



# Structural Analysis & Design Software

[www.dlubal.com](http://www.dlubal.com)



**Dipl.-Ing. (FH) Andreas Höroid**  
Moderator

Marketing & Public Relations  
Dlubal Software GmbH



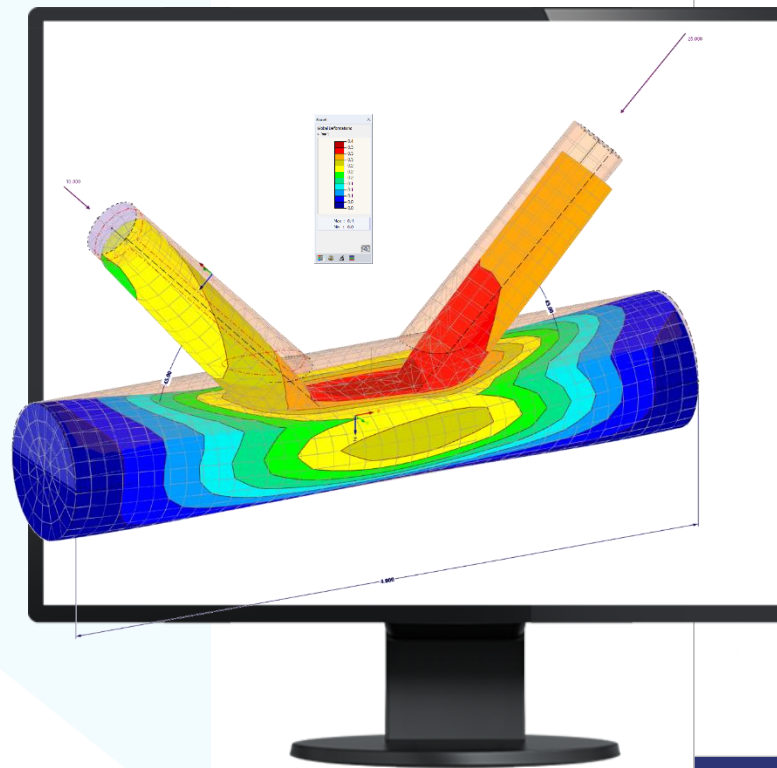
**Dipl.-Ing. (FH) Walter Rustler, M.Eng.**  
Presenter

Sales Director & Customer Support  
Dlubal Software GmbH



**WEBINAR**

# How to Be More Productive Using RFEM



# Questions During the Presentation



GoToWebinar Control Panel  
**Desktop**



E-mail: [info@dlubal.com](mailto:info@dlubal.com)



Show or hide control panel



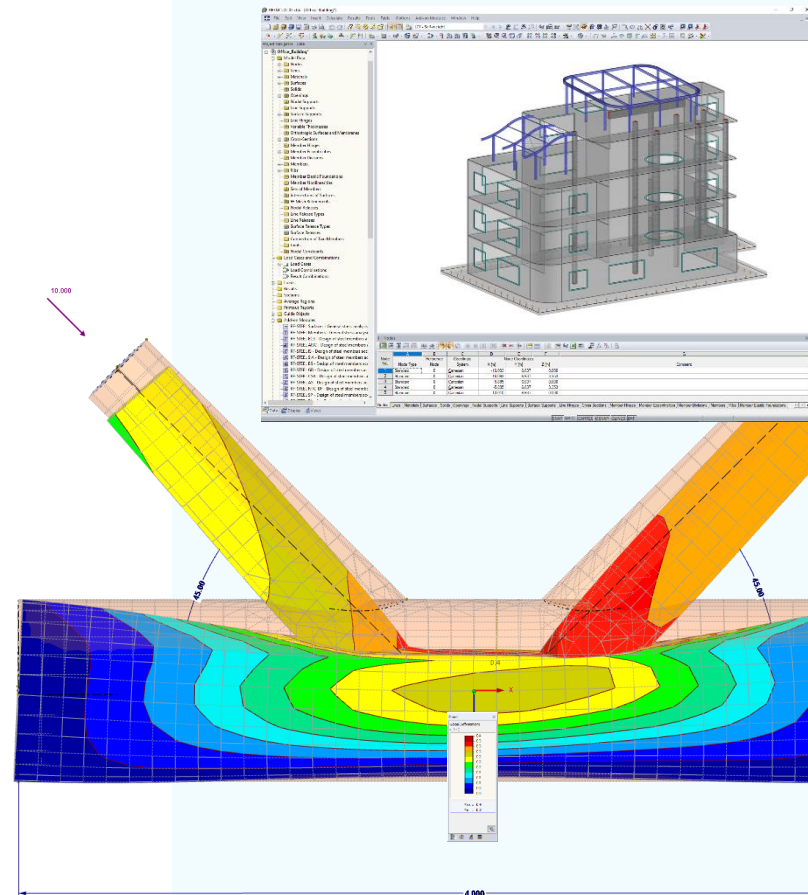
Adjust audio settings

Ask questions



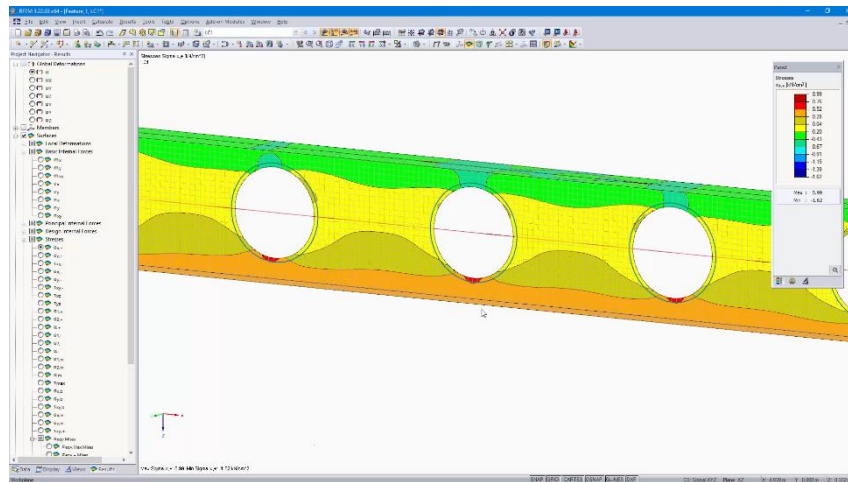
# CONTENT

- 01 Model generation
- 02 Parametric models
- 03 Load input and generation
- 04 Results and documentation



# Generate Surfaces from Members

- Rapid analysis based on shell models
- Takes seconds where others have to involve CAD programs or manually build the model
- Allows for analysis of local effects of load introduction or support conditions





Project Navigator - Data

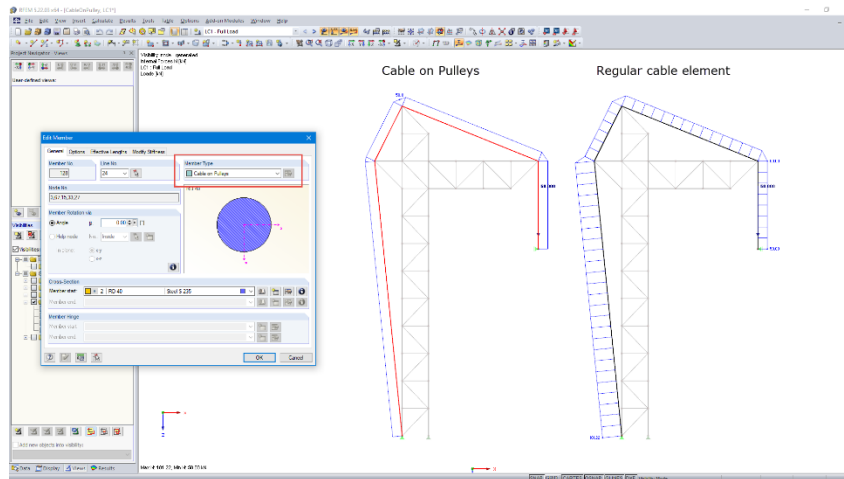
- Feature\_1\* [Webinar]
  - Model Data
    - Nodes
    - Lines
    - Materials
    - Surfaces
    - Solids
    - Openings
    - Nodal Supports
    - Line Supports
    - Surface Supports
    - Line Hinges
    - Variable Thicknesses
    - Orthotropic Surfaces and Membr
    - Cross-Sections
    - Member Hinges
    - Member Eccentricities
    - Member Divisions
    - Members
    - Ribs
    - Member Elastic Foundations
    - Member Nonlinearities
    - Sets of Members
    - Intersections of Surfaces
    - FE Mesh Refinements
    - Nodal Releases
    - Line Release Types
    - Line Releases
    - Surface Release Types
    - Surface Releases
    - Connection of Two Members
    - Joints
  - Nodal Constraints
  - Load Cases and Combinations
    - Load Cases
    - Load Combinations
    - Result Combinations
  - Loads
  - Results
  - Sections
  - Average Regions
  - Printout Reports
  - Guide Objects
  - Add-on Modules
    - RF-STEEL Surfaces - General stress
    - RF-STEEL Members - General stres
    - RF-STEEL EC3 - Design of steel me
    - RF-STEEL AISC - Design of steel m
    - RF-STEEL IS - Design of steel meml
    - RF-STPFI SIA - Design of steel mer





# Cable on Pulley

- Allows for correct modeling of cable gears on cranes
- Results are wrong if you do not use such a feature



Project Navigator - Views

Visibility mode - generated  
 Internal Forces N [kN]  
 LC1 - Full Load  
 Loads [kN]

User-defined views:

**Edit Member**

General Options Effective Lengths Modify Stiffness

Member No. 128 Line No. 24

Member Type: Cable on Pulleys

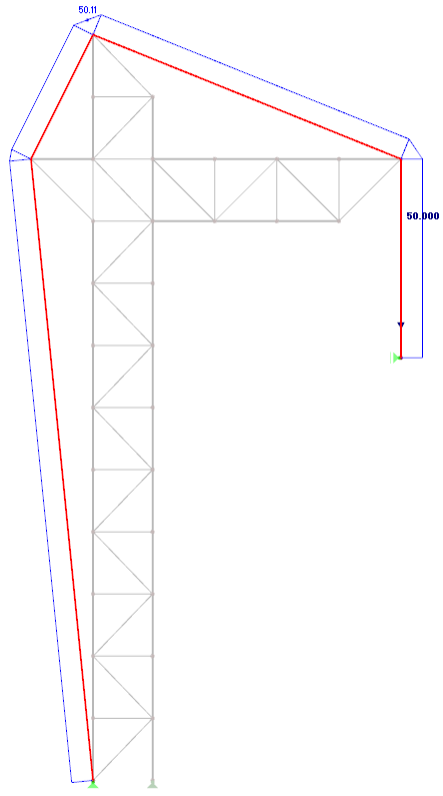
Member Rotation via: Angle  $\beta$ : 0.00 [°]

Cross-Section: Member start: 2 | RD 40 Steel S 235

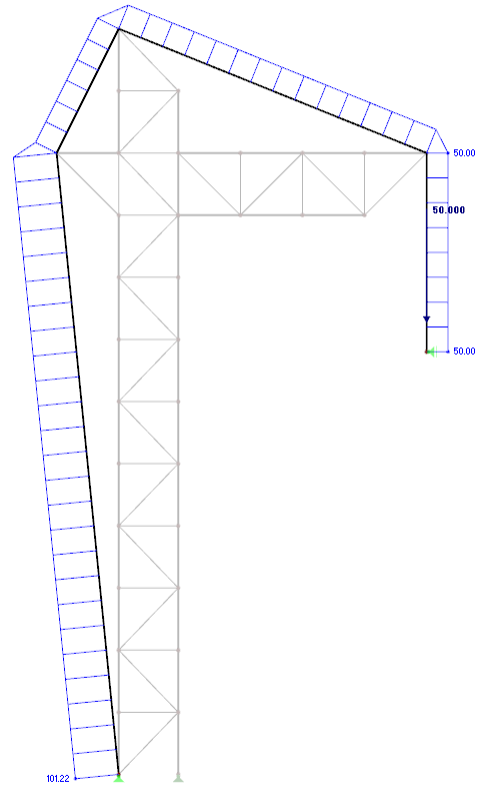
Member Hinge: Member start: Member end:

