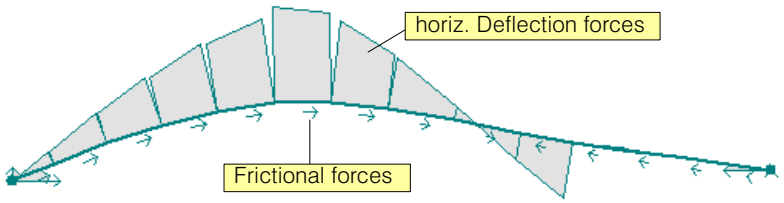
Vertical deflection forces

These forces are proportional to (in elevation given) tendon curvature and the prestressing force. Excluding the friction losses, the deflection forces per section constant at parabolic sections 2nd order linear or parabolic sections 3 in order would be. The forces generated can be checked at the tendon graphic (below) or in the generated load.



Horizontal deflection forces

On model 'slab with normal force' and the resulting deflection forces are introduced as slice forces at horizontal tendon curvature. The following figure shows a portion of a graph of the prestressing load generated with a tendon in the ground plan with the generated deflection and frictional forces.

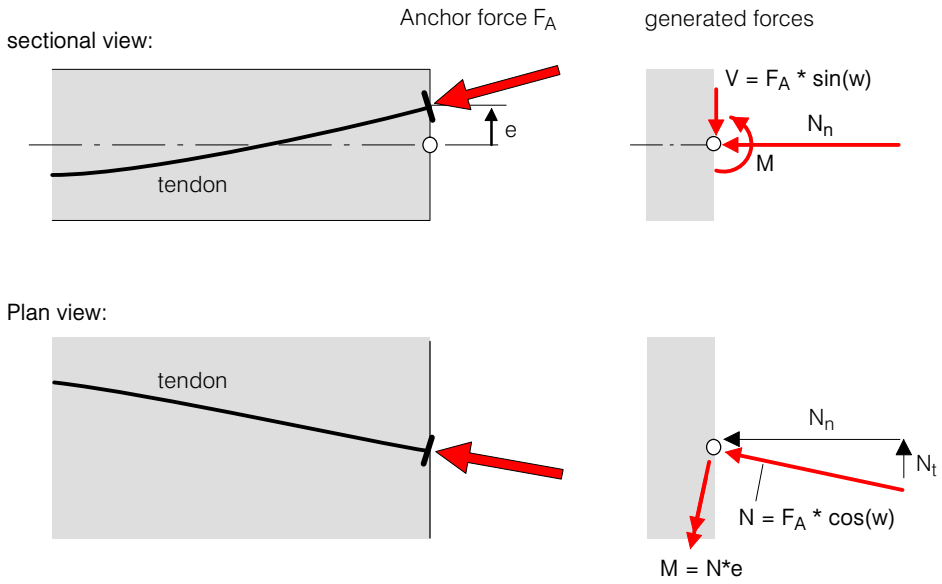


Frictional forces

The frictional forces acting at each point in the direction of the axis of the tendon. Depending on the tendon course arise corresponding forces in vertical and horizontal directions. The latter are generated model only 'slab with normal force' and shown in the graph of the prestressing load (illustration above).

Anchor forces

The in the anchor points of the tendons to the given anchor slab forces the anchor point in the direction of the axis of the tendon. The following forces are introduced:



w = Angle of inclination of the tendon with respect to horizontal plane
N_n produced with 'normal force slab' model only

H 2.8.4 Load generation for slab with different thicknesses

Slab of different thicknesses (ceiling reinforcements, downstand beams) may be treated under certain conditions approximated regardless of the slab model, that is by the way in which the normal forces can be determined. The requirements arising from the approach described below and can approximately be given as follows:

Conditions

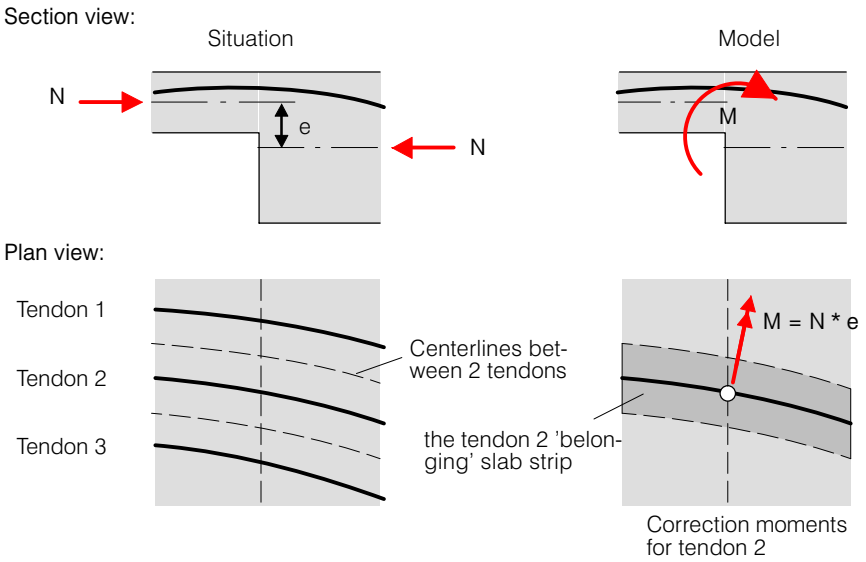
- no significant forces may from the bearings of the slab to the plate act (floating support)
- The tendons in the plan should mainly lie in one or two main directions. With two main directions it must be perpendicular to each other.
- The geometry of the slab should have a clear orthogonality in the main directions of the tendons.
- The tendons should more or less regularly over the plate be distributed (support strips prestressing allowed).
- The main directions of the normal forces in the slab should preferably correspond to the tendon main directions.

Approach solution

The forces described in H 2.8.3 also be introduced here in the same way. In addition, the following corrections are made:

Tendons pass through ceiling leaps of the slab

When eccentricity leaps in the slab (downstand beams are specially treated) are introduced corrective moments, which is illustrated by the following figure:

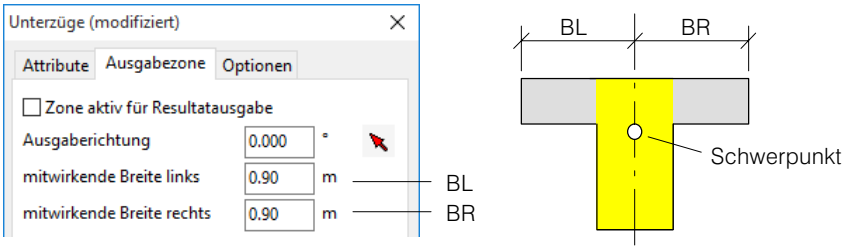


The correction moments are introduced at the points where the tendons cut discontinuity. It is believed that the normal forces N acting in the direction of the tendons.

The normal force N acting in belonging to a tendon slab strip is defined in the tendon dialog tab 'Options' under 'normal force history' (see page H-8).

Tendons in downstand beams

For tendons that run along by downstand beams, will be specially procedure by the gravity of the downstand beams is not accepted at half beam formwork, but the focus of the T-section, which is formed by the downstand beam with its inputted effective width:



In the described eccentricities e above is expected in ceiling leaps or anchor points with the focus of the cut T-section.

When entering the tendon endpoints are by default placed in the middle of the concrete body. Running is prestressing towards the downstand beams, so the endpoints should then still be moved to the gravity point of the T-cross section (> select endpoint <RMB>, properties ...). For already entered tendons the position zs of gravity in the table can be found "Geometry and clamping forces":

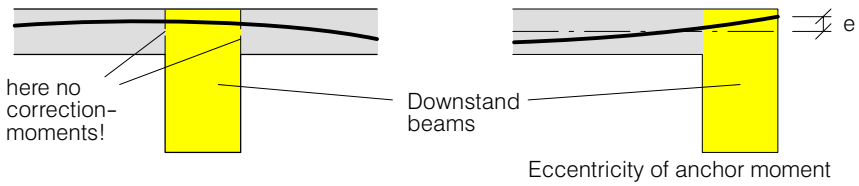
Geometrie und Spannkräfte

Id	x [m]	y [m]	z [m]	x' [m]	dz/dx'	Kote [m]	Dicke [m]	h_{Halter} [cm]	σ_p/f_{pk}
VGX.1			z_s=-0.14					Ø=0	
	0.00	0.50	-0.14				0.28	14.0	0.6445
	5.00	0.50	-0.19				0.28	9.3	0.6688
	10.00	0.50	-0.14				0.28	14.1	0.6868
	15.00	0.50	-0.04	15.00	0	0	0.28	24.0	0.6853
	20.00	0.50	-0.06	20.00	-0.0937	0	0.28	22.3	0.6703

The z in the beginning resp. Endpoint should normally correspond to the amounts reported here with zs gravity

Tendons across downstand beams

For tendons which are transverse to downstand beams, is used as the center of gravity of those of the slab. There is therefore no consideration to the downstand beam:



H 2.9 Dimensioning of the Non-Prestressed Reinforcement

Basically prestressing can be treated in two different ways in the dimensioning of non-prestressed reinforcing steel. Either as an external action on the structure, i.e. only on the loading side, or as a dead load case with the tendons as part of the system. In the second case the tendons are included in the determination of the resistance.

H 2.9.1 Prestressing as an external action

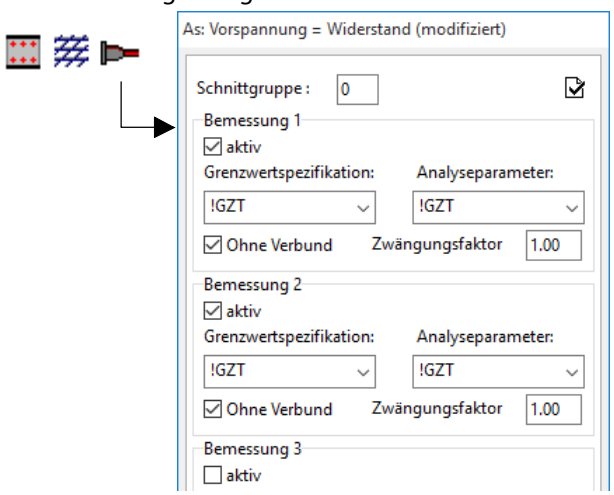
The prestressing load cases, as they are generated in the Prestressing Module, comprise all actions of the tendons on the slab. Therefore, using the standard procedures of CEDRUS the reinforcing steel is dimensioned for ultimate load conditions (i.e. failure). It should be noted that here the load factors for the prestressing case are specified according to the codes. What cannot (without the possibilities described in this chapter) be considered by CEDRUS as a program specifically for slabs, however, are the axial forces in the slab.

H 2.9.2 Prestressing as self-equilibrating stress state

In many publications it is implied that prestressing should be considered as self-equilibrating stress state (due to constrained deformations) and thus the tendons can be considered in the determination of the cross sectional resistances. While this is obvious in the case of beams and there is no problem, in the case of slabs several difficulties have to be overcome. How this is solved in the CEDRUS in dimensioning for prestressing, is described below.

The design criteria

The design can be carried out one or more criteria that are specified in the calculation register in the following dialogue:



The dimensioning conditions

The actions on the slab consist of the components

S_d : Design limits of the sectional forces due to external actions

S_z : Sectional forces due to constrained deformations during prestressing (constrained sectional forces)

The action effects are determined by means of a linear elastic analysis using the FE method. The verification of the ultimate load bearing capacity is carried out at certain points or over sections as follows:

$$S_d + \zeta S_z \leq \frac{R}{\gamma_R}$$

ζ : Constraint factor:
The constrained moments represent a load-free stress state and can be taken into account according to plasticity theory using an arbitrary factor ζ. Practical values of ζ are in the range 0 to 1.

R : Cross sectional resistance taking into account the tendons.

γ_R : Resistance factor.

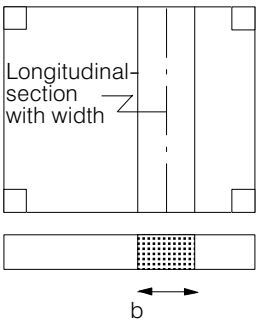
Since the bending action at a point in a slab does not comprise one value only (M_x, M_y, M_{xy}), the above formula cannot be applied directly, but in a appropriate form for the slab.

In CEDRUS the linearized yield conditions are:

$$M_{xB+} \leq R_{x+}/\gamma_R \qquad M_{xB+} : \; Max\,(M_{x+} + \| M_{xy} \|)$$
$$M_{xB-} \geq R_{x-}/\gamma_R \qquad M_{xB-} : \; Min\,(M_{x-} - \| M_{xy} \|)$$
$$M_{yB+} \leq R_{y+}/\gamma_R \qquad M_{yB+} : \; Max\,(M_{y+} + \| M_{xy} \|)$$
$$M_{yB-} \geq R_{y-}/\gamma_R \qquad M_{yB-} : \; Min\,(M_{y-} - \| M_{xy} \|)$$

These conditions apply for two orthogonal reinforcement directions x, y. They cannot be applied for non-orthogonal reinforcement.

Dimensioning as beams



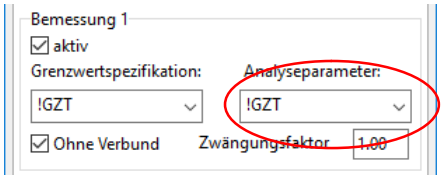
The dimensioning in CEDRUS generally follows at particular points according to the above yield conditions. While, in the determination of the necessary resistance, the reinforcing steel can be considered to be 'smeared' (cm^2/m), this is difficult for individual tendons. Thus the question arises of the width of influence of a tendon in determining the resistance at a point. A point-wise consideration is hardly possible any more.

The dimensioning program, therefore, uses for the prestressing a type of strip method. The user defines these strips in the form of longitudinal sections of a certain width (beam sections of CEDRUS). Such a longitudinal section cuts a **beam** out of a slab, which is considered as such for the dimensioning.

The action effects in such a beam are supplied by the beam sectional results of the basic CEDRUS module, which summarizes the results of the FE calculation for the beam width.

Cross sectional resistance

The determination of the cross sectional resistance is carried out according to the usual design rules for bending and axial force (equilibrium, plane sections remain plane etc.). The following information describes the default settings made by the program. With the aid of the cross section program FAGUS it is possible, to change these for special investigations.



By default a maximum concrete compression of -3.0 ‰ and on the tensile side a maximum strain of the reinforcing steel of 5.0 ‰ are assumed. This limit also applies to the additional strain of the prestressing steel, if this is also in the extreme fibre position. Each tendon is treated with it correct position and initial strain, corresponding to the geometrical input. The concrete is assumed to be cracked in tension.

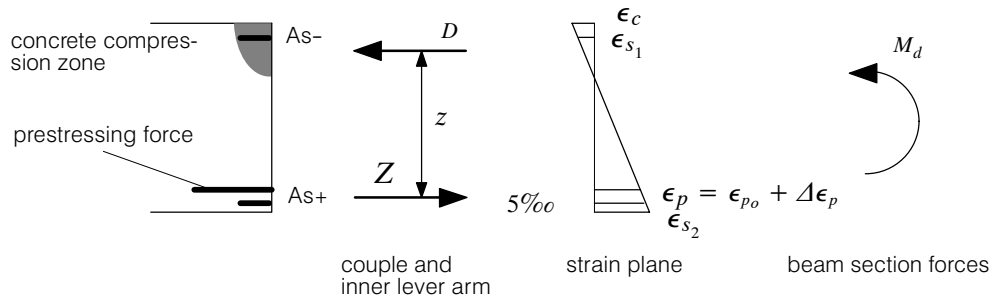
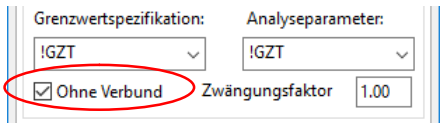


Fig. 9 *Determination of cross sectional resistance*

The initial strain ϵ_{p_o} is calculated in every section according to the stressing programme and the friction losses for each tendon individually. In a section at a distance x from the start of the tendon this is:

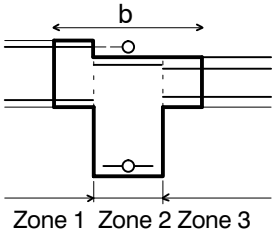
$$\epsilon_{p_o} = \frac{P(x)}{EA_p}$$

The effective contribution of the tendon then depends on whether it is bonded to the surrounding concrete. For unbonded tendons $\Delta\epsilon_p$=0 and therefore the prestressing force depends on the actual strain plane. The composite property is set calculation specification dialog shown below :



For the resistance of the beam all tendons within the beam are considered. Through the choice of the beam width b, therefore, the user also determines the influence width of the tendons.

For the cover of the reinforcing steel the values input for the zone attributes are used. If the dimensioning cross section includes several zones of different thickness and concrete cover, the reinforcement used in dimensioning is placed in the extreme positions (represented by a circle symbol in the diagram on the left). If necessary, compression reinforcement is introduced.



Serviceablility limit state (SLS):

By default the design criterium is the stress limit for the mild reinforcement in tension. The **limiting stress** (depending on bar diameter, spacing etc.) **must be specified by the user** (see menu Settings > Analysis parameter > SLS).

Axial force due to prestressing

For two reasons it may happen that the total prestressing force of all tendons in the beam under consideration are transferred to the concrete cross section of the beam.

- In the case of column strips over the columns the axial force distributes itself more or less uniformly at a certain distance from the anchorage over the whole slab. The cut-out beam therefore only resists a part of it. In general, this part can be estimated quite well.

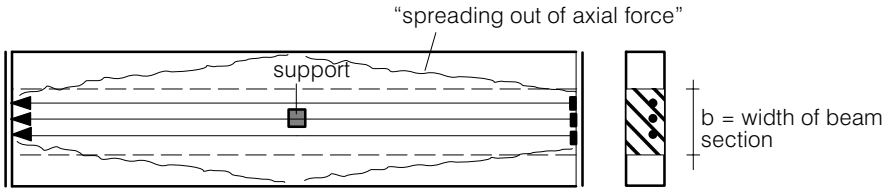


Fig. 10 *Prestressed slab beam*

- The slab supports themselves can also impede the transmission of the prestressing force into the beam. This effect, if present, is difficult to estimate.

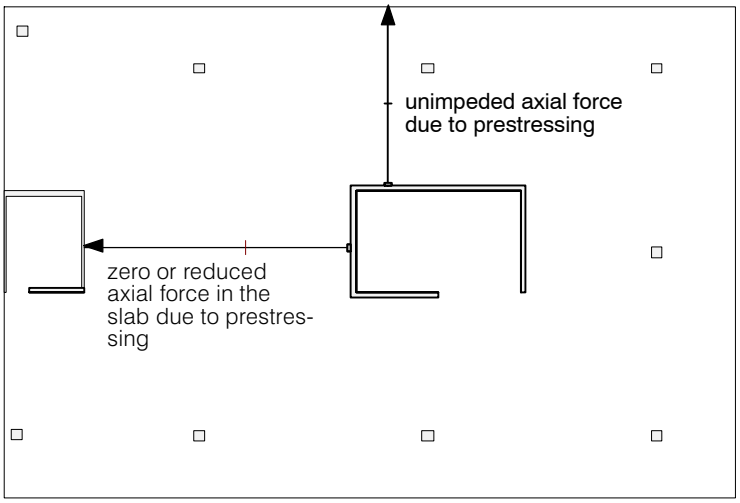


Fig. 11 *Plan view of floor slab*

In the input field 'Portion of Axial Force: Factor' the user can define the expected portion of axial force in the beam using a factor. With the factor = 0.0 there is none, with 1.0 the whole of the axial force is transmitted to the beam.
Since this factor is constant for the whole of the beam section it may be necessary to introduce several sections one after another. But due to the uncertainty of the factor too much time expenditure is not justifiable.

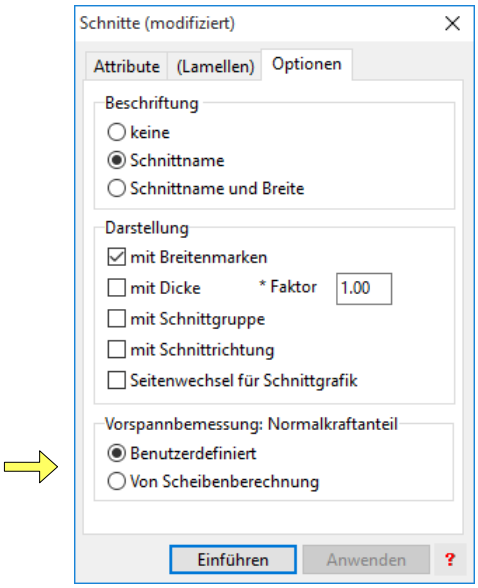
☞ *Tip: It is not always simple to decide how large is the existing axial force in a certain region due to prestressing. This, admittedly, places certain demands on the user. The advantage of a pure slab calculation, however, is that thereby in the calculation assumptions one is forced to adopt simple and clear models. In contrast to apparently more exact calculations (plane stress states, shells) there is the danger of false interpretations (uncertain assumptions regarding stiffness of supports etc.).*

With a stacked slab model the program has no possibility to calculate the normal force from pretension in the beam section. This is determined on the basis of the settings made in the tendon elements (see page H-8).

For the 'slab with normal forces' model, you can choose between

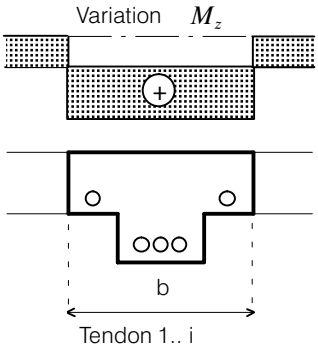
- User defined: N as for the stacked slab model, see above
- From wall analysis

The corresponding setting is made in the sections dialog:



Constraint moments

The constraint moments caused by stressing a group of tendons may be calculated as follows:



$M_P = M_{P \cdot e} + M_z \quad \rightarrow M_z = M_P - M_{P \cdot e}$

M_P : moment in beam due to prestressing as external action, i.e. calculated from the load case deviation forces + anchorage forces.

$M_{P \cdot e}$: $-P \cdot e$ (negative portion of sectional force in the tendon)

M_z : constraint moment (in CEDRUS this constraint moment can be changed using a factor ζ)

This means that in zones where there are no tendons we have directly the constraint moments. In the other zones the question of the influence width arises when introducing the portion of the sectional force in the tendon.

The user also solves this problem by choosing the beam width. All tendons lying within the beam give their portion of the sectional force to the beam.

☞ *Consistent with other actions the moment M_p here too is formed and correctly superimposed by $M_{px} \pm \| M_{pxy} \|$*

Direction of the tendons and of the associated beam sections

The above dimensioning conditions apply to reinforcement in the orthogonal directions x,y. In order to include tendons here, they must also lie in these directions. Small deviations α are permitted, whereby the program multiplies the corresponding steel sections by $\cos^2\alpha$. Tendons with $\alpha > 10°$ are not considered for the resistance. In the case of buildings this condition is usually fulfilled, but rarely for skew bridge deck slabs.

The directions of the beam sections have to agree with the directions of the reinforcement, and the complete dimensioning within a slab zone must always be carried out in these directions. If, for example, in one place only two skew beam sections are considered, the yield conditions would not be fulfilled!

In zones with no tendons the normal dimensioning results of CEDRUS can be used. These are also supplied in the "tendon strips", but there they are not relevant in the tendon direction.

Remarks on the Results Tables

The additional reinforcing steel can be shown for each beam section both in a graphical form and in a numerical form. In the following the relevant numerical outputs are shown, whereby these, depending on the program version, can differ slightly from the form presented.

Limit state specification

The table with the limit state specifications gives information on the loading, as well as the verification type:

With the limit state 'Ultimate Load' the calculations are carried out at the dimensioning level. The material strengths are reduced according to the partial safety factors input with the calculation settings.

In contrast, the dimensioning for serviceability is normally carried out at the working stress level. The cross section is dimensioned for 'Admissible Stresses'. The corresponding load factors can be adjusted, but in this case for the material behaviour no partial safety factors are used.

```
LIMIT STATE SPECIFICATION ACCORDING TO SIA:   !Ultimate (ULS)
Limit state: Ultimate (ULS)
Actions                        Design situations
-------------------------------------------------------------------------
Nr    Name     1               2
0     !Dead load               1.3            0.8
1     !Imposed load            1.5            1.5
2     !Prestressing force      1              1
```

As-Table

The As-Table for the additional reinforcing steel looks as follows:

```
Beam section: A (0.00,1.50 - 23.60,1.50) Beam width 3.00
Limit state specification: !Ultimate (ULS)
As-design ultimate state:B35/25, S500, γc=1.20, γs=1.20
Portion of axial force due to prestressing =  1.00
PT treated as resistance, constraint factor=  1.00
Distance    Md_min    Md_max    Mp_min    Mp_max      -P*e    As-top  As+bottom
   [m]      [kNm]     [kNm]     [kNm]     [kNm]     [kNm]     [mm2]     [mm2]
-------------------------------------------------------------------------------
   0.00      1.23      3.32      0.05      0.05      0.00      0.00      0.00
   0.48     41.50    111.91    -15.45    -15.45     18.06      0.00      2.00
   0.97     81.28    219.17    -30.90    -30.90     36.05      0.00      5.00
              [1]       [1]       [2]       [2]       [3]       [4]       [5]
Explanations:
[1] Limit state moments due to external loading (without prestressing):
[2] Limit state moments due to prestressing (anchorage and deviation forces)
[3] Moments due to -P*e (only for tendons in corresponding section)
[4] Dimensioned top reinforcing steel
[5] Dimensioned bottom reinforcing steel
```

The distance between two neighbouring dimensioning sections along the beam is chosen by the program itself and cannot be changed.

Prestressing record

For a check on the tendons within each dimensioning section the following table is output:

Spannglieder

Distanz [m]	Id.	Stahl	P [kN]	A_p [mm²]	ε [‰]	z_p [cm]	e [cm]	-P*e [kNm]	Winkel [°]	V
1.45	G1.T2	Y1860	126	100	6.46	-23.5	-6.0	7.60	0.0	-
	G1.T3	Y1860	126	100	6.46	-23.5	-6.0	7.60	0.0	-
	G1.T4	Y1860	126	100	6.46	-23.5	-6.0	7.60	0.0	-
	G1.T5	Y1860	504	400	6.46	-23.5	-6.0	30.41	0.0	-
	G1.T1	Y1860	126	100	6.46	-23.5	-6.0	7.60	0.0	-
	Summe		1008	800				60.81		

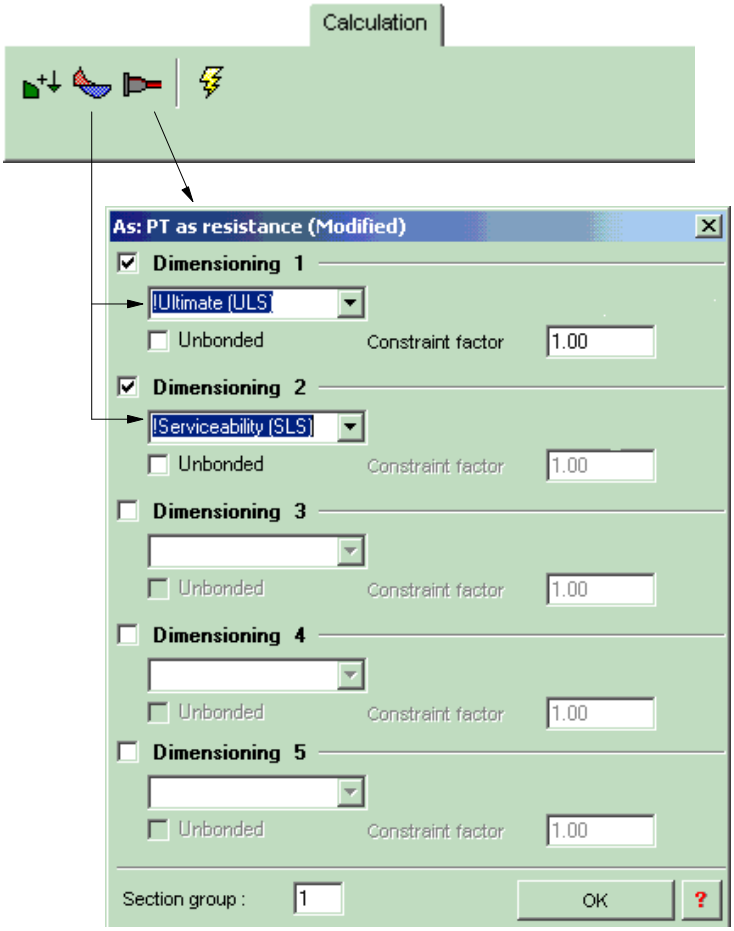
Legend:
Dist : Distance from start of section
zs : Coordinate of centroid of dimensioning cross section
hmax : Max. cross section height
width : Beam width of dimensioning cross section
co,cu : Concrete cover of reinforcing steel (top,bottom)
Id. : Tendon identification
steel : Name of PT steel
P : Tendon force
Ap : Total cross section of prestressing steel
Eps : Strain in prestressing steel due to chosen stressing procedure
zp : Position of tendon with respect to top of slab (+ve upwards)
e : Eccentricity of tendon relative to centroid of dimensioning c. s.

angle : Angle between axis of beam and tendon
(0 = Tendon is parallel to beam axis)

H 2.9.3 Use of program

The calculation method described in the previous section is available only for beam sections for the reasons mentioned. In order to obtain the corresponding results, the input described below is necessary:

Input in the tabsheet 'Calculation'



With the aid of the above dialogue, first of all a special specification has to be made. For each calculation run up to 5 different dimensioning criteria may be decisive. In the above case, e.g., first of all dimensioning is carried out for “failure” with the actions defined under '!Ultimate load state' and then for '!Serviceability state'. Both sets of results are output in tabular form, as also the As limit values from both calculations.

Basically, all automatically or self-defined limit state specifications in dimensioning for prestressing can be used, whereby the action '!Prestress load' in the design situations should normally appear with the factor 1.00. Values which differ from this influence the prestressing force and the initial strain in the prestressing steel, but not the tendon geometry. With a factor = 0.85, e.g., the prestressing losses due to creep and shrinkage can be considered.

Constraint factor ζ:
In dimensioning for the ultimate state the constraints of the prestressing load cases are taken into account using an arbitrary factor. This constraint factor ζ has to be input in the corresponding field on the right hand side, where, as a rule, reasonable values are in the range 0.00 to 1.00. It is recommended when calculating a floor slab to use the same value everywhere!

Without bonding:
With the instruction 'Not bonded' the property input as tendon attribute for this calculation run is deactivated (e.g. assessment of construction states, if the bonding is produced later).

Input in the Tabsheet 'Results'

The 'Prestressing Beam Section Dimensioning' designated as 'As: Prestressing = Resistance' is available in the selection list shown on the left below. As Output Type for such results only beam sections are available.

H 3 Examples

H 3.1 Flat Slabs

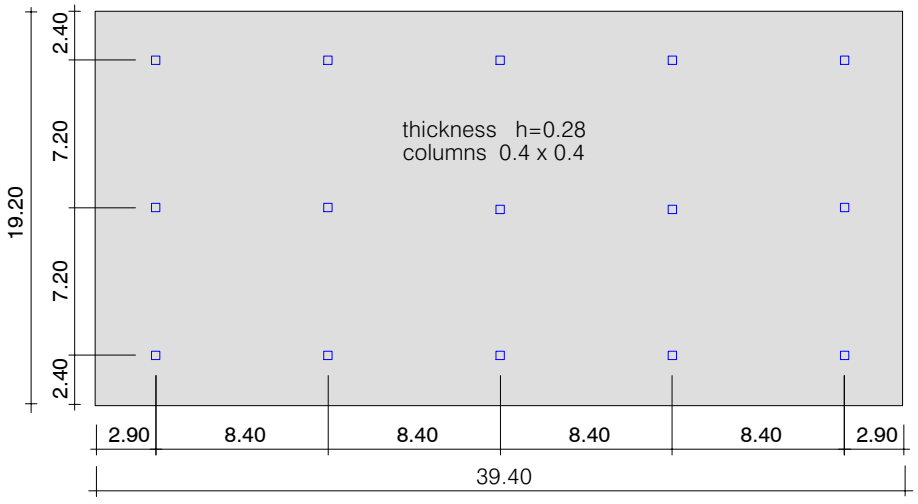
H 3.1.1 Description of problem

By means of the floor slab treated below the the individual input steps for the calculation of a prestressed slab are discussed. This provides above all a basic introduction to the method of working with the prestressing module. Actual numbers are introduced in the second example in connection with the dimensioning of the reinforcing steel.

It is assumed that the user already knows how to use the Graphics Editor. In order to become familiar with the input of geometrical objects it is recommended to work through the introductory example in the basic module of CEDRUS. Further tips are given in the Program's Help System. The input of the prestressing was simplified in this example, in order to make the description easier.

H 3.1.2 Tab sheet 'Structure'

Initially the slab outline, openings and columns are input, whereby all geometrical data may be taken from the diagram given below..



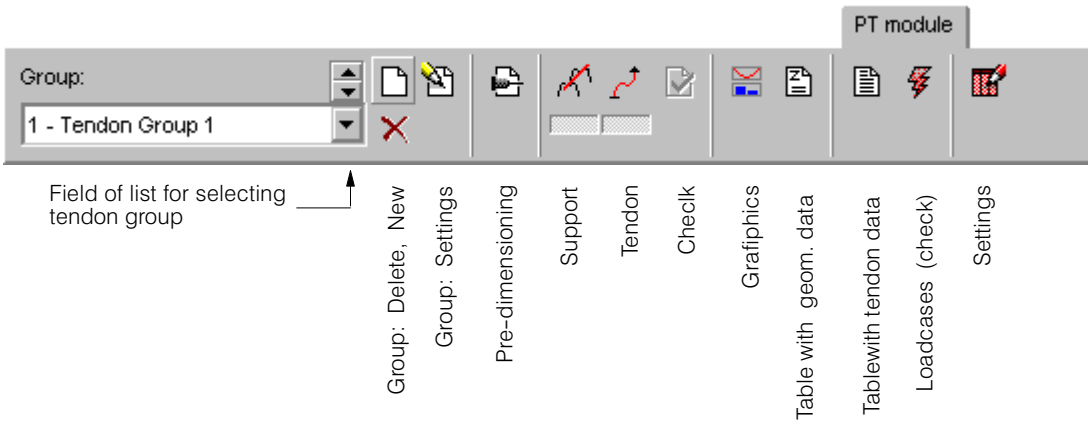
Code	Swisscode (SIA 2xx)
Materials:	Concrete C35/45, Steel B500B, PT steel Y1860.
Prestressing:	A_p =100 mm2 (per strand), $f_y = 0.9 \cdot f_{pk}$ =1670 N/mm^2 , f_{pk} = 1860 N/mm^2

H 3.1.3 Tab sheet 'Loads'

The following two load cases are considered
- Dead load (automatically generated by CEDRUS)
- Live load-general (additive) q = -5 kN/m^2 (whole slab)

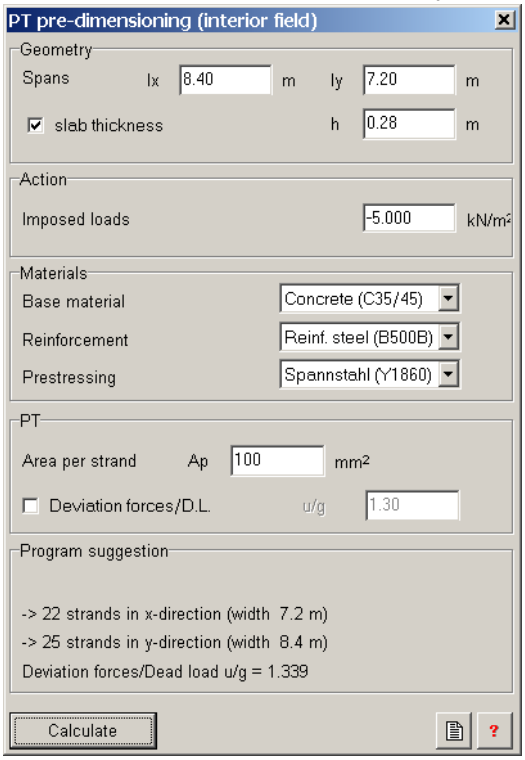
The later has to be input. laster, another load case with the action type 'Prestressing' is automatically added with the input of the prestressing cables.

H 3.1.4 Tab sheet 'Prestressing'



Prestressed flat slabs are probably not often designed by the majority of engineers. In order to quickly obtain an overview, therefore, CEDRUS-5 provides a special program for dimensioning the internal field of a floor slab.

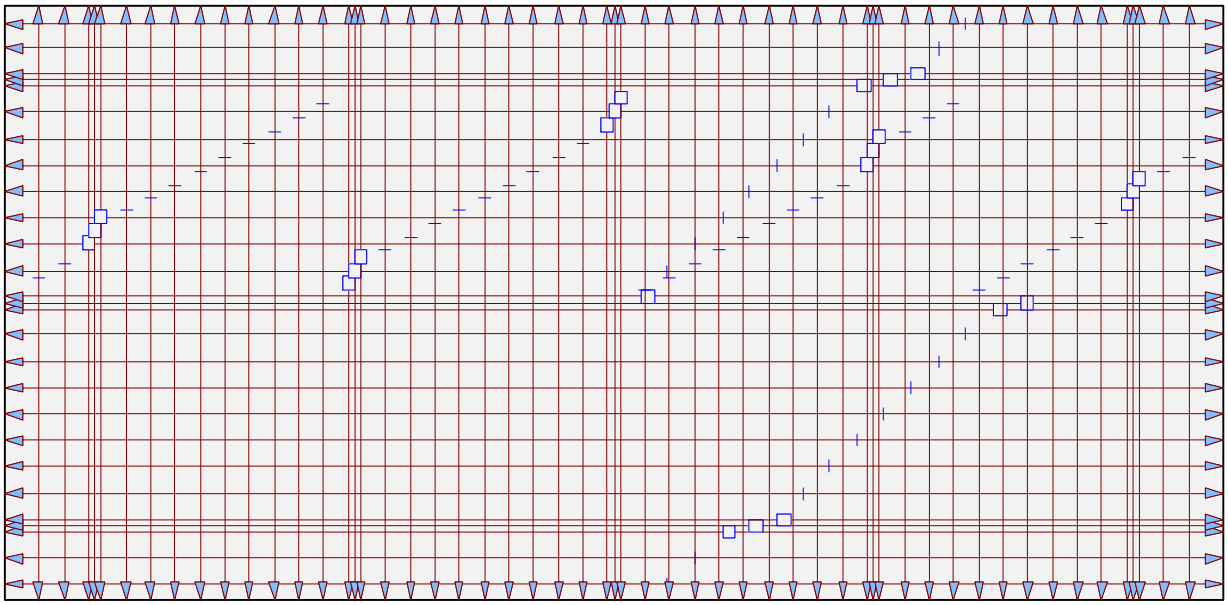
First we select this dialogue field and input as spans for the internal field lx = 8.4 m and ly = 7.2 m. At the moment we do not specify the slab thickness, i.e. it is determined by the program itself. For the loading the value of q = -5 kN/m^2 has to be input and for the prestressing we select the steel 'Y1860', as well as Ap = 100 mm^2.



With [Calculate] we obtain a 'proposal' from the program itself. For different reasons (punching shear, acoustics) we change the suggested slab thickness to h = 0.28 m and we repeat the whole calculation. We now look more closely at the detailed preliminary prestressing sheet (possibly print it out) and then decide upon the following prestressing:

- Column strips: 12 strands (3 "bundles of 4") in the x and y directions
- Field strip: 7 x 1 strand in the x direction, 9 x 1 strand in the y direction

With respect to our introductory example the following arrangement should then be obtained:



Input of tendons and supports

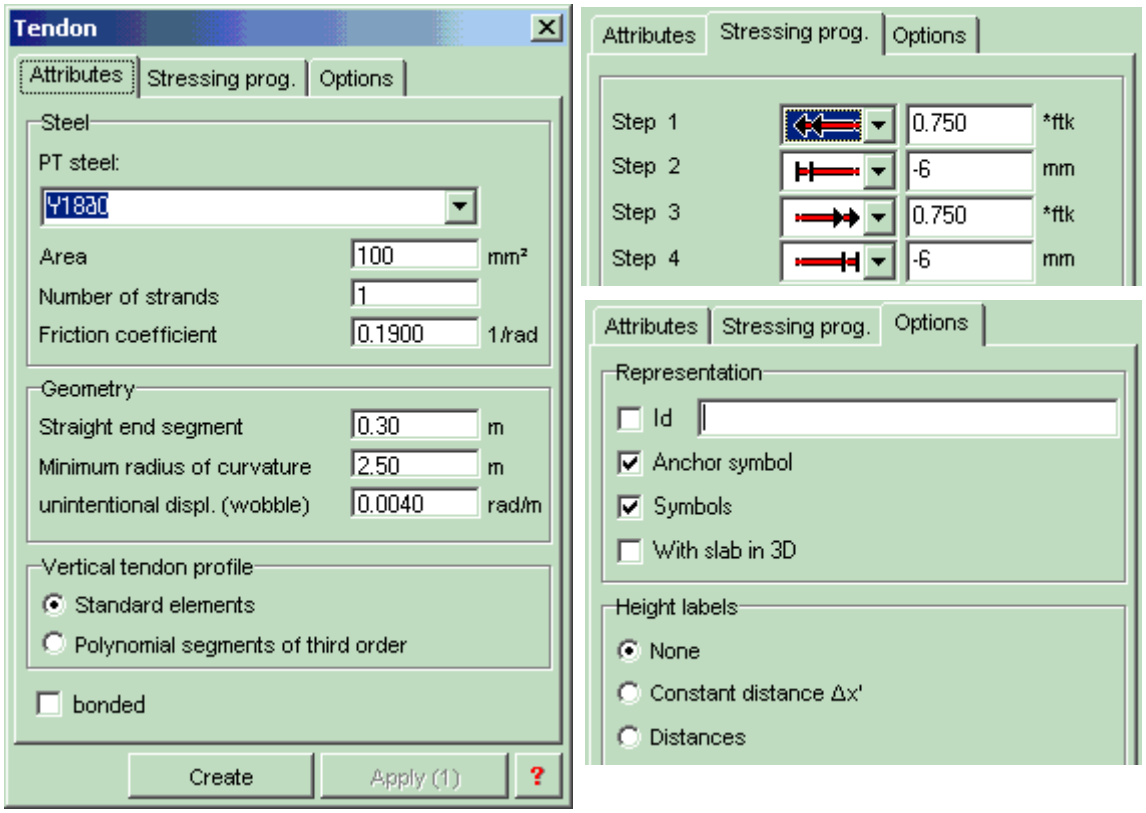
The tendon profile is first drawn with the Graphics Editor in plan view and then brought to the desired vertical position by inputting supports.

In order to give a better overview of this procedure the tendon input is divided into the following two groups:

Group 1: Tendons in the x-direction
Group 2: Tendons in the y-direction

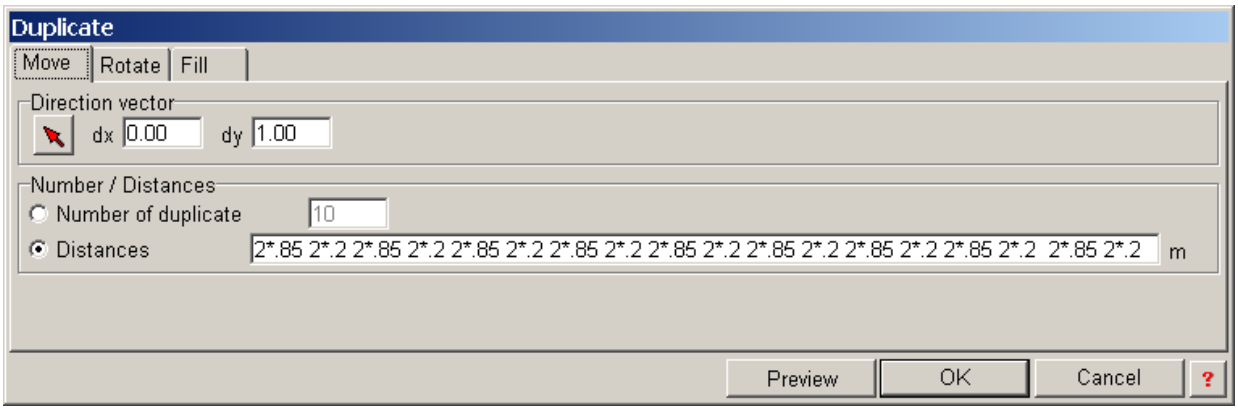
We begin with Group 1:

Now the first x-tendon is input over the bottom row of columns with the following attributes (due to the long tendon lengths stressing is carried out from both sides):



With the button [Create] the polygon input is started with the Graphics Editors and the discrete points of the tendon curve are input geometrically. In our case the starting point is at the left boundary of the slab at (0.0 / 0.5). For the input of the second point we press the x-key, whereby the mouse can only be moved in a horizontal direction, and we click on the right slab boundary. We complete the polygon input by pressing the key <Q>.

Now we duplicate the element just input by first selecting it and then using the function 'Duplicate' from the Context menu with the following parameters:



Distances: 2*0.85 2*0.2 etc.

During the input of the above row, provided the button 'Preview' in the dialogue window is selected, the elements to be created are continuously shown ('dimmed'), i.e. immediate checking is possible at all times.

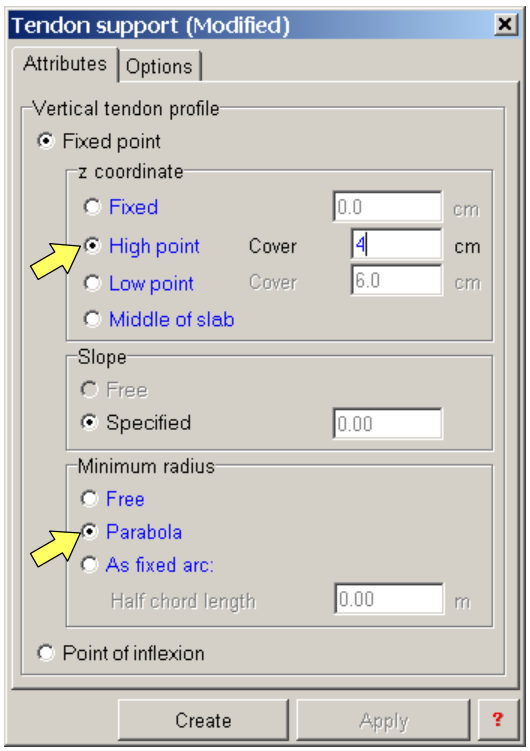
After confirming with [OK] all tendons in the x-direction will be generated.

In the region of the column strips we still have to change the number of strands. For every column strip we select the three tendons and choose 'Attributes' from the Context menu. We change the field ‚No. of strands' to the new value '4' and confirm with 'Apply'.

Now follows the input of the supports. We assume a distance to boundary of 30 mm and that the external diameter of the tendon is 20 mm. The distance from the slab boundary to the line of action of the force in the steel is thus 40 mm for the tendons in the x-direction. In our example, due to the intersection at the middle of the support, this value is increased to 60 mm for the tendons in the y-direction.

In an actual application, the geometrical relationships over the supports have to be planned exactly, whereby in the manufacturers' product documentation of prestressing systems one can find much information regarding this question. For CEDRUS as a calculation program only the centre line of the steel force is of interest. Possible geometrical conflicts,however, may be detected by means of the graphical output (see Checks).

The dialogue window is filled with the following values:



With the button [Create] we can place the supports geometrically. The first point is at (2.9/0) and the last point in the y-direction at the top boundary of the slab.

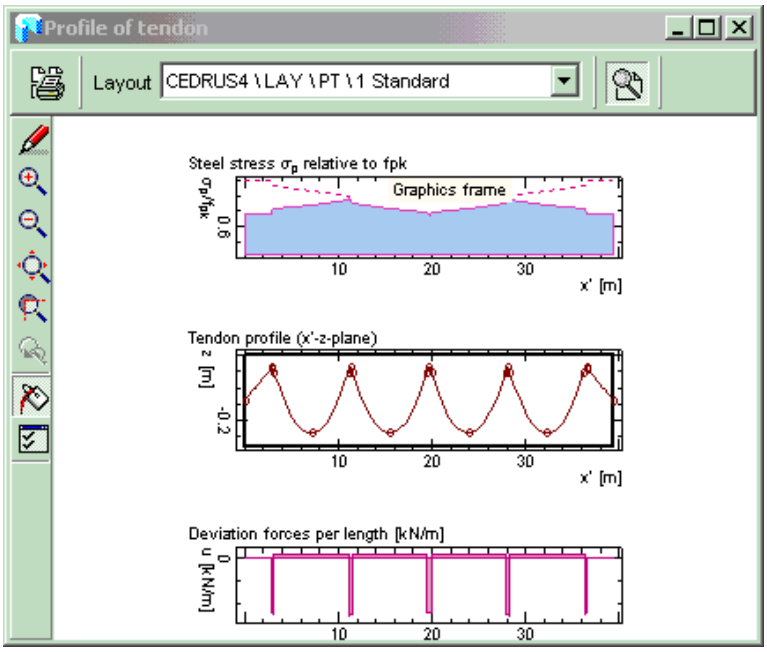
The remaining supports are generated again with the function 'Duplicate' from the Context menu with the following values.
Direction vector: dx = 0.0 dy:=1.0
Distances: 8*4.20 (half span length)

Now we have to change the attributes of all tendon lowest points. We select all supports in the middle of the field and assign them the attribute 'Lowest Point' with distance to boundary of 40 mm, as well as in the section 'Minimum Radius' the value 'free'.

Checks

Now we want to check individual tendons by selecting a particular tendon and then click on the button shown on the left.

In a new dialogue window we should see something like the following diagram



In the above A4 page layout the variation of steel stress, the tendon geometry and the deviations forces transmitted by the tendon to the slab are presented. In our case among other things we should check that the negative deviation forces concentrated in the area of the supports (=highest point) are transmitted to the slab. At the highest point we have specified the minimum radius condition and naturally we want the inflexion point with regard to the punching shear check to be near to the supports.

With the print entry symbol in the upper left corner of this diagram, if required, can be entered in the print list.

Within the input process typical tendons should always be considered and checked in each direction with this function. The dialogue window should never be closed to consider a new tendon, but left in the 'background' any tendon can be selected and the graphics adjusts itself automatically.

The **second tendon group** is created in the same way with the following steps:

- Define the new Group (2)
- Input tendon in y-direction:
  On opening the attributes dialogue reset the input field 'No. of Strands' to 1. The first tendon starts with (1.10/0.0) and ends at the top slab boundary.
- One could once again duplicate all tendons with the following instruction:
  Direction vector: dx = 1.0 dy = 0.0
  Distances:
  2*0.8 2*0.2 10*0.8 2*0.2 10*0.8 2*0.2 10*0.8 2*0.2 10*0.8 2*0.2 2*0.8

  The input of the above line, however, is somewhat lengthy and thus prone to error. As an alternative one could by duplicating just create the first field (Distances: 2*0.8 2*0.2 10*0.8 1*0.2) and then create the rest of the tendons by **mirroring** twice about the corresponding column axes.
- Increase number of strands for the column strips:
  Key words: select relevant tendons, increase attribute 'No. of Strands' once more to 4, [Apply]
- Input supports over the first row of columns (point on left boundary 0/2.4) with following attributes:
  Height: 'Highest Point' with distance to boundary of 60 mm
  Minimum radius: 'Parabola'
- Duplicate supports
  'Distances': 4*3.60 (half span length)
- Change support attributes in middle of field using following attributes:
  Height: 'Lowest Point' with distance to boundary of 60 mm
  Minimum radius: 'free'

  After checking the tendon visually with the input is completed.

Now we leave this window and call the function shown on the left.

By this means the deviation and anchorage forces for all prestressing load cases are formed and at the same time a record file is written. If the program encounters incorrectly input tendons (intersection with slab boundary, violation of minimum radius condition etc.) a corresponding error message is written.

From the information given at the end of the table one can see how large is the sum of the deviation forces. A comparison with the dead load gives:

g (dead load) = 25 kN/m^3 x 39.40 m x 19.20 m x 0.28 m = 5295 kN
u+ (sum of all positive deviation forces) $\approx$ 9524 kN

For each internal field, of the total of 40 strands 24 of them are positioned in the column strips. Only these strands transmit the deviation forces directly to the columns. The ratio u/g therefore can be estimated as follows:

u/g = (24/40) x 9524 /5295 $\approx$ 1.08

If we compare this value with the preliminary dimensioning: Instead of the suggested 47 strands we have only retained 40 of them. In addition, the distances to the boundary for the tendons running in the y-direction are greater than those assumed in the predimensioning.

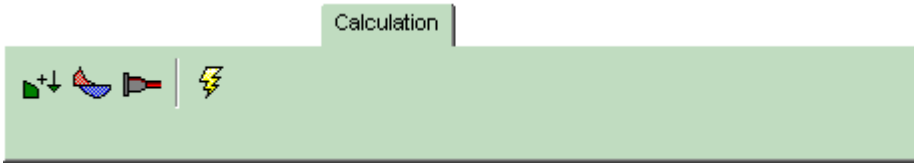
The ratio u/g can also be calculated from a comparison of the two bending deflection diagrams for the load cases dead weight and prestressing.

Since prestressing corresponds to a self-weight stress state, the prestressing load case considered by itself has to be fulfil equilibrium, i.e. the sum of all vertical loads must (up to a certain tolerance value) be equal to zero. This can be checked quite easily in the tabsheet 'Loads' in the tabular output in the last line of the table. The prestressing layout should also be checked once again graphically: Positive and negative deviation forces are presented in different colours and thus it can be seen immediately whether, e.g., the negative deviation forces are actually transmitted to the columns.

Tendons input twice, which lie directly on top of each other, are shown by the program in the table of prestressing data with a warning sign. If the number of strands is shown graphically (Context menu 'Attributes', 'Options'), we can detect such (probably undesired input) immediately, since the corresponding graphical symbols are displayed for each tendon at a different place.

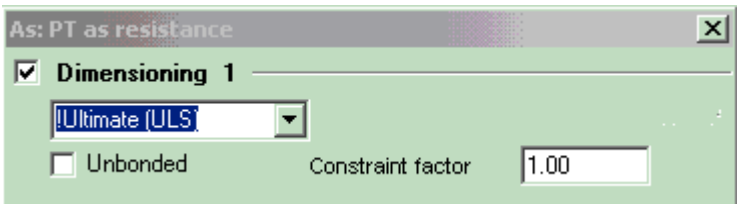
Fig. 12 *Tendon with single strand* *Two tendons each with single strand*

H 3.1.5 Tab sheet 'Calculation'



In our example we are interested above all in the additionally required reinforcing steel, which we want to determine with the aid of a beam section dimensioning. To ensure that the corresponding results are available, first of all we have to make the corresponding specification in the tabsheet 'Calculation' . By default the program creates limit state specifications for '!Ultimate Load' and '!Serviceability'.

We produce now a new specification, stored by the program under the term 'Prestressing=Resistance', by selecting the function on the left and inputting the values shown in the dialogue below:



For the calculation of the action effect the limit state specification '!Ultimate Load' the following factors are used:

```
LIMIT STATE SPECIFICATION   code: SIA 162
ACTION:   !dead load
      permanent                   L1:  [1.00]  Loadcase 1
ACTION:   !imposed load
      plus where decisive         L2:  [1.00]  Loadcase 2
ACTION:   !Prestressing
      permanent                   L3:  [1.00]  Loadcase 3
LIMIT STATE SPECIFICATION:   !Ultimate (ULS)
Limit state: !Ultimate (ULS)
Actions                           Design situations
No.   Name                        1         2
-------------------------------------------------------------------------
0     !Dead load                  1.3       0.8
1     !Imposed load               1.5       1.5
2     !Prestressing               1         1
```

With regard to the last line it should be noted that of the prestressing forces only the constraints are used if the beam section dimensioning is carried out for 'As: Prestressing=Resistance'.

H 3.1.6 Tab sheet 'Results'

Now we change to the tabsheet 'Results', and in the column 'Results for' we choose the line 'As: PT=Resistance'

Then we input over the first row of columns a beam section of width 1.50 m and an axial force factor = 1.00 and examine the corresponding result, i.e. the required additional reinforcing steel.

With this the introductory example comes to a conclusion. In practice, however, one would still have to produce an extensive documentation, but we will leave this out. In the sense of a checklist the most important points in the calculation of a prestressed floor slab are summarized once more in the next section in abbreviated form.

H 3.1.7 Checklist

The following checklist reiterates the most important points which have to be considered in a prestressed floor slab using CEDRUS:

- Input of the outline in plan and of the loads in the basic module of CEDRUS. An 'empty' load case of action type 'Prestressing' also has to be introduced in order to be able later to include the loads generated by the prestressing module.
- Specify reinforcement and prestressing layout qualitatively (width of column strips, gradation of reinforcement etc), clarify geometrical relationships at points of intersection (fix distances to boundaries).
- Floor slab with uniform arrangement of columns:
  - Carry out pre-dimensioning (with the CEDRUS pre-dimensioning program)
  - Select number of strands
  - Provide additional reinforcement possibly in boundary and special areas

  Geometrically complex floor slab:
  determination of a fictitious reinforcing steel using conventional CEDRUS dimensioning methods (beam sections). Then convert to prestressing cross sections corresponding to the ratio of the two steel stresses.
  Reminder: This factor depends on whether a prestressing with or without bonding is chosen (failure stress or existing prestressing steel stress)
- Input prestressing force
- Compare bending deflections due to permanent loads and prestressing in all fields and if necessary adjust prestressing force
- Ultimate load verification: Determine additional reinforcing steel with the means provided by CEDRUS
- Check stresses (Note: CEDRUS only calculates moments, so the axial force has to be estimated). Aim: As few cracks as possible immediately after prestressing
- Check punching shear
- Produce layout plan etc (possibly with export of DXF files to a CAD program)

H 3.2 Two Span Beams

H 3.2.1 Problem description

By means of this example the explanations of chapter H 2.9.2 are documented numerically. For this purpose a simple system was deliberately chosen, so that the values can also be obtained by hand calculations.

Structure type Slab
Code: Swisscode
Materials: Concrete C25/30, (->E_c = 32 kN/mm^2, $\nu = 0.17$) Steel B500B,
Boundary distances for reinforcing steel = 30 mm
Structure: 2 span beam (2 x11.80 m x 3.00 m) with linear bearings (walls:
height = 3 m,Young = 3.5e7 kN/m^2, support: freely rotational,
wall stiffnes) and columns in the middle of the field (0.40 m x 0.40 m,
Area support: settlement = column stiffness, rotations = free)
Slab thickness: d = 0.35 m
Prestressing: According to Fig. 13 and Fig. 14
Prestressing with no bonding, vert. variation with standard elements
4 strands (Y1860) "in field" distributed according to sketch
4 strands as "bundle" (Y1860) over columns according to sketch
Area per strand: Ap = 100 mm^2,
Rmin = 2.5 m, friction coefficient: μ = 0.19/rad, $\Delta\alpha$ = 0.004 rad/m
Stressing procedure: force at start = 0.75 f_{pk} (f_{pk} = 1860 N/mm^2)
Wedge draw-in = -6 mm
FE mesh: Max. size of elements 1 m
Load cases: Dead load (automatically generated)
Live load q = -5 kN/m^2 (whole slab), Cathegory storage area
Prestressing (automatically generated)

Procedure:

- Input slab with above dimensions and loads.
- Input prestressing with prestressing module
- Define longitudinal section (beam of width 3 m) and perform dimensioning

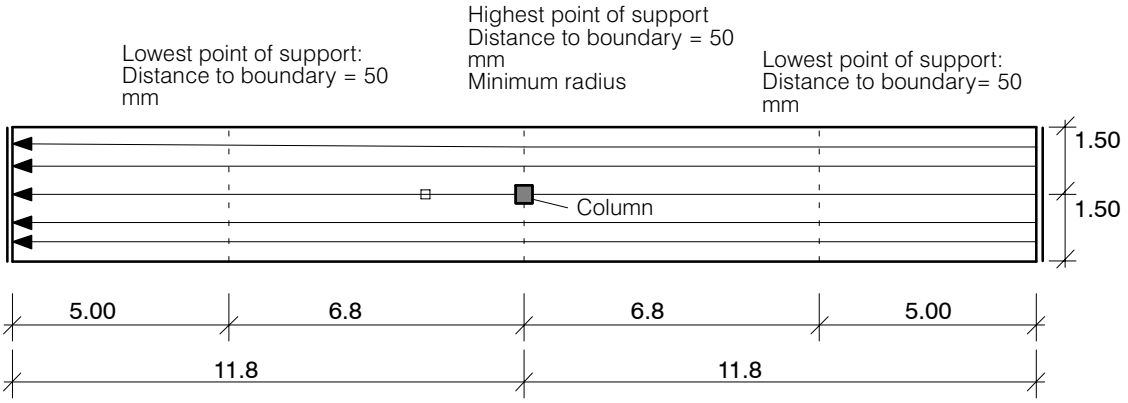


Fig. 13 *Slab outline with tendons*

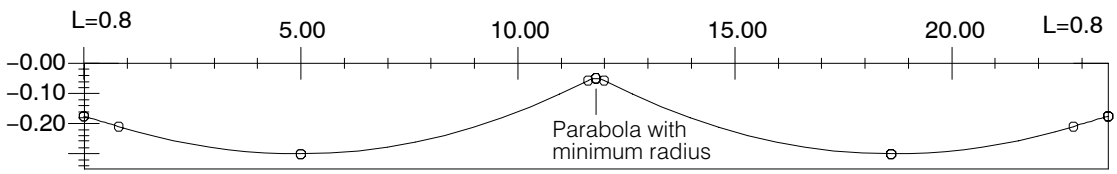
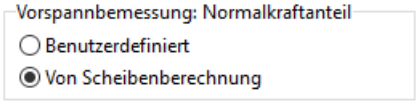


Fig. 14 *Longitudinal section with tendon profile*

☞ ***Alternative calculation: slab with normal force***
If the "Slices" option is present, the sample can be calculated as a slab with normal force. each some hints are given to these results on the following pages. For a calculation of a plate with normal force makes sense, the following setting should be made in the properties of the result section tab 'Options':



H 3.2.2 Results

Results of dimensioning for ultimate load in the field at x = 4.83

Limit state specification:

Actions

act	aut	Name	Fac
☑	☑	Dead load	1
☑	☑	Live load E Storage space	1
☑	☑	Prestressing	1

Action combinations

1	2
1.35	0.8
1.5	1.5
1	1

Beam section: Table of As:

Reinforcement cross sections, !Ultimate limit state/AP2, Beam section: S1 (0.00 , 1.50 - 23.60 , 1.50)
Factor for normal force due to prestressing= 1.00, Beam width 3.00
PT treated as resistance, constraint factor= 1.00

Distance [m]	M_d min [kNm]	M_d max [kNm]	M_p min [kNm]	M_p max [kNm]	-P*e [kNm]	A_{st} [cm²]	A_{sb} [cm²]
4.35	204.20	563.36	-103.20	-103.20	124.79	-0.000	23.949
4.83	204.69	564.71	-104.57	-104.57	127.01	-0.000	23.933
5.32	195.80	540.21	-100.72	-100.72	126.76	-0.000	22.206

Table of tendons

Tendons

Distance [m]	Id.	PT Steel	P [kN]	A_p [cm²]	ε [‰]	z_p [cm]	e [cm]	-P*e [kNm]	Angle [°]	B
4.83	G1.T2	Y1860	127	1.0	6.5	-30.0	-12.5	15.88	0.000	-
4.83	G1.T3	Y1860	127	1.0	6.5	-30.0	-12.5	15.88	0.000	-
4.83	G1.T4	Y1860	127	1.0	6.5	-30.0	-12.5	15.88	0.000	-
4.83	G1.T5	Y1860	509	4.0	6.5	-30.0	-12.5	63.51	0.000	-
4.83	G1.T1	Y1860	127	1.0	6.5	-30.0	-12.5	15.88	0.000	-
	Summe		1017	8.0				127.01		

Summary of section forces (from As-Table)

$M_d\,(g+q)$ = (1.35* 255.9 + 1.5*146.23) = 564.8 kNm
$M_p\,(P\cdot e + M_z)$ = -104.6
$P\cdot e$ = -127.0
M_z (with constraint factor = 1.0) = +22.4
$M_d'\,(g+q+M_z)$ = 587.2 kNm
(= external action)

By comparison: maximum field moment from beam theory

$$M(q) = \frac{9}{128} q\, l^2 = \frac{9}{128} \cdot 15 \cdot 11.8^2 = 146.85$$

Prestressing check (from prestressing record)

$\epsilon_{Po} \approx$ 6.52 o/oo
$\Delta\epsilon = 0.0$ (prestressing without bonding)
$P = 0.00652 \times E_s \times A_s = 1017$ kN (cf. above)
$P * e$ = 0.125 * 1017 = 127.1 kNm (cf. above)
Prestressing forces at dimensioning level: $N_p = \frac{P}{\gamma_p} = 1017$ kN

☞ *According to SIA 262, 4.2.3.3 one must apply the resistance factor of the PT steel to the yield stress and not on the E-modulus. Therefore the prestressing force must not be reduce here.*

Dimensioning (x_c and z estimated or taken from cross section program)
Concrete compression zone : $x_c \approx 62.3$ mm
Inner lever arm : $z \approx 285.5$ mm
Total required tensile force : $\frac{M'_d}{z} = 2057$ kN
Reinforcing steel: $\frac{2057 - 1017}{500/1.15} = 2392$ mm²

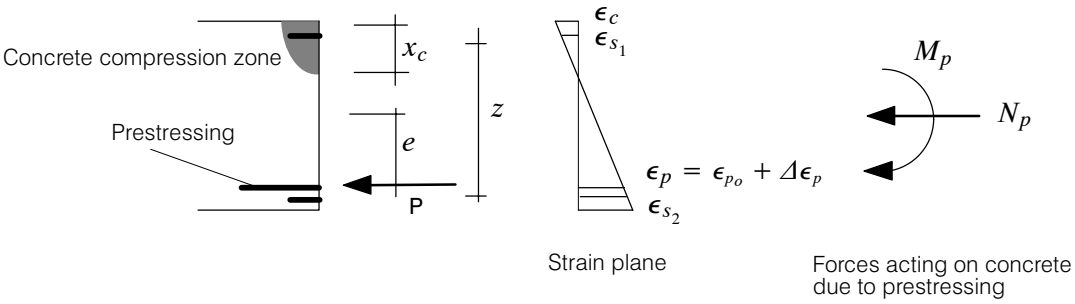


Fig. 15 *Cross section dimensioning*

By comparison: Dimensioning for ultimate load, prestressing with bonding

The difference between this and the previous dimensioning is that the additional strains in the prestressing steel are taken into account:

$\epsilon_{P_o} \approx 6.52$ ‰
$\Delta\epsilon = 4.6$ ‰ Prestressing with bonding: program supplies assumed maximum strain in reinforcing steel = 5 ‰, thus $\Delta\epsilon$ at height of prestressing ca. 4.6 o/oo)
$\epsilon_P \approx 11.1$ ‰ Yield limit exceeded

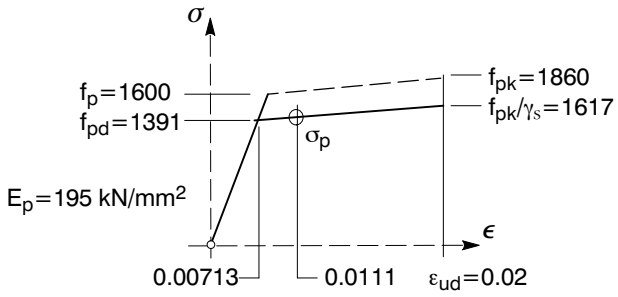


Fig. 16 *Material law for prestressing steel used in this example*

$\sigma_P \approx 1461$ N/mm² (hardening: (1617-1391)*(11.1-7.13)/(20-7.13))
$P = \sigma_P \times A_p = 1169$ kN

Dimensioning:
Concrete compression zone : $x_c \approx 63$ mm
Inner lever arm : $z \approx 284$ mm
Total required tensile force : $M_d/z = 587/0.284 = 2068$ kN
Reinforcing steel: (2068-1169)/(500/1.15) = 2068 mm²

Dimensioning for serviceability, σ_s = 200 N/mm^2
Prestressing without bonding, Results in field at x = 4.83

Limit state specification:

Standard design situation: Serviceability quasi permanent combination Analysis parameter AP1

Actions				Action combinations
act	aut	Name	Fac	1
☑	☑	Dead load	1	1
☑	☑	Live load E Storage space	1	0.8
☑	☑	Prestressing	1	1

This limit state specification was generated automatically with the analysis parameter set 'AP1' associated to it: The specification of $\sigma_{s,adm}$ can be made in the menu 'Settings>Analysis parameter' for the set 'AP1:Serviceability'.

Table of As values:

Reinforcement cross sections, !Serviceability(quasi permanent)/AP1, Beam section: S1 (0.00 , 1.50 - 23.60 , 1.50)
Factor for normal force due to prestressing= 1.00, Beam width 3.00
PT treated as resistance, constraint factor= 1.00
All tendons without bonding

Distance [m]	M_d min [kNm]	M_d max [kNm]	M_p min [kNm]	M_p max [kNm]	-P*e [kNm]	A_{st} [cm²]	A_{sb} [cm²]
4.35	255.24	371.93	-103.20	-103.20	124.79	-0.000	20.485
4.83	255.86	372.82	-104.57	-104.57	127.01	-0.000	20.369
5.32	244.76	356.64	-100.72	-100.72	126.76	-0.000	18.136

Hand calculation or calculation with cross section program (e.g. FAGUS)
Action: Md= Md_max + (Mp_max - P*e) =
256 + 0.8*146 + (-104.6 + 127) = 395.2 kN

Prestressing:
$\epsilon_{P_o} \approx$ 6.52 o/oo
$\Delta\epsilon = 0.0$ (Prestressing without bonding)
$P = 0.00652 \times E_s \times A_s$ = 1017 kN

As-dimensioning:
Concrete compression zone : $x_c \approx$ 84 mm
Inner lever arm : $z \approx$ 278 mm
Total required tensile force : M_d/z =1422 kN
Reinforcing steel: (1422-1014)*10^3/200 = 2040 mm^2

Dimensioning for serviceability, Prestressing with bonding

Prestressing:
$\epsilon_{P_o} \approx$ 6.52 o/oo
$\Delta\epsilon$ = 0.89 o/oo (addit. strain from steel stress 200 N/mm^2 at the level of the tendon)
$P = 0.00741 \times E_s \times A_s$ = 1156 kN

As-dimensioning:
Concrete compression zone : $x_c \approx$ 84.6 mm
Inner lever arm : $z \approx$ 275.7 mm
Total required tensile force : M_d/z =1433 kN
Reinforcing steel: (1433-1156)*10^3/200 = 1385 mm^2

H 3.3 Tips and Tricks

H 3.3.1 Duplication of groups

Often one needs to investigate different alternatives. For this purpose tendons and supports need to be copied from one group to the next. The following key combinations are necessary for this (in abbreviated form):

1. <Ctrl-A> (select all tendons and supports)
2. <Ctrl-C> (copy all tendons)
3. Create new group
4. <Ctrl-V> (insert all objects)

Regarding point 1:
If for example the support dialogue is open, with <Ctrl-A> only the supports (all of them) are selected.

H 3.3.2 Generating possibilities

Compared to CEDRUS-3 the number of specific tendon generating possibilities has been reduced. To compensate for this the functions of the Graphics Editor, which are identical in all Cubus Windows programs, have been considerably extended. Often one direction with the extended 'Duplicate' command can be generated almost in a single operation (in the case of an unsuccessful attempt don't forget the 'Undo' function !). For skew or irregular slab boundaries the 'Trim' function can be useful (in a first step generate tendons over the slab boundary and then select and trim all elements together which are too long).

H 3.3.3 Checks

When generating a load case all necessary checks are carried out (minimum radii, intersection of slab boundary etc).

If an incorrectly input tendon is encountered, a corresponding error message appears on the screen, the object is marked graphically and the reason for the error is given in the corresponding table.

For at least each characteristic tendon a longitudinal section should be considered (parallel tendons intersecting the same support have a similar or identical profile). Besides the x'-z profile, often, it can also be tested whether the desired profile was in fact input correctly by means of the deviation forces (="curvature diagram").

H 3.3.4 Detecting tendons lying on top of each other

See Example 1, Checks (last section)

H 3.3.5 Reference height of supports with different slab thicknesses

For the determination of the z-coordinate of a support both support end points (or the position shifted inwards by a small tolerance value) are decisive, as is shown in the example given below. If these exhibit the same height the input is admissible. For the supports in all the intermediate zones the same z-coordinate is used.

However, if, as in the case shown below, the input is made with reference to the bottom slab boundary, with "Lowest support point ... Distance to boundary ... ", strictly speaking this statement holds only for the two external zones.

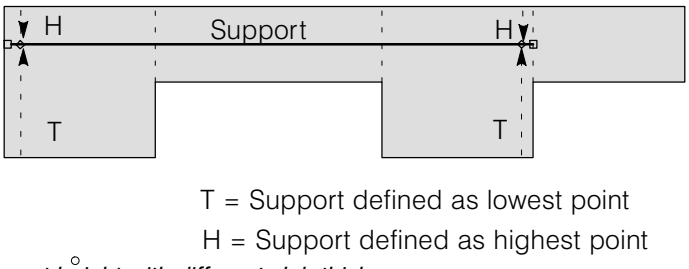


Fig. 17 *Support height with different slab thicknesses*

To enable external prestressing, no check is carried out on whether the tendons measured in the vertical direction lie within the slab. If the end point of the support lies in the area of intersection of two downstanding beams of different thickness, the greater height is used.

Vol. I Externally Bonded Reinforcement

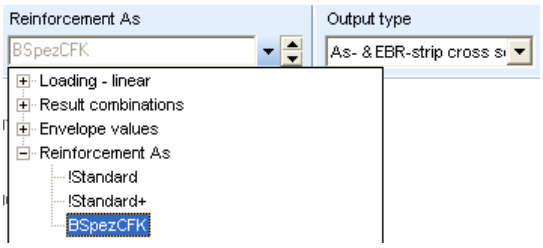
I 1 Introduction

The Externally Bonded Reinforcement (**EBR**) module is optional and designed for analyzing and dimensioning stripped bonded reinforcements based on the **design code SIA 166**. It extends the standard dimensioning procedure of CEDRUS for beam sections. Compared to the standard CEDRUS slab analysis only a few additional input data is necessary to use this module.

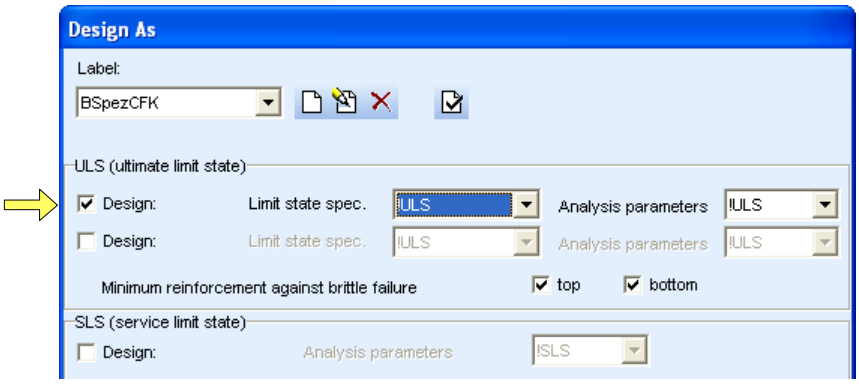
I 2 Running an EBR Analysis

I 2.1 Analysis procedure

An EBR analysis is based on a dimensioning specification (B 12.3) which has to be chosen from the results tabsheet first:



The dimensioning specification can either be a user defined one (as shown above) or a standard specification. In case of a user defined specification it is only allowed to have the first specification belonging to ULS being active (In case of standard specifications this precondition is always ensured):



The used dimensioning specification sets the limit state specification representing the loading as well as the analysis parameter set for dimensioning.