OK Cancel

Cable on Pulleys



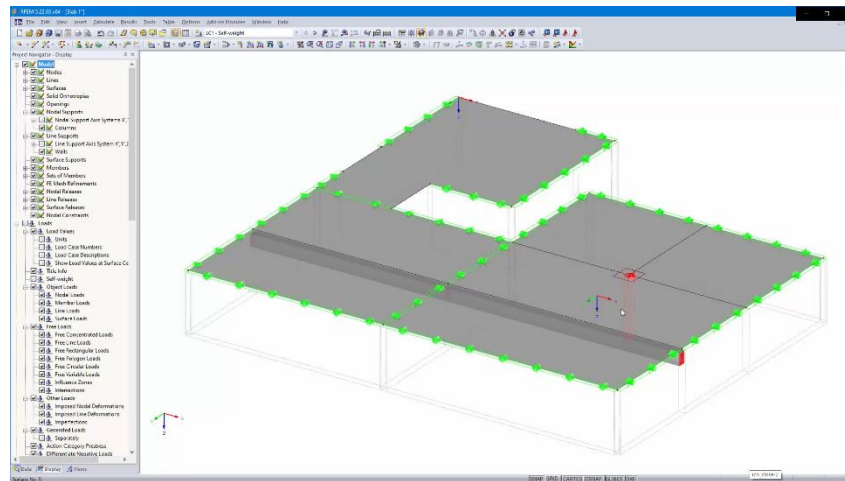
Regular cable element





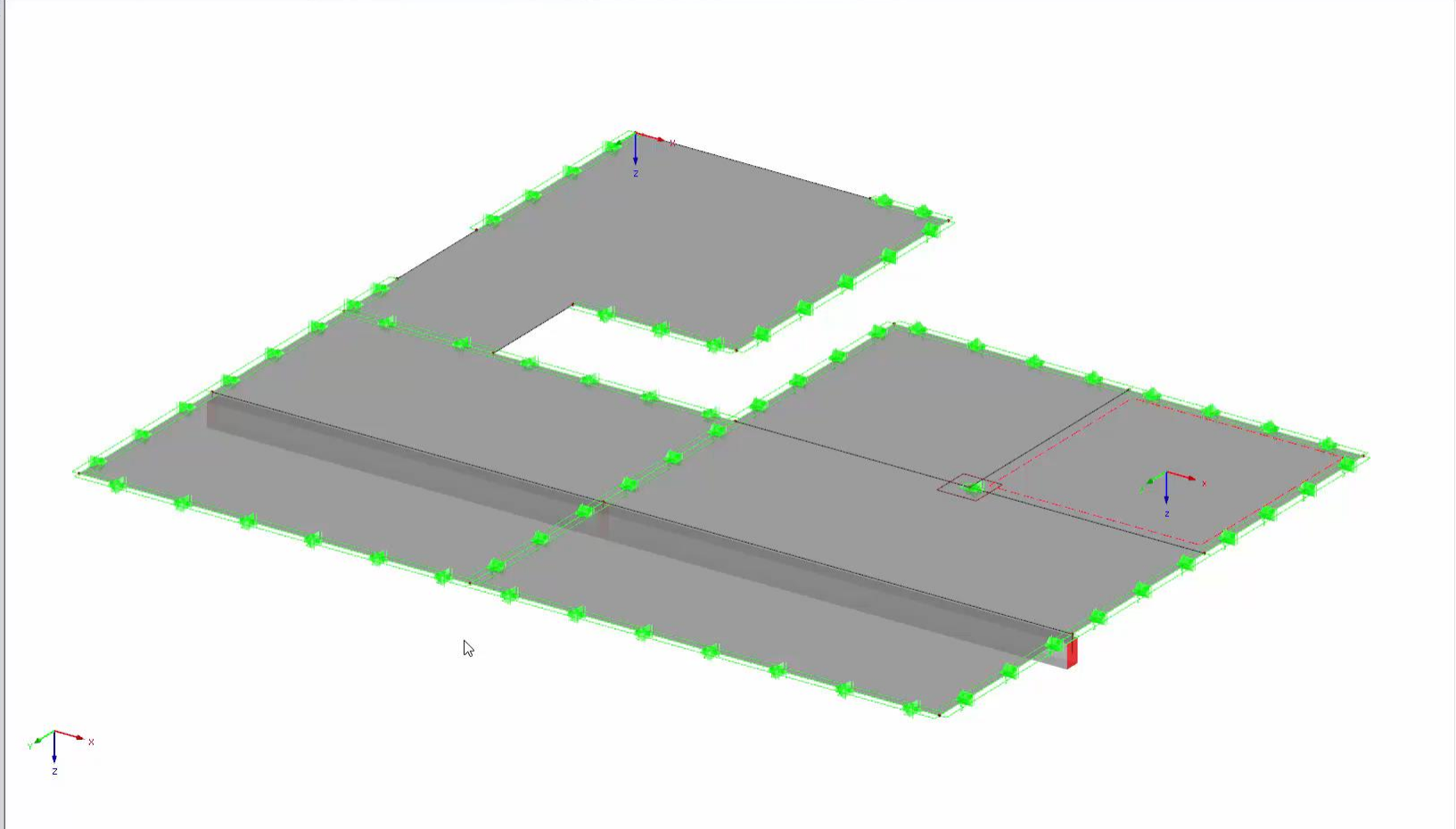
# Supports from Walls and Columns

- Calculate stiffnesses of walls and columns
- Choose from different support conditions
- More realistic stiffness allows to reduce unwanted effect of singularities





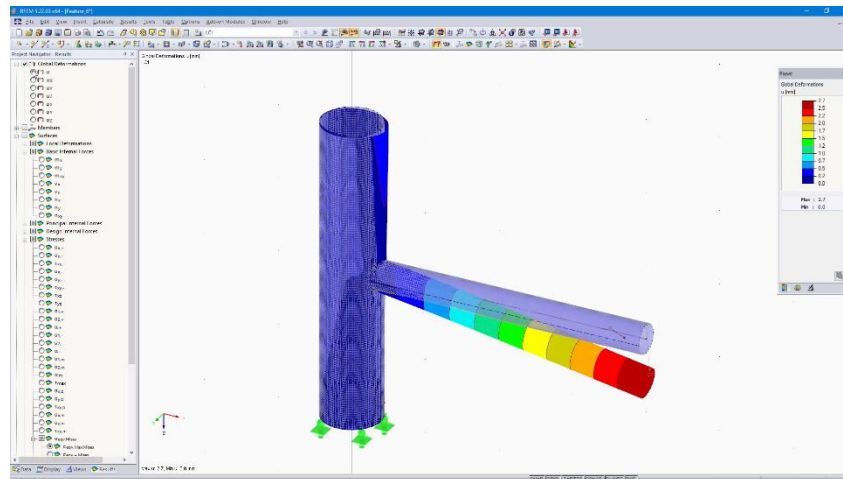
- Project Navigator - Data
  - Slab 1\*
    - Model Data
      - Nodes
      - Lines
      - Materials
      - Surfaces
      - Openings
      - Nodal Supports
      - Line Supports
      - Surface Supports
      - Line Hinges
      - Variable Thicknesses
      - Orthotropic Surfaces and Membr
    - Cross-Sections
    - Member Hinges
    - Member Eccentricities
    - Member Divisions
    - Members
    - Ribs
    - Member Elastic Foundations
    - Member Nonlinearities
    - Sets of Members
    - FE Mesh Refinements
    - Nodal Releases
    - Line Release Types
    - Line Releases
    - Connection of Two Members
    - Joints
    - Nodal Constraints
  - Load Cases and Combinations
    - Load Cases
    - Actions
    - Combination Expressions
    - Action Combinations
    - Load Combinations
    - Result Combinations
  - Loads
  - Results
  - Sections
  - Average Regions
  - Printout Reports
  - Guide Objects
  - Add-on Modules
    - RF-STEEL Surfaces - General stress
    - RF-STEEL Members - General stres
    - RF-STEEL EC3 - Design of steel me
    - RF-STEEL AISC - Design of steel m
    - RF-STEEL IS - Design of steel meml
    - RF-STEEL SIA - Design of steel mer
    - RF-STPFI RC - Design of steel mer





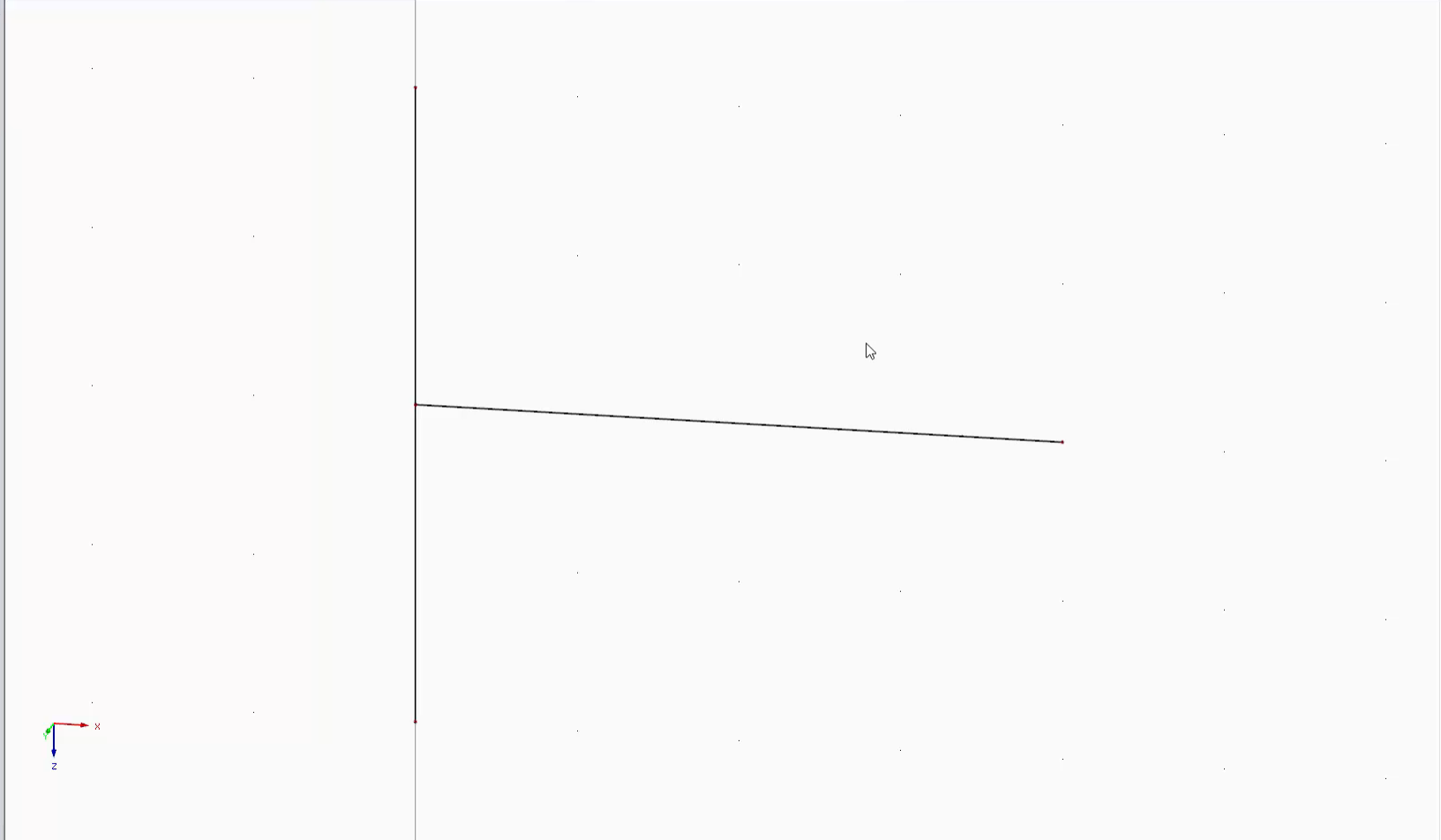
# Intersections

- Keeps surfaces connected with intelligence
- Calculates automatically intersection lines
- No external program needed for geometry





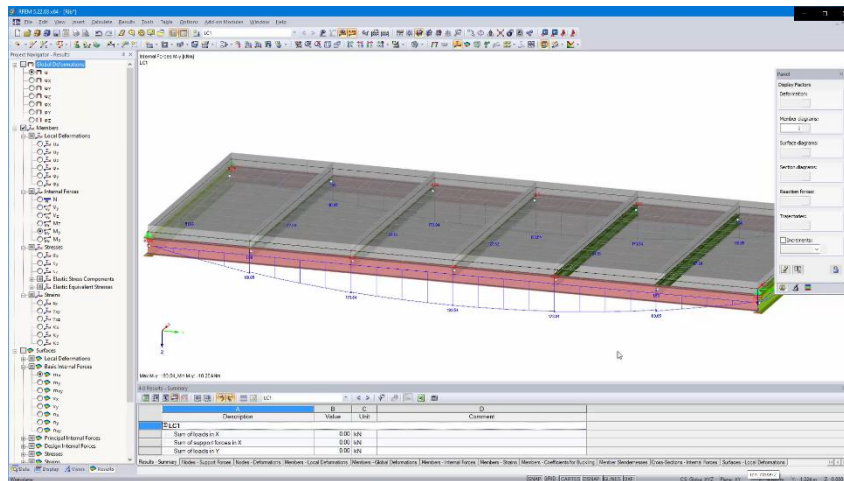
- Project Navigator - Data
  - Feature\_6 [Webinar]
    - Model Data
      - Nodes
      - Lines
      - Materials
      - Surfaces
      - Solids
      - Openings
      - Nodal Supports
      - Line Supports
      - Surface Supports
      - Line Hinges
      - Variable Thicknesses
      - Orthotropic Surfaces and Membr
    - Cross-Sections
    - Member Hinges
    - Member Eccentricities
    - Member Divisions
    - Members
      - Ribs
      - Member Elastic Foundations
      - Member Nonlinearities
    - Sets of Members
    - Intersections of Surfaces
    - FE Mesh Refinements
    - Nodal Releases
    - Line Release Types
    - Line Releases
    - Surface Release Types
    - Surface Releases
    - Connection of Two Members
    - Joints
    - Nodal Constraints
    - Load Cases and Combinations
      - Load Cases
      - Load Combinations
      - Result Combinations
    - Loads
    - Results
    - Sections
    - Average Regions
    - Printout Reports
    - Guide Objects
    - Add-on Modules
      - RF-STEEL Surfaces - General stress
      - RF-STEEL Members - General stress
      - RF-STEEL EC3 - Design of steel mem
      - RF-STEEL AISC - Design of steel me
      - RF-STEEL IS - Design of steel meml
      - RF-STPFI SIA - Design of steel mer