The loading is imposed in two steps:

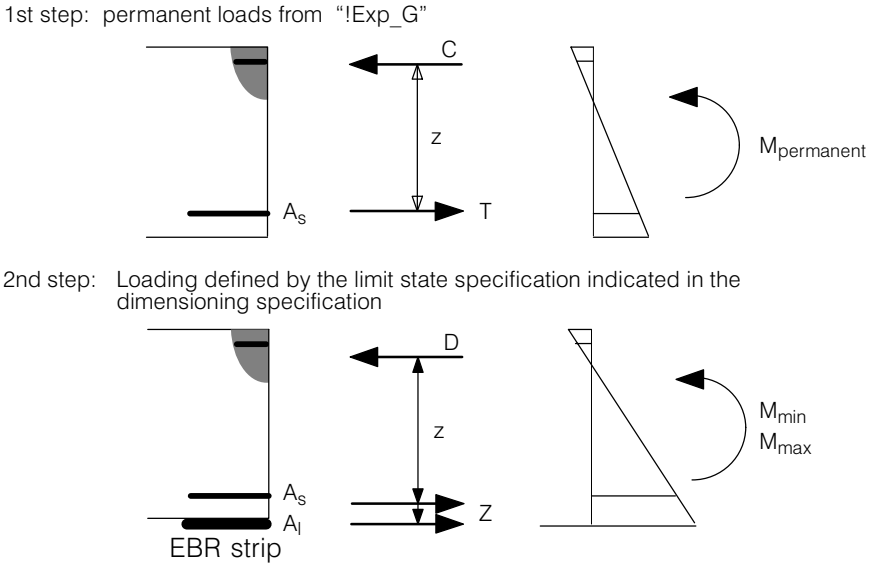
1) In the first step the slab is analyzed for permanent loads without EBR but with the existing reinforcement (see I 2.2).

   The permanent loads are defined by the automatically generated export combination !Exp-G (see B 10.4.3).

   If the existing reinforcement is insufficient to carry the permanent loads a correct bonded reinforcement analysis according to the design code is not possible (even though the structure exists and is capable of carrying the permanent loads). Nevertheless a dimensioning is carried out to obtain a point of reference for these cases already at this stage. Subsequently the strip analysis is run with the minimum required reinforcement values As that exceed the defined reinforcements.

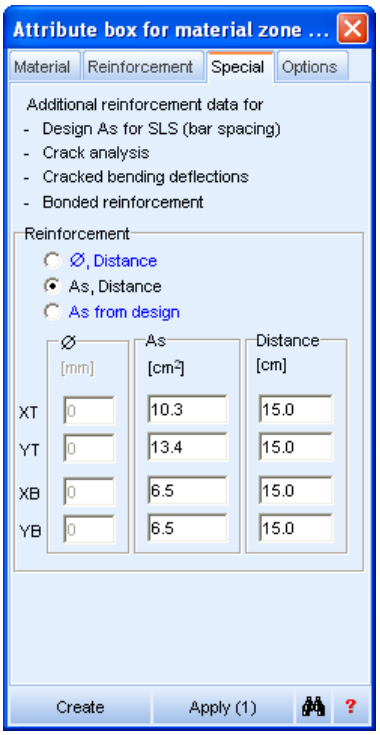
   ☞ *If the existing reinforcement is insufficient for carrying the total loads a part of the permanent loads can be moved to the variable loads.*

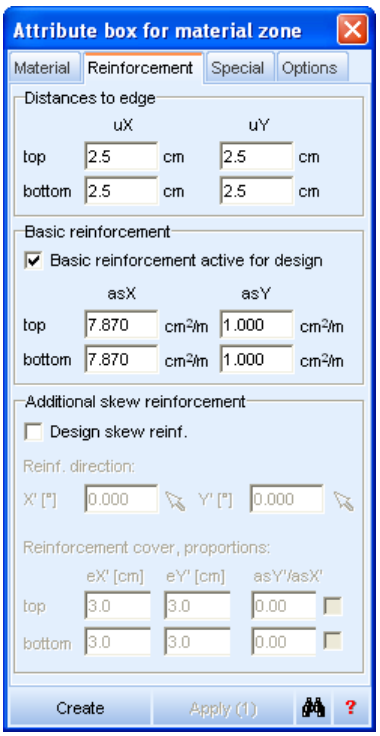
2) In the second step the EBR is activated (see I 2.3) and used to cover the total loads. The total load is specified by the chosen limit state specification of the used dimensioning specification.



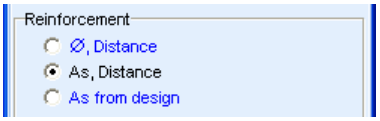
I 2.2 Specification of the reinforcement

The existing reinforcement is defined for each material zone in the related attribute box as follows:





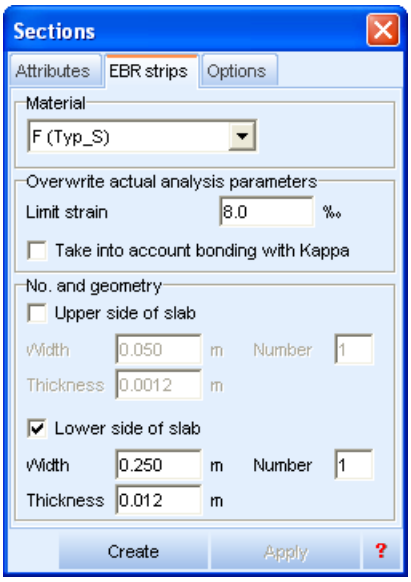
There are three ways to define the reinforcement. All methods can be found in the register 'Special'

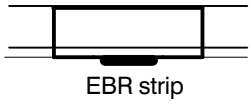


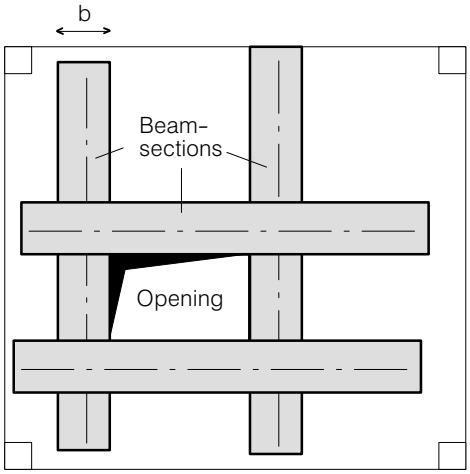
In case of the first and second method the reinforcement is defined by the diameter and bar spacing or by the reinforcement cross section directly underneath the reinforcement type section (in case of the second method the input fields for the bar spacing are active but the data content is not used). In the case 'As from design' the defined value of the basic reinforcement within the attribute box in the register 'Reinforcement' is used. The last two input methods are equal concerning a EBR analysis since identical numerical values are defined in different input fields.

I 2.3 Analysis using Beam Sections

The required verifications for EBR are carried out only in the context of a beam section analysis. The defined section width affects the level of loading which is a result of the finite element analysis for the selected strip. It has to be made sure that always both (orthogonal) directions are being checked and the required structural safeties are assured! For a complete verification the critical zones of the slab should be captured all-over by user defined strips







A new special register titled 'EBR strips' exists in the properties dialogue of sections. It contains:

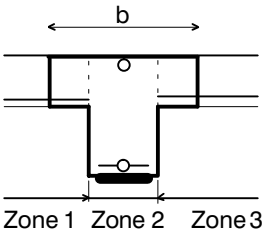
- **(EBR) Material:** Materials which are defined in the menu 'Settings > Materials' are available.
- **Limit strain:** Maximum permissible value of the design code, used as the dimensioning criterion.
- **Take into account bonding with Kappa:** If the box is checked the required strain concentration in the crack zones in respect of the design code SIA 166 is taken into account with $\kappa_s = 0.7, \kappa_l = 0.9$
- **No. and geometry:** By checking the according box a corresponding strip with the specified dimensions is defined at the upper or lower side or at both sides of the slab.
  The program always gives a result which means that if necessary a strip area at the compression side of the section is obtained even though the box was not checked (dimensioning in compression).
  For the final calculation it is recommended to choose the base reinforcement and strip sections such that no increase is necessary by the program.

☞ *To keep the computation time reasonable only the two following calculations are carried out at each results point along the beam section:*

1. $M_{permanent} \rightarrow 2.\ M_{\min}$
1. $M_{permanent} \rightarrow 2.\ M_{\max}$

In fact for the verification of the bond stresses between a strip and the concrete cross section (changes in section forces of the strip) each possible loading combination should be taken into account. In general this a very time consuming analysis and in the current version of the program only the two limit states of the section forces shown above are used.

If the verification of changes in section forces in the strip is decisive several limit state specifications should be analyzed each of them containing only one loading. This is valid especially for large point loads!



Beam section through several zones

The distances to edge of the reinforcement are taken from the zone attributes. If the dimensioning cross section passes several slab zones of different thickness and concrete cover only the reinforcement located at the outer sides is used (symbolized by a circle icon in the adjacent drawing).

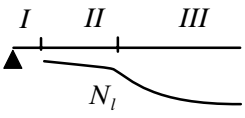
I 2.4 Checklist: Verification procedure for EBR

- Define the needed material in the material dialog ('Settings > Materials')
- Define the existing reinforcement in the attribute boxes of the material zones according to I 2.2 .
- In the first step the automatically generated export combination is used for the loading. The program collects all loadings for this combination which have the action type 'permanent' and are marked for automatic export. The mark for automatic export can either be set in the dialog of the loading attributes or in the loading list. Check: The caption of '!Exp-G' contains all these loadings!
- Create the dimensioning specification for the reinforced slab if no standard specification is being used directly. This way the limit state specification for total load is defined (in general the standard specification !GZT).
- Introduce beam sections (register 'Results') and complete the register 'EBR strips' of the properties dialogue of each beam section.
- Select a reinforcement specification in the branch 'Reinforcement As' of the results tree in the results register and 'As -&EBR-strip cross section' as output type and run the analysis by pressing the flash button.

I 2.5 Results

Table with decisive reinforcements
(= echoprint of the defined sections and dimensioning results respectively):

- Columns with reinforcement and strip sections
  Hint (see above): For a correct verification the base reinforcement as well as the strip sections should be chosen beforehand. If they are insufficient they are automatically increased by CEDRUS and marked in the table accordingly.
- Ratio x/d: This value indicates immediately whether a bonded reinforcement is useful at all.



- Zone information:
  I: State I (concrete non-cracked), anchoring zone
  II: State II (concrete cracked), zone of action
  III: State II (concrete cracked, yielding of the non-prestressed reinforcement), zone of action

Results of the bonded reinforcement analysis separated for both slab sides:

- Strains in the reinforcement
- Strains in the EBR
- Normal forces in the EBR
- Shear stresses between EBR and concrete

Comparison with the regulations of the design code SIA 166:
To satisfy the bending verification according to the design code SIA 166 the following conditions have to be fulfilled:

- Taking into account the bond coefficients (17) (18)
  The bond coefficients $\kappa_s = 0.7$ and $\kappa_l = 0.9$ are predefined in the program. If the analysis has to be done without these bond coefficients the user can set this in the section properties dialogue accordingly.
- Bending verification within unreinforced areas (20)
  Already covered by a normal CEDRUS analysis
- Verification of anchoring at the end of the action zone (21)
  The boundary between zone I and zone II is marked in the table. Moreover, the caption includes the value l_{b0d} so that it is immediately obvious whether the geometry is sufficient for the anchorage.
  l_{b0d}: *Effective anchorage length (increasing this length does not increase the anchorage resistance).*
- Tension force verification in the action zone (21)
  Is fulfilled by defining the limit strain:
  Hint: The attribute of the fiber reinforced plastic materials only contain the Young's modulus so that by defining the limit strain it has to be made sure that the ultimate strain of the strip is not exceeded.
- Verification of changes in tensile forces in the action zone (22)
  This correlates with a maximum shear stress between the EBR strip and the concrete cross section. The allowed value depends on the selected concrete class. If this value is exceeded a corresponding notification is shown in the table.
  Hint: According to the regulations of the design code the changes are the result of the difference of two cross sections. Reasonable values are therefore only obtained with a fine mesh (the distance of the cross sections is about half of the mesh size)!
- Failure of the EBR
  Can be verified with a normal CEDRUS analysis (without EBR).

Vol. J Bewehrung + Traglast

J 1 Einleitung

Die optionalen Module Bewehrung und Traglast bilden eine Erweiterung des Basismoduls von CEDRUS und sind vollständig in dessen Umgebung integriert. Mit dem Bewehrungsmodul kann die Bewehrungsanordnung in Form von Stab- und Mattenlagen bemessen und konstruiert werden. Der Traglastmodul bietet die Möglichkeit, Tragreserven einer bewehrten Platte wirklichkeitsnahe abzuschätzen.

Die im Bewehrungs- und Traglastmodul verwendeten Berechnungsmethoden entsprechen weitestgehend den modernen Normenwerken. Namentlich sind sie zu folgenden Normen kompatibel:

- SIA (Swisscode SIA 260/262)
- EC2 (Eurocode 2, ENV 1992-1-1)
- DIN (E DIN 1045-1)

In den folgenden Abschnitten wird erläutert, weshalb es Sinn macht, die Bewehrung in CEDRUS zu konstruieren, und wie eine realistische Traglastberechnung ohne grossen Modellierungsaufwand möglich ist.

Einschränkung

Fugen mit Gelenkfedern sind nicht erlaubt und müssen gegebenenfalls entfernt werden, bevor mit den Modulen Bewehrung und Traglast gearbeitet werden kann.

Kontexthilfe

Die Informationen, die als Kontexthilfe dienen und im Programm mit <F1> resp. [?] aufgerufen werden, sind in folgenden Kapiteln untergebracht:

zum Bewehrungsmodul ☞ Kapitel J 5.1

zum Dialogfenster Monitor ☞ Kapitel J 5.2

zum Traglastmodul ☞ Kapitel J 5.3

zum Datenaustausch mit CAD ☞ Kapitel J 5.4

J 1.1 Bewehrungsmodul

Je nach Grundriss erfordern die rein elastisch ermittelten Bewehrungsgehalte, wie sie der Grundmodul von CEDRUS liefert, noch eine Menge Ingenieurarbeit, bis sie in eine konkrete Bewehrungsanordnung umgesetzt sind. Dies hängt zu einem grossen Teil mit den hohen Konzentrationen über Stützen, Wandenden oder einspringenden Ecken zusammen. Zwar hat man mit den Balkenschnitten ein Hilfsmittel zum Ausgleich von Beanspruchungsspitzen. Sie sind aber mit Mehrarbeit verbunden und können das Problem nur teilweise lösen. Was zudem fehlt, ist die Behandlung schiefer Bewehrungslagen und vor allem eine automatische Schlusskontrolle, ob ein Bewehrungslayout in allen Punkten genügt. Der Bewehrungsmodul kann diese Probleme beheben und den Weg von den FE-Resultaten zum Bewehrungsplan wesentlich verkürzen helfen. Eine zusätzliche Effizienzsteigerung kann insbesondere mit den von CEDRUS unterstützen Bewehrungsschnittstellen erzielt werden, womit sich die konstruierte Bewehrung direkt in ein CAD-System zwecks Planerstellung übertragen lässt.

Der Bewehrungsmodul zeichnet sich durch folgende Eigenschaften aus:

- konsistente und statisch 'saubere' Lösung (Gleichgewicht)
- Unterstützung von mehreren Lastkombinationen
- grosse Freiheit bei der Bewehrungsführung (->Einbezug der Ingenieurerfahrung)
- rationelle Arbeitsweise durch CAD-Integration
- schiefe Bewehrungsanordnung
- geringe Bewehrungsgehalte (plastische Optimierung)
- kompatibel zu modernen Normenwerken (z.B. SIA, EC2, DIN)

Die Grundlagen und Eigenschaften des Bewehrungsmoduls und dessen Einsatz für praktische Bemessungsaufgaben werden in den Kapiteln J 2 und J 3 näher erläutert.

J 1.2 Traglastmodul

Immer häufiger sehen sich Ingenieure mit der Aufgabe konfrontiert, die Tragfähigkeit bestehender Strukturen bestimmen zu müssen, wobei dies mit einer elastischen Berechnung nur in den seltensten Fällen gelingt. Mit dem Traglastmodul von CEDRUS kann das inelastische (plastische) Materialverhalten von Stahlbeton berücksichtigt werden, womit sich Tragreserven realistisch abschätzen lassen. Ausgehend von einer Bewehrungsanordnung kann das elastische und plastische Strukturverhalten für eine vorgegebene Belastungsanordnung analysiert werden. Im Gegensatz zur Gelenklinientheorie, welche nur obere Grenzwerte für die Traglast angeben kann, liefert dieses Modul selbst für komplexe Strukturen realistische Werte.

Das Traglastmodul zeichnet sich durch folgende Eigenschaften aus:

- realistische Abschätzung von Tragreserven
- einfache Bedienung
- interaktive Verfolgung des Strukturverhaltens
- Abschätzung des erforderlichen Verformungsvermögens
- schiefe Bewehrungsanordnung

Die Grundlagen und Eigenschaften des Traglastmoduls werden in den Kapiteln J 2 und J 4 näher erläutert.

☞ *Es wird jedem Anwender dringend empfohlen, für den Einstieg in die Bewehrungs- und Traglastmodule die folgenden Kapitel vollständig durchzuarbeiten. Vorher sollte man sich nicht an ernsthafte Berechnungen wagen.*

J 2 Grundlagen

In diesem Kapitel werden die Modelle und Lösungsverfahren erläutert, welche sowohl für den Bewehrungs- wie den Traglastmodul von CEDRUS verwendet werden.

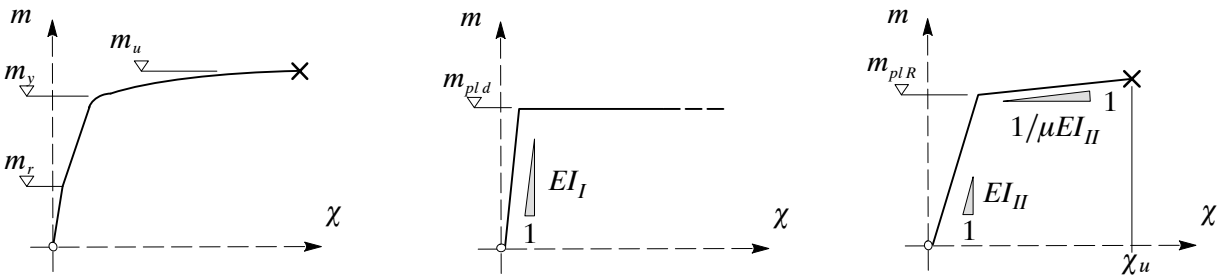
J 2.1 Materialmodell

Sowohl beim Bewehrungs- wie beim Traglastmodul wird das nichtlineare Strukturverhalten von Stahlbetonplatten berücksichtigt. Unter der realistischen Annahme, dass die Verformungen auch im plastifizierten Zustand klein bleiben und Effekte 2. Ordnung keine Rolle spielen, rührt diese Nichtlinearität allein vom elastoplastischen Materialverhalten her. Durch die Begrenzung der Bewehrungsgehalte auf Werte, welche ein sprödes Versagen verhindern, gelingt es, das Materialgesetz einer auf Biegung beanspruchten Platte durch eine elastoplastische Momenten-Krümmungsbeziehung im Querschnitt einheitlicher Breite zu beschreiben.

J 2.1.1 Momenten-Krümmungsdiagramm

Wie folgende Figur zeigt, wird für jeden der beiden Berechnungsmodule eine spezielle Idealisierung des wirklichen Materialverhaltens verwendet:

Bewehrungsmodul (Bemessung) **Traglastmodul** (Analyse)



a) aus Versuchen b) elastisch—plastisch c) elastisch—plastisch mit Verfestigung

Die Diagramme (b) und (c) weisen beide eine anfangs lineare Charakteristik auf, welche nach dem Erreichen der Fliessgrenze m_{pl} in ein ausgeprägtes Fliessplateau mündet. Die Modellierung als linear elastisch-starr plastisches Material (b) mit initialer, ungerissener Steifigkeit (Zustand I) stellt für eine Bewehrungsbemessung ein ausreichend genaues Materialmodell dar, welches die Anwendung der Grenzwertsätze der Plastizitätstheorie erlaubt (s. J 2.1.2). Um eine Abschätzung der erforderlichen, plastischen Verformungen zu ermöglichen, wird für die Traglastanalyse ein erweitertes Modell (c) mit gerissener Steifigkeit (Zustand II) und Verfestigung verwendet. Mit dieser Idealisierung gelingt es, das wirkliche Verhalten (a) genügend genau zu erfassen. Eine weitere Verfeinerung des Modelles vermag die Genauigkeit trotz wesentlich höherem Modellierung- und Rechnungsaufwand kaum entscheidend zu verbessern, da sehr viele Unsicherheiten bei den Modellparametern (Materialkennwerte, Einflüsse Kriechen und Schwinden, Belastungsgeschichte etc.) bestehen.

Die Berechnung der charakteristischen Werte der Diagrammen (b) und (c) wird in den Kapiteln J 3 und J 4 dieses Handbuchs näher erläutert. Dabei gilt es zu beachten, dass Langzeiteffekte nicht berücksichtigt werden.

J 2.1.2 Plastizitätstheorie

Die Plastizitätstheorie versucht das mechanische Materialverhalten durch vereinfachte, nichtlineare Stoffgesetze zu idealisieren, welche Beziehungen zwischen Kraft- und Verformungsgrössen aufstellen, womit sie als Sonderfall einer allgemeinen nichtlinearen Theorie betrachtet werden kann. Sie setzt ausreichendes Verformungsvermögen voraus (s. Abschnitt J 3.1.3 und Kapitel J 4). Ihre Anwendbarkeit auf Stahlbetonplatten ist durch zahlreiche Versuche und den jahrzehntelangen, erfolgreichen Einsatz in der Praxis (z.B. Streifenmethode) belegt. In den modernen Normenwerken wird sie zur Berechnung von Stahlbetonplatten ausdrücklich zugelassen.

Bei plastischen Materialmodellen (s. z.B. Diagramm (b) in J 2.1.1) wächst die Verformung bei steigender Belastung zunächst proportional an bis eine Grenzlast (=Fliessgrenze) erreicht wird. Danach steigen die Verformungen ohne zusätzliche Lasterhöhung, was als plastisches Fliessen bezeichnet wird. Durch diese Vereinfachung des wirklichen Verhaltens beschränkt man sich auf diejenigen Grössen, welche im wesentlichen die Traglast bestimmen, womit einfache Bemessungsregeln gewonnen werden können. Die Plastizitätstheorie postuliert u.a. die folgenden zwei Sätze:

- Statischer (od. unterer) Grenzwertsatz:

  Jede Belastung, zu der sich ein beliebiger Gleichgewichtszustand S angeben lässt, der an keiner Stelle die Fliessgrenze R überschreitet, liegt nicht höher als die Traglast.

  $$S \leq R$$

  Unter einem Gleichgewichtszustand wird dabei ein Spannungszustand verstanden, der überall das Gleichgewicht und die statischen Randbedingungen erfüllt. Der elastische Spannungszustand S_{el} stellt z.B. einen Gleichgewichtszustand dar. Sind mehrere Belastungen zu untersuchen, so kann die obenstehende Ungleichung mit den entsprechenden Grenzwerten formuliert werden:

  $$S^{GW} \leq R$$

- Einspielsatz (nach Melan-Bleich):

  Gegebene Grenzwerte der Belastungsintensität liegen innerhalb des Anpassungsvermögens einer Struktur, wenn die Grenzwerte der elastischen Spannungen S_{el}^{GW} jeder möglichen Belastung, superponiert mit irgendeinem Eigenspannungszustand S_0, nirgends die Fliessgrenze R überschreiten.

  $$S_{el}^{GW} + S_0 \leq R$$

  Als Eigenspannungszustand (oder Zwängungsspannungszustand) S_0 wird dabei ein Spannungszustand bezeichnet, der mit sich selbst (d.h. ohne äussere Lasten) im Gleichgewicht steht. Ein Beispiel für S_0 ist ein Spannungszustand infolge Temperatureinwirkung.

Die Anwendung dieser Theoreme auf praktische Bemessungsaufgaben wird im Kapitel J 3 näher erläutert.

J 2.1.3 Fliessbedingung und Fliessgesetz

Der Spannungszustand einer auf Biegung beanspruchten Platte (Kirchhoff'sche Theorie) wird durch die drei Komponenten m_x, m_y, m_{xy} beschrieben, welche mit den drei Komponenten $\chi_x, \chi_y, \chi_{xy}$ des Verzerrungszustandes assoziert sind. Beim dabei verwendeten, orthogonalen x-y-Koordinatensystem handelt es sich um das lokale System des betrachteten Punktes innerhalb der Platte. Dieses ist in CEDRUS durch die Ausgaberichtung der entsprechenden Materialzone definiert (=Winkel im Gegenuhrzeigersinn ab globaler x-Achse) .

Mit den bekannten Plastizitätsbedingungen für Stahlbetonplatten (s. z.B. EC2 A 2.8) kann die Fliessgrenze (s. Momenten-Krümmungsdiagramme J 2.1.1) in jedem Punkt der Platte anhand von zwei Querschnitten mit je zwei positiven plastischen Widerstandsmomenten m_{pl} angegeben werden:

- Querschnitt in x-Richtung, untere Bewehrung:

$$m_{udx} = m_x \pm m_{xy} \leq m_{plx}$$

- Querschnitt in y-Richtung, untere Bewehrung:

$$m_{udy} = m_y \pm m_{xy} \leq m_{ply}$$

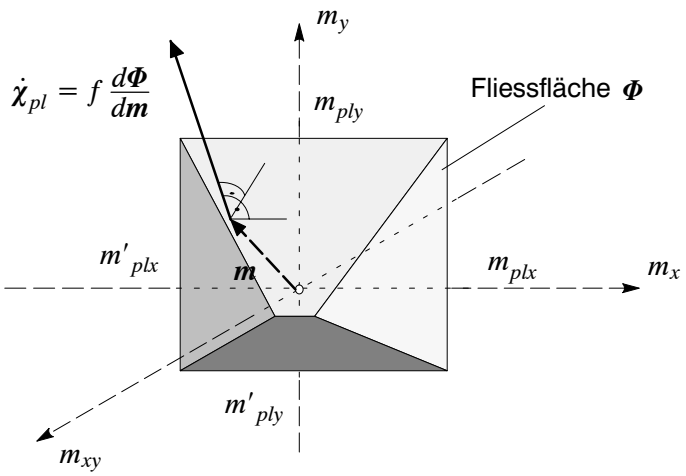
- Querschnitt in x-Richtung, obere Bewehrung:

$$m'_{udx} = -m_x \pm m_{xy} \leq m'_{plx}$$

- Querschnitt in y-Richtung, obere Bewehrung:

$$m'_{udy} = -m_y \pm m_{xy} \leq m'_{ply}$$

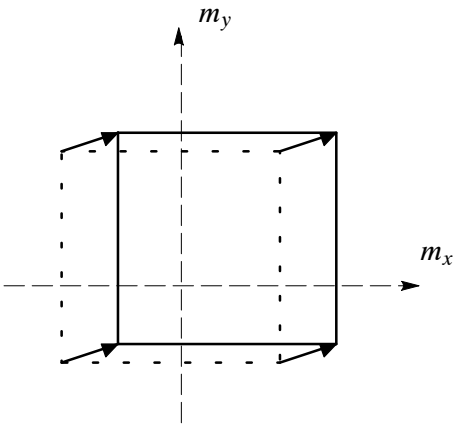
Geometrisch betrachtet bilden die obenstehenden Fliessbedingungen einen durch die konvexe Hülle Φ (=Fliessfläche) begrenzten Körper (=elastischer Bereich) im dreidimensionalen Raum der Momente $\boldsymbol{m} = \langle m_x, m_y, m_{xy} \rangle$:



Wird bei steigender Beanspruchung die Fliessfläche in einem Punkt erreicht, so treten dort plastische Krümmungen $\dot{\chi}_{pl} = (\dot{\chi}_x, \dot{\chi}_y, \dot{\chi}_{xy})$ senkrecht auf Φ auf, was als assoziertes Fliessgesetz bezeichnet wird. Damit kann im Rahmen einer Traglastberechnung die Intensität und Richtung der plastischen Krümmungen berechnet werden (s. Kapitel J 4).

Kinematisches Verfestigungsmodell

Bei Versuchen wird beobachtet, dass sich die Fliessfläche nach dem Eintreten plastischer Verformungen verändert. So nehmen z.B. Querschnitte, welche ins Fliessen geraten sind, mit zunehmenden plastischen Krümmungen mehr Momente auf (s. Diagramm (c) in J 2.1.1), was als Verfestigung bezeichnet wird. Im mehrdimensionalen Spannungsraum kann man zeigen, dass sich der elastische Bereich (=Inhalt von Φ) dabei oft nicht vergrössert. Dieser sog. Bauschinger Effekt wird im Traglastmodul durch eine kinematische Verfestigung modelliert, welche einer Verschiebung von Φ im Raum der Momente $\boldsymbol{m}$ entspricht (s. Skizze unten).

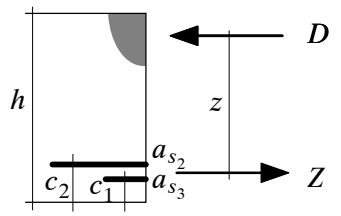


J 2.1.4 Plastisches Widerstandsmoment

Das plastische Widerstandsmoment m_{pl} eines Querschnittes kann aufgrund von Plattendicke h, Betonqualität, Stahlqualität und vertikaler Lage der Biegebewehrung bestimmt werden (s. nachstehende Figur). Dabei werden die in den Normen verankerten Spannungs-Dehnungslinien des Betons und des Betonstahls verwendet. Bedingt durch die unterschiedliche Problemstellung werden für die beiden Berechnungsmodule verschiedene Methoden verwendet.

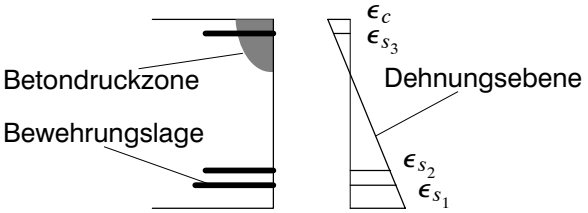
Wie in Kapitel J 3 erläutert, ist für die Bemessung eine lineare Abhängigkeit zwischen plastischem Widerstandsmoment und den unbekannten Bewehrungsflächen a_{s_i} erforderlich. Durch die Annahme eines von a_{s_i} unabhängigen, mittleren Hebelarmes z gelingt es, ein Modell (1) zu formulieren, welches eine ausreichende Genauigkeit bietet. Bei dieser vereinfachten Formulierung nimmt man an, dass die Bewehrung nur auf Zug wirkt und alle Lagen ins Fliessen geraten.

Modell 1: **Bewehrungsmodul**
Berechnung mit mittlerem Hebelarm



$$Z = \sum a_{s_i} f_{y_i} \qquad \Rightarrow m_{pl} = Z\ z$$

Modell 2: **Traglastmodul**
Berechnung mit Dehnungsebene



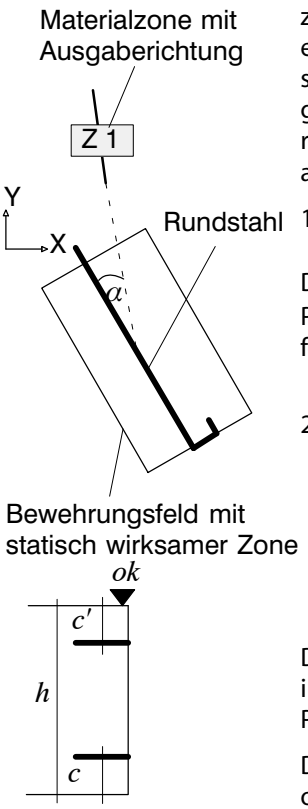
$$\Rightarrow m_{pl} = f(\epsilon)$$

Bei einer Traglastanalyse sind die Bewehrungsflächen jedoch bekannt und das plastische Widerstandsmoment kann mittels Variation der Dehnungsebene iterativ berechnet werden. Dabei werden die Spannungen im Beton und Stahl aufgrund der Dehnungen bestimmt und das plastische Moment durch Integration derselben ermittelt (Modell 2).

J 2.2 Bewehrungsmodell

Neben den Struktur- und Lastdaten des Basismoduls basieren sowohl der Bewehrungs- wie der Traglastmodul auf zusätzlichen Daten, welche die eingelegte **Biegebewehrung** beschreiben.

J 2.2.1 Bewehrungsfelder



Die obere und untere Bewehrung wird getrennt in Form von **Bewehrungsfeldern** spezifiziert, welche einzelnen Rundstahl- und Mattenlagen entsprechen. Diese Felder weisen einen konstanten Widerstand auf und decken einen Bereich der Struktur ab, welcher ihrer statisch wirksamen Zone (d.h. ohne Verankerung) entspricht. Sie werden als beliebige Polygone oder Parallelogramme in der Plattenebene definiert, besitzen eine freie Bewehrungsrichtung und dürfen sich gegenseitig überlappen. Ihre vertikale Lage kann auf zwei Arten angegeben werden:

1. Mittels Eingabe von vier Parametern

Durch die Eingabe von Attributen des sie enthaltenden Betonkörpers (Kote Oberkante ok, Plattendicke h und Randabstände oben c' und unten c) ist die vertikale Lage eines Feldes festgelegt (s. auch nebenstehende Figur).

2. Mittels Zuweisung zu einer Materialzone

Da jede Materialzone (= Ausgabezone) einen Betonkörper inkl. oberem und unterem Randabstand beschreibt, ist die vertikale Position eines Bewehrungsfeldes durch Zuweisung zu einer Zone eindeutig festgelegt. Der Winkel α zwischen Stabrichtung des Feldes und Ausgaberichtung der Materialzone entscheidet, ob der Randabstand in Ausgaberichtung (bei $|\alpha| \leq 45°$) oder derjenige senkrecht dazu übernommen wird.

Die zweite Methode erfordert weniger Aufwand bei der Eingabe der Bewehrung und bietet insbesondere bei Bemessungsaufgaben den Vorteil, dass nach Projektänderungen (z.B. Plattendicke) die Bewehrungsfelder automatisch neu positioniert werden.

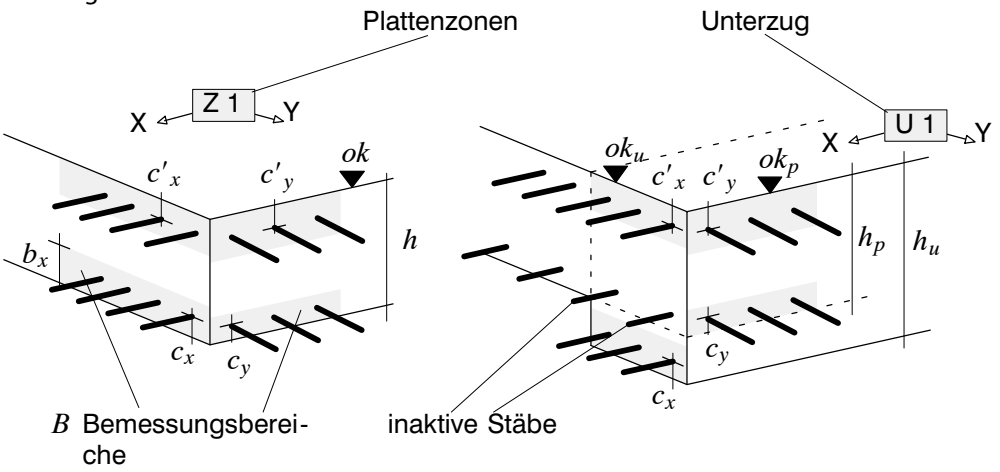
Der verwendete Bewehrungsstahl kann ebenfalls automatisch von der Zone übernommen oder individuell eingestellt werden (s. Register 'Optionen' > 'Stahl' im Dialog 'Bewehrungsfeld').

☞ *Der Randabstand c entspricht dem Abstand des Schwerpunktes eines Bewehrungsstabes vom entsprechenden Betonrand, wobei bei zugeordneter Materialzone für obere Bewehrungslagen der obere und für untere Lagen der untere Randabstand massgebend wird.*

J 2.2.2 Platten- und Unterzugsbewehrung

Aktive Bewehrungslagen

Bei der Bemessung der Biegebewehrung werden zur Berechnung des plastischen Widerstandsmomentes m_{pl} nur diejenigen Stablagen berücksichtigt, welche innerhalb der sog. Bemessungsbereiche B liegen (s. graue Flächen in nachstehender Figur). Jede Plattenzone weist vier dieser Randbereiche auf (in x und y Richtung je oben und unten), deren Höhe b dem zweifachen Randabstand c der Zone entspricht. Ist $c = 0$, so wird b zu 10% der Plattendicke angenommen. Stäbe welche ausserhalb dieser Zone liegen sind inaktiv, wobei dies nicht für die Traglastanalyse zutrifft, wo immer alle Stablagen berücksichtigt werden.



Die obenstehende Figur zeigt eine Plattenzone Z1 (Oberkante ok, Dicke h) und einen Unterzug U1 (Oberkante ok_u, Dicke h_u) mit benachbarter Platte (Oberkante ok_p, Dicke h_p) und den entsprechenden Randabständen c_x, c_y, c'_x, c'_y.

Unterzugsbewehrung

In CEDRUS werden die Unterzüge als orthotrope Plattenzonen modelliert (s. CEDRUS Handbuch Kapitel B 8.7 Materialeigenschaften). Diese bilden eine eigene Materialzone mit vorgegebener Ausgaberichtung, welche der Unterzugsachse entspricht. Sie weisen längs eine erhöhte Steifigkeit auf und wirken quer zur Achse wie die angrenzende Platte. Die Bemessung von Unterzügen erfolgt analog dazu: In Längsachse wird die Bewehrung im Unterzugquerschnitt und quer dazu im Plattenquerschnitt bemessen. Wie die Plattenzonen werden auch die Unterzüge **nur für die Biegebewehrung** bemessen, d.h. die Bügelbewehrung und das entsprechende Versatzmass für die Längsbewehrung müssen nachträglich von Hand bestimmt werden.

✳ **Astuce:** *Wird die untere und obere Plattenbewehrung durch den Unterzug hindurch gelegt, so muss für den Unterzug nur eine untere Längsbewehrung definiert werden. Voraussetzung dafür ist, dass die Angaben zur benachbarten Platte im Unterzugsdialog mit den Daten der Nachbarzonen übereinstimmen. Sinngemäss gilt dies auch für Überzüge, wo nur eine obere Längsbewehrung eingegeben werden muss.*

J 2.2.3 Schiefe Bewehrung

In den Fliessbedingung (J 2.1.3) wird angenommen, dass im betrachteten Punkt eine orthogonale Bewehrung in Richtung des lokalen Koordinatensystems eingelegt ist und die Widerstandsmomente gemäss (J 2.1.4) bestimmt werden können. Dies trifft auf die meisten praktischen Fälle zu, vorausgesetzt der Benutzer stellt sicher, dass das lokale Koordinatensystem (d.h. Ausgaberichtung der Materialzone) mit den Bewehrungsrichtungen übereinstimmt. Ist die Bewehrungsanordnung jedoch nicht orthogonal, so muss die Fliessbedingung erweitert werden.

Dies gelingt durch eine verfeinerte Berechnung der plastischen Momente. Mittels Gleichgewichtsbedingungen lassen sich aus allen vorhandenen Bewehrungslagen in Abhängigkeit von ihrer Abweichung β_i zur lokalen x-Achse sechs Widerstandsmomente bestimmen, wobei jedes untere und obere Bewehrungsfeld ein Widerstandsmoment von m_{pl_i} bzw. m'_{pl_i} aufweist:

$$m_{plX} = \sum m_{pl_i} \cos\beta_i^2 \qquad m'_{plX} = \sum m'_{pl_i} \cos\beta_i^2$$

$$m_{plY} = \sum m_{pl_i} \sin\beta_i^2 \qquad m'_{plY} = \sum m'_{pl_i} \sin\beta_i^2$$

$$m_{plXY} = \sum m_{pl_i} \sin\beta_i \cos\beta_i \qquad m'_{plXY} = \sum m'_{pl_i} \sin\beta_i \cos\beta_i$$

Daraus können anschliessend die äquivalenten Widerstandsmomente bestimmt und in den Fliessbedingungen (J 2.1.3) eingesetzt werden.

$$m_{plx} = m_{plX} \pm m_{plXY} \qquad m_{ply} = m_{plY} \pm m_{plXY}$$

$$m'_{plx} = m'_{plX} \pm m'_{plXY} \qquad m'_{ply} = m'_{plY} \pm m'_{plXY}$$

Diese Widerstandstransformation ermöglicht die Behandlung beliebig orientierter Bewehrungsfelder. Es gilt dabei jedoch zu beachten, dass die Resultate nicht richtungsinvariant sind, d.h. sie hängen von der Orientierung des lokalen x-y-Koordinatensystems ab. Der Tragwiderstand schief gestellter Bewehrungsfelder kann zudem nur sehr schlecht genutzt werden, weshalb eine schiefe Bewehrung wenn immer möglich verhindert werden sollte.

Für eine detaillierte Herleitung dieses Verfahrens sei hier auf folgende Literatur verwiesen:

- R. Wolfensberger: "Traglast und optimale Bemessung von Platten", Institut für Baustatik und Konstruktion, ETH Zürich, Dissertation Nr. 3451, 1964.

☞ *Um gute Resultate zu erzielen gilt es zu beachten, dass sich die Richtungen einer orthogonalen Bewehrung mit dem Koordinatensystem (d.h. Ausgaberichtung der Materialzone) decken und bei nichtorthogonalen Bewehrungslagen eine Achse des Koordinatensystems mit der Hauptbewehrungsrichtung übereinstimmt, damit die entsprechende Lage voll ausgenutzt werden kann.*

J 2.3 Lösungsverfahren

Die im Bewehrungs- und Traglastmodul verwendeten Rechenverfahren basieren auf dem FE-Modell (Elementeinteilung, Knoten, Freiheitsgrade, Lagerungsbedingungen) des Basismoduls, wobei anstelle des hybriden Elementmodelles (s. CEDRUS Handbuch Kapitel B 2: Grundlagen) ein kinematisches Modell nach der sog. *Free-Formulation* Theorie verwendet wird. Für ein eingehendes Studium dieses Modelles sei auf folgende Literatur verwiesen:

- P. G. Bergan, M.K. Nygard: "Finite Elements with Increased Freedom in Choosing Shape Functions", Int. Journal for Num. Meth. in Eng., Vol. 20,1985.

Implementierung des Materialgesetzes

In den meisten FE-Programmen wird das Materialgesetz für die Spannungen in den sog. Integrationspunkten aller Elemente formuliert. Im Gegensatz dazu ist das Materialverhalten beim Bewehrungs- und Traglastmodul von CEDRUS für jedes Element im Raum seiner Elementknotenkräfte beschrieben, was insbesondere für die Bewehrungsbemessung grosse Vorteile bringt. Für ein eingehendes Studium dieser Formulierung sei auf nachstehende Literatur verwiesen:

- P.N. Steffen: "Elastoplastische Dimensionierung von Stahlbetonplatten mittels Finiter Bemessungselemente und Linearer Optimierung", Institut für Baustatik und Konstruktion, ETH Zürich, Bericht Nr. 220, 1996 Birkhäuser Verlag Basel.
- E. Anderheggen, G. Glanzer, P.N. Steffen: "Yield Surfaces in the Element Nodal Force Space", Computers & Structures, 1999.

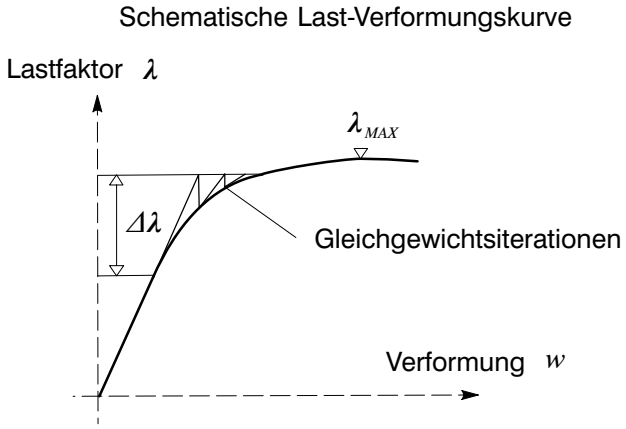
Zur Bestimmung des Widerstandes innerhalb eines Finiten Elementes werden alle Bewehrungsfelder berücksichtigt, welche mehr als die Hälfte des Elementes bedecken.

J 2.3.1 Nichtlineare Analyse

Im Traglastmodul kann das elastische und plastische Verhalten einer Platte mit bekannter Bewehrungsanordnung für eine vorgegebene Belastung analysiert werden.

Inkrementelle, iterative Berechnung

Die Traglastberechnung erfolgt mittels des inkrementellen, interativen Verfahrens nach Newton-Raphson. Ausgehend von einem vorgegebenen Zustand (d.h. Spannungs- und Verformungszustand, initiale Steifigkeit) wird die Belastung in Inkrementen sukzessive erhöht. Dabei werden die Momente in der Platte verfolgt und plastische Krümmungen in Zonen mit erschöpftem plastischem Widerstand eingeführt (s. J 2.1.3).



Um den zugehörigen Punkt auf der Last-Verformungskurve zu bestimmen, sind innerhalb eines Lastinkrementes $\Delta\lambda$ Gleichgewichtsiterationen notwendig (s. obenstehende Figur). Ist dies nicht möglich, so liegt die aufgebrachte Last oberhalb der Traglast und $\Delta\lambda$ muss verkleinert werden. Dieser inkrementelle, iterative Prozess wird beendet, sobald das Lastinkrement sehr klein wird, womit der Traglastfaktor λ_{MAX} erreicht wurde. In diesem Moment verliert die Stahlbetonplatte ihre Stabilität und es bildet sich ein Kollapsmechanismus aus, welcher die Struktur schliesslich zum Einsturz bringt. Ein Mechanismus kann sich jedoch nur ausbilden, falls die Modellannahmen (s. J 2.1) zutreffen. Um die Einhaltung dieser Annahmen sicherzustellen werden nach jedem Lastinkrement zusätzliche sog. Abbruchkriterien (s. J 4.1.2) kontrolliert, um die Berechnung allenfalls frühzeitig abbrechen zu können.

Einfluss von Ausgangszustand und Lastgeschichte

Der Traglastfaktor für eine gegebene Belastung (bei Vernachlässigung der Verfestigung und unter Annahme eines ausreichenden Rotationensvermögen) ist theoretisch nur von den Betonabmessungen, den Lagerungsbedingungen und der Bewehrung der Platte abhängig. Damit üben die Steifigkeitsverhältnisse, die Reihenfolge der verschiedenen Einwirkungen und evtl. vorhandene Zwängungen keinen Einfluss auf die Traglast aus, weshalb diese Faktoren beim Tragsicherheitnachweis gemäss den modernen Normen vernachlässigt werden dürfen.

Da das Rotationensvermögen von Stahlbetonplatten jedoch beschränkt ist, müssen bei einer Traglastberechnung auch die Verformungen zwecks Begrenzung der Rotationen bestimmt werden (s. J 4.1.2 Rotationsvermögen). Diese hängen jedoch stark vom initialen Spannungs- und Verformungszustand und der Intensität und Reihenfolge der verschiedenen Einwirkungen (=Lastgeschichte) ab.

Der initiale Zustand einer Stahlbetonplatte kann aber nie genau angegeben werden, da er durch viele unbekannte Faktoren (z.B. Betonqualität, Rissbildung, Umwelteinflüsse, Schwinden etc.) bedingt ist. Bei einer Traglastanalyse interessieren jedoch nicht die effektiv auftretenden Verformungen, sondern nur maximale Werte zur Kontrolle des Verformungsvermögen. Eine brauchbare Abschätzung für diesen Grenzwert gelingt, indem als Ausgangszustand die unverformte, spannungslose Konfiguration mit gerissener Anfangssteifigkeit angenommen wird (s. J 2.1.1).

Auch die genaue Intensität und Reihenfolge der Einwirkungen ist bei Bauwerken nicht bekannt. CEDRUS erlaubt die näherungsweise Berücksichtigung des Einflusses der Lastgeschichte, indem die Belastung als Serie von unterschiedlichen Laststufen spezifiziert werden kann (s. J 4.1.1).

J 3 Bewehrungsmodul

Der Bewehrungsmodul ist ein umfangreiches Werkzeug zur Bemessung und Konstruktion der Bewehrung einer Platte. In diesem Kapitel werden zuerst die Grundkonzepte dieses Moduls erläutert und danach die Bewehrungsbemessung anhand eines konkreten Beispiels aufgezeigt.

J 3.1 Grundkonzepte

J 3.1.1 Bemessung auf Tragsicherheit

Gemäss den modernen Normenwerken gilt die Tragfähigkeit eines Querschnittes als nachgewiesen, falls

$$S_d \leq R_d$$

wobei unter S_d der Bemessungswert der Schnittgrösse und unter R_d der zugehörige Bemessungswert des Widerstandes verstanden wird.

Bemessungswerte für Widerstände

Bei Stahlbetonplatten unter reiner Biegebeanspruchung entspricht R_d dem plastischen Widerstandsmoment $m_{pl\,d}$ (s. Figur in J 2.1.1), welches aus den mit Teilsicherheitsfaktoren (γ_c ,γ_s für Beton und Betonstahl) reduzierten, charakteristischen Baustoffkennwerten (f_{ck},f_{yk} für Beton und Betonstahl) ermittelt wird. CEDRUS verwendet beim Bewehrungsmodul immer diejenigen Teilsicherheitsbeiwerte γ_c und γ_s, welche unter den **Analyseparametern '!GZT (Tragsicherheit)'** gerade eingestellt sind (s. Hauptmenu 'Einstellungen>Analyseparameter'). Der Baustoffkennwert von Beton wird vom in der aktuellen Materialzone eingestellten Körper-Baustoff (s. Dialog Matrialbox) übernommen, wobei f_{ck} aus der diesem Baustoff zugewiesenen Baustoffklasse stammt. Die Zuweisung von Baustoffen zu Baustoffklassen kann der Benutzer im Baustoffdialog (Hauptmenu 'Einstellungen>Baustoffe') beeinflussen.

$$R_d = m_{pl\,d} = \frac{1}{\gamma_s} \sum a_{s_i} f_{yk_i} z$$

Entsprechend J 2.1.4 wird das Widerstandsmoment $m_{pl\,d}$ aus den Bewehrungsflächen a_{s_i} und Festigkeiten f_{yk_i} aller aktiven Bewehrungslagen bestimmt (s.J 2.2.2). Der innere Hebelarme wird innerhalb eines Finiten Elementes oben und unten in x- und y-Richtung näherungsweise mit $z = 0.9\,(h - c)$ angenommen (s. Figuren in J 2.1.4und J 2.2.2), wobei für c der Randabstand der entsprechenden Ausgabezone verwendet wird. Für schiefe Bewehrungslagen werden die Formeln aus J 2.2.3 verwendet.

Bemessungswerte für Schnittgrössen

Die Normenwerke verlagen, dass als Bemessungswerte für Schnittgrössen die Bemessungsgrenzwerte S^{GW} aller untersuchten Gefährdungsbilder verwendet werden, welche mit dem Basismodul automatisch generiert werden können (s. CEDRUS Handbuch: Kapitel 2.3.):

$$S_d = GW(\gamma_j\,,\, S_i) = S^{GW}$$

Mit dem Bewehrungsmodul können die Schnittgrössen linear-elastisch (=*elastisch*) oder nach Plastizitätstheorie (=*plastisch*) berechnet werden. Beiden Verfahren ist gemeinsam, dass die Schnittgrössenverteilung eine Gleichgewichtslösung darstellt, d.h. sie steht mit den aufgebrachten Lasten im Gleichgewicht und erfüllt die statischen Randbedingungen.

☞ *Gemäss SIA, EC2 und DIN kann die Schnittgrössenermittlung linear-elastisch, nach Plastizitätstheorie oder mit nichtlinearen Verfahren erfolgen. CEDRUS (mit Bewehrungs- und Traglastmodul) kann Schnittgrössen nach allen drei Verfahren bestimmen. Eine Bewehrungsbemessung mittels den äusserst rechenaufwendigen, nichtlinearen Verfahren ist für praktische Probleme jedoch ungeeignet, da die Bewehrung bereits vor einer nichtlinearen Analyse bekannt sein muss und jede Lastkombination einzeln zu analysieren ist.*

J 3.1.2 Elastische Bemessung als Optimierungsaufgabe

Mit dem FE-Modell des CEDRUS Basismoduls lassen sich die elastischen Bemessungsgrenzwerte S_{el}^{GW} aufgrund einer vorgegebenen Grenzwertspezifikation (s. CEDRUS Handbuch: Kapitel 2.3.3.) direkt berechnen. Hat der Programmbenutzer überall in der Platte Bewehrungsfelder gemäss J 2.2 mit noch undefinierten Bewehrungsflächen a_{s_i} eingelegt, so kann die Bemessungsaufgabe wie folgt formuliert werden:

$$S^{GW} = S_{el}^{GW} \leq R_d \qquad \Rightarrow S_{el}^{GW} \leq \frac{1}{\gamma_S} \sum a_{s_i} f_{yk_i} z$$

Diese Ungleichung weist als Unbekannte nur die Bewehrungsflächen a_{s_i} auf. Da sich die Bewehrungsfelder gegenseitig überlappenden dürfen, stellt die Wahl der a_{s_i} ein lineares Optimierungsproblem dar, welches im Bewehrungsmodul für minimales Bewehrungsgewicht gelöst wird. Die sog. Zielfunktion der Optimierung G, welche minimiert wird, entspricht dem Gesamtgewicht der eingelegten Bewehrung und lässt sich aus den zu bestimmenden Bewehrungsflächen a_{s_i}, den Flächen der einzelnen Bewehrungsfelder F_i und dem spezifischen Gewicht des Stahles ρ berechnen:

$$G = \rho \sum a_{s_i} F_i \rightarrow Minimum$$

Nachdem der Benutzer das Strukturmodell samt Belastung mit dem Basismodul von CEDRUS erstellt, eine Grenzwertspezifikation vorgegeben und Bewehrungsfelder definiert hat, so ermittelt der Bewehrungsmodul auf Knopfdruck die optimalen Werte für die Bewehrungsflächen a_{s_i}. Diese werden automatisch in Durchmesser und Abstände bzw. Netztypen umgerechnet, womit eine vollständige und tragsichere Bewehrung resultiert, welche anschliessend in Form einer Bewehrungsskizze ausgedruckt oder zwecks Weiterbearbeitung zu einem Bewehrungsplan in ein CAD-System übertragen werden kann (s. Kapitel 5).

☞ *Der Bemessungsalgorithmus kann nur dann eine konsistente Bemessung durchführen, wenn in allen beanspruchten Zonen Bewehrungsfelder definiert wurden.*

Vorgabe minimaler Bewehrungsgehalte

In weiten Teilen einer Platte muss z.B. aus Gründen der Gebrauchstauglichkeit (s. Abschnitt J 3.1.4) eine Minimalbewehrung eingelegt werden. Diese Bedingungen können im oben beschriebenen Optimierungsverfahren direkt berücksichtigt werden, indem für jedes Bewehrungsfeld ein minimaler Bewehrungsgehalt angegeben werden kann, der bei der optimalen Wahl der Bewehrungsflächen nicht unterschritten wird.

Grenzen der elastischen Bemessung

Eine rein elastische Bemessung der Plattenbewehrung liefert in vielen Fällen übermässig hohe Bewehrungsgehalte. Dies hängt einerseits mit den unrealistischen Konzentrationen über Stützen, Wandenden oder einspringenden Ecken zusammen, welche durch die elastische Plattentheorie bedingt sind (Singularitäten). Andererseits muss an vielen Stellen Minimalbewehrung eingelegt werde, welche statisch nicht genutzt werden kann. Um unvernünftige Bewehrungsanordnungen zu vermeiden, decken Ingenieure die Momentenspitzen oft nicht vollständig mit Bewehrung ab und legen als Kompensation dafür in den benachbarten Zonen etwas mehr Bewehrungsstahl ein. Dieses Vorgehen erfordert Intuition und einen beträchtlichen Nachbearbeitungsaufwand. Im Prinzip ist eine solche Lösung mit den Normvorschriften nicht vereinbar, kann doch kaum mehr von elastischer Schnittkraftermittlung gesprochen werden. Was noch schwerer wiegt ist die Tatsache, dass diese 'Umlagerung von Hand' theoretisch als fragwürdig bezeichnet werden muss, da kein Gleichgewichtszustand mehr garantiert werden kann.

J 3.1.3 Plastische Bemessungsoptimierung

Die Problematik der hohen Bewehrungskonzentrationen (s. J 3.1.2) kann entschärft werden, indem das inelastische Verhalten von Stahlbetonplatten, welches dieses Material auszeichnet, bei der Schnittgrössenermittlung einbezogen wird. Der Bewehrungsmodul erlaubt den Abbau dieser hohen Momentenspitzen durch Einführung **kontrollierter Schnittkraftumlagerungen**. Anders als bei der oben erwähnten, oft problematischen 'Umlagerung von Hand' geschieht dies auf theoretisch absicherter Basis:

Gemäss Einspielsatz der Plastizitätstheorie (s. J 2.1.2) gilt:

$$S_{el}^{GW} + S_0 \le R_d \qquad \Rightarrow S_{el}^{GW} + S_0 \le \frac{1}{\gamma_S} \sum a_{s_i} f_{yk_i} z$$

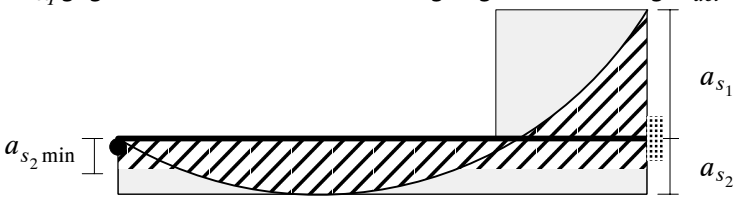
Die obenstehende Bemessungsungleichung entspricht bis auf den Eigenspannungszustand S_0 (s. J 2.1.2) der elastischen Bemessung des letzten Abschnitts, wobei S_0 eine Umlagerung der Schnittgrössen bewirkt. Bei geschickter Wahl von S_0 gelingt es, die elastische Bemessung zu optimieren, indem die Momentenspitzen umgelagert werden. CEDRUS generiert diesen Eigenspannungszustand S_0 aus einer Serie von vielen Eigenspannungszuständen S_{0_j}, womit sich die plastische Bemessungsoptimierung als lineares Optimierungsproblem mit zwei Typen von Unbekannten formulieren lässt:

$$S_0 = \sum S_{0_j} \qquad \Rightarrow S_{el}^{GW} + \sum S_{0_j} \le \frac{1}{\gamma_S} \sum a_{s_i} f_{yk_i} z$$

Ausgehend von einer elastischen Bemessung (J 3.1.2) wird die Wahl der Bewehrungsgehalte in einem automatischen Verfahren optimiert, indem schrittweise Eigenspannungszustände S_{0_j} generiert und superponiert werden. Der Programmbenutzer kann dabei am Bildschirm mitverfolgen, wie die Momente umgelagert werden, wobei er die Optimierung jederzeit abbrechen kann.

Effizienzparameter

Die Bemessungsoptimierung ist ein numerisch aufwendiges, inkrementelles Verfahren, welches besonders bei grossen Systemen sehr hohe Rechenzeiten zur Ermittlung der optimalen Lösung erfordert. Dank ausgefeilten Methoden gelingt es CEDRUS jedoch, innerhalb kurzer Zeit eine suboptimale Lösung zu finden, welche schon recht nahe am absoluten Optimum liegt (z.B. 90%) und in der Folge trotz langer Berechnung nur noch wenig verbessert werden kann. Damit kann der Programmbenutzer die Berechnung abbrechen, sobald ihm die erzielte Lösung gefällt. Der Verlauf des sog. Effizienzparameters hilft ihm dabei abzuschätzen, ob sich eine weitere Berechnung lohnt. Dieser skalare Parameter ist definiert durch das Verhältnis von statisch und konstruktiv notwendigem Bewehrungsvolumen V_{req} gegenüber dem Volumen der eingelegten Bewehrung V_{act}.



Wie in der obenstehenden Zeichung an einem gleichmässig belasteten Träger mit zwei Bewehrungslagen aufzeigt, entspricht V_{req} (schraffierte Fläche) der minimal, vorgegebenen und der zusätzlich darüberhinaus erforderlichen Bewehrung. Liegt der Effizienzparameter in der Nähe von 100%, so ist die Bewehrung optimal ausgenützt.

Anpassung der Tragwirkung an die Bewehrungsführung

Obwohl die Bemessung weitgehend automatisch abläuft, bestimmt der Ingenieur massgeblich die Güte der erzielten Bewehrung. Das Optimierungsverfahren versucht nämlich, den Widerstand der eingeführten Bewehrungsfelder optimal zu nutzen, indem die Tragweise der Platte durch Momentenumlagerung der vorgegebenen Bewehrungsanordnung soweit wie möglich angepasst wird. So kann z.B. der Widerstand von Bewehrungsfeldern in gemäss elastischer Berechnung nur schwach beanspruchten Zonen mobilisiert werden, welche aus Gründen der Gebrauchstauglichkeit jedoch einen Minimalgehalt aufweisen. Dies erlaubt dem Ingenieur eine grosse Freiheit bei der Bewehrungsführung: Bei der Eingabe der Bewehrungsfelder muss er sich nicht genau an die elastisch ermittelten Bewehrungsgehalte richten, sondern kann ausführungstechnische Aspekte einfliessen lassen und eine einfachere Bewehrungsanordnung wählen. Da die konstruktive Durchbildung der Bewehrung einen massgeblichen Einfluss auf das spätere Tragverhalten (s. nächsten Abschnitt) und die Ausführungskosten ausübt, können dadurch wirtschaftliche Lösungen erzielt werden.

Grenzen der plastischen Bemessung

Man kann zeigen, dass plastische Verformungen vor dem Erreichen der Traglast bei allen statisch unbestimmten Tragwerken auftreten, selbst wenn diese nach der Elastizitätstheorie bemessen wurden. Deshalb benötigen alle statisch ausgenützten Stahlbetonplatten ein gewisses Verformungsvermögen (=Duktilität). Das Rotationsvermögen hängt dabei zu einem grossen Teil von der konstruktiven Durchbildung der Bewehrung ab. Der Ingenieur muss dies unabhängig von der Art der Schnittgrössenermittlung sicherstellen in dem er die Bewehrungsfelder entsprechend festlegt. Dies bedeutet dass kein Querschnitt über- oder unterbewehrt und die Bewehrung überall gut verteilt (d.h. der Plattendicke angemessene, relativ geringe Stabdurchmesser und kleine Teilung) und ebenso verankert wird. Eine sorgfältige Konstruktion der Bewehrung ist insbesondere bei einer plastischen Bemessungsoptimierung notwendig, da diese ausgiebig von plastischen Rotationen Gebrauch macht. Glücklicherweise ist das Rotationsvermögen von mehrachsig gespannten Stahlbetonplatten (im Gegensatz zu Stabtragwerken) allgemein viel grösser als angenommen.

In den modernen Normen finden sich entsprechende Vorschriften zur Sicherstellung der Querschnittsduktilität (z.B. vorgeschriebene Stahlqualität, maximaler Bewehrungsgrad) und Beschränkung der Schnittkraftumlagerung, welche bei der Eingabe der Bewehrungsfelder beachtet werden sollten. Insbesondere nach der neuen DIN-Norm kann es in Einzelfällen jedoch notwendig sein, die Rotationsfähigkeit nachzuweisen. Diese Aufgabe kann mit dem Traglastmodul gelöst werden, welcher in Kapitel J 4 näher erläutert wird.

J 3.1.4 Bemessung auf Gebrauchstauglichkeit

Neben dem Nachweis der Tragsicherheit erfordern die modernen Normenwerke auch einen Gebrauchstauglichkeitsnachweis, welcher auf einem anderen Beanspruchungsniveau durchgeführt werden muss. Hierfür reicht im Normalfall eine Berechnung mit ungerissener, elastischer Steifigkeit, wie sie der Basismodul von CEDRUS zur Verfügung stellt, etwa zur Bestimmung von Durchbiegungen (mit entsprechend angepasstem E-Modul) oder zum Nachweis maximaler Stahlspannungen zwecks Rissesicherung.

Im Zusammenhang mit der Momentenumlagerung infolge einer plastischen Bemessung auf Tragsicherheit muss jedoch sichergestellt sein, dass unzulässig grosse Verformungen und Rissbreiten im Gebrauchszustand verhindert werden. Dies gelingt einerseits durch eine saubere, konstruktive Durchbildung der Bewehrung (s. z.B. letzten Abschnitt 'Grenzen der Bemessung') und andererseits durch die Verhinderung von grossen Momentenumlagerungen unter Gebrauchslasten.

Vorgabe einer Minimalbewehrung für Tragsicherheitsbemessung

Die zweite Anforderung kann mit dem Bewehrungsmodul direkt im Rahmen einer Bemessung auf Tragsicherheit erfüllt werden, indem vor der plastischen Bemessungsoptimierung für die Bewehrungsfelder minimale Bewehrungsgehalte definiert werden, welche aus einer elastischen Bemessung für Gebrauchslasten resultieren. Damit kann es sinnvoll sein, nach der Eingabe einer Bewehrungsanordnung die Gehalte der Bewehrungsfelder für Gebrauchslasten in einem ersten Schritt elastisch zu ermitten. Diese Bewehrungsgehalte können anschliessend (ev. beschränkt auf kritische Felder) als minimale Gehalte vorgegeben werden. Danach kann diese Bewehrungsanordnung auf Tragsicherheit mittels plastischer Optimierung bemessen werden, wobei sich dann oft zeigt, dass nur noch wenig Zusatzbewehrung notwendig ist.

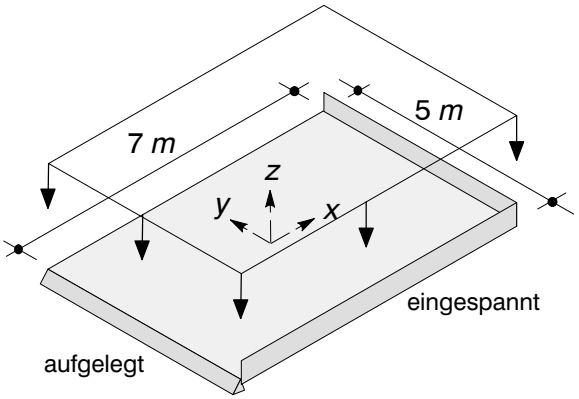
J 3.2 Bemessungsablauf

In diesem Abschnitt wird der Ablauf einer Bemessung mit dem Bewehrungsmodul anhand eines einfachen Beispiels aufgezeigt. Um diese Einführung übersichtlich und verständlich zu halten, wird auf die Behandlung vieler Details verzichtet. Für eine Beschreibung aller Funktionen sei auf das umfangreiche **Hilfesystem von CEDRUS** verwiesen, welches Ihnen jederzeit über das Hauptmenü "Hilfe" oder kontextbezogen durch Drücken der Taste <F1> bzw. der Schaltfläche [?] zur Verfügung steht.

☞ *Für diesen Abschnitt wird vorausgesetzt, dass Sie mit dem Grundmodul von CEDRUS vertraut sind und damit die Funktionsweise der Steuerregister, der Dialoge und des Grafikeditors kennen.*

Einführungsbeispiel

Das Einführungsbeispiel besteht aus einer rechteckigen, dreiseitig auf Wänden gelagerten Platte konstanter Dicke h, welche zusätzlich zum Eigengewicht durch eine gleichförmig verteilte Nutzlast q beansprucht wird. Die Bewehrung dieser Platte soll nach der SIA Norm (SIA Swisscode) unter Verwendung eines Betons der Güteklasse C20/25 und eines Stahls der Klasse B500B bemessen werden:



$h = 22\ cm$

$q = 5\ \frac{kN}{m^2}$

Um die Struktur- und Lastdaten der Platte einzugeben, eröffnen Sie zuerst eine neue Berechnung und wählen als Norm 'SIA'. Danach stellen Sie im Baustoff-Dialog (d.h. bei Hauptmenu>Einstellungen>Baustoffe) sicher, dass für den Baustoff 'C' die Klasse 'C20/25' (E-Modul=3.04e7 kN/m^2, ν=0.17) und für 'R' die Klasse 'B500B' eingestellt sind. Danach geben Sie im Register/Struktur/ den Umriss ein (Rechteck-Werkzeug, absolute Koordinaten 0↵ 0↵ und 7↵ 5↵) und positionieren eine Attributbox für Materialzone (Dialog: isotrop, Dicke=0.22 m, Baustoffe 'C' und 'R', Steifigkeitsfaktor fE=1). Nun definieren Sie die Auflager durch die Eingabe einer Wand mit frei drehbarer Lagerung (Dialog: Einsenkung=blockiert) am linken Rand und einer Eckwand mit allgemeiner Lagerungsart (Dialog: Einsenkung=blockiert, Rotation X=blockiert, Rotation Y=frei) am unteren und rechten Rand. Im Register/Lasten/ erzeugen Sie zum automatisch erzeugen Eigengewicht 'EG' eine zweite Belastung 'NL' zur Einwirkung 'Nutzlast'-'Allgemein' und setzen eine Flächenlastbox (Dialog: -5 kN/m^2, [Ganze Struktur]). Zum Abschluss geben Sie im Register/FE-Netz/ der Netzattributbox eine maximale Seitenlänge=0.5 m vor. Von hier können Sie ohne die Berechnung zu starten direkt ins Register/Resultate/ wechseln, um zur Kontrolle z.B. die Grenzwerte der Verschiebungen für '!GZT' grafisch darzustellen (Maximalwert des unteren Grenzwertes 'D<' = -5.4 mm). Nun welchseln Sie ins Register /Bewehrung/ .

J 3.2.1 Vorbereitung der Ausgabezonen

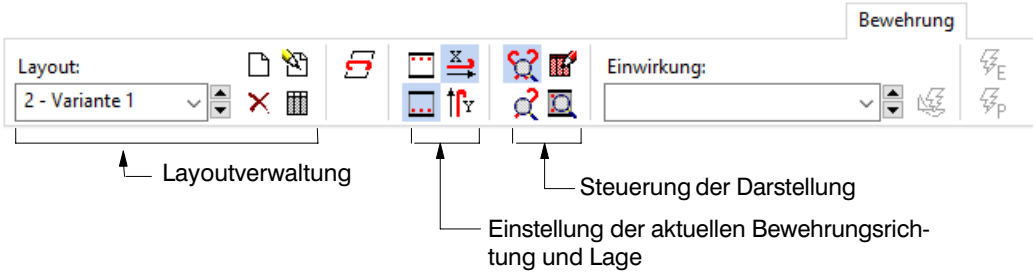
Bei einer Bemessungsaufgabe muss der Programmbenutzer in einem ersten Schritt überall in der Platte Bewehrungsfelder in Form von Stab- und Mattenlagen mit noch undefinierten Bewehrungsflächen a_s einlegen.

Die vertikale Lage eines Bewehrungsfeldes wird am einfachsten durch Zuweisung zu einer Ausgabezone (=Materialzone) definiert (s. Kapitel J 2.2). Um dieses Verfahren zu ermöglichen, müssen Ausgaberichtung und Randabstände der Materialzonen richtig eingestellt sein. Dies kann im Register/Struktur/ (evtl. bereits bei der Struktureingabe) im Dialogfenster der Materialzone (Register/Ausgabezonen/) geschehen. Die Beispielplatte weist nur eine Zone auf, für welche Sie die Ausgaberichtung zu 0 (=entspricht X-Achse) und die Randabstände je zu 3 cm setzen.

☞ *Wie in J 2.2.3 ausführlich erörtert, sollte die Ausgaberichtung wenn immer möglich mit der Hauptbewehrungsrichtung übereinstimmen. Damit kann es bei komplizierten geometrischen Verhältnissen notwendig werden, dass Sie mehrere Materialzonen mit unterschiedlichen Ausgaberichtungen eingeben müssen, obwohl die ganze Platte die gleichen Materialeigenschaften (d.h. Höhe Oberkante, Dicke, Beton etc.) aufweist.*

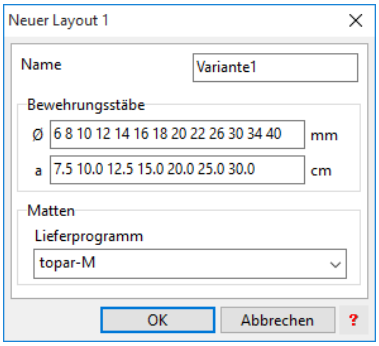
J 3.2.2 Erzeugen eines Layouts

Die Eingabe der Bewehrung erfolgt im Register/Bewehrung/. Um Variantenvergleiche zu unterstützen, wird die Bewehrung einer Platte in voneinander unabhängigen Layouts verwaltet.



Um einen neuen Layout zu erzeugen, klicken Sie auf die abgebildete Schaltfläche, womit ein Dialogfenster erscheint:

Hier können Sie dem Layout einen Namen geben und die bei der Bemessung zu verwendenden Stabdurchmesser und -abstände spezifizieren. Für das Bemessungsbeispiel löschen Sie den kleinsten Stababstand (7.5 cm) aus den zulässigen Werten. Wollen Sie mit Bewehrungsnetzen arbeiten, so könnten Sie hier ein Mattenprogramm (=Liste von Mattentypen) wählen.



Nach dem Schliessen des Dialogs mit [OK] wird für den Layout der neue Layer '1' in der Layergruppe 'Bewehrung' erzeugt und sichtbar gemacht. Zudem ist der neue Layout im Register als aktiv gesetzt (s. Registerbild oben, Layoutverwaltung).