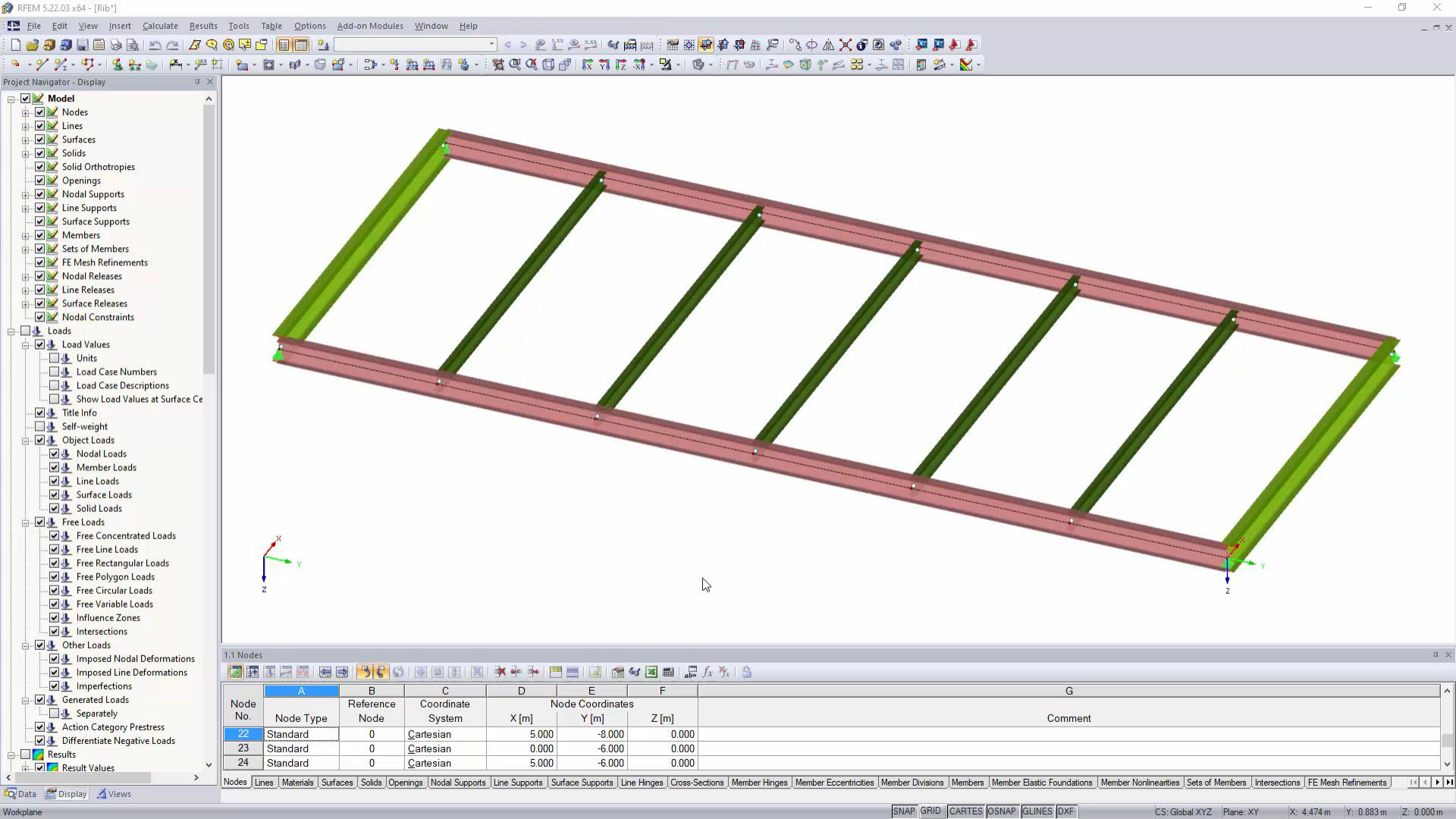




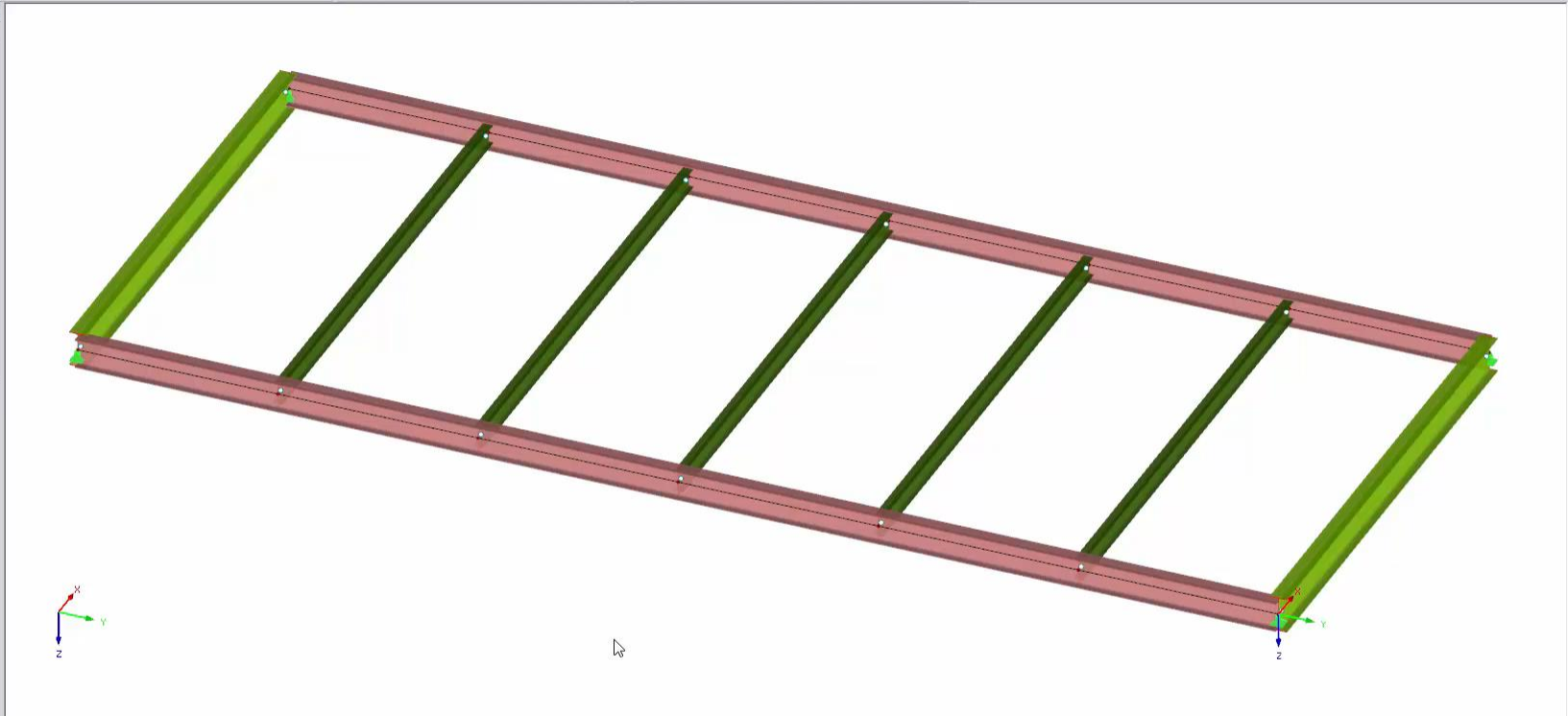
# Ribs

- Sets members eccentric based on depth of member and surface
- Takes eccentricities into account
- Easy and fast plus more realistic modeling without manual calculation of eccentricities or rigid links





- Project Navigator - Display
- Model
    - Nodes
    - Lines
    - Surfaces
    - Solids
    - Solid Orthotropies
    - Openings
    - Nodal Supports
    - Line Supports
    - Surface Supports
    - Members
    - Sets of Members
    - FE Mesh Refinements
    - Nodal Releases
    - Line Releases
    - Surface Releases
    - Nodal Constraints
  - Loads
    - Load Values
      - Units
      - Load Case Numbers
      - Load Case Descriptions
      - Show Load Values at Surface Ce
    - Title Info
    - Self-weight
    - Object Loads
      - Nodal Loads
      - Member Loads
      - Line Loads
      - Surface Loads
      - Solid Loads
    - Free Loads
      - Free Concentrated Loads
      - Free Line Loads
      - Free Rectangular Loads
      - Free Polygonal Loads
      - Free Circular Loads
      - Free Variable Loads
      - Influence Zones
      - Intersections
    - Other Loads
      - Imposed Nodal Deformations
      - Imposed Line Deformations
      - Imperfections
    - Generated Loads
      - Separately
      - Action Category Prestress
      - Differentiate Negative Loads
  - Results
    - Result Values

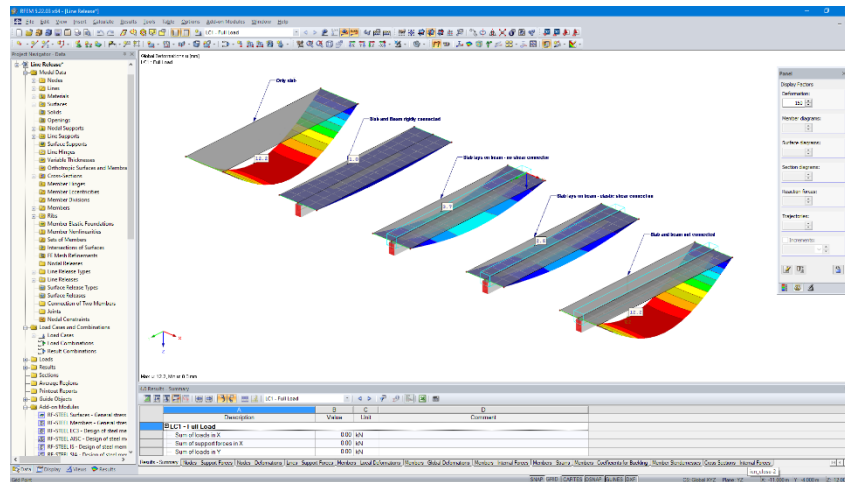


1.1 Nodes

Node No.	A Node Type	B Reference Node	C Coordinate System	D Node Coordinates			G Comment
				X [m]	Y [m]	Z [m]	
22	Standard	0	Cartesian	5.000	-8.000	0.000	
23	Standard	0	Cartesian	0.000	-6.000	0.000	
24	Standard	0	Cartesian	5.000	-6.000	0.000	

# Line Releases

- Allows to model elastic connections of slabs and beams or walls
- Connection forces as result
- Ideal for e.g. slabs that rest on walls (connection acts only if in compression or tension)



- Materials
- Surfaces
- Solids
- Openings
- Nodal Supports

### Edit Line Release Type

No.

**Translational Release**

	Spring constant	Nonlinearity
<input checked="" type="checkbox"/> $u_x$	$C_{ux} : 50000.000$ [kN/m <sup>2</sup> ]	None
<input type="checkbox"/> $u_y$	$C_{uy} : $ [kN/m <sup>2</sup> ]	None
<input type="checkbox"/> $u_z$	$C_{uz} : $ [kN/m <sup>2</sup> ]	None

**Rotational Release**

	Spring constant	Nonlinearity
<input type="checkbox"/> $\phi_x$	$C_{\phi x} : $ [kNm/rad/m]	None

Comment

Only slab

### Edit Line Release

No.  Line No.

**Release**

Line release type:

Local axis system from:

Original line

Member on original line

Z-axis perpendicular to surface No.:

Help node

No.:  In plane:  x-y  x-z

Line release rotation via angle  $\beta$ :  [°]

**Released Objects**

Members No.:

Surfaces No.:

Solids No.:

Comment

**Location of Line Release**

Original line

Released line

**Generated Released Line No.**

Use these nodes as the definition nodes:

- Load Cases
- Load Combinations
- Result Combinations
- Loads
- Results
- Sections

Max u: 12.2, Min u: 0.0 mm

4.0 Results - Summary

LC1 - Full Load

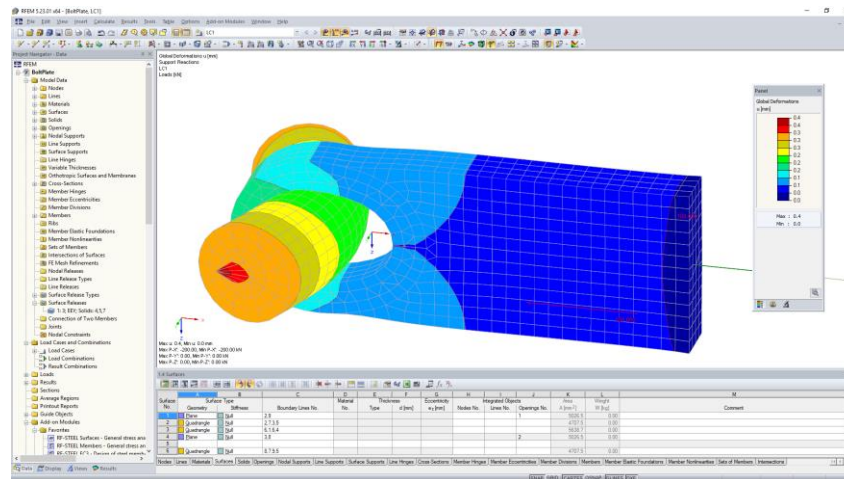






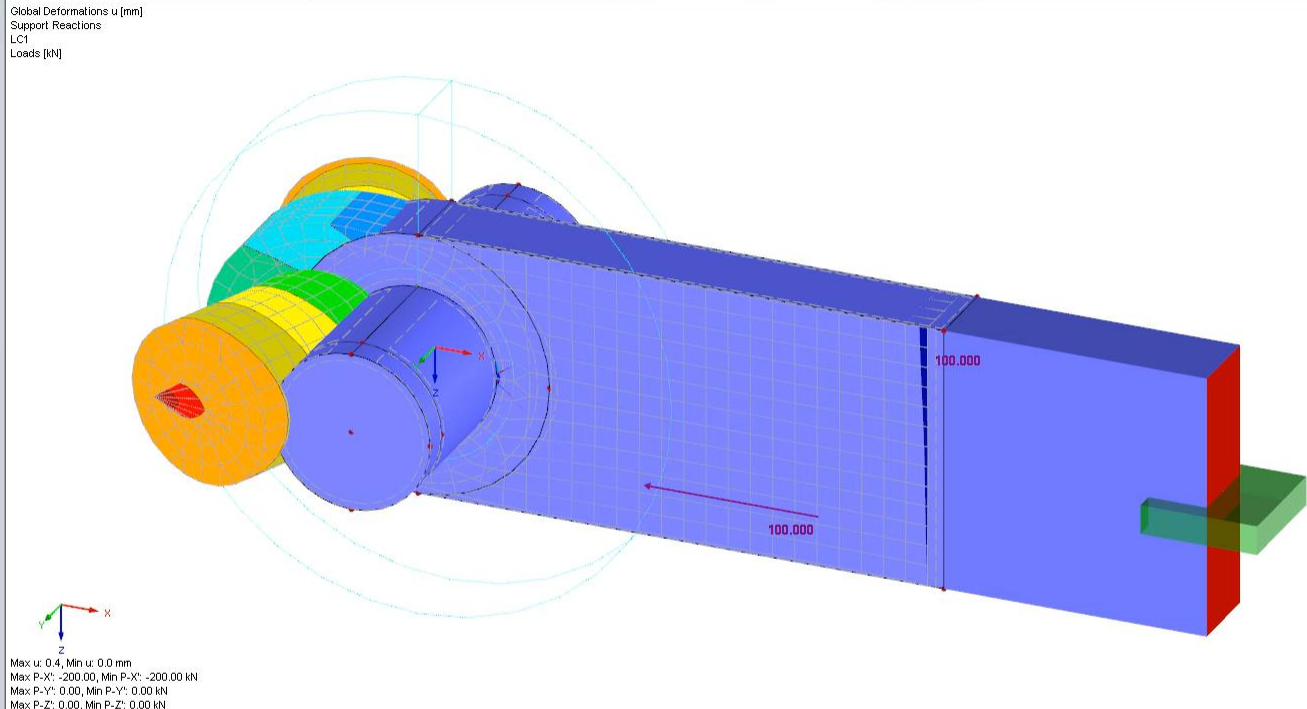
# Surface Releases

- Allow to model contact properties
- Includes Yielding and Tearing
- Slip
- Compression or Tension only
- Ideal for connection design

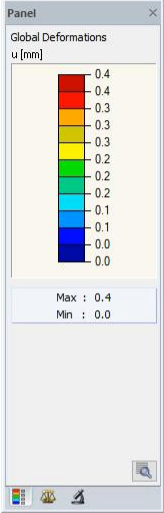




- Project Navigator - Display
- Model
    - Nodes
    - Lines
    - Surfaces
    - Solids
    - Solid Types
      - Solid Axis Systems x,y,z
      - Split Solids
    - Solid Orthotropies
    - Openings
    - Nodal Supports
    - Line Supports
    - Surface Supports
    - Members
    - Sets of Members
    - FE Mesh Refinements
    - Nodal Releases
    - Line Releases
    - Surface Releases
    - Nodal Constraints
  - Loads
    - Load Values
      - Units
      - Load Case Numbers
      - Load Case Descriptions
      - Show Load Values at Surface Center
    - Title Info
    - Self-weight
    - Object Loads
      - Nodal Loads
      - Member Loads
      - Line Loads
      - Surface Loads
      - Solid Loads
    - Free Loads
      - Free Concentrated Loads
      - Free Line Loads
      - Free Rectangular Loads
      - Free Polygon Loads
      - Free Circular Loads
      - Free Variable Loads
      - Influence Zones
      - Intersections
    - Other Loads
      - Imposed Nodal Deformations
      - Imposed Line Deformations
      - Imperfections
      - Generated Loads
      - Separately
      - Action Category Prev Press



Max u: 0.4, Min u: 0.0 mm  
 Max P-X: -200.00, Min P-X: -200.00 kN  
 Max P-Y: 0.00, Min P-Y: 0.00 kN  
 Max P-Z: 0.00, Min P-Z: 0.00 kN



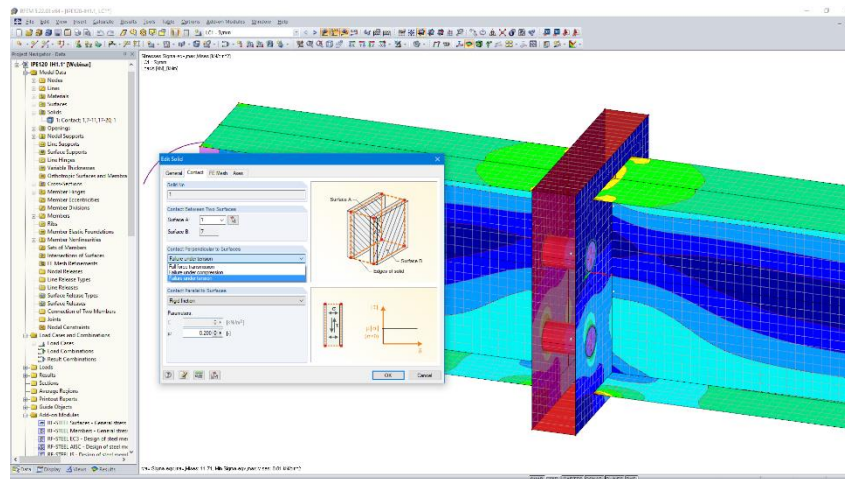
1.4 Surfaces

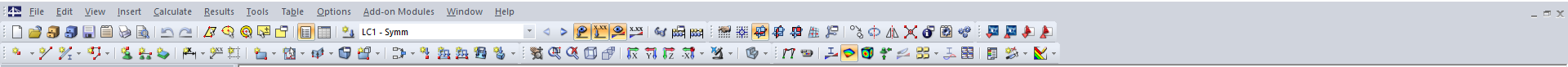
Surface No.	Geometry	Surface Type	Stiffness	Boundary Lines No.	Material No.	Thickness Type	Thickness d [mm]	Eccentricity e <sub>z</sub> [mm]	Nodes No.	Integrated Objects Lines No.	Openings No.	Area A [mm <sup>2</sup> ]	Weight W [kg]	Comment
1	Plane	Null		2,9							1	5026.5	0.00	
2	Quadrangle	Null		2,7,3,5								4707.5	0.00	
3	Quadrangle	Null		6,1,6,4								5638.7	0.00	
4	Plane	Null		3,8							2	5026.5	0.00	
5														
6	Quadrangle	Null		8,7,9,5								4707.5	0.00	



# Solids with contact properties

- Not a standard feature in other software
- Not expected in building software
- You do not need ANSYS, Abaqus or other typical FEA Software next to RFEM





Project Navigator - Data

- IP120-IH1.1\* [Webinar]
  - Model Data
    - Nodes
    - Lines
    - Materials
    - Surfaces
    - Solids
    - 1: Contact, 1,7-11,17-20; 1
    - Openings
    - Nodal Supports
    - Line Supports
    - Surface Supports
    - Line Hinges
    - Variable Thicknesses
    - Orthotropic Surfaces and Membr
  - Cross-Sections
  - Member Hinges
  - Member Eccentricities
  - Member Divisions
  - Members
    - Ribs
    - Member Elastic Foundations
  - Member Nonlinearities
  - Sets of Members
  - Intersections of Surfaces
  - FE Mesh Refinements
    - Nodal Releases
    - Line Release Types
    - Line Releases
    - Surface Release Types
    - Surface Releases
    - Connection of Two Members
    - Joints
  - Nodal Constraints
  - Load Cases and Combinations
    - Load Cases
    - Load Combinations
    - Result Combinations
  - Loads
  - Results
  - Sections
  - Average Regions
  - Printout Reports
  - Guide Objects
  - Add-on Modules
    - RF-STEEL Surfaces - General stress
    - RF-STEEL Members - General stres
    - RF-STEEL EC3 - Design of steel me
    - RF-STEEL AISC - Design of steel me
    - RF-STPFI IS - Design of steel mem

Stresses Sigma-*eqv*,max,Mises [kN/cm<sup>2</sup>]  
LC1 : Symm  
Loads [kN], [kNm]

Edit Solid

General Contact FE Mesh Axes

Solid No.: 1

Contact Between Two Surfaces

Surface A: 1

Surface B: 7

Contact Perpendicular to Surfaces

- Failure under tension
- Full force transmission
- Failure under compression
- Failure under tension

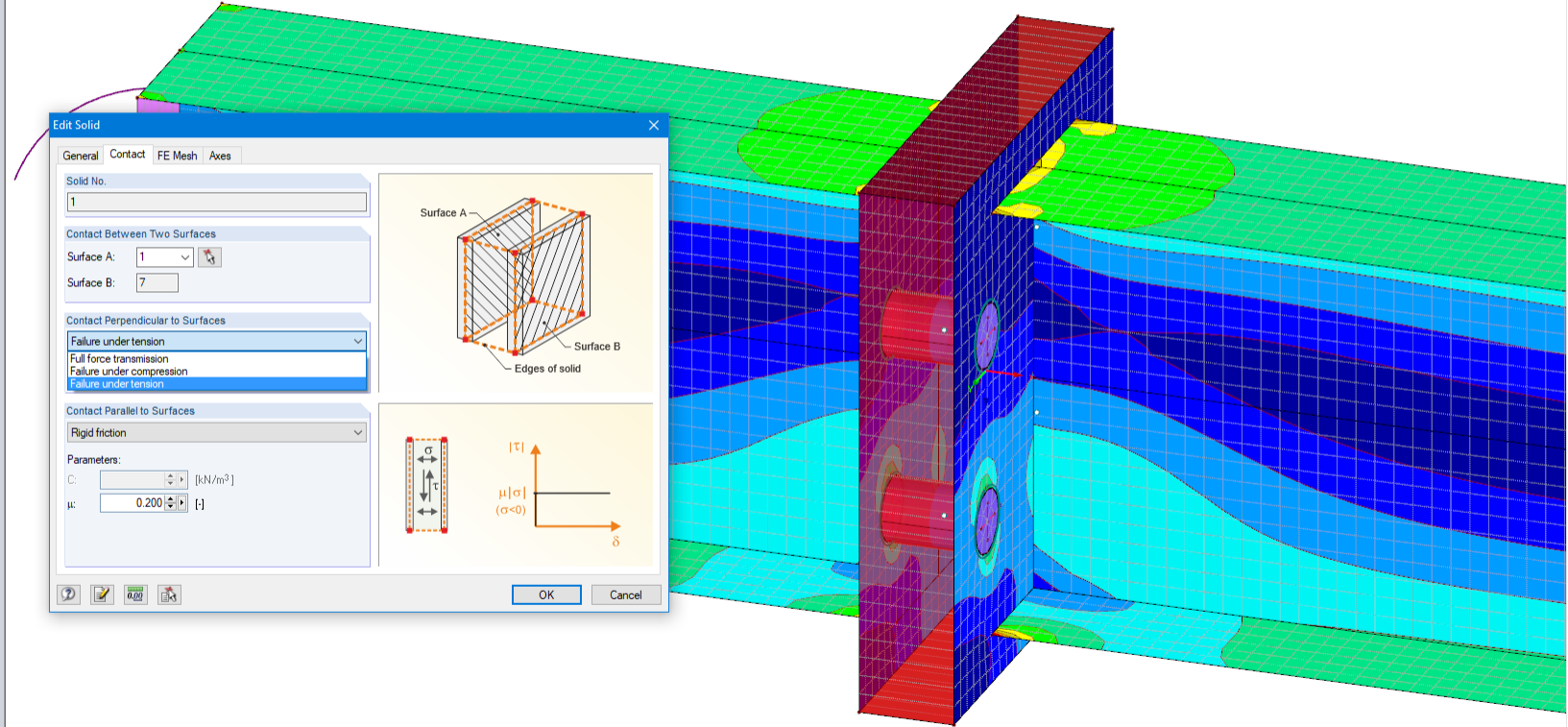
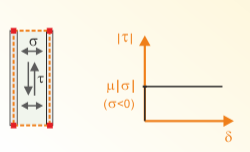
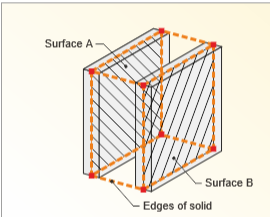
Contact Parallel to Surfaces

Rigid friction

Parameters:

C: [kN/m<sup>3</sup>]

$\mu$ : 0.200 [-]

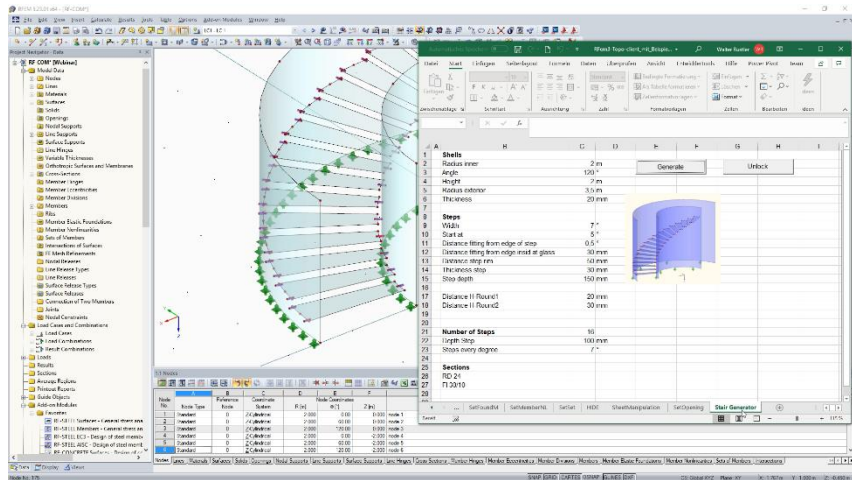


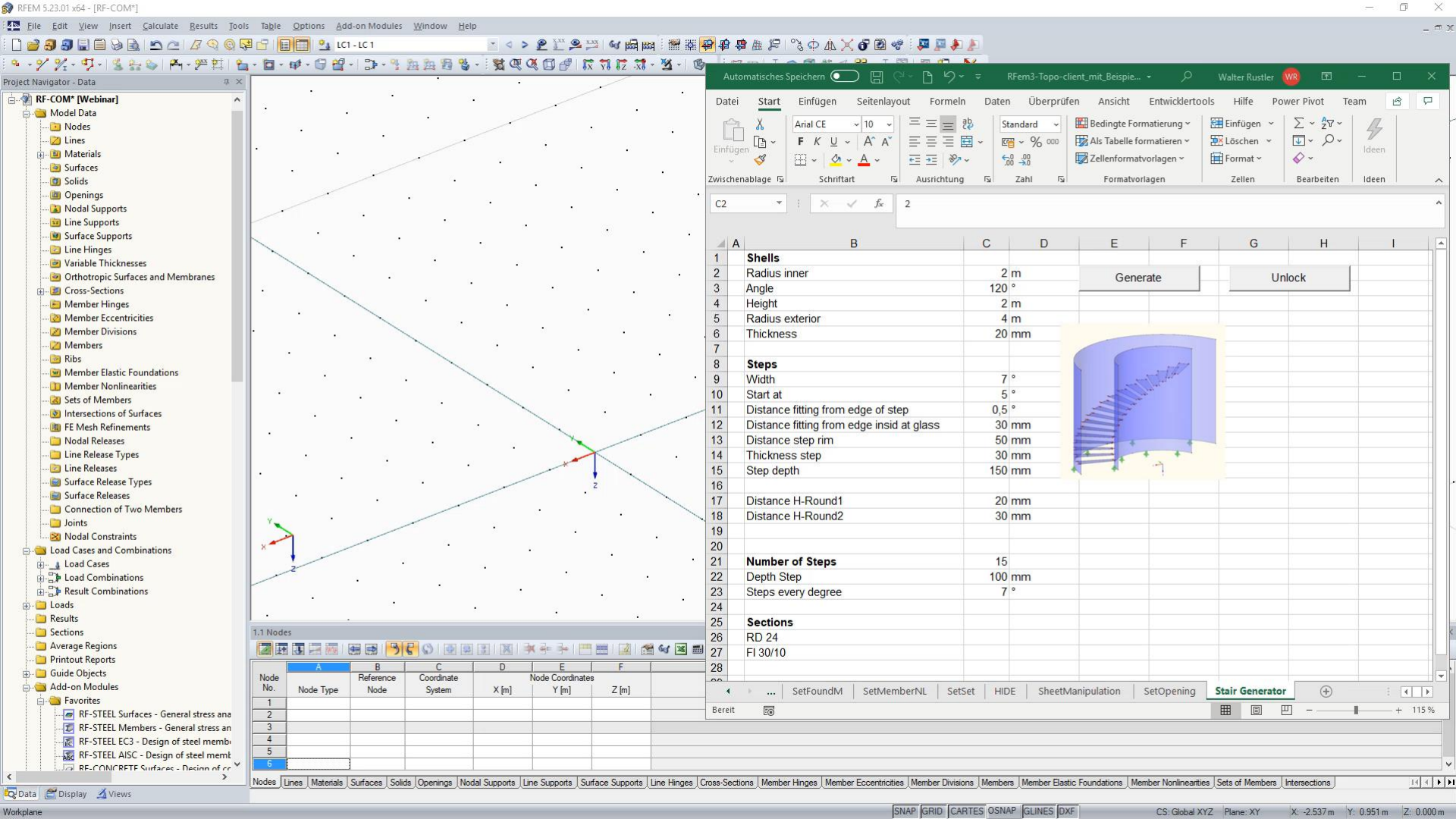
Max Sigma-*eqv*,max,Mises: 11.71, Min Sigma-*eqv*,max,Mises: 0.01 kN/cm<sup>2</sup>



# Programable Interface API (RF-COM)

- Allows to automatize workflows
- Create, modify and get data
- Ideal for program customization and your own structure generators or design modules
- Compatible with MS-Excel, C++, C# etc.