☞ *CEDRUS verwendet bei neuen Berechnungen immer die oben angezeigten Vorgabewerte. Wenn Sie andere Vorgabewerte (z.B. für die zulässigen Stabdurchmesser) definieren wollen, so können Sie (wie übrigens bei den meisten Dialogen in Cubus-Programmen) mit der Tastenkombination Ctrl-F9 die aktuell eingestellten Werte des Dialogs als Vorgabe für neue Berechnungen definieren. Mit Ctrl-F10 können Sie die zuletzt (mit Ctrl-F9) abgespeicherte Vorgabe in den Dialog laden.*

J 3.2.3 Eingabe Bewehrungsfelder

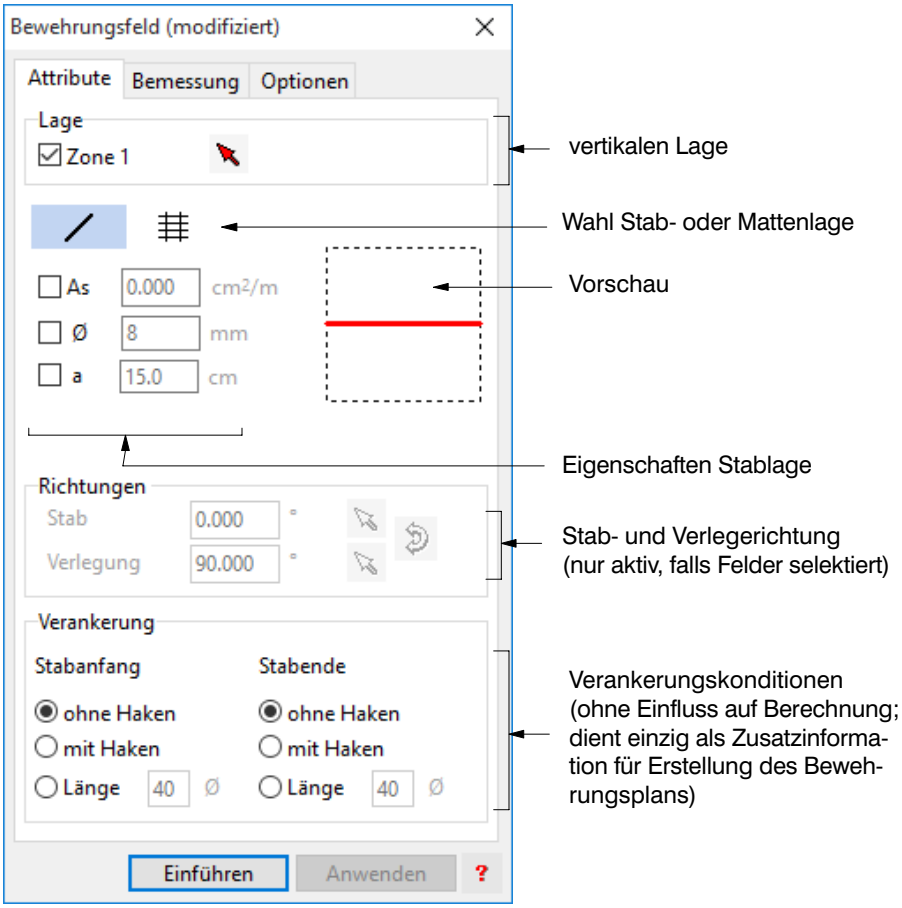
Untere Bewehrung

Nun kann die Eingabe der Bewehrungsfelder beginnen, welche wie oben erwähnt getrennt für die untere und obere Bewehrung erfolgt. Durch Klicken auf die nebenstehende Schaltfläche wählen Sie für die Eingabe zuerst die untere Bewehrung.

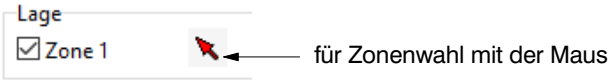
Angesichts der Plattengeometrie soll in beiden Richtungen eine durchgehende untere Bewehrung eingelegt werden. Um mit der Bewehrung in X-Richtung anzufangen, stellen Sie sicher, dass die X-Bewehrung aktiviert ist (nebenstehende Schaltfläche). Damit ist alles für die Eingabe der unteren Bewehrung in X-Richtung vorbereitet.

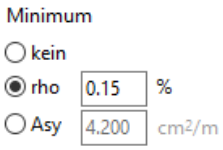
☞ *Die hier gewählte X-Richtung(bzw. Y-Richtung) entspricht dabei der aktuellen Koordinaten-Achse, welche im Dialog 'Eingabeoptionen' (Schaltfläche* *am linken Rand) definiert werden kann. Da die X- und Y-Achsen beliebig orientiert sein dürfen, ist somit auch die Eingabe von schiefer Bewehrung möglich.*

Klicken Sie auf die Schaltfläche 'Eingabe Bewehrungsfeld'. Es erscheint folgendes Dialogfenster:

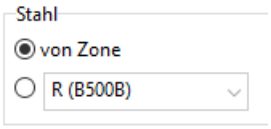


Dieser Dialog zeigt alle Attribute eines Bewehrungsfeldes an. Für die Beispielplatte sollen keine Matten verwendet werden. Wählen Sie deshalb die Einstellung Stabfeld. Vor der Konstruktion der geometrischen Ausdehnung in der Plattenebene, definieren Sie zuerst die vertikale Lage durch Zuordnung zur Zone 1:





Aus konstruktiven Gründen muss oft eine Minimalbewehrung eingelegt werden. Dies kann direkt bei der Eingabe der Bewehrungsfelder geschehen, indem Sie im Register/Bemessung/ einen minimalen Gehalt vorgeben. Für das Beispiel wählen Sie einen geometrischen Bewehrungsgehalt von 0.15%, was einer minimalen Bewehrungsfläche von 3.3 cm^2/m entspricht.



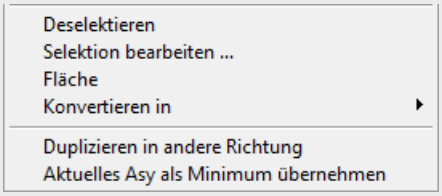
Jedes Bewehrungsfeld kann eine andere Stahlgüte aufweisen, welche Sie im Register/Optionen/ einstellen können. In den meisten Fällen verwendet man jedoch einen einheitlichen Betonstahl (hier hellgrau geschrieben), welchen Sie in der Baustoffverwaltung (Hauptmenu>Baustoffe) konfigurieren können.

Beim Öffnen des Dialogs wurden nebenstehende Zeichenwerkzeuge aktiviert, womit man rechteckförmige und polygonale Felder konstruieren kann. Die untere Bewehrungslage in X-Richtung kann nun als Rechteckfeld eingegeben werden, indem Sie entweder auf die Schaltfläche [Einführen] des Dialogs oder auf den Schalter drücken und anschliessend auf die linke untere und rechte obere Ecke des Umrisses klicken. Das eben konstruierte Bewehrungsfeld wird jetzt am Bildschirm als gestricheltes Rechteck (=statisch wirksame Zone) mit beschriftetem, durchlaufendem Bewehrungsstab gezeichnet. Die Beschriftung ist auf dem Stab positioniert, erfolgt in dessen Richtung und zeigt die aktuelle Bewehrungsfläche an. Weil noch keine Bemessung durchgeführt wurde, entspricht diese momentan dem minimal vorgegebenen Gehalt.

☞ *Die Beschriftung zeigt neben der aktuellen Bewehrungsfläche noch weitere Attribute eines Feldes an. So kann z.B. anhand eines unterstrichenen Textes erkannt werden, dass ein minimaler Gehalt vorgegeben wurde. Weitere Erläutertungen zu diesem Thema finden Sie im Kapitel J 3.2.7*

Um die deckungsgleiche Y-Bewehrung einzugeben, drücken Sie ohne den Dialog zu schliessen auf die nebenstehende Register-Schaltfläche. Dabei werden die Vorschaugrafik und die Werte für die Richtungen im Dialog angepasst, während die übrigen Einstellungen unverändert bleiben. Analog zur X-Bewehrung konstruieren Sie jetzt das Bewehrungsfeld in Y-Richtung durch Drücken auf die Schaltfläche [Einführen] und anschliessendes Klicken auf zwei gegenüberliegende Umrissecken.

Duplizieren in andere Richtung



Für den häufigen Fall von deckungsgleichen Feldern in beiden Bewehrungsrichtungen steht Ihnen alternativ zur oben beschriebenen Eingabeart die bequeme Duplizieren-Funktion zur Verfügung:

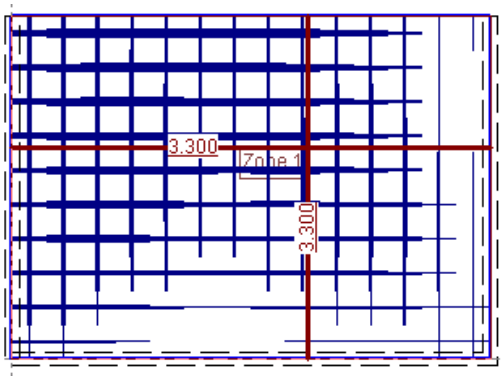
Um diese hier anzuwenden, machen Sie zuerst die Eingabe des zweiten Bewehrungsfeldes rückgängig (CTRL-Z=Undo). Anschliessend selektieren Sie das Feld in X-Richtung (Klick mit linker Maustaste). Im Objektmenu (Klick mit rechter Maustaste) wählen Sie danach 'Duplizieren in andere Richtung'. Damit wurde ein Feld in Y-Richtung mit gleichen Attributen erzeugt.

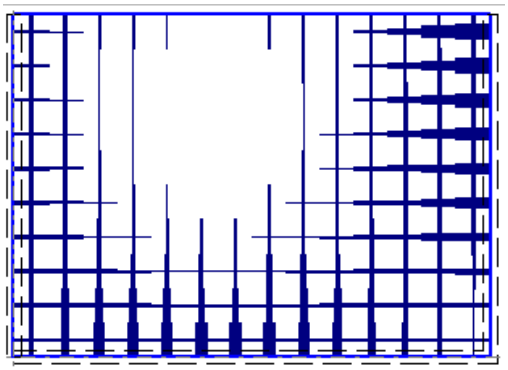
☞ *Die Funktion 'Duplizieren in andere Richtung' kann auch auf mehrere Bewehrungsfelder gleichzeitig angewendet werden, sofern nur Rechteck- (bzw. Parallelogramm)-Felder selektiert sind. Dabei gilt es zu beachten, dass sich die neu erzeugten Felder nur durch die vertauschten Stab- und Verlegerichtung von den originalen unterscheiden und die aktuell eingestellten Eingaberichtungen somit keinen Einfluss haben.*

Konstruktion aufgrund elastischem Bewehrungsbedarf

Bis jetzt wurden die Bewehrungsfelder ohne Rücksicht auf die Beanspruchung, rein aufgrund geometrischer und konstruktiver Überlegungen definiert. Um basierend auf den elastisch erforderlichen Bewehrungsgehalten konstruieren zu können, muss zuerst die massgebende Einwirkung angegeben werden, wobei alle vorhandenen Grenzwertspezifikationen zur Verfügung stehen.

Wählen Sie die Spezifikation 'GZT', welche von CEDRUS automatisch erzeugt wurde. Anschliessend klicken Sie auf die Schaltfläche 'Berechnen Bewehrungsbedarf', womit (falls noch nicht erfolgt) die FE-Masche generiert, die Lastfälle gelöst und die elastischen Grenzwerte des Bewehrungsbedarfs berechnet werden.

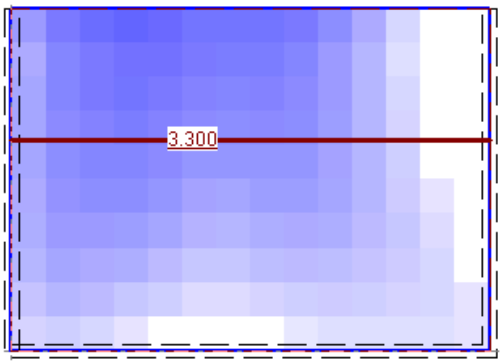




Am Bildschirm sind jetzt die statisch erforderlichen Bewehrungsflächen elementweise als blaue Linien in den Ausgaberichtungen grafisch dargestellt, wobei die Linienstärke mit ansteigender Fläche zunimmt. Die erforderliche Bewehrungsfläche wird dabei aufgrund des in der Materialzone angegebenen Bewehrungsstahls (Dialog Attributbox Materialzone>Baustoffe>Bewehrung) ermittelt.

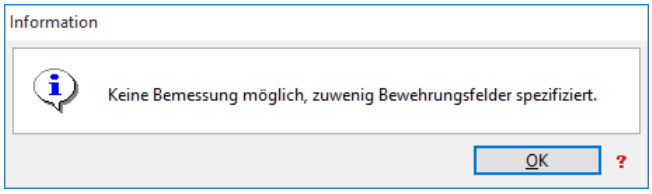
Um die numerischen Werte zu erfahren, bewegen Sie den Mauszeiger über die Farbgrafik und CEDRUS zeigt Ihnen die der Mauszeigerposition entsprechenden Bewehrungsflächen in der Statuszeile an (untere Fensterrand). Mit dem nebenstehenden Schalter können Sie auch den Bedarf an oberer Bewehrung darstellen (s. oben). Wie zu erwarten war, benötigt man aus elastischer Berechnung v.a. oben bei den eingespannten Lagern und unten entlang des freien Randes Bewehrung. Stellen sie anschliessend wieder auf die untere Bewehrung um.

Um die Übersichtlichkeit zu erhöhen, kann die gleichzeitige Darstellung der Felder und Resultate beider Bewehrungsrichtungen unterdrückt werden. Stellen Sie über die nebenstehenden Registerschalter die aktuelle Bewehrungsrichtung auf X und die Anzeige auf 'Felder in Bewehrungsrichtung zeigen'. Damit ist nur noch das Feld in X-Richtung sichtbar und die erforderlichen Bewehrungsflächen in dieser Richtung werden nun als Farbverlaufsgrafik dargestellt.

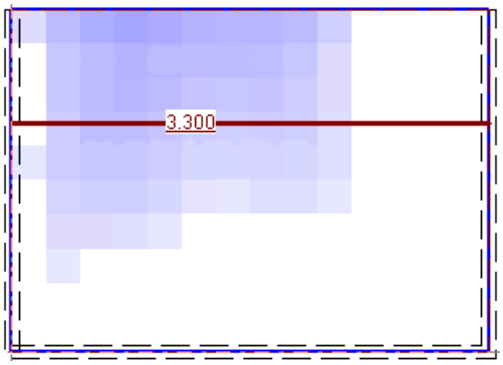


Zulagen

Hier stellt sich die Frage, ob und wo Zulagen einzuführen sind. Diese sind sicher in denjenigen Bereichen sinnvoll, wo der erforderliche Gehalt weit über die minimale Bewehrung hinausgeht. Solche Bereiche können von CEDRUS dargestellt werden, falls Felder mit Minimalbewehrung eingegeben wurden. Drücken Sie zu diesem Zweck auf die Schaltfläche 'elastische Bemessung' E, worauf der folgende Dialog erscheint:



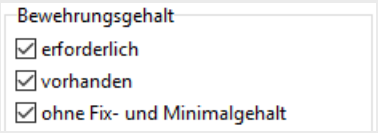
Da keine obere Bewehrung eingegeben wurde, ist zu diesem Zeitpunkt keine elastische Bemessung möglich. Die Darstellung der erforderlichen Bewehrungsgehalte wird jedoch angepasst (s. nebenstehendes Bild), womit nur noch diejenigen Zonen farblich hervorgehoben werden, deren Beanspruchung über den Widerstand der minimal vorgegebenen Bewehrung hinausgeht.

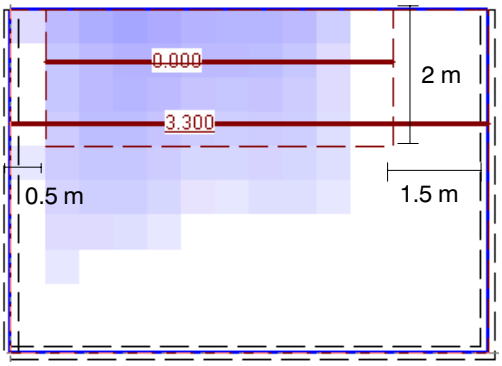


☞ *Sobald die Schaltfläche für 'elastische Bemessung' gedrückt wurde, sind in der Resultatdarstellung nur noch die Zonen sichtbar, deren Beanspruchung die Minimalbewehrung überschreiten. Wie im folgenden Kasten erläutert, kann dieses Verhalten über den Dialog 'Einstellungen Darstellung' konfiguriert werden.*

Einstellung der Resultatdarstellung

Der Dialog zur Beeinflussung der Darstellung von Resultaten und Bewehrungsfeldern kann über nebenstehende Schaltfläche geöffnet werden. Als Vorgabe ist im Register/Resultate/ der aktuelle Bewehrungsgehalt (erforderlich und vorhanden) gewählt. Zudem ist die Schaltfläche 'ohne Fix- und Minimalgehalt' aktiviert, womit die fest vorgegebenen und minimalen Gehalte vor der Darstellung abgezogen werden.



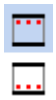


Aufgrund dieser Darstellung und konstruktiver Überlegungen sollen nun entlang des freien Randes eine Zulage definiert werden. Legen Sie deshalb ein neues Bewehrungsfeld ein, indem Sie zuerst im Dialog Bewehrungsfeld (Register/Bemessung/) den Minimalgehalt deaktivieren, den Schalter [Einführen] drücken und anschliessend ein zusätzliches Bewehrungsfeld konstruieren (s. nebenstehendes Bild).

Obere Bewehrung

Aus konstruktiven Gründen soll auch oben eine durchgehende Minimalbewehrung eingelegt werden. Sie können dies analog zur unteren Bewehrung tun, indem Sie in beiden Richtungen je ein Bewehrungsfeld mit vorgeschriebenem Minimalgehalt konstruieren.

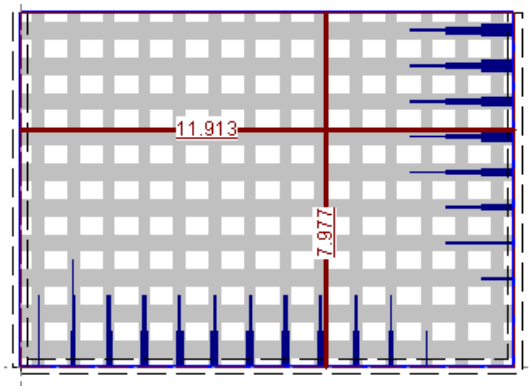
Das selbe können Sie einfacher und schneller erreichen, wenn Sie zuerst alle Felder der unteren Bewehrung mit dem Schalter sichtbar machen, die beiden grossen Bewehrungsfelder selektieren (linke Maustaste: Klick auf ersten Stab und SHIFT-Klick auf zweiten Stab) und diese auf die Ablage kopieren (via Objektmenu oder CTRL-C). Danach wechseln Sie auf die obere Bewehrung und fügen beide Felder ein (via Objektmenu 'Einfügen' oder CTRL-V).

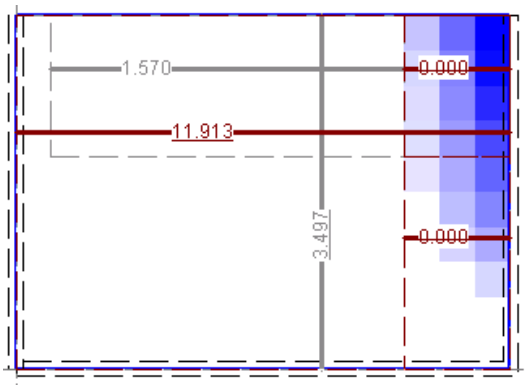


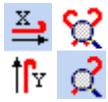
Zulagen

Um die Zulagen konstruieren zu können, drücken Sie wiederum auf die Schaltfläche .

Da nun überall in der Platte eine Bewehrung definiert wurde, welche einen variablen Bewehrungsgehalt aufweist, ist diesmal eine elastische Bemessung möglich. Am Bildschirm wird nun neben dem erforderlichen (blau) auch der aufgrund der elastischen Bemessung eingelegte Bewehrungsgehalt (grau) dargestellt (s. linkes Bild):







Das dabei erzielte Resultat von über 95 kg/m^3 ist jedoch von rein theoretischer Natur, da in der oberen Bewehrung keine Abstufungen eingegeben wurden. Für die Konstruktion der Zulagen in X-Richtung stellen Sie die Sichtbarkeit gemäss nebenstehenden Schaltflächen ein und drücken zusätzlich den Register-Schalter , welcher die untere Bewehrung hellgrau zeichnet (s. Bild oben rechts). Damit können Sie nun eine obere Zulage (ohne Minimalgehalt!) anschliessend an die untere konstruieren, indem sie im Dialog [Einführen] drücken und dann die rechte untere Ecke jener Zulage und die rechte obere Ecke des Umrisses anklicken. Analog dazu definieren Sie ein zweites Bewehrungsfeld direkt daneben.

In Y-Richtung geben Sie nur ein zusätzliches Bewehrungsfeld ein, welches mit einer Breite von ca. 1.5 m der gesamten Einspannung entlang verläuft. Dazu machen Sie mit zuerst die Y-Richtung zur aktuellen Bewehrungsrichtung, drücken im Dialog [Einführen] und konstruieren dieses Feld mit dem Grafikeditor (z.B. mit absolute Koordinaten 0 ↵ 0 ↵ und 7 ↵ 1.5 ↵).

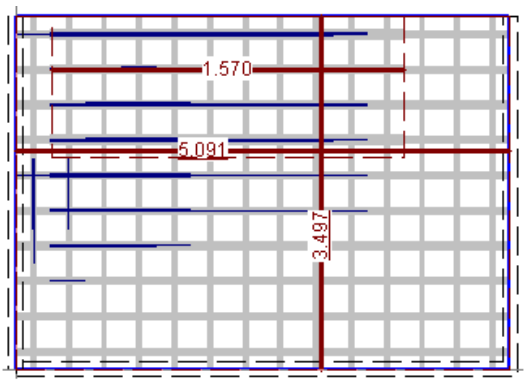
Positionierung der Feldbeschriftung

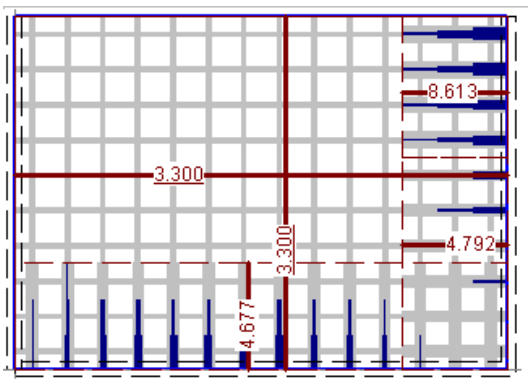
Bei der Konstruktion eines Bewehrungsfeldes wird der Stab und die Beschriftung automatisch positioniert. Dies kann wie im vorliegenden Beispiel (bei der Zusatzlage in Y-Richtung) dazu führen, dass zwei Stäbe gerade übereinander zu liegen kommen. Das Verschiebung einer Beschriftung ist eine Standardfunktion aller Eingabeobjekte und geschieht wie folgt:

- Selektieren Sie das gewünschte Bewehrungsfeld.
- Wählen Sie über das Objektmenü (=Klick auf rechte Maustaste) die Funktion 'Beschriftung verschieben' und setzen Sie die Beschriftung an eine neue Position.

J 3.2.4 Elastische Bemessung

Damit ist die Eingabe der Bewehrung abgeschlossen und die elastische Bemessung kann mit der Schaltfläche erneut ausgelöst werden. Diesmal resuliert ein realistisches Bewehrungsgewicht von ca. 62 kg/m^3 aus den statisch erforderlichen Bewehrungsflächen (s. Bild unten). Diese werden anschliessend (s. Kapitel J 3.2.7) in Stablagen mit Durchmesser und Teilung umgerechnet, womit sich das Gesamtgewicht etwas erhöhen wird. Dabei gilt es zu beachten, dass in dieser Berechnung die Verankerungslängen, Stösse und Randeisen noch nicht eingerechnet sind, welche das Gewicht erfahrungsgemäss nochmals um ca. 10-15% anheben können.





Wenn Sie mit alle Felder sichtbar machen und einmal die untere (links) und dann die obere Bewehrung aktivieren, so präsentieren sich Ihnen die obenstehenden Darstellungen. Diese zeigen auf, dass die eingelegte Minimalbewehrung einen grossen Teil des elastisch erforderlichen Widerstandes abzudecken vermag und der Widerstand der Zulagen nicht optimal ausgenützt ist. Durch eine feinere Abstufung der zusätzlichen Lagen könnte das Gesamtgewicht der Bewehrung reduziert werden. Diese geringe Einsparung würde jedoch durch die Zunahme der Bewehrungspositionen und Verankerungslängen zunichte gemacht.

☞ *Möchten Sie auf eine plastische Bemessungsoptimierung verzichten, so können Sie den nachfolgenden Abschnitt überspringen und direkt zur Erstellung der Bewehrungsskizze übergehen*

J 3.2.5 Bemessungsoptimierung

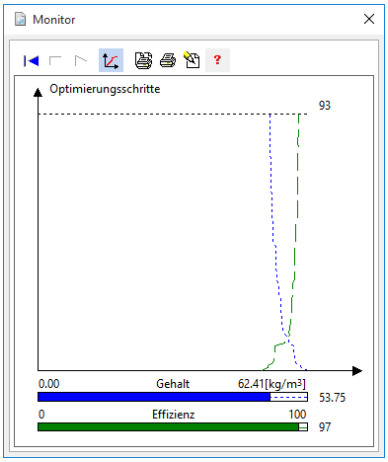
Mittels plastischer Bemessungsoptimierung kann auch für dieses eigentlich unproblematische Plattenbeispiel eine verbesserte Lösung erreicht werden:

Bemessungsmonitor

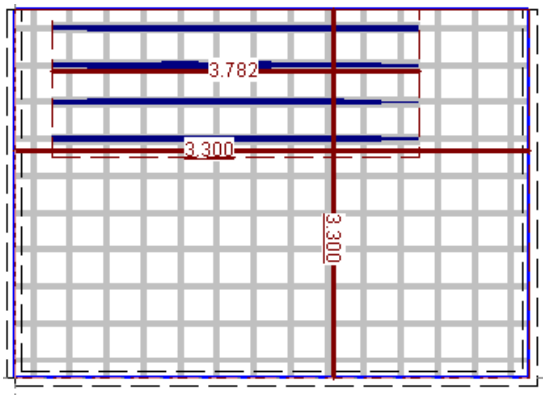
Um die Optimierung zu starten, drücken Sie nun den Register-Schalter , womit der

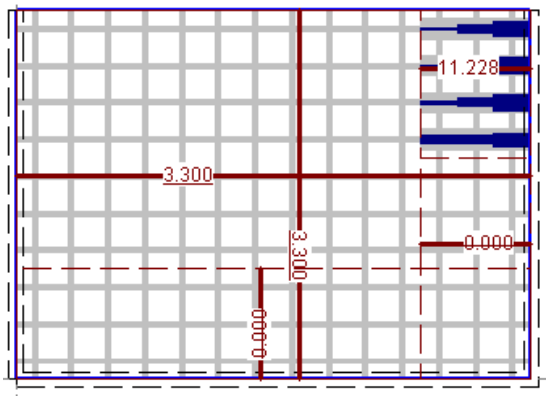
Dialog 'Monitor' erscheint. Dort klicken Sie auf nebenstehende Schaltfläche , was die automatische Bemessungsoptimierung in Gang setzt. Sie können die Berechnung am Bildschirm mitverfolgen, wo nach jedem Schritt die Grafik der Bewehrungsgehalte neu gezeichnet wird.

Im Monitor ist der Verlauf der Bemessungsoptimierung durch zwei Parameter dokumentiert, welche in Funktion der Optimierungsschritte aufgezeichnet werden: Der erste Parameter zeigt den erforderlichen Bewehrungsgehalt an, während der zweite die Effizienz der Bemessung illustriert (s. J 3.1.3). Schon nach kurzer Zeit verlaufen die beiden Kurven sehr flach. Dies ist ein Anzeichen dafür, dass der Algorithmus in die Nähe des Optimums gelangt ist und weitere Berechnungen das Bewehrungsgewicht nicht mehr entscheidend reduzieren können.



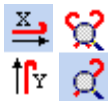
Damit können Sie die Berechnung durch Drücken der nebenstehenden Schaltfläche anhalten und den Monitordialog schliessen. Die resultierende Lösung weist ein Bewehrungsgewicht von unter 53 kg/m^3 auf, was ungefähr 86% des elastischen Bedarfs entspricht:



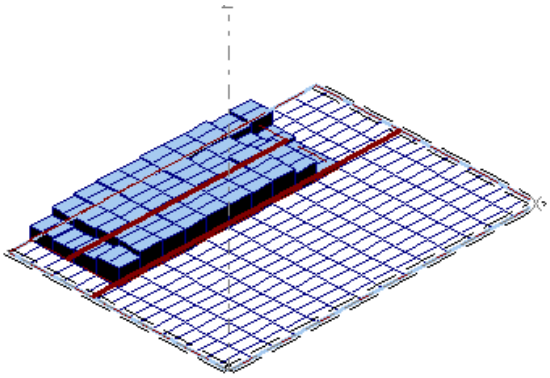


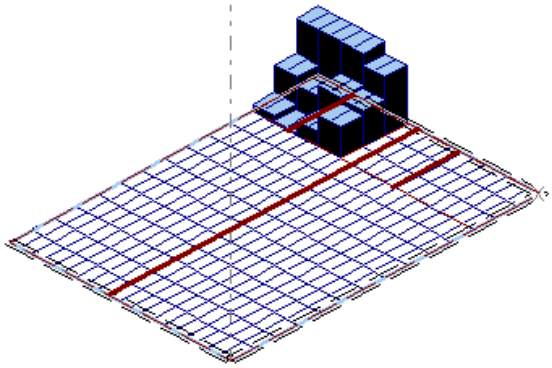
☞ *Bei allen oben dargestellten Resultatgrafiken sind die der Grafik entsprechenden nummerischen Werte des Elementes an der aktuellen Mausposition immer in der Statuszeile ersichlich (z.B. erforderliche Bewehrungsflächen).*

J 3.2.6 Interpretation der Resultate



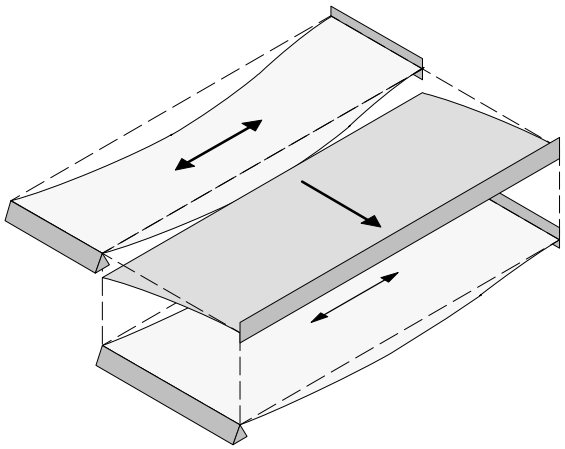
Aus der obenstehenden Darstellung wird ersichtlich, dass von allen spezifizierten Zulagen nur jene benötigt werden, welche entlang des freien Randes verlaufen. Im Vergleich zur elastischen Lösung, wo der Kraftfluss kontinuierlich über die ganze Platte verläuft, weist die optimierte Lösung dort eine vorherrschende Tragrichtung auf. Diese Tragweise wird noch deutlicher sichtbar, wenn Sie nur die X-Richtung einschalten und mittels Anklicken der Schaltfläche (am linken Fensterrand) auf 3D-Darstellung umschalten:





Die obenstehende Grafik zeigt den zusätzlich zur Minimalbewehrung erforderlichen Bewehrungsbedarf in Form einer Säulengrafik. Leicht erkennbar ist das Stützmoment (rechts) und das Feldmoment (links). Die resultierende Tragweise der Platte kann etwa wie folgt interpretiert werden:

Zwischen der Zone am freien Rand und der übrigen Platte zeichnet sich eine deutliche 'Lastscheide' ab. Provoziert durch die Anordnung der Bewehrungsfelder, gleicht die Tragweise in der Randzone einem einfachen Balken, der auf einer Seite aufgelegt und auf der anderen eingespannt ist. Die restlichen Lasten werden teilweise über einen Kragarm entlang der langen Einspannung und teilweise wiederum über einen Balken in Längsrichtung abgetragen. Der Optimierungsalgorithmus hat dabei die Lastaufteilung so gewählt, dass die Minimalbewehrung ausreicht.



☞ *Diese Tragweise erinnert an eine Lösung nach der Streifenmethode. Dies ist kein Zufall, basiert jene doch genau auf den selben theoretischen Grundlagen. Im Gegensatz zur Streifenmethode erfolgt die Berechnung hier jedoch an einem Kontinuum, womit die Verträglichkeit implizit erfüllt wird. Zudem kann CEDRUS mit Grenzwerten arbeiten und wählt die optimale Lastaufteilung selbständig aufgrund der spezifizierten Bewehrungsfelder.*

Diese Betrachtung zeigt deutlich, dass die Wahl der Bewehrungslagen das Tragverhalten der Struktur entscheidend beeinflusst, da der Optimierungsalgorithmus die Tragwirkung der vorgegebenen Bewehrungsanordnung anzupassen versucht. Vom ökonomischen Standpunkt aus gesehen, trägt neben dem geringeren Gesamtgewicht auch die Vereinfachung der Bewehrungsanordnung zur Wirtschaftlichkeit der optimierten Lösung bei.

J 3.2.7 Erstellung Bewehrungsskizze

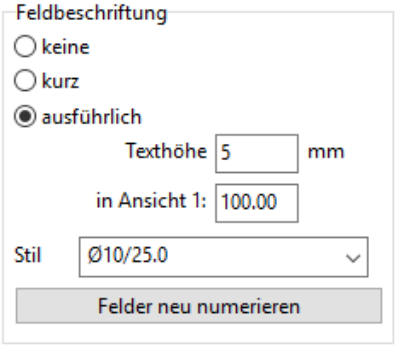
Als letzten Schritt im Bemessungsablauf erfolgt die Konstruktion der einzelnen Stablagen und die Erstellung der Bewehrungsskizze.

Wechseln Sie dazu wieder auf 2D-Darstellung und stellen Sie sicher, dass beide Richtungen der unteren Bewehrung gezeigt werden. Da die Resultatdarstellung nicht mehr benötigt wird, können Sie diese ausschalten, indem Sie den Resultatlayer über den nebenstehenden Schalter in der Layergruppe Bewehrung betätigen und anschliessend das Bild neu zeichnen lassen (F2). Den Bewehrungsfeldern sollen nun Durchmesser und Teilungen zugeordnet werden. Selektieren Sie dazu die beiden grossen Felder, öffnen den Bewehrungsfeld-Dialog (z.B. über Eigenschaften im Kontextmenu) und wechseln ins Register/Bemessung/. Die Eingabefelder Durchmesser ⌀ und Teilung *a* sind noch nicht aktiv, enthalten jedoch einen Vorschlag (hellgrau), den Sie durch Klick auf die beiden Schaltflächen übernehmen können:

Als Zusatzinformation für die Erstellung des Bewehrungsplanes können Sie bei der Verankerung im Register/Attribute/ je ein Haken und eine Verankerungslänge spezifizieren. Drücken Sie nun die Schaltfläche [Anwenden], womit beide Felder festgelegt und die effektiv vorhandene Bewehrungsfläche neben der statisch erforderlichen gezeigt wird (s. oben). Nun selektieren Sie die Zulage und wählen eine Teilung von 15 cm (durch Klick auf die Schaltfläche *a* mit anschliessender Eingabe von 15) und aktivieren den vorgeschlagenen Durchmesser 8 mm. Dann wählen Sie an beiden Enden eine Verankerungslänge und drücken erneut [Anwenden].

Jetzt wechseln Sie in die obere Bewehrung und gehen bei den grossen Feldern analog zu den unteren Lagen (⌀8mm alle 15 cm) vor. Dann selektieren Sie die Zulage am oberen Rand, wählen eine Teilung von 15 cm und aktivieren den vorgeschlagenen Durchmesser 14 mm. Auch hier wählen Sie eine Verankerungslänge für beide Enden und drücken [Anwenden]. Die restlichen, statisch nutzlosen Felder können Sie jetzt selektieren und mit der DEL-Taste löschen. Damit ist die Bewehrungsanordnung konstruiert und Sie können den Bewehrungsdialog schliessen.

Um die Darstellung noch etwas zu verfeinern, öffnen Sie den Dialog 'Einstellungen Darstellung' mit der nebenstehenden Schaltfläche und wechseln ins Register/Bewehrung/. Dort wählen Sie die ausführliche Feldbeschriftung und geben für die Ansicht 1:100 eine Texthöhe 3 mm vor. Drücken Sie zudem 'Felder neu numerieren', womit den eingegebenen Feldern eine aufsteigende Positionsnummer, beginnend mit den unteren Lagen vergeben wird. Danach schliessen Sie den Dialog wieder.

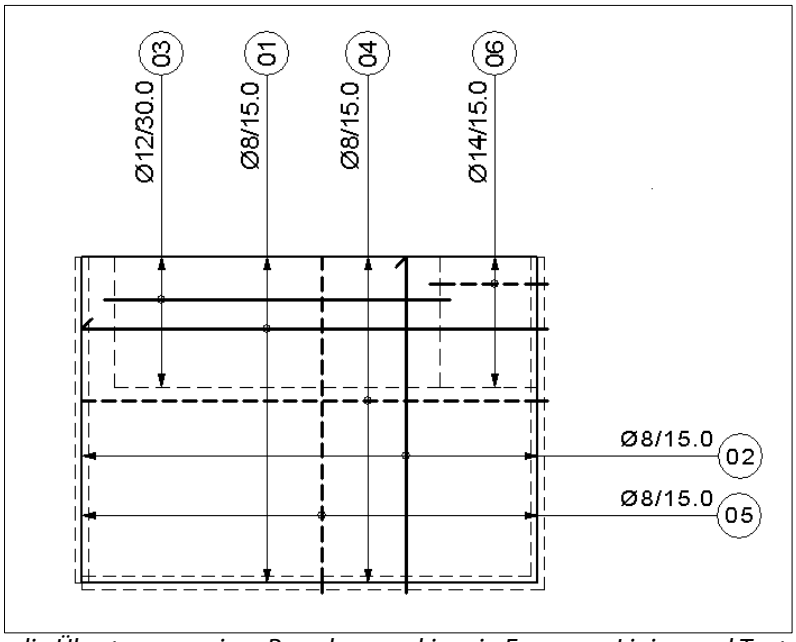


Am Bildschirm erscheint jetzt eine Bewehrungsskizze, welche einer Plandarstellung ähnlich ist. Bereiten Sie diese durch Verschieben der Beschriftungspunkte (s. oben, Kasten 'Positionierung der Feldbeschriftung') optisch noch etwas auf. Damit ist die Bemessung abgeschlossen und Sie können die Skizze ausdrucken (s. Bild unten).