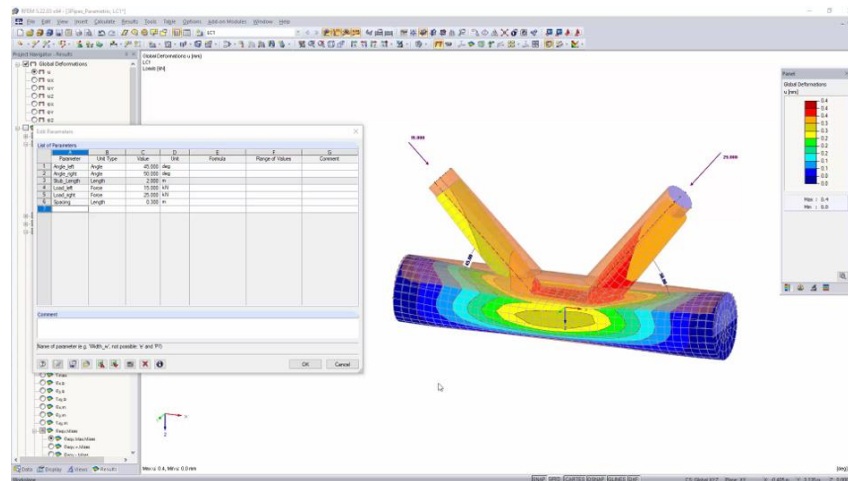






# Parametric models

- For structural and load data
- Allows intelligent and automated modification of structures
- No programming skills required
- For simple reoccurring analysis tasks





Project Navigator - Results

- Global Deformations
  - u
  - ux
  - uy
  - uz
  - qx
  - qy
  - qz

Global Deformations u [mm]  
LC1  
Loads [kN]

Edit Parameters

List of Parameters

Parameter	Unit Type	Value	Unit	Formula	Range of Values	Comment
1 Angle_left	Angle	45.000	deg			
2 Angle_right	Angle	50.000	deg			
3 Stub_Length	Length	2.000	m			
4 Load_left	Force	15.000	kN			
5 Load_right	Force	25.000	kN			
6 Spacing	Length	0.300	m			

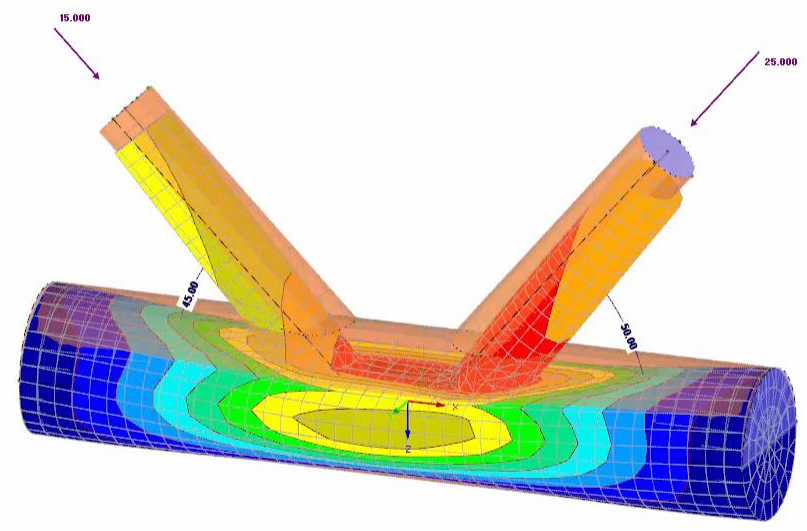
Comment

Name of parameter (e.g. 'Width\_w', not possible: 'e' and 'PI')

OK Cancel

- tmx
- Gx.b
- Gy.b
- txy.b
- Gx.m
- Gy.m
- txy.m
- Geqv.Mises
- Geqv.Max.Mises
- Geqv.+Mises
- Genv.-Mises

Max u: 0.4, Min u: 0.0 mm



Panel

Global Deformations  
u [mm]

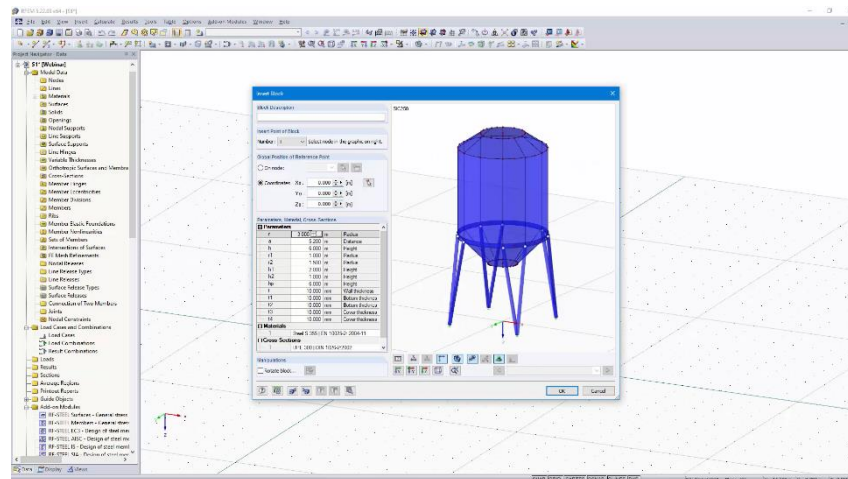
Max : 0.4  
Min : 0.0

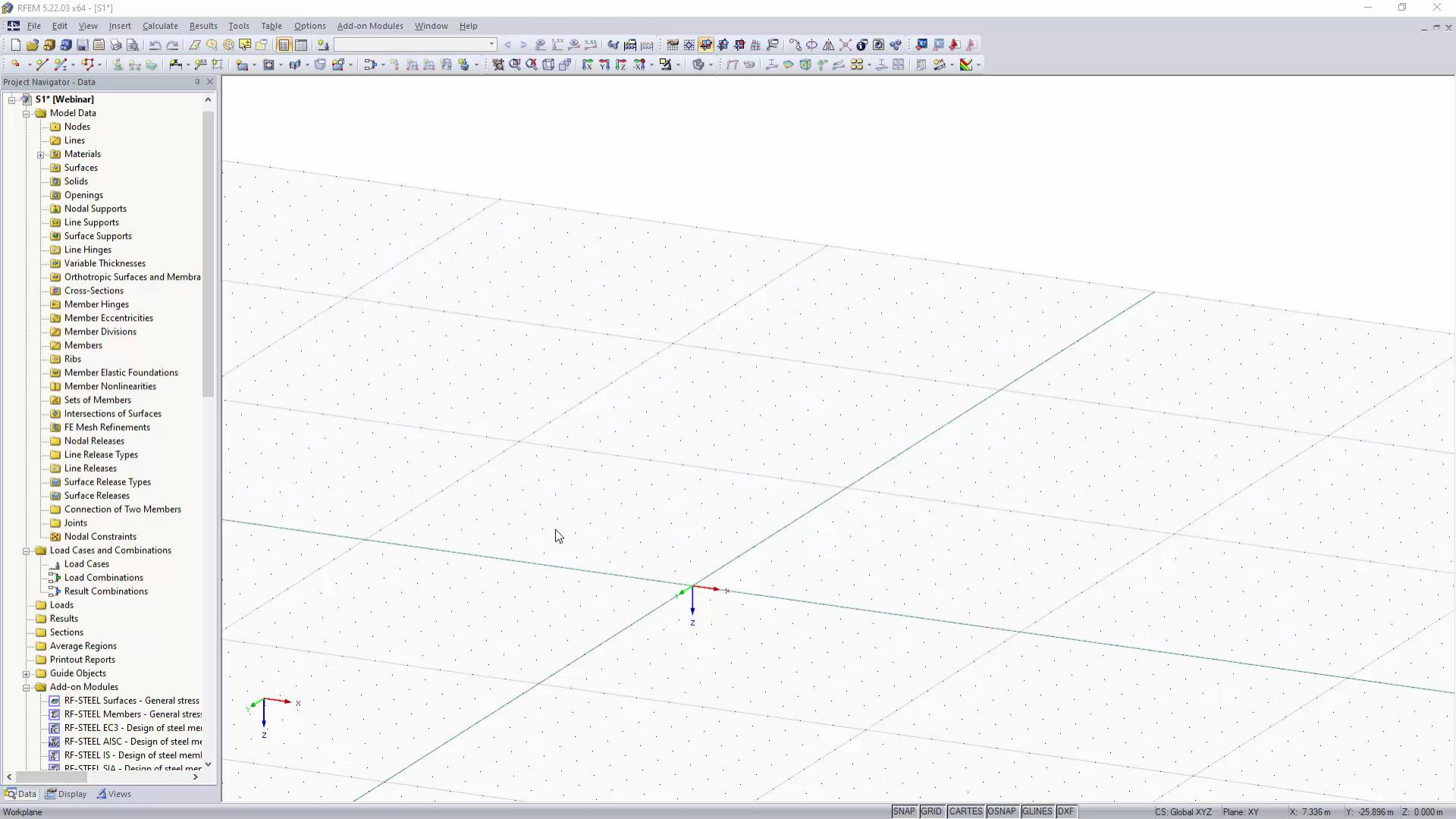




# Parametric models library

- Part of standard installation
- Full parametric models of typical structures like frames, trusses, silos
- Optional use for merging of structures

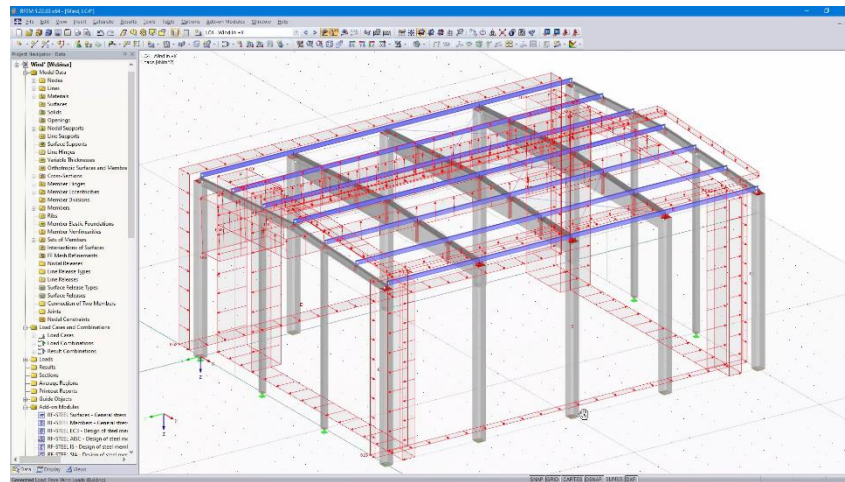






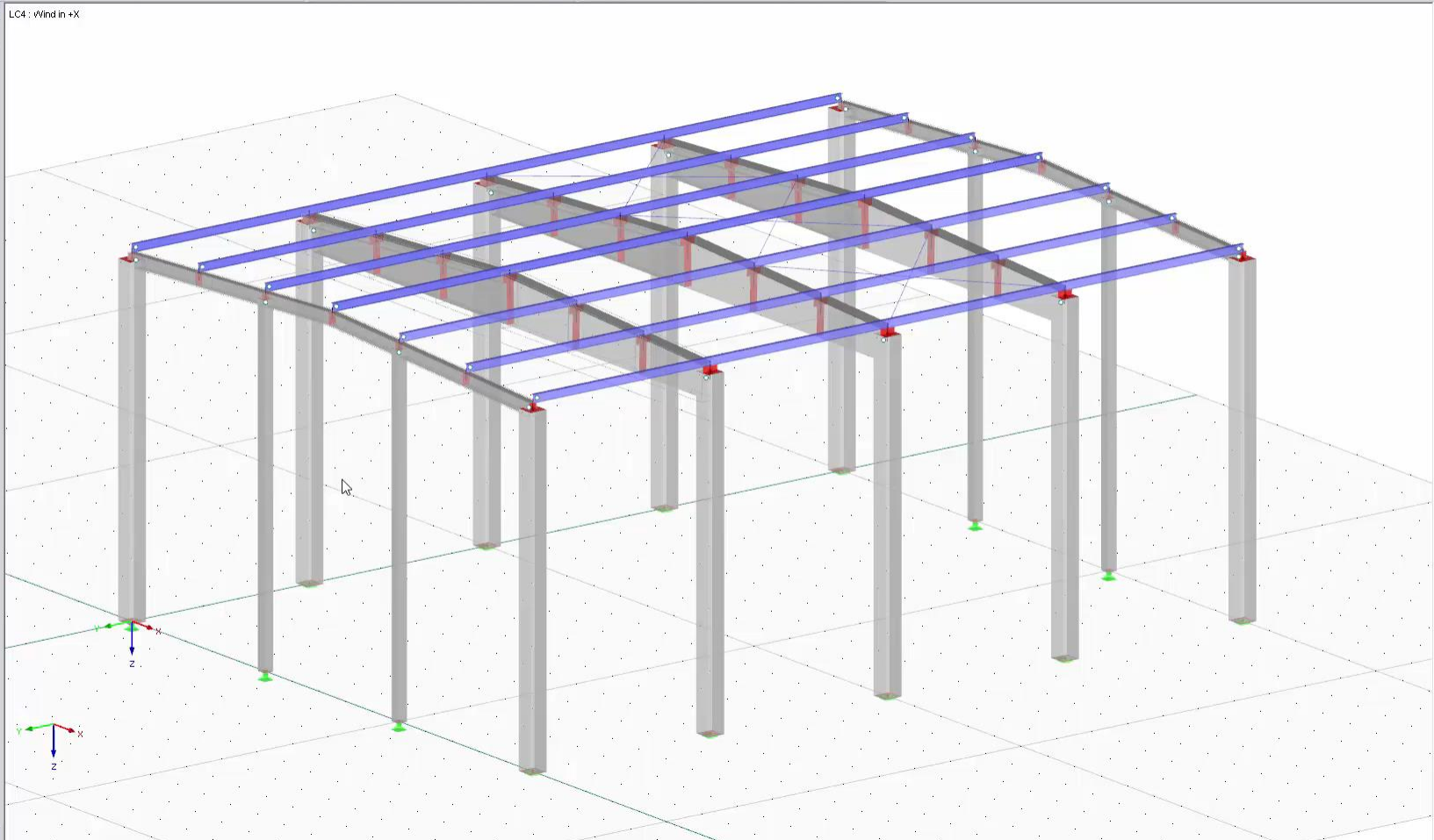
# Windload generators for 3D buildings

- Creates wind profiles for regular buildings
- Distributes area loads to member loads based on Eurocode et al...
- Intelligent – when structure changes load is recalculated





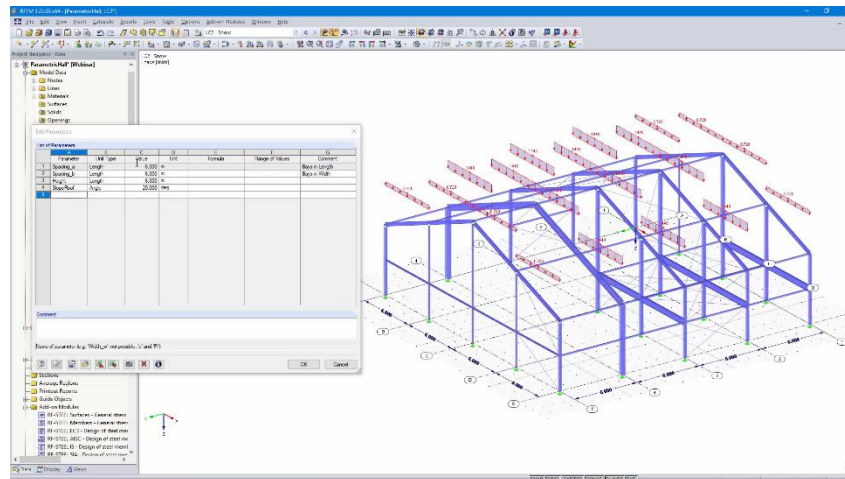
- Project Navigator - Data
- Wind\* [Webinar]
    - Model Data
      - Nodes
      - Lines
      - Materials
      - Surfaces
      - Solids
      - Openings
    - Nodal Supports
      - Line Supports
      - Surface Supports
      - Line Hinges
      - Variable Thicknesses
    - Orthotropic Surfaces and Membr...
    - Cross-Sections
    - Member Hinges
    - Member Eccentricities
    - Member Divisions
    - Members
      - Ribs
      - Member Elastic Foundations
      - Member Nonlinearities
    - Sets of Members
    - Intersections of Surfaces
    - FE Mesh Refinements
    - Nodal Releases
      - Line Release Types
      - Line Releases
      - Surface Release Types
      - Surface Releases
    - Connection of Two Members
    - Joints
    - Nodal Constraints
    - Load Cases and Combinations
      - Load Cases
      - Load Combinations
      - Result Combinations
    - Loads
    - Results
    - Sections
    - Average Regions
    - Printout Reports
    - Guide Objects
    - Add-on Modules
      - RF-STEEL Surfaces - General stress
      - RF-STEEL Members - General stres
      - RF-STEEL EC3 - Design of steel me
      - RF-STEEL AISC - Design of steel me
      - RF-STEEL IS - Design of steel meml
      - RF-STEEL SIA - Design of steel mer





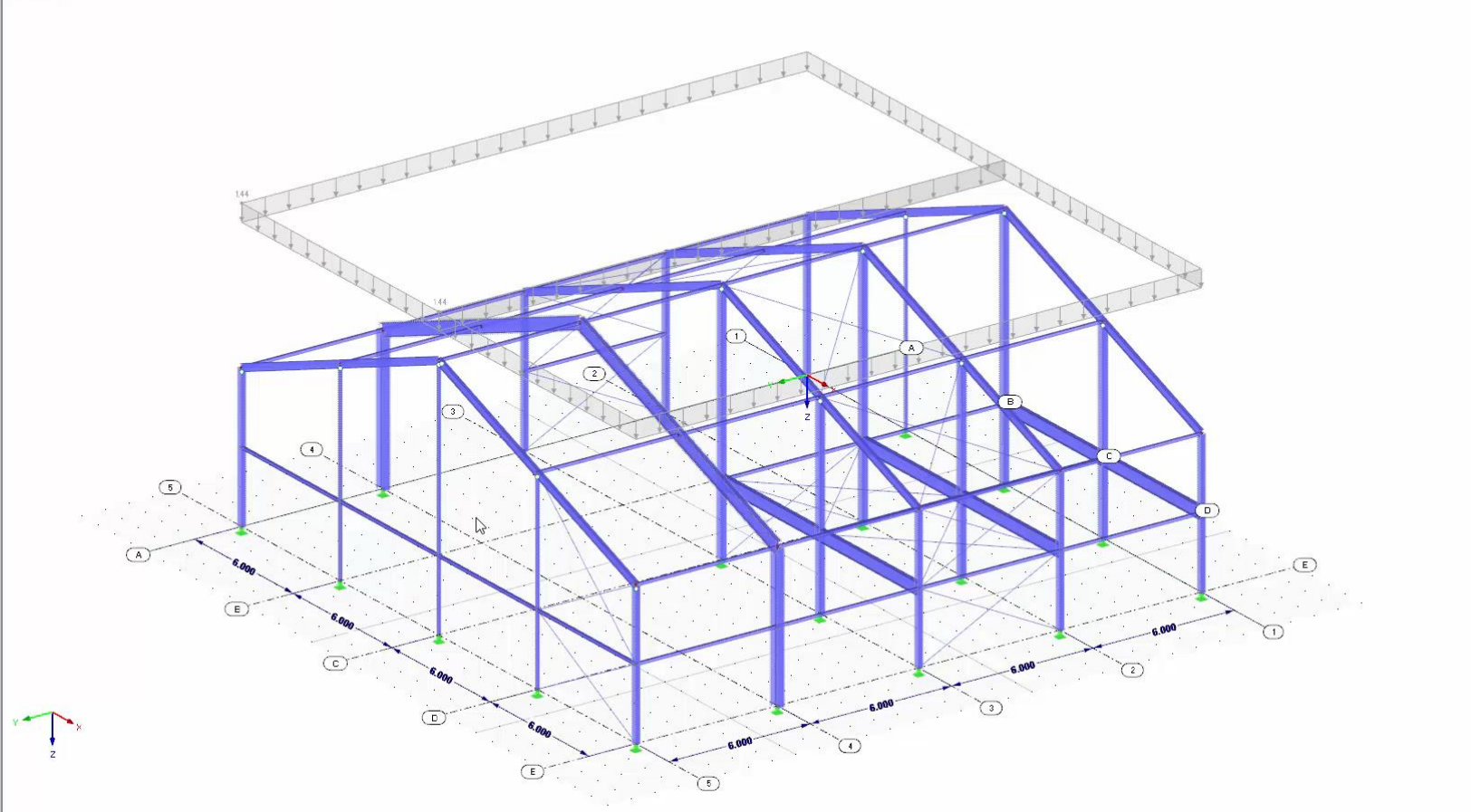
# Snow load generators for 3D buildings

- Creates snow loads for regular buildings
- Distributes area loads to member loads based on Eurocode et al...
- Intelligent – when structure changes load is recalculated



- ParametricHall [Webinar]
  - Model Data
    - Nodes
    - Lines
    - Materials
    - Surfaces
    - Solids
    - Openings
    - Nodal Supports
      - Line Supports
      - Surface Supports
      - Line Hinges
      - Variable Thicknesses
      - Orthotropic Surfaces and Membr
    - Cross-Sections
    - Member Hinges
    - Member Eccentricities
    - Member Divisions
    - Members
    - Ribs
    - Member Elastic Foundations
    - Member Nonlinearities
  - Sets of Members
    - Intersections of Surfaces
    - FE Mesh Refinements
    - Nodal Releases
    - Line Release Types
    - Line Releases
    - Surface Release Types
    - Surface Releases
    - Connection of Two Members
    - Joints
    - Nodal Constraints
  - Load Cases and Combinations
    - Load Cases
    - Load Combinations
    - Result Combinations
  - Loads
  - Results
  - Sections
  - Average Regions
  - Printout Reports
  - Guide Objects
  - Add-on Modules
    - RF-STEEL Surfaces - General stress
    - RF-STEEL Members - General stres
    - RF-STEEL EC3 - Design of steel me
    - RF-STEEL AISC - Design of steel m
    - RF-STEEL IS - Design of steel meml
    - RF-STEELI SIA - Design of steel mer

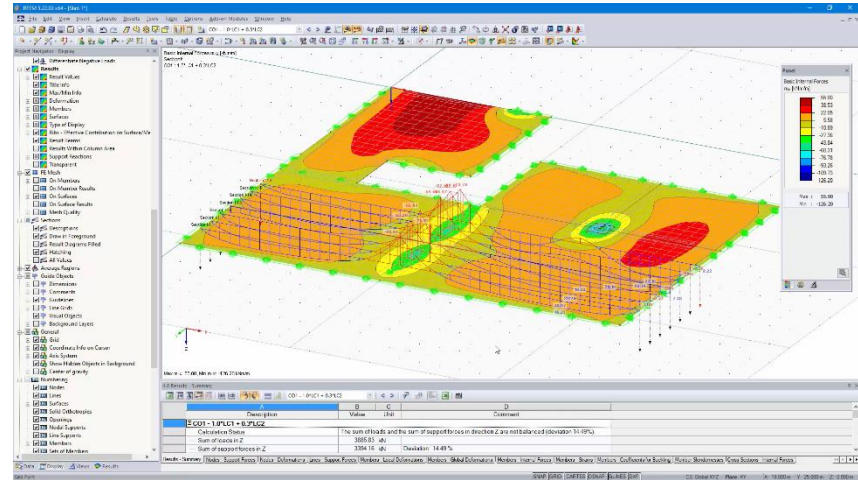
LC2 : Snow Loads [kN/m<sup>2</sup>]



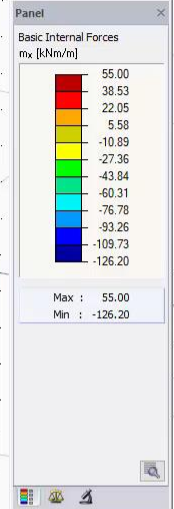
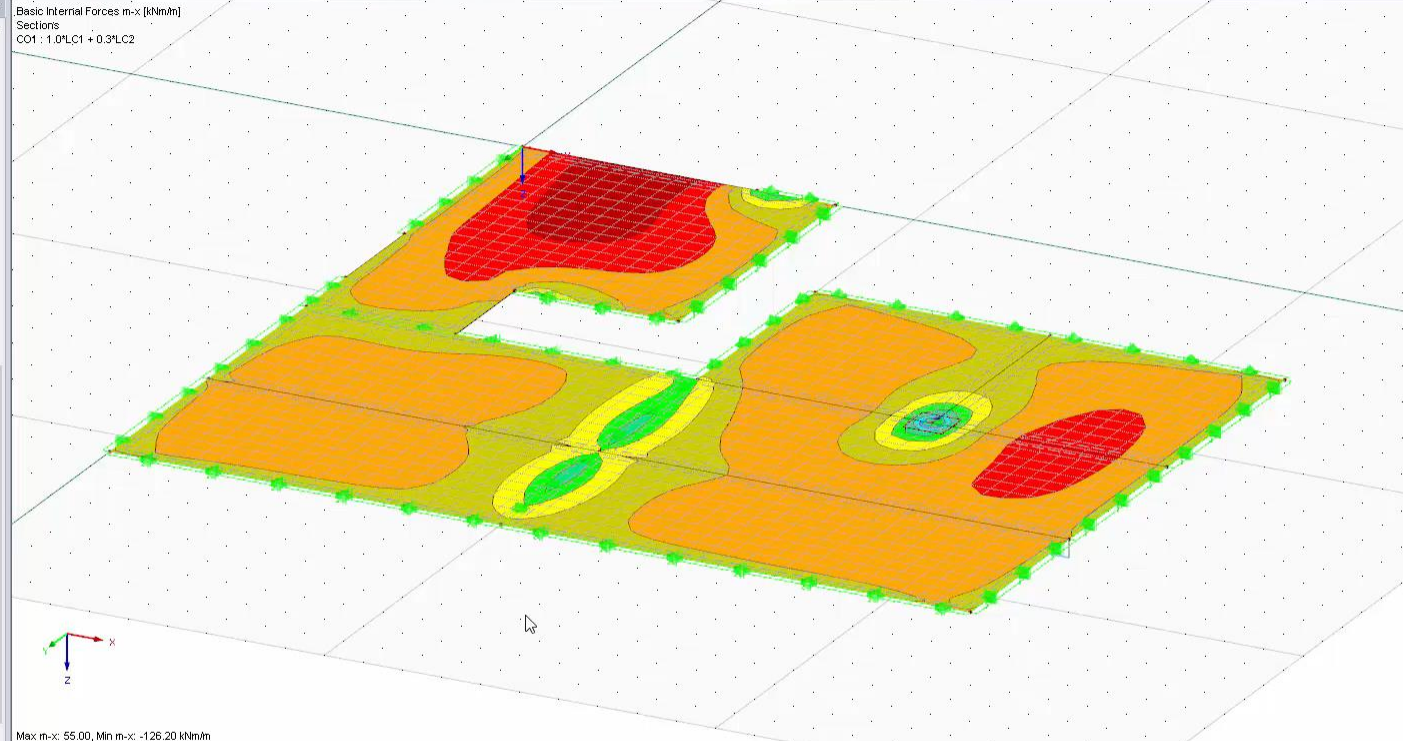