Falls Sie über ein CAD-System verfügen, welches mit CEDRUS Bewehrungsdaten austauschen kann, so haben Sie jetzt die Möglichkeit, die Bewehrungsanordnung zu exportieren. Dabei werden alle Bewehrungslagen mit ihren Attributen (Durchmesser, Teilung, Verankerungskondition etc.) auf eine Datei geschrieben, welche Sie anschliessend im CAD-System einlesen können, um dort die Bewehrung fertig zu konstruieren (d.h. konstruktive Zulagen, Stösse etc.) und den Bewehrungsplan zu erstellen.

Dazu öffnen Sie mit nebenstehender Register-Schaltfläche den Dialog 'Bewehrungslayout exportieren' und wählen die gewünschte Schnittstelle.

☞ *CEDRUS unterstützt mehrere Bewehrungsformate. Standardmässig haben Sie die Wahl zwischen dem CEDRUS eigenen ECB-Format und dem Industriestandard DXF, welcher je-*



doch nur die Übertragung einer Bewehrungsskizze in Form von Linien und Texten erlaubt und somit nur bedingt geeignet ist. CEDRUS unterstützt auch einige proprietäre Formate von CAD-Herstellern (z.B. ALLPLAN mit der Serviceoption 'ALLPLAN').

J 4 Traglastmodul

Der Traglastmodul erlaubt die nichtlineare Berechnung einer Stahlbetonplatte mit vorgegebener Bewehrungsanordnung. In diesem Kapitel werden die Grundkonzepte der Berechnung näher erläutert und der Ablauf einer Traglastberechnung anhand einer Beispielplatte aufgezeigt.

J 4.1 Grundkonzepte

Die modernen Normenwerke erlauben die Anwendung von nichtlinearen Berechnungsverfahren, um die Grenzzustände der Gebrauchstauglichkeit und Tragsicherheit nachzuweisen. Der Traglastmodul von CEDRUS *beschränkt sich auf Berechnungen zur Tragsicherheit* und kann deshalb keine Informationen über Rissbreiten und Durchbiegungen unter Gebrauchslasten geben. Da die Bewehrung vor Beginn der Berechnung angegeben werden muss und nur eine Lastkombination gleichzeitig untersucht werden kann (d.h. es ist keine Grenzwertbildung möglich), eignet er sich nur indirekt für Bemessungsaufgaben. Mit dem Traglastmodul können jedoch Tragreserven bestehender Strukturen ermittelt oder der Rotationsbedarf im Anschluss an eine plastische Bemessung abgeschätzt werden.

Nichtlineare Berechnungsverfahren

Unter nichtlinearen Verfahren werden FE-Berechnungen verstanden, welche die nichtlinearen Eigenschaften der Baustoffe ausnützen können. Ausgehend von einem initialen Spannungs- und Verformungszustand wird das Verhalten einer Struktur mit vorgegebener Bewehrung unter schrittweiser Erhöhung der Belastung analysiert (s. J 2.3.1). Bisher fanden diese äusserst rechenintensiven Verfahren jedoch noch wenig Anwendung in der Praxis, obwohl sie in der Forschung schon seit langem eingesetzt werden. Dies liegt zu einem grossen Teil an der Natur von nichtlinearen Berechnungen, wo schon kleine Variationen der Eingabeparameter grosse Veränderungen der Resultate bewirken können. Angesichts der nur ungenau erfassbaren Eigenschaften von Stahlbetonstrukturen (Ausgangszustand, Materialkennwerte, Lagerungsbedingungen, Belastung etc.) erfordern diese Verfahren vom Benutzer viel Erfahrung bei der Problemmodellierung und Resultatinterpretation. Diese Problematik wächst mit der Komplexität der verwendeten Modelle, weshalb im Traglastmodul von CEDRUS ein einfaches Materialmodell mit wenigen Parametern verwendet wird (s. J 2.1.1), welche im wesentlichen die Traglast bestimmen. In Kombination mit der einfachen Berechnungssteuerung (s. J 4.1.3) und der anschaulichen Grafikausgabe erleichtert die Einfachheit des verwendeten Modelles die Interpretation der Resultate. Beim Traglastmodul von CEDRUS beschränken sich die notwendigen Benutzereingaben auf die Bewehrungsanordnung und die Belastung. Die Spezifikation eines Materialmodelles entfällt. Damit werden Traglastberechnungen auch Ingenieuren zugänglich, welche nicht über Spezialwissen auf dem Gebiet der nichtlinearen FE-Methode verfügen.

J 4.1.1 Berechnungsmodell

Die Berechnungen des Traglastmoduls (s. J 2.3) basieren auf dem FE-Modell des Grundmoduls und dem Bewehrungsmodell (s. J 2.2), welche aus den Benutzereingaben für Strukturgeometrie und Bewehrungsanordnung automatisch generiert werden. Dabei werden folgende, grundlegenden Annahmen getroffen:

- kleine Verformungen (auch im plastifizierten Zustand)
- keine Effekte 2. Ordnung
- duktiles Verhalten (s. J 4.1.2 Rotationsvermögen)
- ausreichender Schubwiderstand (s. J 4.1.2 Durchstanzen)
- konstante oder lineare Lagerungsbedingungen

Damit beschränkt sich die nichtlineare Modellierung auf das nichtlineare Materialgesetz.

Nichtlineare Momenten-Krümmungsbeziehung

Wie im Kapitel J 2.1 beschrieben, kann das nichtlineare Verhalten einer auf Biegung beanspruchten Platte durch eine elastoplastische Momenten-Krümmungsbeziehung beschrieben werden. Das im Traglastmodul verwendete Diagramm für einen Querschnitt einheitlicher Breite ist in J 2.1.1 c) dargestellt und wird durch vier charakteristische Werte definiert: Die gerissene Steifigkeit des Querschnitts EI_{II}, das plastische Moment (=Fliessmoment) $m_{pl\,R}$, die Verfestigungszahl μ und die Grenzkrümmung χ_u. CEDRUS ermittelt die Werte für EI_{II}, $m_{pl\,R}$ und χ_u automatisch aufgrund des Betonquerschnittes und der eingelegten Bewehrung (s. J 2.1.4) in jedem Element in zwei orthogonalen Querschnitten (s. J 2.2.3). Dabei wird der Widerstand der eingelegten Bewehrung sowohl auf Zug wie auf Druck berücksichtigt. Die Berechnung erfolgt nach der Methode der Variation der Dehnungsebene (s. z.B. Handbuch Querschnittsprogramm FAGUS). Für die Kennwerte f_{ck}, f_{yk} von Beton und Betonstahl werden analog zum Bewehrungsmodul (s. D 4.1) die entsprechenden Werte der in der Materialzone bzw. in jedem Bewehrungsfeld gewählten Baustoffe verwendet (einstellbar via Hauptmenu 'Einstellungen>Baustoffe'). Bei diesen Berechnungen werden die normenabhängigen Widerstandsfaktoren (d.h. γ_c ,γ_s für Beton und Betonstahl), die nichtlinearen Spannungs-Dehnungslinien und die Grenzdehnungen immer den aktuell eingestellten **Analyseparametern 'GZN** (Nichtl. Schnittgrössenermittlung) entnommen (s.Hauptmenu Einstellungen>Analyseparameter). Beim Eurocode wird zudem die Mitwirkung des Betons zwischen den Rissen (insbesondere bei der Bestimmung von χ_u) durch eine wirksame, mittlere Spannungs-Dehnungslinie des Stahls berücksichtigt wird (s. z.B. EC2 Bild A 2.1 oder E-DIN Bild 7 und 8). Die DIN-Norm erlaubt die direkte Berechnung von χ_u aus den Querschnittsdaten, wogegen die EC2-Norm nur Begrenzungen für die plastische Gelenkrotation angibt. Da dabei angenommen wird, dass sich ein Gelenk über eine Länge von ca. 1.2-facher Plattenstärke erstreckt, kann daraus der plastische Krümmungsanteil χ_{pl} bestimmt werden, womit sich zusammen mit dem elastischen Anteil χ_{el} (ermittelt aus $m_{pl\,R}$ und EI_{II}) die Grenzkrümmung χ_u berechnen lässt. Mangels entsprechender Vorschriften wird dieses Verfahren nur für diese Normen verwendet.

Verfestigung

Das verfestigende Verhalten von Stahlbetonquerschnitten verhindert u.a. die Lokalisierung der Verformungen nach Beginn der Plastifizierung. Dadurch können sich die plastifizierten Zonen in der Platte ausbreiten (d.h. der Gelenkbereich wird breiter), was die dafür erforderlichen Krümmungen reduziert. Als willkommenen Nebeneffekt erhöhen Materialmodelle mit Verfestigung die Stabilität von numerische Berechnungen. Die positive Verfestigungszahl μ ist der einzige Berechnungsparameter, der vom Benutzer vorgegeben werden kann. Sie ist im bilinearen Materialdiagramm (s. J 2.1.1 c) als Verhältnis zwischen der Steifigkeit nach der Plastifizierung des Querschnittes und der initialen Steifigkeit EI_{II} definiert und ist konstant über die Platte. Das Verfestigungsverhalten ist als kinematische Verfestigung implementiert (s. J 2.1.3), womit sich das Verhalten im plastischen Bereich bei Belastung und Entlastung zutreffend modelliert lässt.

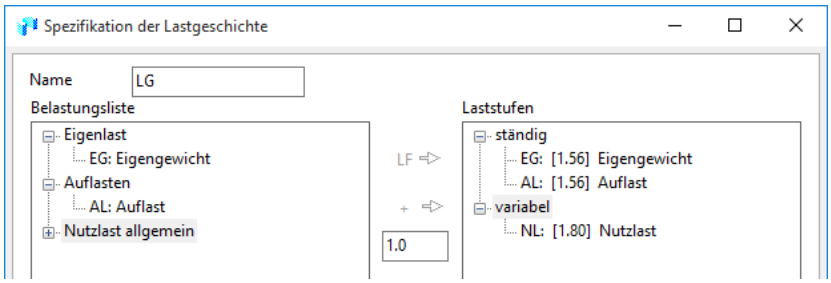
Lastgeschichte

In CEDRUS muss der Benutzer die Lastgeschichte als Serie von unterschiedlichen Laststufen spezifizieren, welche nacheinander auf die Platte aufgebracht werden und jeweils einer beliebigen Kombination von Lastfällen entsprechen. Sobald das vorgegebene Lastniveau einer Laststufe erreicht ist, wird die nächste aufgebracht, wobei der Lastfaktor für die letzte Stufe so weit wie möglich gesteigert wird. Bei Traglastberechnungen an realen Bauwerken ist man oft nicht an einem globalen Traglastfaktor für alle aufgebrachten Lasten interessiert, sondern möchte z.B. für eine bestimmte Lastanordnung, welche zusätzlich zu den ständigen Lasten wirkt, einen Sicherheitsfaktor berechnen. Dieses Problem kann mit unterschiedlichen Laststufen gelöst werden.

Teilsicherheitsbeiwerte

Das Sicherheitskonzept der modernen Normen basiert auf probabilistischen Ansätzen, welche in Form von Teilsicherheitsbeiwerten für Einwirkungen und Baustoffe in die Berechnungen einfliessen. Die Teilsicherheitsbeiwerte für die Einwirkungen sind immer bei der Lastgeschichte zu berücksichtigen. Die Teilsicherheitsbeiwerte für die Baustoffe kann der Benutzer auf zwei Arten berücksichtigen:

- **Reduktion der Baustoffkennwerte**
  Sind die Widerstandsfaktoren γ_c und γ_s für Beton und Betonstahl bei den Analyseparametern > 1.0, so resultiert ein reduzierter plastischer Querschnittswiderstand (Standardfall). Damit ist das rechnerische Kraft-Verformungsverhalten sehr weit vom realen Tragverhalten entfernt (d.h. die Struktur fliesst viel zu früh, die Verformungen sind sehr unrealistisch). Der Programmbenutzer muss in diesem Fall in der Lastgeschichte jedoch nur die Sicherheitsbeiwerte der Einwirkungen berücksichtigen, um das von den Normen geforderte Sicherheitsniveau zu erreichen.
- **Erhöhung der angreifenden Belastung**
  Werden die Widerstandsfaktoren γ_c und γ_s für Beton und Betonstahl bei den Analyseparametern alle zu 1.0 gesetzt, so resultiert ein realistischeres Kraft-Verformungsverhalten. Um das von den Normen geforderte Sicherheitsniveau zu erreichen, müssen die Teilsicherheitsbeiwerte für die Baustoffe jedoch auf der Lastseite berücksichtigt werden. Der Programmbenutzer muss in diesem Fall bei der Definition der Lastgeschichte sowohl die Teilsicherheitsbeiwerte für die Einwirkungen als auch diejenigen für die Baustoffe spezifizieren (s. Bild unten). Dies ist jedoch nur möglich, wenn γ_c und γ_s den gleichen Wert aufweisen.



☞ *Beispiel: Soll die Tragsicherheit eines durch Eigengewicht, Auflast und eine Nutzlast beanspruchten, bestehenden Tragwerkes gemäss (alter) SIA Norm 160/162 berechnet werden, so kann man die Eigenlasten und Auflasten, erhöht durch die Sicherheitsbeiwerte für ständige Einwirkungen (d.h. 1.3) und für die Materialien (d.h. 1.2) zu einer ersten Laststufe zusammenfassen. Die Nutzlast wird, erhöht durch die Beiwerte für Materialien und veränderliche Einwirkungen (d.h. 1.5), als zweite Laststufe spezifiziert. Resultiert in der anschliessenden Berechnung ein Traglastfaktor für die zweite Laststufe von mehr als 1.0, so gilt die Tragfähigkeit als nachgewiesen (s. obenstehendes Bild).*

Bei beiden Verfahren sollte (bis auf nummerisch bedingte Unterschiede) der gleiche Traglastfaktor resultieren, sofern ideal-plastisches Materialverhalten spezifiziert wurde (d.h. keine Verfestigung).

J 4.1.2 Abbruchkriterien

Wie im Kapitel J 2.3.1 erwähnt, werden bei der inkrementellen, iterativen Berechnung neben dem Kriterium der Systemstabilität zusätzliche Abbruchkriterien kontrolliert. Dies ist notwendig um sicherzustellen, dass die getroffenen Modellannahmen (s. J 4.1.1) eingehalten werden. In CEDRUS werden nach jedem Lastinkrement vier skalare Parameter berechnet, welche den akuellen Zustand des Systems anschaulich beschreiben. Jede dieser Grössen wird automatisch kontrolliert, damit die Berechnung bei Überschreitung vorgegebener Grenzwerte abgebrochen werden kann. Um dem Benutzer jederzeit einen Überblick über das Systemverhalten zu geben, wird der Verlauf dieser Systemparameter in einer Grafik aufgezeichnet. Der Benutzer hat zudem die Möglichkeit, die Kontrolle dieser Parameter zu deaktivieren und einzelne Grenzwerte zu konfigurieren. Die vier Zustandsgrössen werden in der Folge kurz beschrieben:

☞ *Da die rechnerische Traglast bei einer Berechnung mit Verfestigungszahl $\mu > 0$ ohne zusätzliches Abbruchkriterium theoretisch unendlich gross werden kann, dürfen nie alle Kontrollparameter deaktiviert werden. In der Regel empfiehlt es sich, alle Kontrollen eingeschaltet zu lassen (Vorgabe).*

Durchbiegung

Mit zunehmender Erhöhung der Belastung wird die Last-Verformungskurve (s. J 2.3.1) 'flacher' und die Durchbiegungen steigen in Traglastnähe rasant an. Um die Grenzen des Berechnungsmodelles (s. J 4.1.1) nicht zu verlassen, muss sichergestellt werden, dass die Verformungen nicht zu gross werden. Aus diesem Grund vergleicht CEDRUS den Wert der maximalen Knotendurchbiegung $w_{\max}$ mit einem vom Benutzer vorgegebenen Wert w_u. Wird dieser überschritten, so wird die Berechnung unterbrochen.

$$w_u \geq w_{\max}$$

Hier gilt es zu beachten, dass die berechnete Durchbiegung in der Regel einem oberen Grenzwert der (ohne Langzeiteffekte) zu erwartenden Werte entspricht (s. dazu J 2.3.1 'Einfluss von Ausgangszustand und Lastgeschichte'). Der Knoten der maximalen Durchbiegung wird während der Berechnung nach jedem Lastschritt zusammen mit dem Verschiebungsbild (nur in 3D-Sicht) am Bildschirm dargestellt.

Systemsteifigkeit

Die Systemsteifigkeit K_s ist ein Mass für die aktuelle Reststeifigkeit des Systems. Sie wird nach jedem Lastinkrement $\Delta\lambda$ (s. J 2.3.1) mittels folgender Formel ermittelt:

$$K_s = \frac{W(0)}{W(\lambda)} > 0 \qquad \text{mit } W(\lambda) = \frac{W_{\Delta\lambda}}{\Delta\lambda^2}$$

Dabei bedeutet $W_{\Delta\lambda}$ die Arbeit des letzten Lastinkrementes und $W(0)$ die normierte Arbeit des initialen Lastinkrementes. Am Anfang der Berechnung gilt $K_s = 100\%$. Wird die Struktur bei zunehmender Belastung 'weicher' bzw. 'steifer' , dann ist $K_s < 100\%$ bzw. $K_s > 100\%$. Beim Erreichen der Traglast liegt K_s nahe bei Null. Der Bereich zwischen dem sich abzeichnenden und dem vollständig ausgebildeten Mechanismus ist oft sehr fliessend und letzterer numerisch nicht immer genau berechenbar. Um einen konservativen Wert für die Traglast zu erhalten, muss angesichts eines geringen Wertes von K_s auf die Erschöpfung des Tragwiderstandes geschlossen werden. Deswegen bricht CEDRUS die Berechnung ab, sobald die Systemsteifigkeit unter ein Prozent fällt oder den Wert der Verfestigungszahl erreicht hat.

Rotationsbedarf

Wie in J 4.1.1 erläutert, wird im Berechnungsmodell angenommen, dass sich die Struktur duktil verhält. Im Gegensatz zu Stabtragwerken trifft diese Annahme in der Regel auf mehrachsig gespannte Stahlbetonplatten zu, da diese meist schwach bewehrt sind und eine mehrdimensionale Lastabtragung aufweisen. Der zur Umlagerung der Momente notwendige Rotationsbedarf ist i.a. viel kleiner als die vorhandene Rotationsfähigkeit. Anschliessend an eine plastische Bemessung der Bewehrung (s. J 3.1.3) kann eine Ermittlung des Rotationsbedarfs jedoch sinnvoll sein, da ausgiebig von Momentenumlagerungen Gebrauch gemacht wird.

Um einen Rotationsfähigkeitsnachweis zu führen müssen die Verformungen bekannt sein. Wie in J 2.3.1 erläutert (s. 'Einfluss von Ausgangszustand und Lastgeschichte'), lassen sich diese nur grob bestimmen, womit eine genaue Berechnung des Rotationsbedarfs unmöglich ist. Mit dem Berechnungsmodell von CEDRUS kann jedoch eine brauchbare Abschätzung erreicht werden. Die erforderlichen Rotationen werden nach jedem Lastinkrement kontrolliert, indem in allen Querschnitten die Grenzkrümmung χ_u (s. J 4.1.1) mit der mittleren Krümmung χ_m verglichen werden:

$$\chi_u \geq \chi_m = \chi_{el} + \chi_{pl}$$

Die mittlere Querschnittskrümmung χ_m wird aus der Summe des elastischen χ_{el} und plastischen Anteils χ_{pl} berechnet (s. J 2.1.3), welche aus den mittleren Elementkrümmungen in Bewehrungsrichtung bestimmt werden. Um den aktuellen Verformungsbedarf des Systems als skalare Grösse auszudrücken, ist der Rotationsbedarf K_r als Maximalwert aller Querschnitte definiert:

$$K_r = MAX(\frac{\chi_m}{\chi_u})$$

Bei $K_r < 100\%$ verfügt die Platte über ausreichendes Rotationsvermögen. Wird $K_r > 100\%$, so ist die Rotationsfähigkeit nicht gewährleistet und die Berechnung wird unterbrochen. Die plastifizierten Zonen (=Elemente) mit rechnerisch erschöpftem Rotationsvermögen werden am Bildschirm mit einer schwarzen, jene mit ausreichender Duktilität mit einer gelben Schraffur unterlegt. In jedem Element werden zudem die plastischen Hauptkrümmungen χ_{pl} als Hauptwertgrafik (rot=positiv, blau=negativ) dargestellt. Die entsprechenden numerischen Werte für χ_{pl} und das Verhältnis von χ_m zu χ_u (in%) wird bei Positionierung des Mauszeiger in die Mitte eines Elementes in der Statuszeile angeschrieben.

Durchstanzen

Beim Überschreiten der Schubtragfähigkeit der Platte kommt es zu einem spröden Versagen, wobei das Berechnungsmodell diesen Zustand nicht erfassen kann (s. J 4.1.1). Da auch die Schubspannungen aus den Verformungen berechnet werden, ist ihre exakte Bestimmung nicht möglich. Die grösste Gefährdung durch Schubversagen geht jedoch von Auflagern (Stützen, Wandenden etc.) und konzentrierten Lasten aus, welche bei einer Bemessung einem Durchstanznachweis unterzogen werden (s. CEDRUS Handbuch Kapitel 2.6.). Dieses Problem wird von CEDRUS angegangen, indem nach jedem Lastinkrement die vom Benutzer vorgängig generierten Durchstanzobjekte kontrolliert werden. Die Kontrolle beschränkt sich dabei nur auf diejenigen Durchstanzobjekte, welche mit Auflagern verknüpft sind. Hier gilt es zu beachten, dass die berechneten Auflagerreaktionen für eine bestimmte Belastung zwischen einer elastischen Lösung (Grundmodul) und einer nichtlinearen Lösung (Traglastmodul) grosse Unterschiede aufweisen können. Um die aktuelle Gefährdung durch Schubversagen mit einem skalaren Parameter auszudrücken, ist der Durchstanzparameter K_p als Maximum des Verhältnisses zwischen der aktuellen Durchstanzlast V und dem zugehörigen Widerstand V_R aller Durchstanzobjekte definiert:

$$K_p = MAX(\frac{V}{V_R})$$

Wie in J 4.1.1 (Lastgeschichte) erörtert, sollten alle Sicherheitsbeiwerte (Material- und Lastbeiwerte) bei der Definition der Belastung berücksichtigt werden. Aus diesem Grund wird V_R für alle Normen wiefolgt berechnet:

$V_R = \gamma_c \; V_{Rd}$ $\qquad\qquad$ V_{Rd} = Widerstand aus Durchstanznachweis

Die Platte verfügt über einen ausreichenden Durchstanzwiderstand, falls $K_p < 100\%$. Wird $K_p > 100\%$, so besteht die Gefahr eines spröden Schubversagens und die Berechnung wird unterbrochen. Jedes kontrollierte Durchstanzobjekt wird am Bildschirm als Kreisscheibe dargestellt, welche vor dem Erreichen des Widerstandes V_{Rd} grün und danach rot gezeichnet wird, wobei das Verhältnis von V zu V_{Rd} ihre Grösse bestimmt. Durch Positionieren des Mauszeigers auf diese Grafik werden die numerischen Werte für V und das Verhältnis von V zu V_{Rd} in der Statuszeile angeschrieben.

✳ **Astuce:** *Durchstanzobjekte für konzentrierte Lasten, welche vom Traglastmodul nicht kontrolliert werden, lassen sich ohne nichtlineare Berechnung direkt aufgrund der aufgebrachten Gesamtlast überprüfen.*

J 4.1.3 Berechnungssteuerung

Eine nichtlineare Analyse stellt betreffend Modellierung, Berechnungssteuerung und Resultatinterpretation höhere Anforderungen an die Ingenieure als eine linear elastische Berechnung. Wie im Abschnitt J 4.1.1 ausgeführt, ist der zusätzliche Modellierungsaufwand (nach einer elastischen Berechnung) mit CEDRUS auf die Eingabe der Bewehrung, der Belastung und einer Verfestigungszahl beschränkt. Um dem Programmanwender auch die nichtlineare Berechnung soweit wie möglich zu vereinfachen, wurde eine automatische Berechnungssteuerung implementiert, welche ohne Benutzereingabe auskommt. Damit kann sich der Ingenieur auf die Interpretation und Verifikation der Berechnung konzentrieren, wobei ihm CEDRUS dafür Unterstützung bietet. Der Benutzer kann das Strukturverhalten während der Berechnung direkt am Bildschirm mitverfolgen. Nach jedem Lastinkrement werden die Strukturverformungen und die plastifizierten Zonen (inkl. Risskonzentrationen) grafisch dargestellt und die Verlaufsgrafik der Steuerungsparameter (siehe oben) aufgezeichnet. Damit ist der aktuelle Systemzustand und das allgemeine Strukturverhalten leicht erkennbar. Die Berechnung kann jederzeit unterbrochen werden, z.B. falls das Systemverhalten dem Ingenieur nicht plausibel erscheint oder die Berechnung sehr lange dauert. Der erforderliche Rechenaufwand für eine inkrementelle, iterative Berechnung liegt nämlich weit höher als für eine elastische Berechnung. Je nach vorhandener Hardware, untersuchtem Problem (Systemgrösse, Bewehrungsanordnung, Belastung etc.) und verwendetem FE-Modell (Masche) kann die Rechenzeit zur Ermittlung der Traglast sehr gross werden.

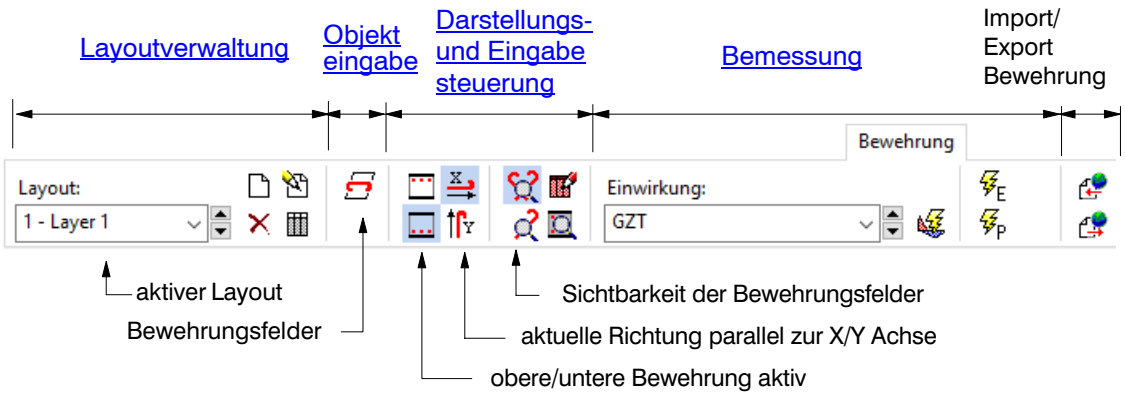
☞ *Die maximale Grösse des FE-Modelles hängt nur vom zur Verfügung stehenden Arbeitsspeicher des Computers ab. Da jedoch die erforderliche Rechenzeit mit zunehmender Anzahl Knoten und Elemente sehr stark steigt, ist es empfehlenswert, eine allzufeine Elementeinteilung zu vermeiden.*

J 5 Kontexthilfe

J 5.1 Das Registerblatt /Bewehrung/ (Option)

Im Registerblatt /Bewehrung/ werden die Daten der schlaffen Bewehrung und die Schalungsgeometrie verwaltet. Hier kann die Bewehrung in Form von Stab- und Mattenlagen eingegeben, bemessen und bearbeitet werden.

Hinweis: Für eine Einführung in die theoretischen Grundlagen und die Funktionsweise des Bewehrungsmoduls sie hier auf das Handbuch des Bewehrungsmoduls verwiesen.



Um Variantenvergleiche zu ermöglichen, wird die Bewehrung einer Platte in voneinander unabhängigen **Layouts** verwaltet, wobei jeweils nur einer aktiv sein kann.

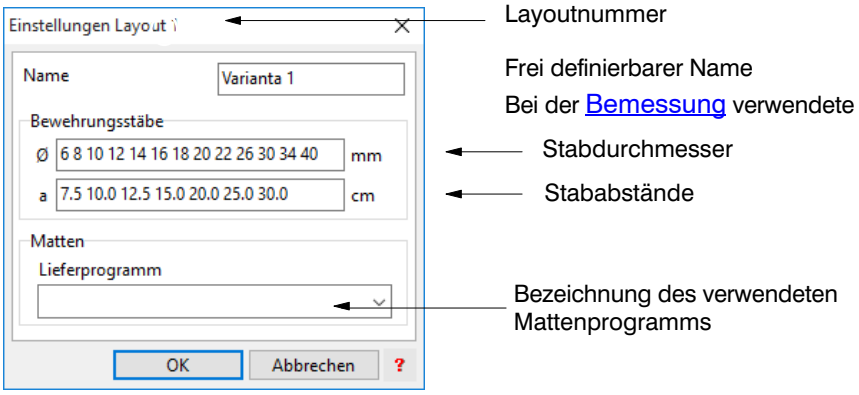
Mit den Schaltern oder wird der aktive Layout ausgewählt. Die Bewehrungsdaten sind pro Layout in einem separaten Layer gespeichert, womit über die Layerschaltflächen auch inaktive Layouts dargestellt werden können.

J 5.1.1 Layoutverwaltung

Zur Verwaltung der Layouts sind folgende Schaltflächen im Registerblatt /Bewehrung/ vorhanden:

Neuen Layout erzeugen

Ein neuer Layout wird durch Drücken dieser Schaltfläche erzeugt, womit der Dialog /Einstellungen Layout/ erscheint.



Jedem neuen Layout wird dabei eine eindeutige Nummer zugeordnet, welche im Layerschalter angezeigt wird. Bei der Wahl des Mattenprogrammes stehen alle im Projekt- (User) und Programmverzeichnis (Cedrus\Platon) gespeicherten Mattendateien zur Auswahl (*Norm_*.RNL, z.B. SIA_artec.RNL) .

Layout löschen

Mit diesem Befehl kann der aktive Layout gelöscht werden.

Layouteinstellungen ändern

Zur nachträglichen Änderung der Layouteinstellungen kann mit diesem Befehl des Dialogfenster /Einstellungen Layout/ dargestellt werden.

Liste der Bewehrungsfelder

Dieser Befehl erstellt eine Liste aller Bewehrungsfelder des aktuellen Layouts..

J 5.1.2 Eingabe der Bewehrungsobjekte

Bewehrungsfeld (Schaltfläche im Registerblatt /Bewehrung/)

Die Bewehrung wird in Form von frei definierbaren Feldern spezifiziert, welche sich gegenseitig überlappen dürfen (s. auch Grundlagen: Bewehrungmodell). Diese Bewehrungsfelder sind statisch aktive Zonen mit vorgegebener Stabrichtung und konstantem Bewehrungsgehalt. Sie entsprechen einzelnen Stab- oder Mattenlagen. Die Felder können als Parallelogramme oder Polygone eingeben werden. Sobald Sie auf diese Schaltfläche klicken erscheint das Dialogfenster Bewehrungsfeld, wo Sie die Eigenschaften der Felder definieren können.

Hinweis: Bei der Eingabe eines neuen Bewehrungsfeldes entspricht die Stabrichtung der gerade eingestellen Richtung (siehe Eingabeoptionen>Koordinaten) der Koordinatenachse X oder Y (je nach Zustand der nachfolgend beschriebenen Schaltflächen zur Darstellungs- und Eingabesteuerung).

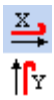
J 5.1.3 Darstellungs- und Eingabesteuerung

Die Eingabe und Darstellung der Bewehrungsfelder wird über folgende Schaltflächen im Registerblatt /Bewehrung/ gesteuert:

Obere/Untere Bewehrung

(Schaltfläche im Registerblatt /Bewehrung/)

Die Verwaltung der Bewehrungsfelder erfolgt getrennt für die oberen und unteren Bewehrungslagen. Mit diesen Schaltflächen kann zwischen den beiden Seiten hin und her geschalten werden, wobei allenfalls vorhandene, lageabhängige Bemessungsresultate ebenfalls angepasst werden.



Aktuelle Bewehrungsrichtung

(Schaltfläche im Registerblatt /Bewehrung/)

Diese Schaltflächen bestimmen die aktuelle Bewehrungsrichtung, welche der Koordinatenachse X bzw. Y entspricht (siehe Dialog Eingabeoptionen>Koordinaten). Die aktuelle Richtung definiert die Stabrichtung für ein neues Bewehrungsfeld und steuert die Sichtbarkeit der vorhandenen Felder.

Sichtbarkeit der Bewehrungsfelder

(Schaltfläche im Registerblatt /Bewehrung/)

Sichtbar sind diejenigen Felder, deren Stabrichtung mit der aktuellen Bewehrungsrichtung innerhalb eines Toleranzwinkels übereinstimmen. Ist die obere Schaltfläche gedrückt, so werden zusätzlich alle Felder dargestellt, welche senkrecht zur aktuellen Richtung stehen. Diese Schaltfläche bestimmt zudem die Darstellung der Bemessungsresultate.

Durch Klicken auf diese Schaltfläche öffnet sich ein Dialogfenster mit Darstellungsoptionen.

Falls diese Schaltfläche gedrückt ist, werden alle Felder der gegenüberliegenden Seite (d.h. oben bzw. unten) hellgrau gezeichnet. Diese sind im Grafikeditor fangbar, aber nicht selektierbar.

J 5.1.4 Das Dialogfenster Darstellung

Die Darstellung der Bewehrungsfelder wird durch diesen Dialog gesteuert. Die Einstellungen gelten dabei für den aktuellen Layout. Das Dialogfenster verfügt über die Registerblätter Bewehrung und Resultate.

Die Felder des Registerblattes *Bewehrung*:

Toleranzwinkel

Sichtbar sind diejenigen Felder, deren Stabrichtung mit der aktuellen Bewehrungsrichtung innerhalb des hier definierten Tolwinkels übereinstimmen.

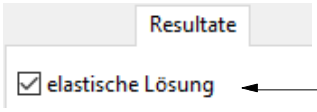
Feldbeschriftung

Die Bewehrungsfelder können ausführlich, kurz oder gar nicht beschriftet werden. Die Grösse der Kurzbeschriftung ist konstant und entspricht der Einstellung des Grafikeditors (**Menu Optionen>Schriftgrösse**). Die Schriftgrösse bei ausführlicher Beschriftung ändert sich je nach Anzeigemassstab. Sie wird in diesem Dialogfenster via Texthöhe und entsprechendem Massstab spezifiziert.

Die Felder des Registerblattes *Resultate* (Bemessung):
(in Dialogfenster Darstellung)

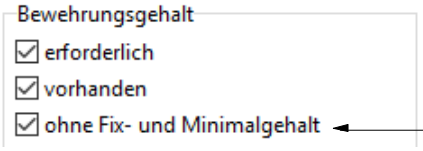
Die Resultate der Bemessung (z.B. erforderlicher Bewehrungsbedarf) werden je nach Wert als Farbverteilungen, Pfeile, Säulen oder als Kreuze elementweise dargestellt. Ihre

Präsentation wird durch dieses Registerblatt gesteuert.



dargestellt wird immer nur die elastische Lösung

Darstellungsoptionen Bewehrungsgehalt



dargestellt wird nur der variable, über das spezifizierte Minimum hinausgehende Bewehrungsgehalt

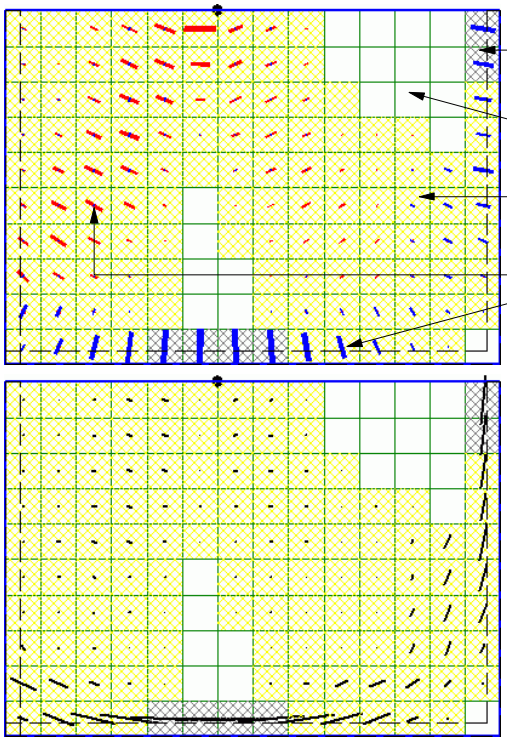
Die Felder des Registerblattes *Resultate* (Analyse):
(in Dialogfenster Darstellung)

Plastifizierung

Trajektoriendarstellung der plastischen Hauptkrümmungen (blau negativ, rot positiv)

Wahrscheinlichkeit von grossen Rissen (Line = Rissrichtung)

Resultatdarstellung im Traglastmodul:



grau:
hohe plastische Rotationen

weiss:
ohne plastische Verformung

gelb:
mit plastischer Verformung

rote Linie=positive
blaue Linie=negative
plastische Hauptkrümmungen

Rissekonzentrationen
(Linien = Rissrichtung)

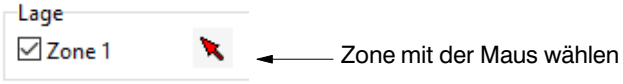
J 5.1.5 Das Dialogfenster Bewehrungsfeld

Das Dialogfenster verfügt über drei Registerblätter: Das erste umfasst die allgemeinen Attribute, das zweite enthält die bemessungsspezifischen Einstellungen und das dritte beherbergt zusätzliche Optionen.

Die Felder des Registerblattes *Attribute*:

Zuweisung zu Materialzone oder Unterzug

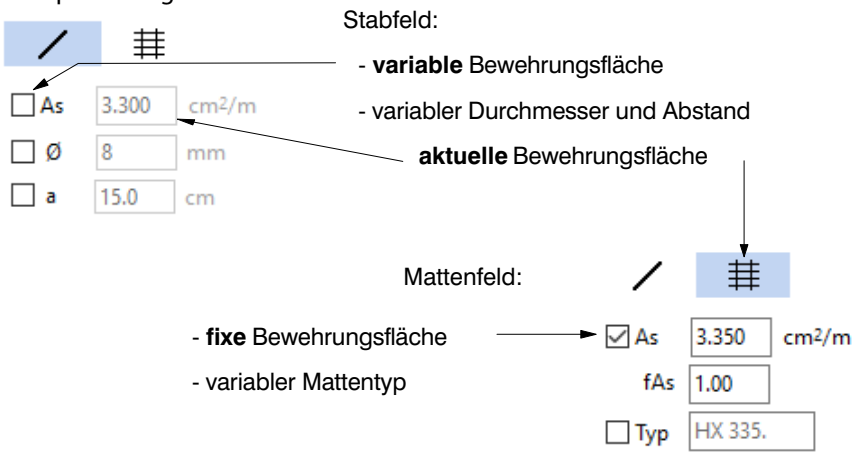
Jedes Bewehrungsfeld kann einer Materialzone oder einem



Unterzug zugewiesen werden, womit seine vertikale Position durch deren Eigenschaften (d.h. Dicke, Überdeckung etc.) eindeutig festgelegt ist. Alternativ dazu kann die vertikale Lage im Registerblatt Optionen unabhängig von einer Zone definiert werden.

Feldtyp (in Dialogfenster Bewehrungsfeld)

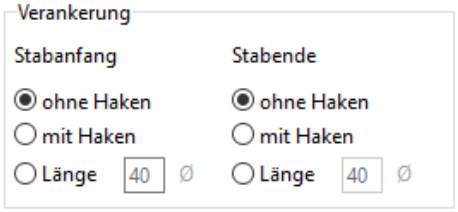
Ein Bewehrungsfeld kann als Stab- oder Mattenfeld spezifiziert werden. Ein Stabfeld erzeugt Bewehrungswiderstand nur in Stabrichtung, während ein Mattenfeld senkrecht dazu ebenfalls einen Widerstand aufweist, welcher durch den Faktor **fAs** zum Widerstand in Hauptrichtung definiert ist.



Ist der Bewehrungswiderstand **As** eines Feldes als variabel definiert, so wird bei einer Bemessung ein optimaler Wert dafür bestimmt. Soll jedoch der Bewehrungswiderstand vorgegeben werden, so kann dies über eine fixe Bewehrungsfläche, einen fixen Stabdurchmesser mit festem -abstand oder einen definierten Mattentyp geschehen.

Verankerung (in Dialogfenster Bewehrungsfeld)

Die Spezifikation der Verankerung am Stabanfang und -ende wird nicht in die Berechnung



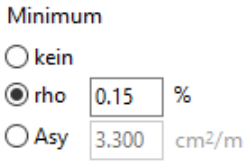
einbezogen, da ein Bewehrungsfeld über seine gesamte Ausdehung einer statisch aktive Zonen mit konstantem Bewehrungsgehalt entspricht. Diese Angaben dienen einzig als Illustration der vom Ingenieur gewünschten Verankerungsbedingungen.

Die Felder des Registerblattes *Bemessung*:

In diesem Registerblatt sind die bemessungspezifischen Attribute zusammengefasst.

Minimalgehalt (in Dialogfenster Bewehrungsfeld)

Neben den Eingaben zum Feldtyp (analog Registerblatt Attribute) kann jedem Feld ein minimaler Bewehrungsgehalt (z.B. aus Gründen der Gebrauchstauglichkeit) zugeordnet werden, welcher bei einer Bemessung nicht unterschritten wird.



Hinweis: Über das Bearbeiten-Kontextmenü (drücken rechte Maustaste innerhalb Grafikfläche) kann die aktuelle Bewehrungsfläche Asy von ausgewählten Bewehrungsfeldern als Minimum übernommen werden. Dies kann nützlich sein, um den für Gebrauchslasten elastisch bemessenen Bewehrungsgehalt in der anschliessenden plastischen Bemessungsoptimierung auf Tragsicherheitsniveau nicht zu unterschreiten.

Die Felder des Registerblattes *Optionen*:

Lage (in Dialogfenster Bewehrungsfeld)

Alternativ zur Zuweisung zu einer Materialzone oder zu einem Unterzug kann die vertikale Position eines Feldes über die Eingabefelder Oberkante, Plattendicke und Randabstände definiert werden.

Hinweis: Dabei gilt es jedoch zu beachten, dass die Lage des Bewehrungsfeldes bei einer Änderung der Materialeigenschaften (z.B. Plattendicke) nicht mehr automatisch angepasst werden kann.

Gruppe (in Dialogfenster Bewehrungsfeld)

Mehrere Felder können einer **Gruppe** zugeordnet werden, indem die Gruppenzuordnung aktiviert und die gewünschte Gruppennummer eingegeben wird. Alle Felder einer Gruppe

weisen (unabhängig von ihrer Bewehrungsrichtung) den selben Bewehrungsgehalt auf, womit es z.B. im Rahmen einer Bemessung möglich ist aufgebogene Eisen oder einen Mattentyp für mehrere Plattenzonen zu definieren.

Hinweis: Bei einer Bemessung wird für eine Gruppe von Feldern der grösste, spezifizierte minimale und fixe Bewehrungsgehalt massgebend.

Darstellung der Feldbeschriftung (in Dialogfenster Bewehrungsfeld)

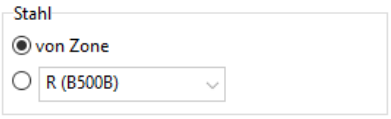
Das Aussehen der Beschriftung ist im Dialogfenster Darstellung layoutweise definiert. Po-

sition und Richtung der Beschriftung wird bei Aktivierung des entsprechenden Schalters automatisch festgelegt. Ist der Schalter deaktiviert, so kann die Schriftrichtung als Winkel zur X-Achse angegeben werden.

Stahleigenschaften (in Dialogfenster Bewehrungsfeld)

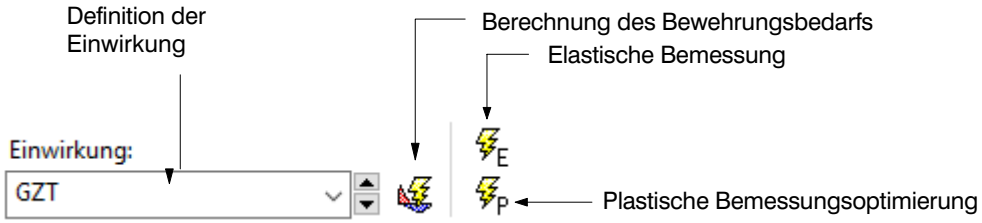
Die Stahlqualität kann entweder von der Vorgabe, d.h. vom Baustoff Bewehrung der Ma-

terialzone, übernommen oder hier separat spezifiziert werden.



J 5.1.6 Bemessung

Ist eine Lizenz für den (optionalen) Bewehrungsmodul von CEDRUS vorhanden, so ermöglicht das Registerblatt /Bewehrung/ eine elastische und/oder plastische Bemessung der Bewehrung. Die Bemessung wird durch die unten gezeigten Schaltflächen gesteuert:



Ablauf einer Bemessung

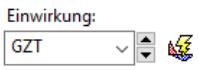
Eine Bemessung erfolgt grundsätzlich in fünf Schritten:

- 1. Definition der Einwirkung
- 2. Berechnung des Bewehrunsgbedarfs
- 3. Eingabe der Bewehrungsfelder
- 4. Elastische Bemessung
- 5. Plastische Bemessungsoptimierung

Basisdaten einer Bemessung

Die Bemessung basiert auf teilweise normenabhängigen Basisdaten, welche pro Projekt bzw pro Layout definiert werden:

- Betoneigenschaften (Festigkeit, Widerstandsbeiwert etc.)
  normenabhängig, aus Berechnungseinstellungen>Norm
- Stahleigenschaften (Festigkeit, Widerstandsbeiwert etc.)
  normenabhängig, aus Berechnungseinstellungen>Norm oder direkt aus Dialog Bewehrungsfeld
- Verwendeter Rundstahl (Abstände, Durchmesser)
  aus Layouteinstellungen
- verwendete Matten
  aus Layouteinstellungen



1. Definition der Einwirkung

Der aktuelle Layout wird für die hier angegebene Grenzwertspezifikation bemessen. Zur Auswahl stehen alle automatisch und manuell erzeugten Spezifikationen (siehe Registerblatt /Berechnung/).

2. Berechnung Bewehrungsbedarf

Mit der Betätigung dieser Schaltfläche werden die Bemessungsgrenzwerte der Beanspruchung für die angegebene Einwirkung neu berechnet und der dafür notwendige, elastische Bewehrungsbedarf elementweise grafisch dargestellt.

Hinweis: Allfällig vorhandene elastische oder plastische Elementresultate gehen verloren, während die Bewehrungsgehalte der Felder unverändert bleiben.

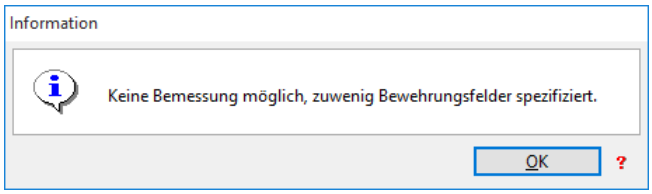
3. Eingabe der Bewehrungsfelder (s. auch Grundlagen: Bewehrungsmodell)

Die Eingabe der Bewehrung erfolgt über die Objekteingabe Bewehrungsfeld. Ist eine plastische Bemessungsoptimierung vorgesehen, so sollten hier neben statischen unbedingt auch konstruktive Aspekte berücksichtig werden.

4. Elastische Bemessung
(im Registerblatt /Bewehrung/)

Mit diesem Befehl werden die vorgängig spezifizierten Bewehrungsfelder elastisch bemessen, d.h. deren (unbekannter) Bewehrungsgehalt so gewählt, dass der elastische Bewehrungsbedarf überall abgedeckt wird. Da sich Felder gegenseitig überlappen können ist dies i.a. eine Optimierungsaufgabe, welche für minimales Bewehrungsgewicht gelöst wird (s. Bewehrungmodul: Grundkonzepte).

Eine elastische Bemessung ist nicht in jedem Fall möglich, wobei eine solche Situation durch folgende Meldung angezeigt wird:



Keine gültige Lösung resultiert, falls beanspruchte Plattenzonen existieren, wo in einer Richtung

- keine Bewehrungsfelder spezifiziert oder
- nur Felder mit fixem Gehalt definiert sind und deren Gesamtwiderstand kleiner ist als die Beanspruchung

Da bei einer Platte mehrere Bewehrungsrichtungen und -lagen exisiteren, werden diese problematischen Zonen grafisch hervorgehoben.

Hinweis: Bereits vorhandene elastische oder plastische Resultate gehen bei dieser Aktion verloren.

5. Plastische Bemessungsoptimierung
(im Registerblatt /Bewehrung/)

Nach erfolgter, erfolgreicher, elastischer Bemessung kann die **Bewehrung optimiert** werden, indem von der plastischen Schnittkraftumlagerung Gebrauch gemacht wird. Diese automatische Bemessungsoptimierung versucht durch geschickte Momentenumlagerungen den Widerstand der eingelegten Bewehrung möglichst gut auszunützen und dabei das Gesamtgewicht der Bewehrung zu minimieren (s. Bewehrungsmodul: Grundkonzepte).

Mit der Betätigung dieser Schaltfläche öffnet sich das Monitorfenster, wo die Bemessungsoptimierung gestartet und jederzeit vom Benutzer abgebrochen werden kann.

Damit können **unrealistische Bewehrungskonzentrationen** aus der elastischen Lösung (z.B. bei einspringenden Ecken, über Punktstützen oder Lastangriffspunkten etc.) physikalisch korrekt abgebaut und die Tragwiderstände der aus konstruktiven Gründen ohnehin notwendigen Bewehrungslagen (z.B. zwecks Rissesicherung) aktiviert werden.
Je nach Problem lässt sich dadurch gegenüber einer elastischen Bemessung eine **Bewehrungsersparnis von bis zu 30%** erzielen.

Hinweis: Eine Bemessungsoptimierung ist nur möglich, falls mindestens ein Bewehrungsfeld mit variabler Bewehrungsfläche vorhanden ist.

J 5.1.7 Darstellung der Resultate (im Registerblatt /Bewehrung/)

Darstellung der Bewehrungsfelder

Je nach Feldtyp und Feldattributen (z.B. Minimalgehalt) werden die Bewehrunsgfelder anders dargestellt.

Die **Feldbeschriftung** wird durch den Darstellungsdialog gesteuert. Aus dem Beschriftungstext und der Schriftart sind die wichtigsten Feldattribute ersichtlich:

Felder mit kurzer Beschriftung:

4.24	**variabler** Bewehrungsgehalt (aktueller Wert = 4.24)
4.24	**fixer** Bewehrungsgehalt durch Vorgabe von As (4.24)
ø6/15.0	**fixer** Bewehrungsgehalt durch Vorgabe von Abstand (15.0) und Durchmesser (6)
4.24	Feld mit minimalem Gehalt
4.24(ø6)	Feld mit variablem Gehalt aber fixem Durchmesser
4.24 [1]	Feld gehört zur Gruppe 1

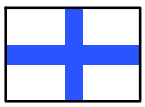
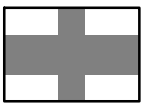
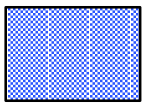
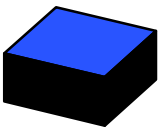
Elementweise Darstellung der Resultate

Die Resultate der Bemessung werden elementweise gezeichnet, wobei die **Darstellung** von folgenden Einstellungen abhängt:

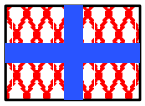
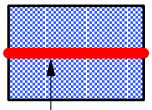
- Steuerungsschaltflächen im Registerblatt /Bewehrung/
- Dialogfenster /Darstellung/

Hier eine **Zusammenfassung der Darstellungsarten** des Bewehrungsgehaltes anhand

eines einzelnen Finiten Elementes:

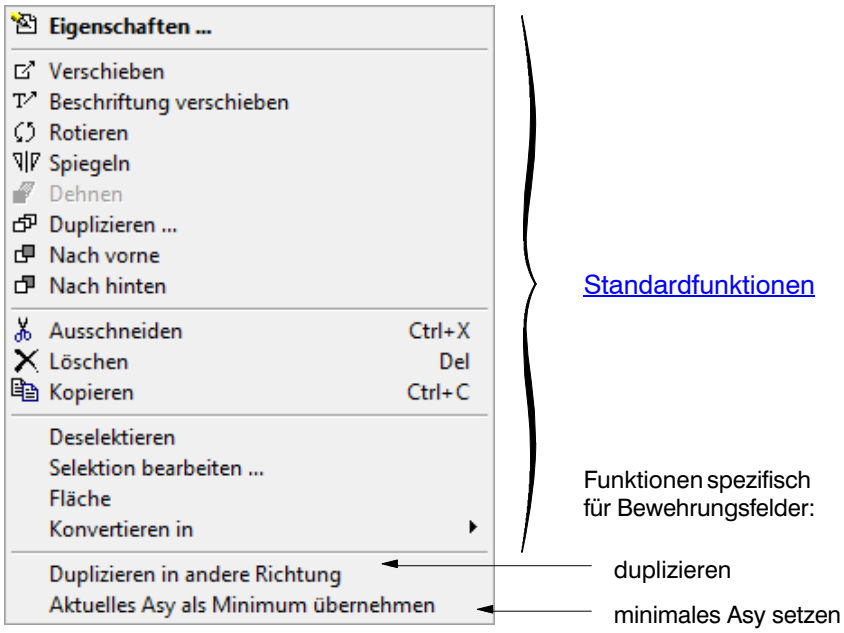
	Bewehrungsgehalt		
	erforderlich		vorhanden
	2D	3D	2D
zwei Richtungen	 *Strichdicke*	–	 *Strichdicke*
eine Richtung	 *Farbintensität*	 *Säulenhöhe*	–

Durch Resultatdarstellung wird in jedem einzelnen Finiten Element ersichtlich, falls **keine gültige Lösing** bei der elastische Bemessung resultierte:

zwei Richtungen 	eine Richtung  Richtung mit zuwenig Bewehrung

J 5.1.8 Das Bearbeiten-Kontextmenü (Bewehrungfelder)

Wird die rechte Maustaste bei selektierten Bewehrungsfeldern gedrückt, so erscheint das



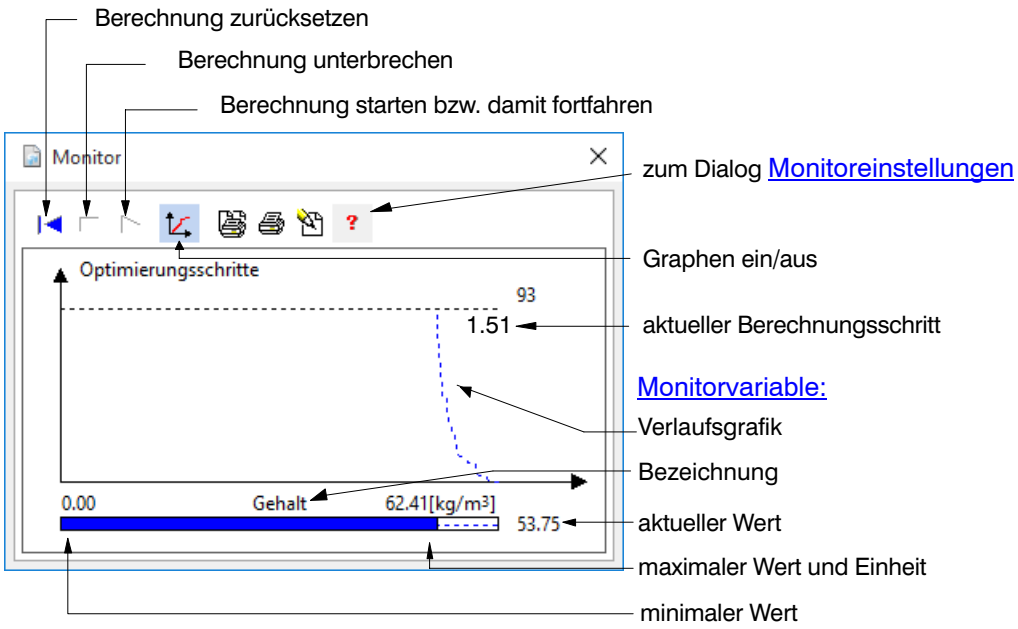
Bearbeiten-Kontextmenü. Dieses Menü entspricht dem Standardmenü mit zwei zusätzlichen Funktionen:

Mit dem Befehl **'Duplizieren in andere Richtung'** werden alle selektierten Stabfelder dupliziert und die Bewehrungsrichtung mit der Verlegerichtung vertauscht.

Der Befehl **'Aktuelles Asy als Minimum übernehmen'** definiert für alle selektierten Felder mit variablem Bewehrungsgehalt einen minimalen Wert für Asy in der Grösse der aktuellen Bewehrungsfläche, welche z.B. aus einer elastischen Bemessung hervorging.

J 5.2 Das Dialogfenster Monitor

Das Dialogfenster Monitor dient zur Steuerung und Dokumentation der inkrementellen Berechnungen bei der plastischen Bemessungsoptimierung und der Traglastanalyse.



J 5.2.1 Monitorvariablen

Eine Monitorvariable ist ein Zahlenwert (z.B. der aktuelle Lastfaktor), der sich in jedem Berechnungsschritt ändern kann und dessen Verlauf im Monitor gezeigt wird. Die Darstellung kann dabei im Dialog Monitoreinstellungen beeinflusst werden.

- Monitorvariablen für Bemessungsoptimierung
- Monitorvariablen für Traglastanalyse

Monitorvariablen Bemessungsoptimierung

(im Dialogfenster /Monitor/)

Folgende Bemessungsresultate werden nach jedem Optimierungsschritt für die gesamte Platte berechnet:

- **Gewicht** = Gesamtgewicht der Bewehrung, berechnet aufgrund der aktuell erforderlichen Bewehrungsflächen (ohne Zusatzbewehrung für Umwandlung in Stabdurchmesser und Abstand bzw. Mattentyp)
- **Gehalt** = **Gewicht** geteilt durch Betonvolumen, welches aus der rechnerischen Strukturgeometrie (nicht Schalungsgeometrie!) ermittelt wird
- **Effizienz** = Volumen der angegeben Minimal- und Fixbewehrung plus die (aktuell) statisch darüberhinaus erforderliche Bewehrung über dem Volumen der (aktuell) bemessenen Bewehrung (siehe Effizienzparameter).

Folgende Zustandswerte der Optimierung sind im Monitor verfügbar:

- **Tableaugrösse** = Grösse der Optimierungsmatrix im Hauptspeicher
- **Iterationen** = Anzahl Umtauschschritte für letztes Optimierungsinkrement

Monitorvariablen Traglastanalyse

(im Dialogfenster /Monitor/)

Folgende Resultate werden nach jedem abgeschlossenen Lastschritt berechnet:

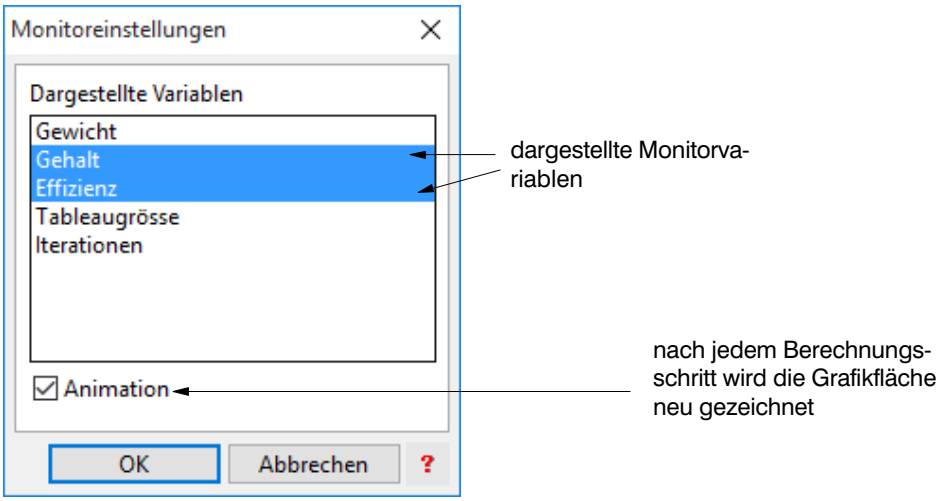
- **Steifigkeit** = Parameter der vorhandenen Systemsteifigkeit: 100% = Anfangssteifigkeit, nahe bei 0% = Instabilität (s. Traglastmodul: Grundkonzepte).
- **Durchbiegung** = vertikale Einsenkung des Knotens mit dem maximalen Wert (absolut). Vergleich mit der vorgegebenen Maximaldurchbiegung.
- **Rotationsfähigkeit** = maximales Verhältnis der aktuellen, mittleren Krümmung zur Grenzkrümmung der Querschnitte aller Elemente (s.Traglastmodul: Grundkonzepte): >100% = Rotationsvermögen erschöpft
- **Durchstanzen** = Maximum des Verhältnisses zwischen aktueller Durchstanzlast und dem zugehörigen Widerstand aller Durchstanzobjekte (s. Traglastmodul: Grundkonzepte): >100% = Durchstanzwiderstand erschöpft

Folgender Zustandswerte der Analyse ist im Monitor verfügbar:

- **Iterationen** = Anzahl Gleichgewichtsiterationen für den letzten Lastschritt

J 5.2.2 Das Dialogfenster Monitoreinstellungen

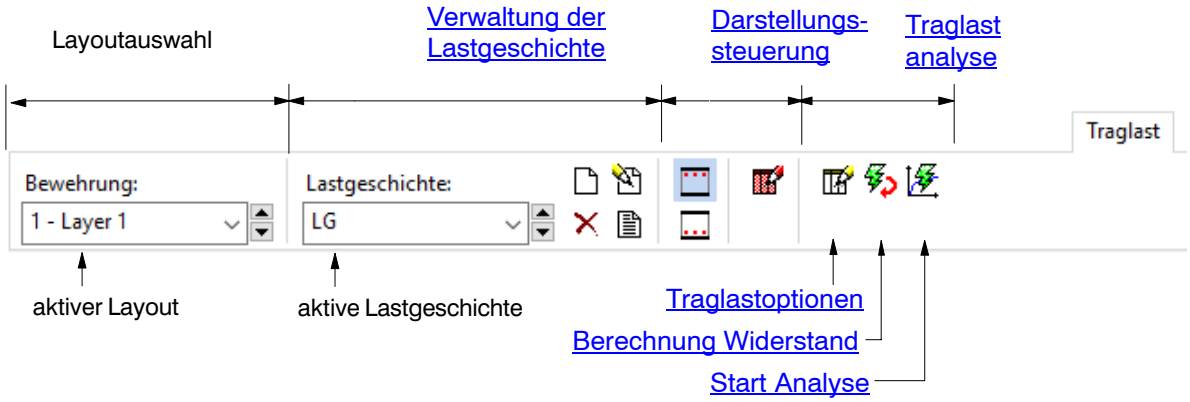
Das Dialogfenster Monitoreinstellungen steuert die Darstellung des Berechnungsmonitors.



J 5.3 Das Registerblatt /Traglast/ (Option)

Das Registerblatt /Traglast/ dient zur Analyse der Tragfähigkeit einer Platte, wobei die vorhandene Bewehrung vorgängig im Registerblatt /Bewehrung/ konstruiert werden muss.

Hinweis: Für eine Einführung in die theoretischen Grundlagen und die Funktionsweise des Bewehrungsmoduls sie hier auf das Handbuch des Traglastmoduls verwiesen.



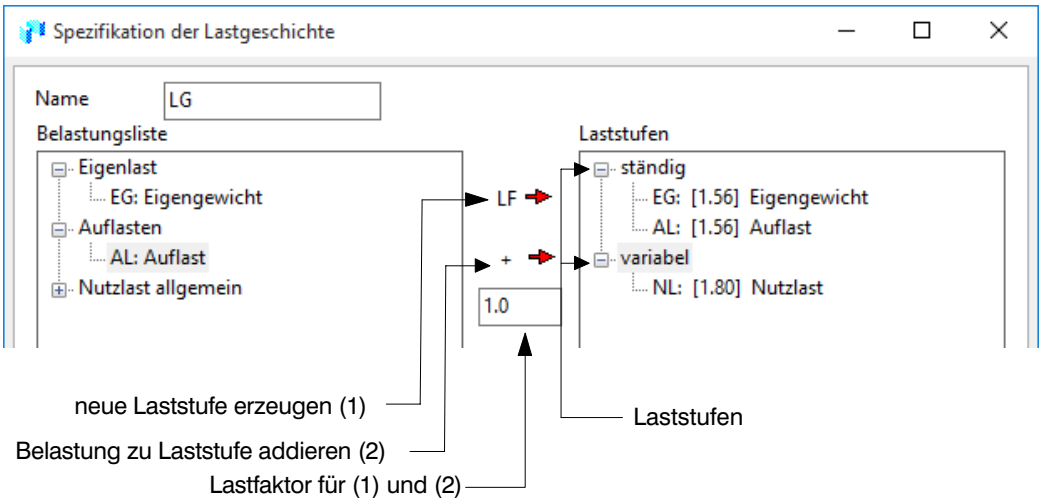
Mit den Schaltern oder wird der zur Traglastanalyse verwendete Bewehrungslayout ausgewählt.

J 5.3.1 Verwaltung der Lastgeschichte

Zur Verwaltung der Lastgeschichte sind folgende Schaltflächen im Registerblatt /Traglast/ vorhanden:

Neue Lastgeschichte erzeugen

Eine neue Lastgeschichte wird durch Drücken dieser Schaltfläche erzeugt, womit der Dialog /Lastgeschichte/ erscheint.



Dort kann die Lastgeschichte in Form von beliebig vielen, frei definierbaren **Laststufen** spezifiziert werden, welche freie Kombinationen der vorhandenen Belastungen sind. Bei der Traglastanalyse wird die Struktur mit diesen Laststufen belastet.

***Hinweis**: Nähere Erläuterungen zur Definition der Lastgeschichte finden Sie in den Abschnitten 'Lastgeschichte' und 'Teilsicherheitsbeiwerte' des Handbuches.*

Lastgeschichte löschen

Mit diesem Befehl kann die aktive Lastgeschichte gelöscht werden.

Lastgeschichte ändern

Dieser Befehl dient zur nachträglichen Änderung der aktiven Lastgeschichte.

J 5.3.2 Darstellungssteuerung

Die Darstellung der Bewehrung und der Analyseresultate wird über folgende Schaltflächen im Registerblatt /Traglast/ gesteuert:

Obere/Untere Bewehrung

(Schaltfläche im Registerblatt /Traglast/)

Die Darstellung der Bewehrungsfelder erfolgt getrennt für die oberen und unteren Bewehrungslagen. Mit diesen Schaltflächen kann zwischen den beiden Seiten hin und her geschalten werden.

Optionen

Durch Klicken auf diese Schaltfläche öffnet sich das Dialogfenster mit Darstellungsoptionen.

J 5.3.3 Traglastanalyse (im Registerblatt /Traglast/)

Die Traglastanalyse erfolgt durch eine nichtlineare, inkrementelle Finite Element Berechnung, wo eine vorgegebene Struktur (inkl. Bewehrung) schrittweise belastet wird, bis das System instabil und die Analyse abgebrochen wird. Diese inkrementelle Berechnung kann auch aufgrund anderer Kritertien gestoppt werden, welche im Dialogfenster /Traglastoptionen/ konfiguriert werden.

Ablauf einer Traglastanalyse

Eine Analyse erfolgt grundsätzlich in vier Schritten:

- 1. Eingabe der Bewehrungsfelder
  (im Registerblatt /Bewehrung/)
- 2. Definition der Lastgeschichte
- 3. Berechnung des Widerstandes
- 4. Bestimmung des Traglastfaktors

Basisdaten einer Traglastanalyse

Die Analyse basiert auf teilweise normenabhängigen Basisdaten, welche pro Projekt bzw pro Layout definiert werden:

- Betoneigenschaften (Festigkeit, Widerstandsbeiwert etc.)
  normenabhängig, aus Berechnungseinstellungen>Norm
- Stahleigenschaften (Festigkeit, Widerstandsbeiwert etc.)
  normenabhängig, aus Berechnungseinstellungen>Norm oder direkt aus Dialog Bewehrungsfeld

Traglastoptionen

(im Registerblatt /Traglast/)

Mit der Betätigung dieser Schaltfläche erscheint das Dialogfenster /Traglastoptionen/.

Berechnung des Widerstandes

(im Registerblatt /Traglast/)

Mit der Betätigung dieser Schaltfläche wird der Bewehrungswiderstand (plastisches Moment) des gewählten Layouts in jedem Punkt der Platte berechnet (s. Traglastmodul: Grundkonzepte).

Bestimmung des Traglastfaktors

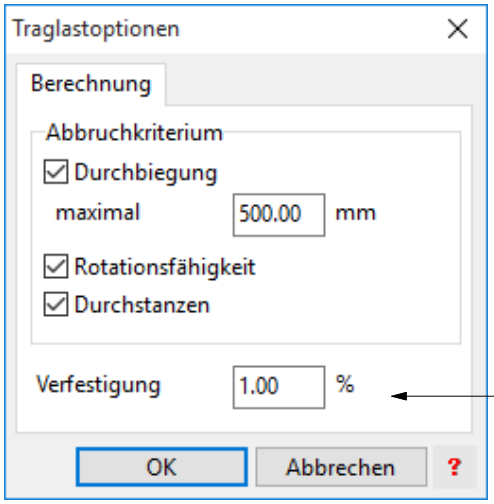
(im Registerblatt /Traglast/)

Nach erfolgter Berechnung des Bewehrungswiderstandes kann die nichtlineare, inkrementelle Traglastanalyse gestartet werden. Mit der Betätigung dieser Schaltfläche öffnet sich das Monitorfenster, wo die Analyse gestartet und jederzeit vom Benutzer abgebrochen werden kann.

Wichtige **Steuerungsparameter** der inkrementellen Analyse können im Dialogfenster /Traglastoptionen/ eingestellt werden.

J 5.3.4 Das Dialogfenster Traglastoptionen

In diesem Dialogfenster können Steuerungsparameter für die Traglastanalyse eingestellt werden.



Aktivieren der Abbruchkriterien:
– 'Durchbiegung'.
– Rotationsfähigkeit
– Durchstanzen

Verfestigung nach Erreichen der Fliessgrenze als Prozentsatz der Anfangssteifigkeit

Verfestigung

Beim einem bewerten Querschnitt steigt das Moment unter Zunahme der Krümmungen auch nach dem Erreichen des Fliessmomentes weiter an (siehe Handbuch). Die Verfestigung, definiert als Steigung der Momenten-Krümmungslinie im Fliessbereich, liegt in der Regel bei ca. 1- 3% des E-Moduls des gerissenen Querschnittes und bewirkt das charakteristische Verfestigungsverhalten des Stahlbetonquerschnittes. Falls keine Verfestigung angenommen wird, resultiert eine konservativere Traglast. Falls die Verfestigung zu hoch angesetzt ist, kann die Traglast überschätzt werden. In der Praxis empfiehlt es sich den Einfluss der Verfestigung zu untersuchen, indem einmal ohne und einmal mit wenig Verfestigung (z.B. 2%) gerechnet wird.

***Achtung**: Ist eine Verfestigung vorgegeben, dann kann es vorkommen, dass die Struktur nie instabil wird, da nach dem Fliessen eine Reststeifigkeit in der Grössenordnung der Verfestigung existiert. Somit kann die Traglast unendlich gross werden, falls die Berechnung nicht durch andere Abbruchkriterien gestoppt wird*

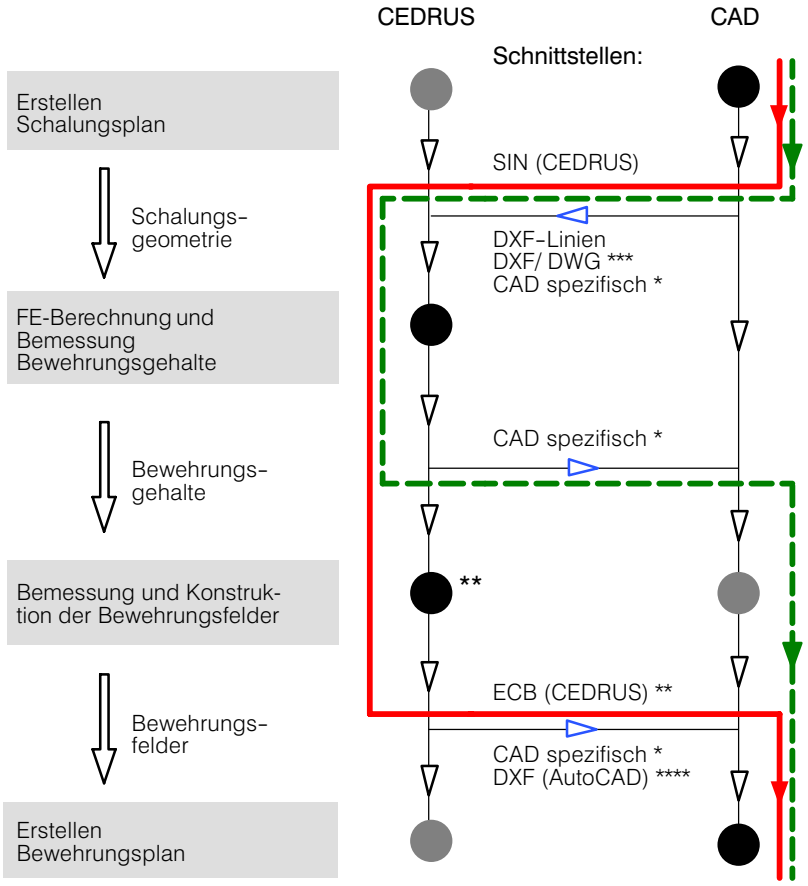
Hinweis: Wird keine Verfestigung angenommen, so kann dies in Einzelfällen zu numerischen Problemen führen, welche in einer viel zu tiefen Traglast resultieren. In diesem Fall sollte man eine geringe Verfestigung (z.B. 0.5%) vorgeben, und die Berechnung wiederholen.

J 5.4 Datenaustausch mit CAD-Systemen

J 5.4.1 Einleitung

Siehe dazu auch Teil L des Handbuchs.

J 5.4.2 Bemessung: Schema des Datenflusses



Legende:
* Serviceoption AP od. IS erforderlich
** Option Bewehrung erforderlich
*** Serviceoption CV, evtl. CX erforderlich
**** nur Zeichnungsdaten (Nachbearbeitung notwendig)

● sehr gut geeignet ● nur bedingt geeignet

Die obenstehende Darstellung zeigt schematisch den Datenfluss bei einer Bemessung. Der rot eingezeichnete Ablauf ist ideal, wenn Sie über eine Lizenz des Bewehrungsmoduls verfügen, da Sie am Ende im CAD-System direkt eine dreidimensionale Beschreibung der Bewehrung erhalten, welche konstruktiv und zeichnerisch zum Bewehrungsplan aufgearbeitet werden kann. Ist dies nicht der Fall, so haben Sie die Möglichkeit, die Bewehrungsgehalte von CEDRUS in Ihr CAD-System zu übertragen (grün, gestrichelt), wobei Sie anschliessend selbst die Umrechnung in Bewehrungslagen und deren Konstruktion vornehmen müssen. Für beide Abläufe müssen Sie sicherstellen, dass CEDRUS und Ihr CAD-System über gemeinsam unterstützte Schnittstellenformate verfügen.