# Results Diagram on surfaces

- Easy to generate, object oriented
- Option to smooth results in region
- Evaluate results like engineers expect it



- Project Navigator - Display
- Differentiate Negative Loads
  - Results
    - Result Values
    - Title Info
    - Max/Min Info
    - Deformation
    - Members
    - Surfaces
    - Type of Display
    - Ribs - Effective Contribution on Surface/Me
    - Result Beams
    - Results Within Column Area
    - Support Reactions
    - Transparent
  - FE Mesh
    - On Members
    - On Member Results
    - On Surfaces
    - On Surface Results
    - Mesh Quality
  - Sections
    - Descriptions
    - Draw in Foreground
    - Result Diagrams Filled
    - Hatching
    - All Values
  - Average Regions
  - Guide Objects
    - Dimensions
    - Comments
    - Guidelines
    - Line Grids
    - Visual Objects
    - Background Layers
  - General
    - Grid
    - Coordinate Info on Cursor
    - Axis System
    - Show Hidden Objects in Background
    - Center of gravity
  - Numbering
    - Nodes
    - Lines
    - Surfaces
    - Solid Orthotropies
    - Openings
    - Nodal Supports
    - Line Supports
    - Members
    - Sets of Members



4.0 Results - Summary

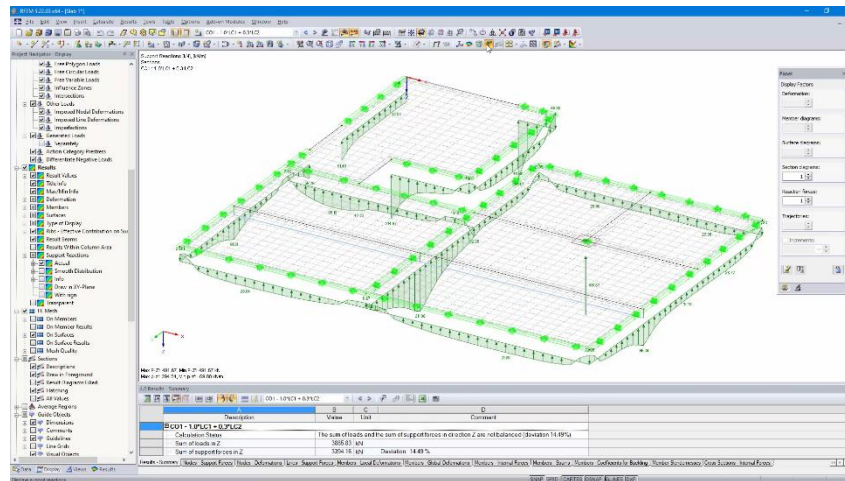
A	B	C	D
Description	Value	Unit	Comment
<b>CO1 - 1.0*LC1 + 0.3*LC2</b>			
Calculation Status	The sum of loads and the sum of support forces in direction Z are not balanced (deviation 14.49%).		
Sum of loads in Z	3885.83	kN	
Sum of support forces in Z	3394.16	kN	Deviation: 14.49 %





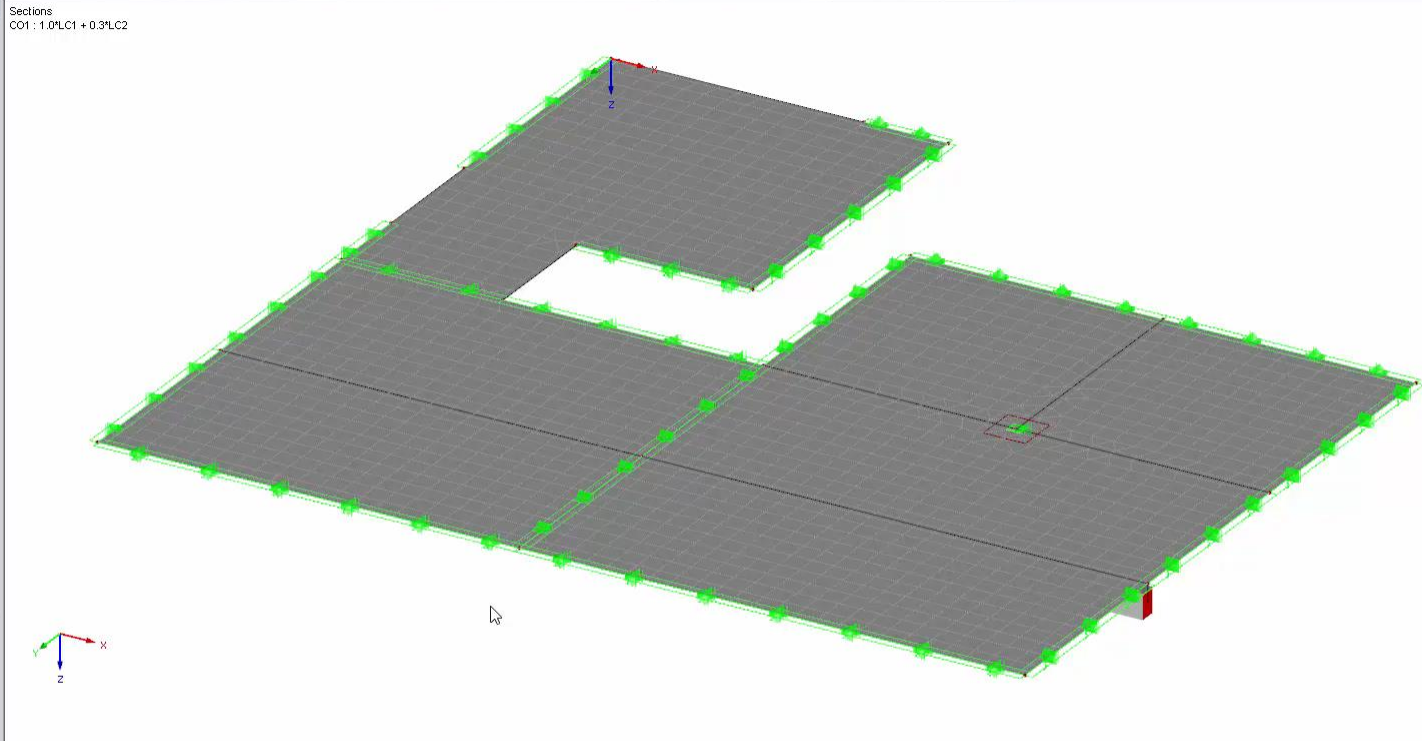
# Support Reactions

- Average values on line supports
- Display of results in plane
- Sum and center of forces
- Results Diagram on supports





- Project Navigator - Display
- Free Polygon Loads
  - Free Circular Loads
  - Free Variable Loads
  - Influence Zones
  - Intersections
  - Other Loads
  - Imposed Nodal Deformations
  - Imposed Line Deformations
  - Imperfections
  - Generated Loads
  - Separately
  - Action Category Prestress
  - Differentiate Negative Loads
  - Results
    - Result Values
    - Title Info
    - Max/Min Info
    - Deformation
    - Members
    - Surfaces
    - Type of Display
    - Ribs - Effective Contribution on Surfaces
    - Result Beams
    - Results Within Column Area
    - Support Reactions
      - Actual
      - Smooth Distribution
      - Info
      - Draw in XY-Plane
      - With sign
    - Transparent
  - FE Mesh
    - On Members
    - On Member Results
    - On Surfaces
    - On Surface Results
    - Mesh Quality
  - Sections
    - Descriptions
    - Draw in Foreground
    - Result Diagrams Filled
    - Hatching
    - All Values
  - Average Regions
  - Guide Objects
  - Dimensions
  - Comments
  - Guidelines
  - Line Grids
  - Visual Objects



Panel

Display Factors

Deformation: [Dropdown]

Member diagrams: [Dropdown]

Surface diagrams: [Dropdown]

Section diagrams: [Dropdown] 1

Reaction forces: [Dropdown]

Trajectories: [Dropdown]

Increments: [Dropdown]

[Icons]

4.0 Results - Summary

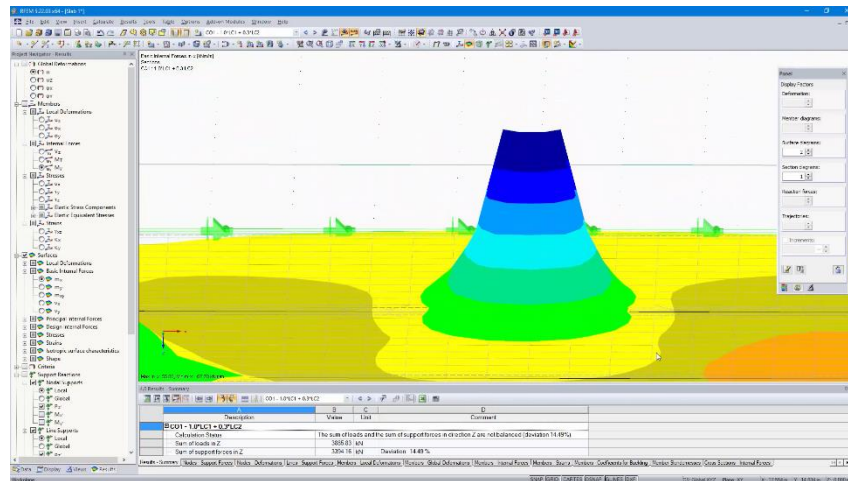
A	B	C	D
Description	Value	Unit	Comment
<b>CO1 - 1.0*LC1 + 0.3*LC2</b>			
Calculation Status	The sum of loads and the sum of support forces in direction Z are not balanced (deviation 14.49%).		
Sum of loads in Z	3885.83	kN	
Sum of support forces in Z	3394.16	kN	Deviation: 14.49 %

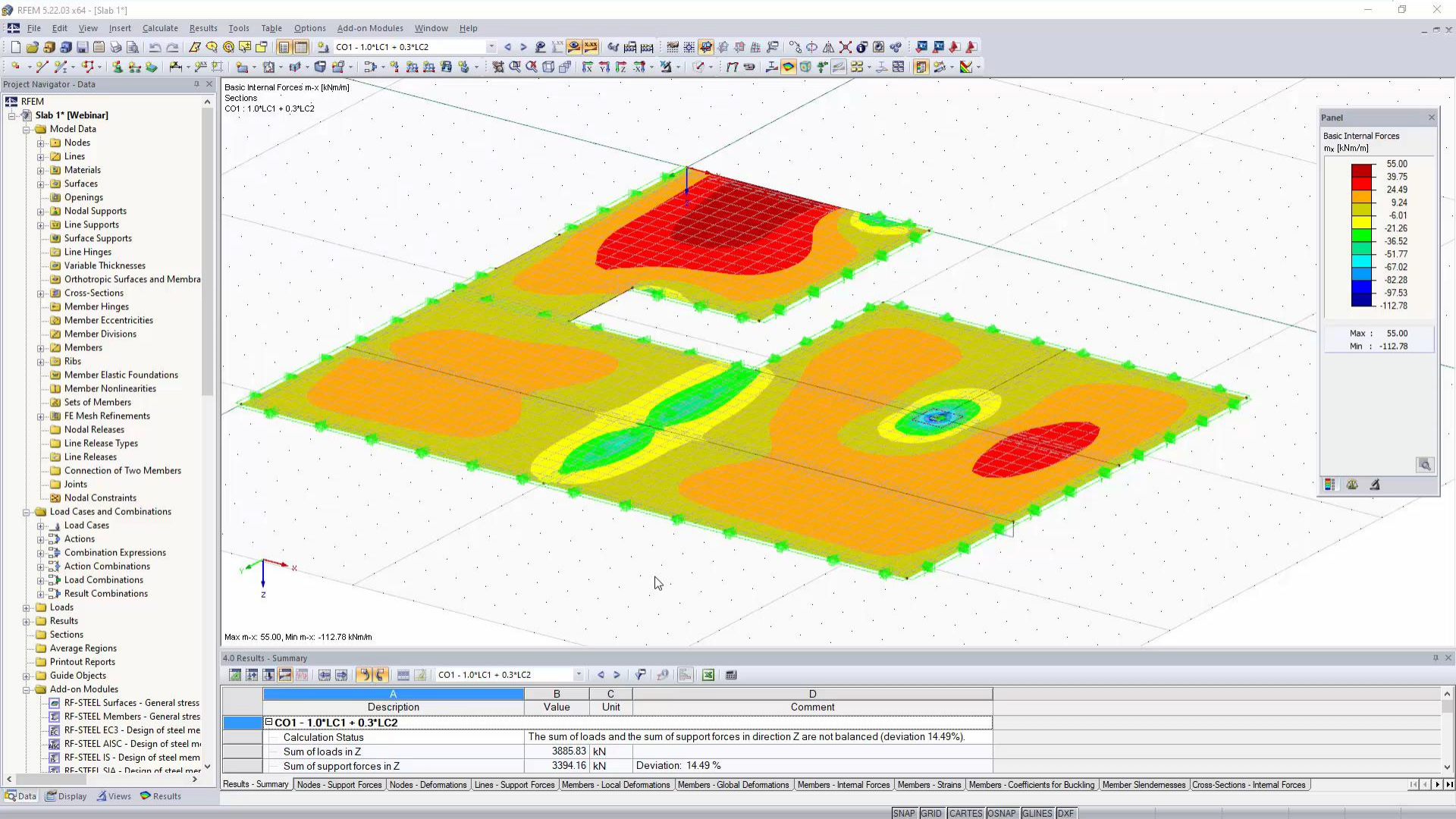
Results - Summary | Nodes - Support Forces | Nodes - Deformations | Lines - Support Forces | Members - Local Deformations | Members - Global Deformations | Members - Internal Forces | Members - Strains | Members - Coefficients for Buckling | Member Slendernesses | Cross-Sections - Internal Forces