J 5.4.3 Austausch der Struktur/Schalungsgeometrie

Für die Übertragung von Schalungsdaten stehen Ihnen folgende Formate zur Verfügung, welche beim Einlesen in CEDRUS direkt Eingabeobjekte (Umriss, Stütze etc.) erzeugen:

- SIN = *Beschreibung der CEDRUS-Eingabeobjekte*
  - gehört zum Lieferumfang des Basismoduls
  - muss von Ihrem CAD-System explizit unterstützt werden
- CAD spezifisch = *Format des CAD-Herstellers*
  CEDRUS unterstützt diverse CAD spezifische Formate, welche allenfalls den Erwerb einer entsprechenden Serviceoption erfordern

  Neben dem direkten Einlesen von Strukturdaten, können auch Zeichnungsdaten importiert werden:
- DXF-Linien
  - gehört zum Lieferumfang des Basismoduls
  - wird von den meisten CAD-Systemen unterstützt

  Die DXF-Zeichnung wird in CEDRUS als Hintergrundzeichnung abgelegt, womit die Strukturdaten mit dem Grafikeditor sehr effizient durch Abgreifen der Koordinaten eingeben werden können.
- DXF/DWG
  - wie DXF-Linien, jedoch erweitert, effizienter und nach aktuellem Stand
  - erfordert Servicoption CV und
  - evtl. CX für das halbautomatische Generieren von CEDRUS-Grundrissen

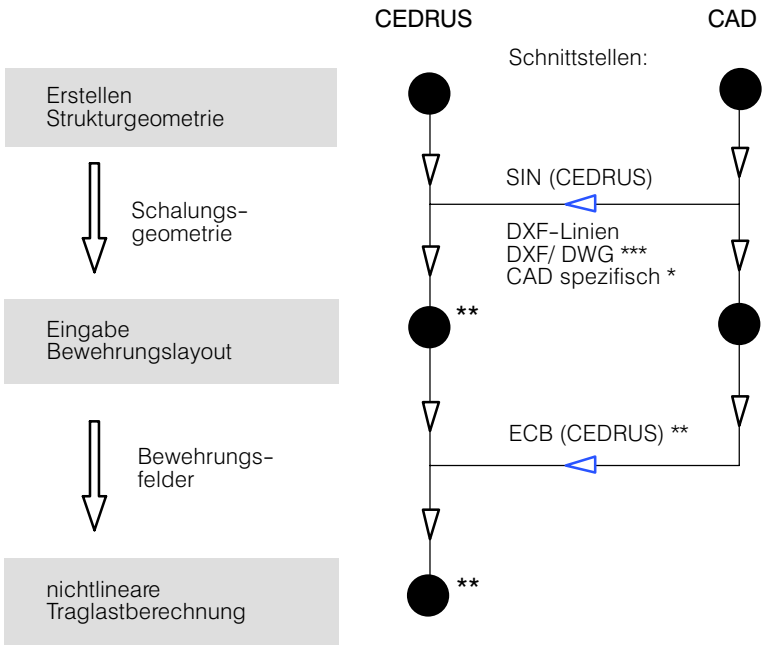
J 5.4.4 Austausch Bewehrungsfelder

Für die Übertragung von Bewehrungsfeldern (d.h. Stab- und Mattenlagen) stehen Ihnen folgende Formate zur Verfügung:

- ECB = *Beschreibung der CEDRUS-Bewehrungsfelder*
  - gehört zum Lieferumfang der Bewehrungs- und Traglastmodule
  - muss von Ihrem CAD-System explizit unterstützt werden
- DXF = *Industriestandard für Zeichnungsobjekte (Linie, Punkte etc.)*
  - gehört zum Lieferumfang der Bewehrungs- und Traglastmodule
  - wird von den meisten CAD-Systemen unterstützt

  Da die Bewehrungsfelder nur als Zeichnung übertragen werden, gehen viele Daten verloren (Durchmesser, Teilung etc.), was einen grossen Nachbearbeitungsaufwand im CAD-System nach sich zieht
- CAD spezifisch = *Format des CAD-Herstellers*
  CEDRUS unterstützt diverse CAD spezifische Formate, welche allenfalls den Erwerb einer entsprechenden Serviceoption erfordern

J 5.4.5 Traglastanalyse: Schema des Datenflusses



Legende:
* Serviceoption AP oder IS erforderlich
** Option Traglast erforderlich
*** Serviceoption CV und evtl. CX erforderlich

Die obenstehende Darstellung zeigt schematisch den Datenfluss bei einer Traglastanalyse. Analog zum Vorgehen bei der Bemessung kann die Schalungsgeometrie entweder in CEDRUS eingegeben werden oder von einem CAD-System übernommen werden, sofern eine gemeinsame Schnittstelle besteht. Das selbe gilt auch für die Bewehrungsdaten, welche ebenfalls von CAD-Systemen eingelesen werden können.

Vol. K Foundations

The optional module 'F' is an extension of the building module 'G' aimed at the analysis of soil-structure interaction. In comparison to the base module of CEDRUS, this option allows a more realistic dimensioning of the foundation of a building.

In this chapter the foundation module is explained in detail. It consists of the following parts:

Introduction
Modelling the Soil-Structure Interaction
Process of a Foundation Analysis
Story with LARIX-Foundation
Building Model with LARIX-Foundation

K 1 Introduction

The foundation module is an extension of the building module of CEDRUS and is based on the 3D-soil option of LARIX. In conjunction with LARIX this optional module of CEDRUS can model the soil-structure interaction, allowing a realistic design of the foundation of a building with bottom slabs, possibly standing on piles. In order to activate the soil-structure interaction, the user has to specify a so-called LARIX foundation for the bottom slab, introducing special boundary conditions for the slab. With a LARIX foundation one can model a raft, a pile or even a piled raft foundation.

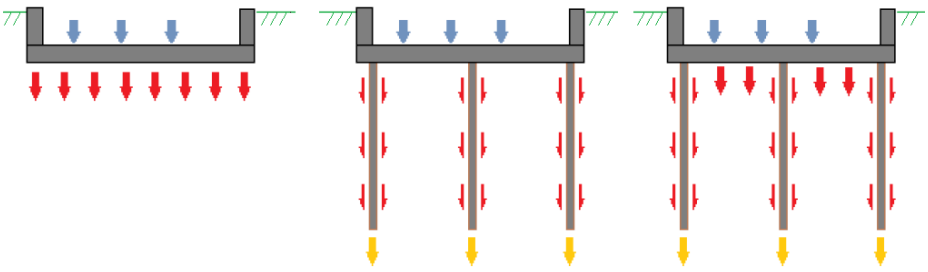
In order to model the soil-structure interaction, the following program options have to be licensed:

- Program CEDRUS: base module, option 'G' (building) , option 'F' (foundation)
- Program LARIX: option '3' (3D ground and settlements)

The calculations in CEDRUS are performed with the stacked slabs model. Therefore LARIX foundations are supported in building models only.

Types of Foundations

With LARIX foundations you can analyze raft, pile or piled raft foundations. Corresponding to the type of foundation, the following loads are acting in and on the ground:



Raft foundation | Pile foundation | Piled raft foundation

Analysis Model

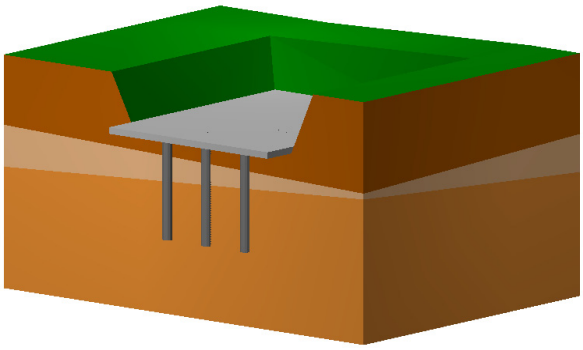
The simulation of the soil-structure interaction is based on the settlement calculation of LARIX. As explained in the LARIX manual (Part F), this analysis targets long term loading situations and can only handle vertical, static loads. Therefore no dynamic analysis can be performed with this module.

K 2 Modelling the Soil-Structure Interaction

The soil-structure interaction is a very complex, physical problem: the interaction between a slab, possibly founded on piles, and the ground is of highly nonlinear characteristic. An exact simulation of this interaction could be achieved by means of nonlinear finite element analysis methods, based on the knowledge of soil properties and conditions of the three dimensional ground. Unfortunately, these methods are very demanding with respect to model input and calculation times. They require very accurate soil data (including the complete loading history!) and the results are extremely sensitive with respect to this data. Because soil data of this quality is usually not available in engineering practice, extensive parametric studies are necessary in order to obtain reliable results. Therefore such a costly nonlinear finite element can hardly ever be justified for the design of a building foundation.

Because of these shortcomings the analysis method implemented in CEDRUS and LARIX is based on a much simpler method, which can nevertheless simulate the soil-structure interaction with acceptable accuracy. It is based on classical, geotechnical theories that have a proven record in practice. Although the quality of the required input of soil properties and conditions remains a problem, there is only one well understood soil parameter needed for the analysis. Because the model is simple enough to be understood by practicing engineers and the computing times are modest, parametric studies are very feasible.

The implemented method combines the functionality of CEDRUS and LARIX in order to achieve a realistic design of the bottom slab: from the calculated settlement of the loaded slab in LARIX a bedding of the bottom slab is found and introduced as support condition in CEDRUS. With this bedding the required reinforcement of the bottom slab can be calculated.



Slab with piles in LARIX

The implemented method can model the following aspects of the soil-structure interaction:

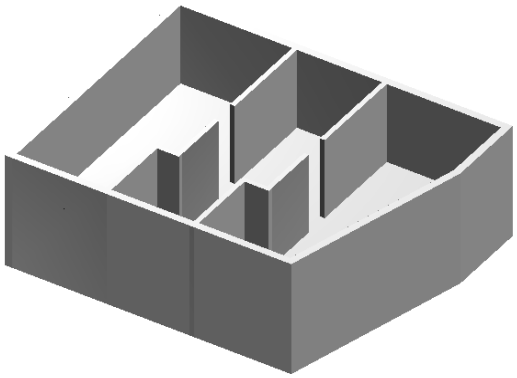
Influence of the Ground

The settlements of a slab depend on the existing soil layers and their material properties. In LARIX the ground is modeled in 3D as a series of discrete soil layers.

☞ *For a detailed description of the ground model see the LARIX manual Vol. E '3D ground model'.*

Influence of the Stiffness of the Slab

In addition to the condition of the ground, the settlements of a slab depend on the stiffness of the building as well. Especially the stiffness of the lowest (i.e. foundation) story has a massive influence on the observed settlements. Therefore the elastic stiffness of the bottom slab, specified in the building model of CEDRUS, is taken into account. In order to get a realistic slab stiffness, make sure to model the (reinforce concrete) walls standing on the bottom slab as upstanding beams in CEDRUS (see B 8.7.1).



Bottom slab with walls modeled as upstanding beams

Influence of the Loading of the Slab

The settlements of a slab depend heavily on the applied load acting on the slab. In contrary to the stiffness of the slab, which is (relatively) easy to determine, the correct loads are not known, because they are varying in time. In this method however, only the long-term settlements are of interest, which depend on the permanent loads of the building. These loads are reliably be obtained from the quasi-permanent loads (according the national codes). Therefore the loading of the bottom slab corresponds to the automatically generated load combination '!GQ2' in the CEDRUS building model.

☞ *The load of the slab used in the settlement analysis of LARIX is available in CEDRUS as the automatically generated load combination '!L'.*

Influence of Piles

For a pure pile foundation or a combined piled raft foundation the behavior of the pile is dominant in the settlement calculation. In the implemented analysis method the characteristic of the piles is given as a load-settlement diagram for each individual pile. These diagrams are very common in engineering practice (e.g. obtained from a pile test). Based on the characteristic of each pile, the special behavior of grouped piles and the interaction between the slab and the attached piles can be modeled.

☞ *For a detailed description of the pile modelling see Vol. F of the LARIX manual.*

Influence of Multiple Slabs

In some building projects not only one but multiple bottom slabs have to be founded close to each other. Because of the three dimensional nature of the implemented method, the interaction between these slabs can be modeled as well. In order to analyze such situations, all bottom slabs have to be part of the same building model.

Limits of the implemented Method

As mentioned at the start of this chapter, the soil-structure interaction is an extremely complex, physical problem. Although the implemented method can take into account a variety of influencing effects, the obtained results are always just an approximation of the reality. Because the implemented method is based both on the building model (i.e. stacked slabs model) of CEDRUS and on the 3D ground model (i.e. settlement analysis) of LARIX, a user must ensure that none of the limits of these modules are exceeded. As an example, buildings on a sloped terrain are allowed in a CEDRUS building analysis, but are not supported by the settlement analysis of LARIX. Please check the limiting conditions of both modules in the manual.

K 3 Process of a Foundation Analysis

For a foundation analysis both programs CEDRUS and LARIX are required. A foundation analysis is performed in 3 major steps:

Step	1	2	3
Program	CEDRUS	LARIX	CEDRUS
Actions	● Foundation slab(s): ○ input piles if any ○ take care of the stiffnesses (upstanding beams for walls...) ○ mark as LARIX foundation(s) ● Calculation of stacked slabs model ● Data are stored for step 2	● Definition of the 3D-ground model ● Definition of the pile properties ● Start settlement calculation (algorithm for calculating the bedding stiffnesses for which the displacements are equal to the settlements)	● Import of the LARIX solution (bedding stiffnesses) in the stacked model tabsheet ● RC design of the foundation slab(s) as usual

Step 1: Calculation of the Slab Stiffness in CEDRUS

The first step is a standard building analysis with the stacked slabs model. By assigning a so-called LARIX foundation to a slab (s. page K-6), it can be prepared with or without piles for the foundation analysis. After a successful analysis of the building model in CEDRUS, the geometry, the stiffness and the loads of the foundation slab is stored for the next step.

Step 2: Calculation of the Slab Bedding in LARIX

The second step is performed in LARIX and corresponds to a settlement analysis of a loaded slab located in the ground. Started directly from CEDRUS (s. dialog 'LARIX settlement calculation'), in LARIX a 3D project is created and the stored data from the first step is loaded into the model. Here the user has to specify the properties of the 3D ground by defining boreholes. If the bottom slab has piles, the characteristic pile diagrams have to be defined as well. Next a settlement calculation is performed, simulating the soil-structure interaction. In order to optimize the foundation, this analysis can be repeated with different pile configurations. As a result, corresponding to the settlements, the bedding of the bottom slab (i.e. elastic support conditions) is obtained. Finally this bedding and the resulting piles are stored for the next step.

Step 3: RC design of the Slab in CEDRUS

The last step is executed again in CEDRUS, where the resulting slab bedding and the piles of the last step are imported (s. dialog 'LARIX settlement calculation'). Now a standard analysis of the bottom slab is performed, e.g. by calculating the required reinforcement and analyzing the punching shear problem.

Design of the Bottom Slab for Limit States

According to the national codes a slabs has to be designed based on limit states. The (elastic) bedding obtained from the LARIX settlement calculation of step 2 corresponds to the assumed quasi-permanent loads. Since this loading is more or less equal to the serviceability limit state (SLS), all the calculations with respect to the SLS (e.g. displacements, stresses) are perfectly suitable for the bedded slab.

The ultimate limit states (ULS) is based on a number of short-time load combinations, which generally differ from the load pattern and load amplitudes of the SLS. However, because of the dominating contribution of the dead load for a bottom slab, the load patterns of ULS and SLS are very similar and thus the ULS loads are almost identical to the SLS loads multiplied by a constant factor. Because of the similarities of ULS and SLS and the fact, that short-time load do not change the settlements in any significant way, the bedding obtained for SLS loads can be used for the ULS as well.

State of the LARIX-Foundation

As mentioned above the LARIX foundation depends directly from the properties of the slab (i.e. geometry, stiffness, loading) of CEDRUS, because this data is used in the settlement calculation in LARIX. Therefore, if one of the properties is changed, the previously calculated bedding in LARIX is no more valid and the settlement calculation must be repeated. In this situation, the state of the LARIX foundation is set to 'not up to date'. After a successful settlement analysis and an import of the result into CEDRUS, the state is set to 'undefined'. Only after successfully solving the stacked slab model the state is set to 'up to date' and the user can analyze and design the bottom slab.

State of the LARIX-Foundation	Slabs can be calculated*	Description
Undefined	No	The stacked slab model has not been solved after the last change of building data. It is therefore not clear if the LARIX foundation is up to date or not.ell
Up to date	Yes	After the last import of the foundation from LARIX no relevant changes in the building data have been made. All LARIX foundations in the building are up to date.
Not up to date	No	After the last import of the foundation from LARIX some relevant changes in the building data have occurred, making the LARX foundation invalid. In this case the settlement calculation must be repeated.

* restriction applies to slabs with LARIX foundation only

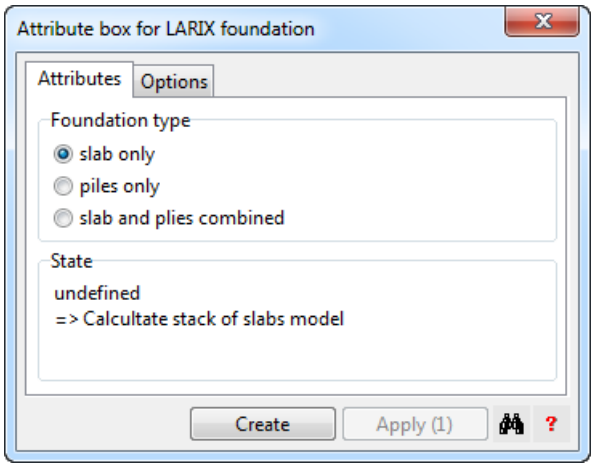
K 4 Story with LARIX-Foundation

If the soil-structure interaction should be taken into account for a bottom slab, the following restrictions apply to the input of the slab:

- No area supports allowed
- No joints allowed
- No walls allowed (exception: linear longitudinal spring, s. B 8.10)
- No nonlinear supports
- No LARIX foundations in a master story

Specification of a LARIX-Foundation

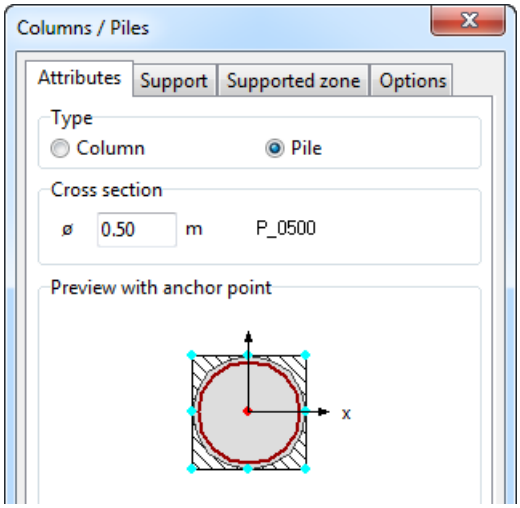
In order to activate the soil-structure interaction, in addition to a normal slab input, a so-called 'LARIX foundation' must be specified. This is done by clicking on the button 'LARIX foundation' in the tabsheet 'Structure' and by placing the corresponding box with the graphics editor inside the outline of the slab. This box is then stored in the drawing layer 'LARIX foundation'. In the attribute dialog shown, the property 'foundation type' can be set. You can later change this attribute in LARIX if needed. The actual state of the object is shown in the dialog.



For a slab only one box can be specified, but more than one slab with LARIX foundation is allowed in a building.

Specification of Piles

If you plan to place a bottom slab on piles, you have to define all possible pile locations in CEDRUS. In the design process performed in LARIX, you can later select the optimal pile location together with diameter and length.



Because piles and columns have many properties in common (e.g. support conditions, punching analysis) they share the same dialog. Piles are independent graphic objects, which are manipulated just like columns. There are a few differences between piles and columns. The following restrictions apply to piles:

- No element support allowed
- No supports for rotation allowed
- No inclination allowed
- Circular cross-section only

The height and the diameter specified in the attribute dialog are starting values only: you can change them later in LARIX.

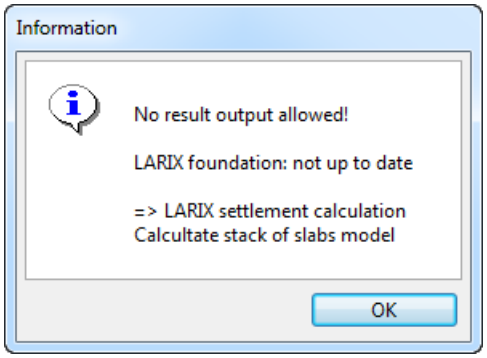
Columns and piles are stored in the same drawing layer. With the layer button shown here you can control the visibility of piles in CEDRUS.

Analysis of a LARIX-Foundation

A LARIX foundation is defining the bedding (springs) of the slab plus all the active piles. In order to analyze or design it, you have to use LARIX. You can start LARIX in the tabsheet 'Stack of slabs model' (s. next Chapter) of the CEDRUS building module. As soon as the foundation has been analyzed in LARIX you can import the resulting data in CEDRUS. After the import you can design the reinforcement of all the bottom slabs.

As described in the next chapter, a LARIX foundation can have three different states, which are shown in the graphic. If the state is 'up to date' no relevant changes in the building data have been made since the last import of the LARIX solution. In this state you can obtain all results of a slab.

If the state is 'not up to date' or 'undefined (see page K-5) you cannot obtain any results for a slabs with LARIX foundation, because the bedding is not valid yet. If a user tries to get a result output the following error message appears:



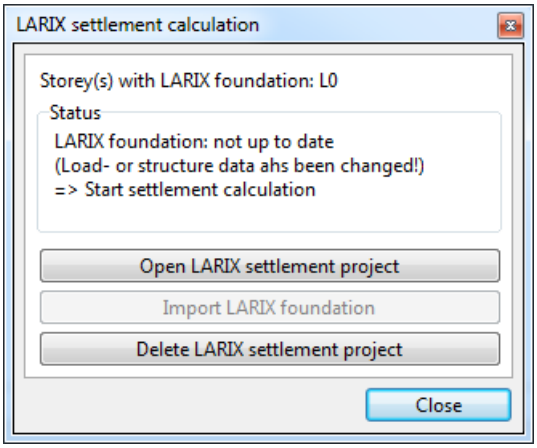
Error message in case of an invalid LARIX foundation

K 5 Building Model with LARIX-Foundation

If a soil-structure analysis should be executed for a building, you have to specify a LARIX foundation for all the bottoms slabs (s. K 4). In the building model only a few simple steps are needed for such an analysis.

Dialog LARIX settlement calculation

The soil-structure analysis is controlled by the Dialog 'LARIX settlement calculation', which is accessible from the tabsheet 'Stack of slabs model'. By clicking on the button shown to the left the following dialog is opened. You can only this dialog only if a valid solution of this model has been calculated. If this is not the case (e.g. due to an input error), you have to fix this problem first.



In the dialog the current state of the LARIX foundation is presented. Depending on the state you can execute the following steps:

Opening a LARIX settlement project

By clicking on the button below you can open the corresponding LARIX project.

Open LARIX settlement project

Now the application LARIX is launched, in case you have a valid license for option '3' of LARIX. If a corresponding project is already existing, the project is loaded and updated with the actual data from CEDRUS regarding the bottom slabs and the piles. If no project is found, a new one is created.

In LARIX you have to specify the 3D ground model. It is possible to add additional terrain loads or isolated footing to the model. If the slab has piles, you have to define their properties. Now you can execute the settlement calculation. If the results look realistic, you can export the obtained foundation (i.e. bedding of the slab, piles) to CEDRUS as a so-called 'LARIX foundation'.

Import the LARIX-Foundation

Back in CEDRUS you can click on the button below

Import LARIX foundation

in order to import the foundation calculated in the previous step. The button is active only if a valid result has been exported in LARIX before.

With the import of the LARIX foundation the bedding and the piles of the bottom slabs are adjusted. The building model is now complete and you can obtain any result from the bottom slabs as you like. You can also generate the mass, volume and length tables of the building, which include the corresponding values for the piles.

Delete the LARIX settlement project

If the LARIX project is not needed anymore, you can delete it by clicking on the button below.

Delete LARIX settlement project

This will not delete the calculated bedding of the bottom slabs, but all the data of the 3D ground model will be lost.

Visualization and Documentation of the LARIX foundation

For the visualization of the LARIX foundation, a special layer button in the group 'Geometry' is introduced.

If this layer is active, all the bottom slabs and their piles are displayed.

For the documentation of the LARIX foundation, the corresponding slabs and piles are marked in the output tables of the building geometry.

Vol. L CAD Interface

L 1 Import

Overview:

	for which	where	required option	description
DXF-Lines	Transfer lines from DXF files to the graphics editor	Menu File > Import > DXF-Lines...	none	Chap. L 1.1.1
DXF/DWG Plans	To display as background plans for tracing or for CAD-Extract	tab sheet CAD button	Service option CV (CADView)	Chap. L 1.1.2 and manual Vol. C
SIN	structural data	tab sheet Structure button	none	Chap. L 1.2.1
LIN	loading data	tab sheet loads button	none	Chap. L 1.3.1
Allplan	structural data	as SIN	none	as SIN
Allplan 3D	building data	building module, tab sheet CAD button	Option G (buildings) Service option AP	manual, Chap. D 7.2
isbCAD	structural data	tab sheet Structure button	Service option IG	Chap. L 1.2.3
isbCAD	loading data	tab sheet Loads button	Service option IG	Chap. L 1.3.2
ECB	Reinforcement layouts for reinforcement module	Tab sheet reinforcement button	Option reinforcement or ultimate load	

L 1.1 Background drawings

L 1.1.1 DXF-Lines

Using the menu command 'File> Import> DXF Lines ...', lines from a DXF file can be imported into the graphics editor. A layer group called DXF is created, in which the imported layers are stored. A detailed description can be found in the manual of the graphics editor in chap. 10.1.

L 1.1.2 DXF/DWG-Drawings

The CADView program module allows the import of DXF and DWG files according to the current standard of these interfaces defined by AutoCAD. In contrast to the limited possibilities of the import of DXF lines described above, this allows very large files with all important attributes such as colors, fillings, etc. to be imported and efficiently handled. The import is carried out in the CAD tab of the slab program using the adjacent button.

For performance reasons the import is not directly in the data structure of the graphics editor. However, if necessary, individual layers of imported plans can be transferred to user layers of the graphics editor.

The imported plans can, on the one hand, be used to snap coordinates when entering structure and loads. On the other hand, they are also used as a starting point for the application of CADExtract for the semi-automatic generation of CEDRUS slab floor plans.

The detailed description can be found in Vol. C of this manual.

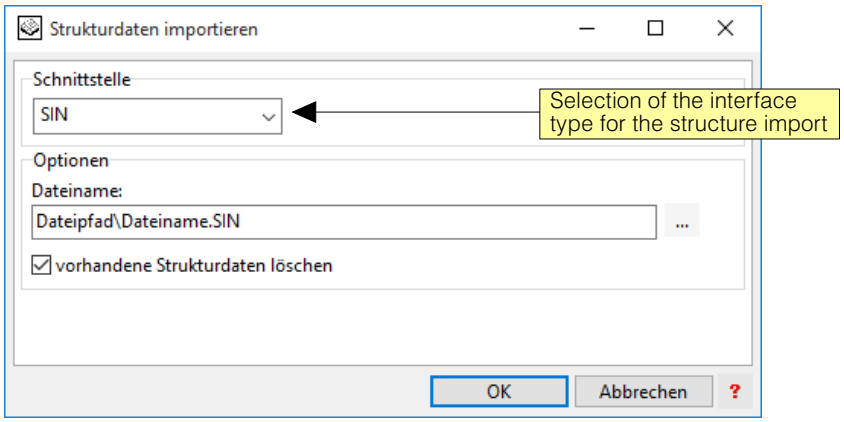
L 1.2 Structure

L 1.2.1 SIN Interface

The SIN interface is a text interface, which describes the entire structure data of a CEDRUS calculation (slab or wall). A current description of the interface can be requested from Cubus.

Import of structural data

Files according to this standard can be imported in the Structure tab on the button shown left. The following dialog appears:



Depending on the selected setting, an existing structure can be deleted in the current calculation or can be supplemented by the data of a SIN file.

Structure recovery

CEDRUS always stores a backup file according to the SIN standard (several generations with the file names STRUKTUR_Recovery.SIN, ... 1.SIN is always the newest file). This file can be used with the menu command 'File'> 'Recovery' to regenerate a data structure that is no longer consistent. For more information, see Chap. B 4, page B-4.

Menu File > Import

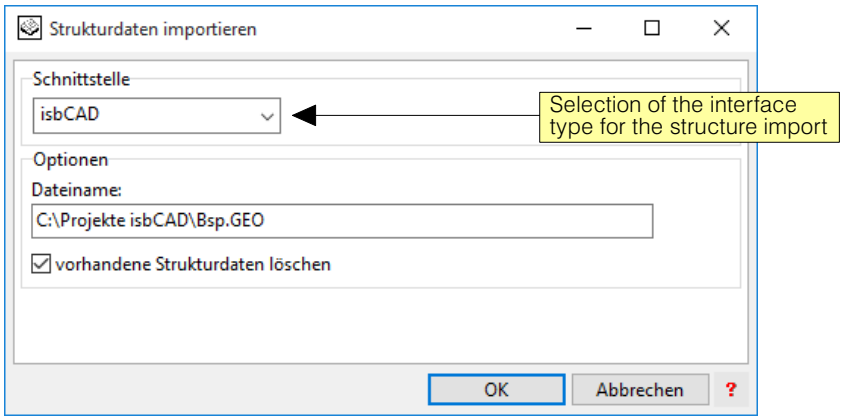
In the case of an open CEDRUS calculation, the data file of a different calculation selected via the CubusExplorer can be imported using the menu command 'File' > 'Import'. Any existing data for the current calculation is deleted. The STRUKTUR_Recovery1.SIN file described in the section above is proposed for importing the structure data. If desired, the loads (LIN files, see below) of the relevant calculation are also imported with this menu command.

L 1.2.2 Structure import from Allplan

Allplan is able to create a SIN file with the CEDRUS structure data. These SIN files can be imported as described in L 1.2.1.

L 1.2.3 IsbCAD Interface

CEDRUS also supports the geometry interface of the isbCAD program from Glaser. Files according to this standard can be imported as SIN files in the structure register via the button shown left. The following dialog appears:



Loadings, which can also be contained in an isbCAD GEO file, must be imported into the load register.

L 1.2.4 Allplan 3D

A 3D building model defined in Allplan can be imported and transformed into CEDRUS building model. The import takes place in the CAD register of the building module with the usual import button. Read more in Chap. D 7.2 of the manual.

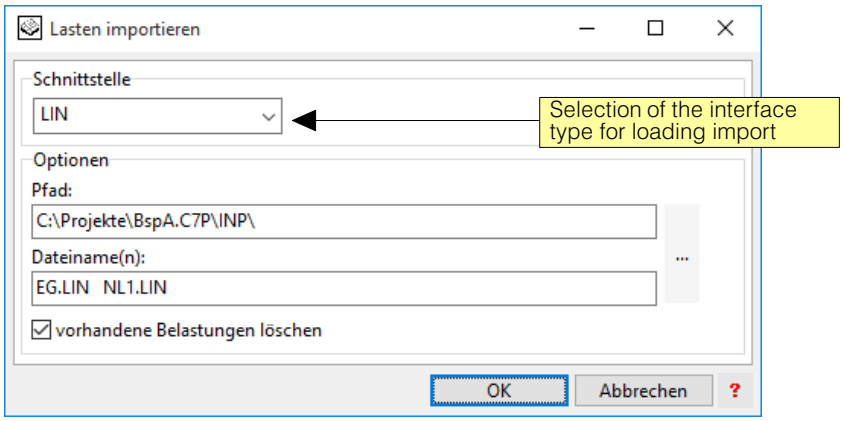
L 1.3 Loadings

L 1.3.1 LIN Interface

The LIN interface is a text interface with which loads are described. One load is recorded per LIN file. CEDRUS automatically generates a LIN file called 'Loading-ID'.LIN the INP folder upon release of calculation per load.

A current description of the interface can be requested from Cubus.

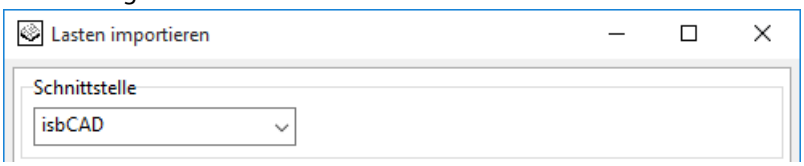
The import of loads according to the LIN standard is carried out in the tab sheet 'load' via the button shown left. The following dialog appears:



The LIN files may originate from other CEDRUS calculation or be generated in a different way and stored somewhere.

L 1.3.2 IsbCAD Interface

As described above in L 1.2.3, isbCAD GEO files can also contain load data. The import is started in the tab sheet 'loads' with the button shown left. In this dialog the interface has to be changed to 'isbCAD'.



L 1.4 Reinforcement for the reinforcement module

L 1.4.1 ECB Interface

With this interface reinforcement fields (ie bar and mesh layers) can be imported into the reinforcement module of CEDRUS. The import is carried out by means of the button shown left on the tab sheet 'Reinforcement' of CEDRUS.

Currently, this interface is supported by BACAD from Help! Informatique (www.bacad.ch).

L 2 Export

Overview:

Structure- and Loading data:

	where	Required Options	Description
SIN (Structure data)	Calculation folder, subfolder INP: STRUKTUR_Recovery1.SIN	none	Chap. L 1.3.1
LIN (Loading data)	Calculation folder, subfolder INP: BelName.LIN	none	Chap. L 1.3.1
IFC (Building geometry)	Main menu: „File“ > „Export“ > „IFC“	keine	Kap. D 7.1

Reinforcement results from basic module:

Allplan / BAMTEC	Tab sheet Results button	Service option AP	
Allplan / BAMTEC dat		Service option AP	
isbCAD		Service option IG	

Reinforcement layouts from reinforcement module:

ECB	Tab sheet reinforcement	Option Reinforcement or ultimate load	
ALLPLAN	Tab sheet reinforcement	Service option AP	
DXF	Tab sheet reinforcement	none	

L 2.1 SIN/LIN Structure- und Loading data

There is no explicit export function for these interfaces described above under Import. However, the corresponding files are created by default in the INP subfolder of the calculation folder and can be read from there.

L 2.2 Reinforcement results from Base module

The reinforcement results determined with the basic module of CEDRUS can be exported with the button shown left in the tab sheet 'Results'.

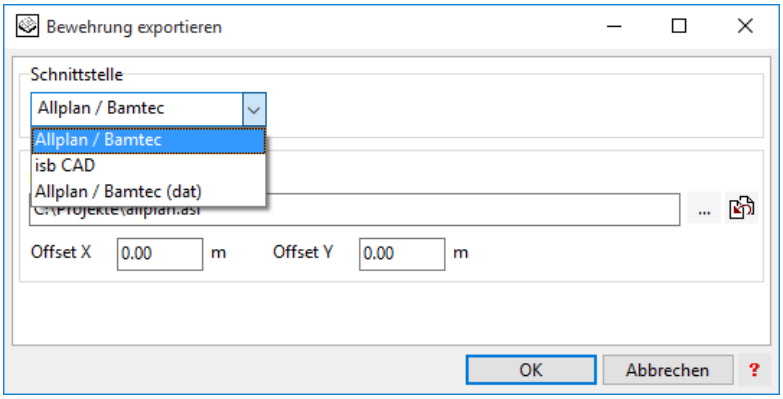
The dialog offers three interfaces to choose from:

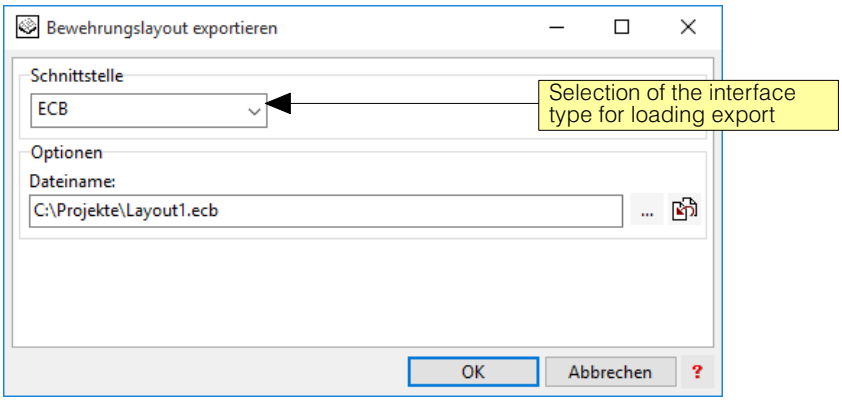
- **Allplan:** It is a text interface, which contains the reinforcement results in the element centers. The text file generated has the extension '.asf' and can be used to create a reinforcement plan with Allplan or for the treated about Allplan reinforcement mesh system **BAMTEC**.
- **Allplan (dat):** This is an old interface definition no longer supported by current Allplan versions.
- **IsbCAD:** The reinforcement results in the element centers are written in binary form to a file according to the isbCAD standard.

For Allplan: The location of the result points is described in the export files by default in CEDRUS coordinates. For an adaptation to the coordinate system of the CAD program, an offset (correction values) for the X and Y values of the coordinates can be specified in the dialog.

L 2.3 Reinforcement from reinforcement module

The reinforcement layout defined in the reinforcement module with the determined reinforcement cross sections can be exported with the export button in the reinforcement register for further processing in a CAD system.





The interfaces of Allplan and ECB as well as the DXF format of AutoCAD are supported.