# Average Regions

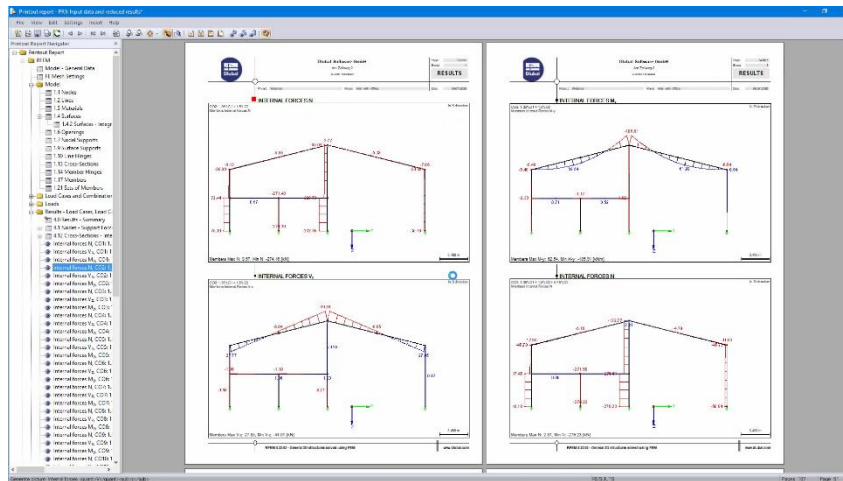
- Smooth values on singular regions
- Avoid undesignable regions
- Practical solution for engineers
- Avoid manual result explanations





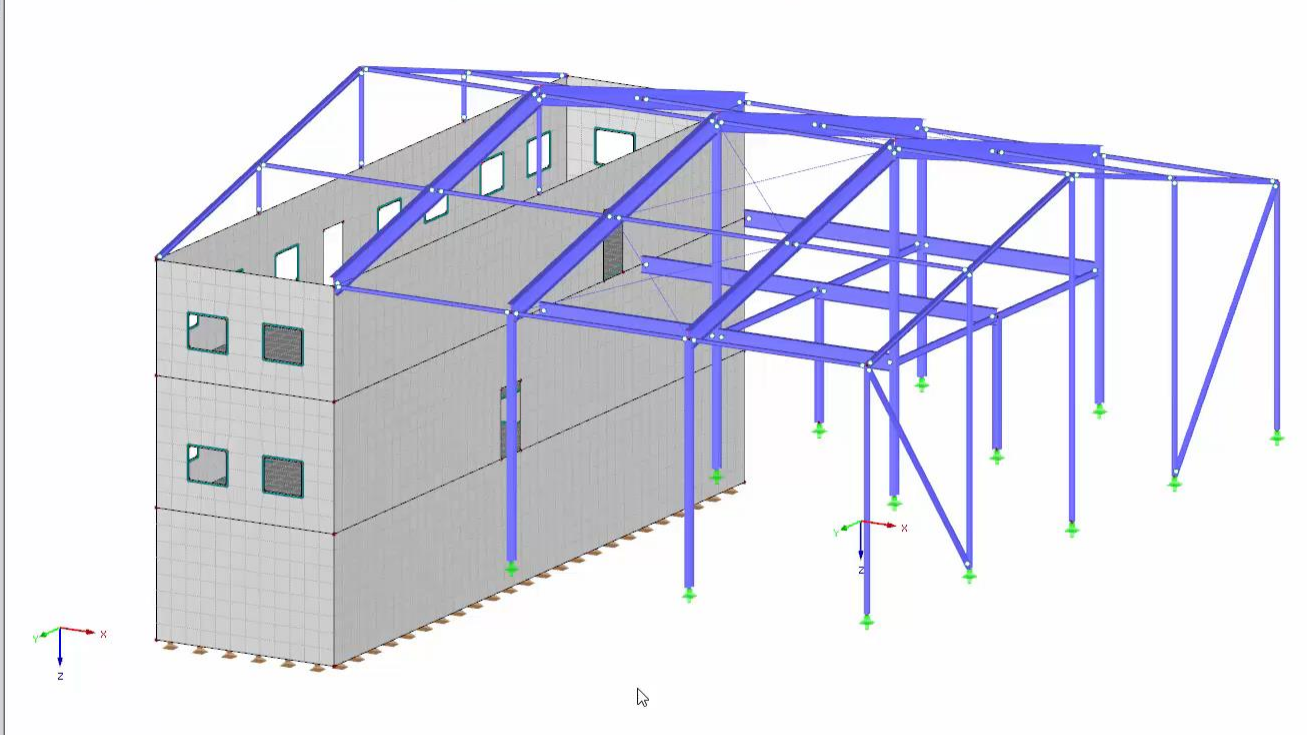
# Mass Print

- Creates automatically graphics for report
- Automatic and optimal arrangement on paper
- Graphics are linked to model and always up to date
- Avoids time consuming individual printing of images





- Project Navigator - Data
- Hall\_with\_Office\* [Webinar]
    - Model Data
      - Nodes
      - Lines
      - Materials
      - Surfaces
      - Solids
      - Openings
      - Nodal Supports
      - Line Supports
      - Surface Supports
      - Line Hinges
      - Variable Thicknesses
      - Orthotropic Surfaces and Membr
      - Cross-Sections
      - Member Hinges
      - Member Eccentricities
      - Member Divisions
      - Members
      - Ribs
      - Member Elastic Foundations
      - Member Nonlinearities
      - Sets of Members
      - Intersections of Surfaces
      - FE Mesh Refinements
      - Nodal Releases
      - Line Release Types
      - Line Releases
      - Surface Release Types
      - Surface Releases
      - Connection of Two Members
      - Joints
      - Nodal Constraints
    - Load Cases and Combinations
      - Load Cases
      - Actions
      - Combination Expressions
      - Action Combinations
      - Load Combinations
      - Result Combinations
    - Loads
    - Results
    - Sections
    - Average Regions
    - Printout Reports
    - Guide Objects
    - Add-on Modules
      - RF-STEEL Surfaces - General stress
      - RF-STEEL Members - General stress
      - RF-STEEL EC3 - Design of steel m



4.0 Results - Summary

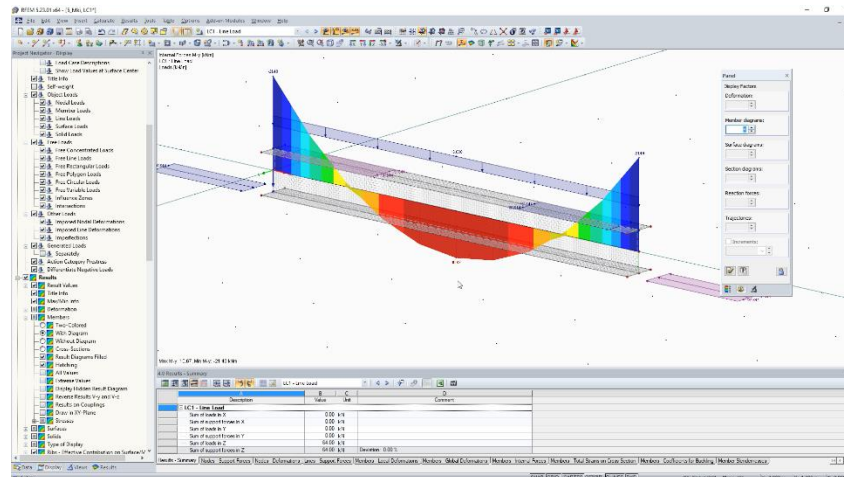
A	B	C	D
Description	Value	Unit	Comment
<b>CO1 - 1.35*LC1</b>			
Sum of loads in X	0.00	kN	
Sum of support forces in X	0.00	kN	
Sum of loads in Y	0.00	kN	

Results - Summary | Nodes - Support Forces | Nodes - Deformations | Members - Local Deformations | Members - Global Deformations | Members - Internal Forces | Members - Strains | Members - Coefficients for Buckling | Member Slendernesses | Set of Members - Internal Forces | Cross-Sections - Internal Forces



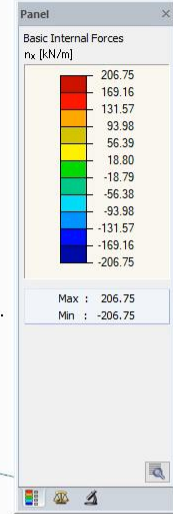
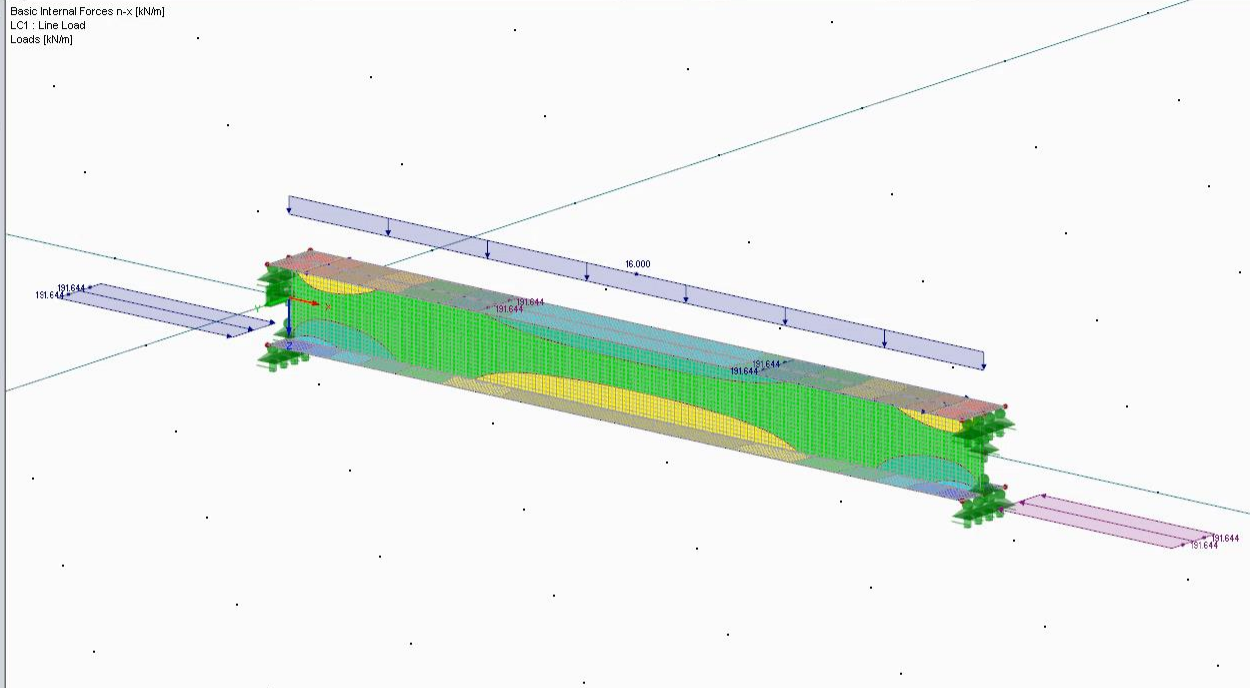
# Result Beam

- Integrates results of beams, surfaces, solids to a common beam result (N, M, V)
- Better understanding of forces and option to cross-check the analysis results
- Optional design as a member for surfaces/solids in RF-STEEL





- Project Navigator - Display
- Compression
  - Buckling
  - Cable
  - Cable on Pulleys
  - Result Beam
  - Definable Stiffness
  - Coupling Rigid-Rigid
  - Coupling Rigid-Hinge
  - Coupling Hinge-Rigid
  - Spring
  - Dashpot
  - Null
  - Member Elastic Foundations
  - Member Axis Systems x,y,z
  - Member Axis Systems x,u,v
  - Member Orientations
  - Member Hinges
  - Material Descriptions
  - Bottom Fibers
  - Cross-Section Outlines
  - Cross-Section Descriptions
  - Line Mode at Member Ends
  - Eccentricities
  - Cross-Sections of Result Beams
  - Integration Areas
  - Front Areas Distinguished by Color
  - Spring Members
  - Dashpot
  - Sets of Members
  - FE Mesh Refinements
  - Nodal Releases
  - Line Releases
  - Surface Releases
  - Nodal Constraints
  - Loads
    - Load Values
      - Units
      - Load Case Numbers
      - Load Case Descriptions
      - Show Load Values at Surface Center
    - Title Info
    - Self-weight
    - Object Loads
      - Nodal Loads
      - Member Loads
      - Line Loads
      - Surface Loads
      - Solid Loads
    - Free Loads



Max n-x: 206.75, Min n-x: -206.75 kN/m

4,0 Results - Summary

LC1 - Line Load

A	B	C	D
Description	Value	Unit	Comment
<b>LC1 - Line Load</b>			
Sum of loads in X	0.00	kN	
Sum of support forces in X	0.00	kN	
Sum of loads in Y	0.00	kN	
Sum of support forces in Y	0.00	kN	
Sum of loads in Z	64.00	kN	
Sum of support forces in Z	64.00	kN	Deviation: 0.00 %



# — Get further details about Dlubal



Visit website  
[www.dlubal.com](http://www.dlubal.com)

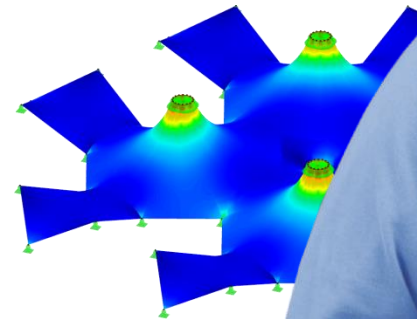
- Videos and recorded webinars
- Newsletters
- Events and conferences
- Knowledge Base articles



See Dlubal  
Software in  
action in a  
webinar



Download  
free trial  
license



**Dlubal Software GmbH**  
Am Zellweg 2,  
93464 Tiefenbach, Germany

Phone: +49 9673 9203-0  
E-mail: [info@dlubal.com](mailto:info@dlubal.com)



[www.dlubal.com](http://www.dlubal.com)