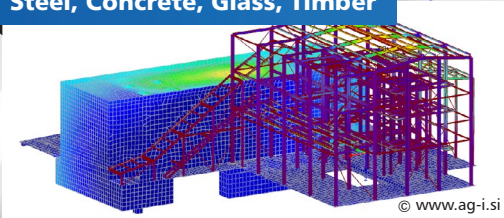


RFEM 5

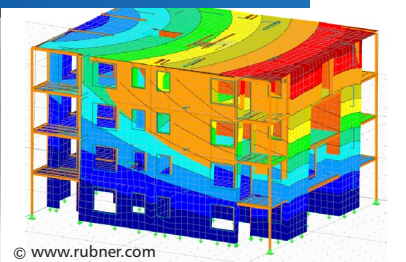
FEM Software for Structural and Dynamic Analysis

Steel, Concrete, Glass, Timber

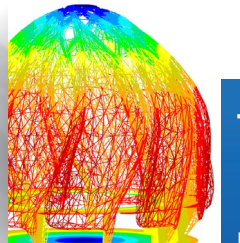


© www.ag-i.si

Stability and Dynamics



© www.rubner.com



© www.josef-gartner.de

3D Frameworks

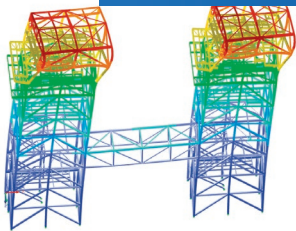
RFEM 5 – For Calculations in Civil, Mechanical and Plant Engineering.

Intuitive, efficient, universal, powerful.

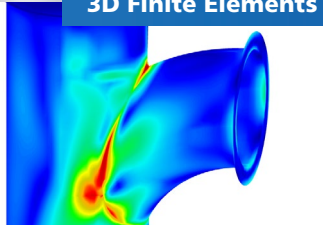
The RFEM program family is based on a modular system. The main program RFEM is used to define structures, materials and loads for planar and spatial structural systems consisting of plates, walls, shells and members. Creating combined systems as well as modeling solid and contact elements is also possible.

RFEM provides deformations, internal and support forces as well as soil contact stresses. For the subsequent design RFEM offers various add-on modules taking into account material- and standard-specific conditions. The modular approach allows you to combine all programs individually according to your needs. Upgrades at a later time are always possible. RFEM offering numerous interfaces represents the perfect tool for a smooth interaction between CAD and structural analysis in Building Information Modeling (BIM).

Plant Construction



3D Finite Elements

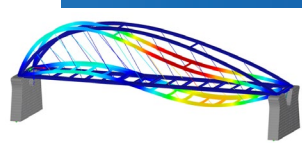


Solid Construction

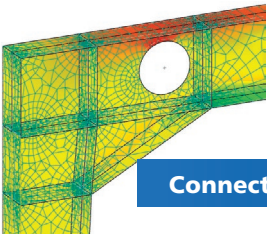


© www.ssp-muc.com

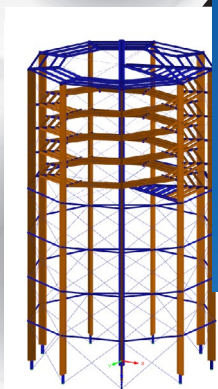
Steel Construction



Connections



With CAD Connection



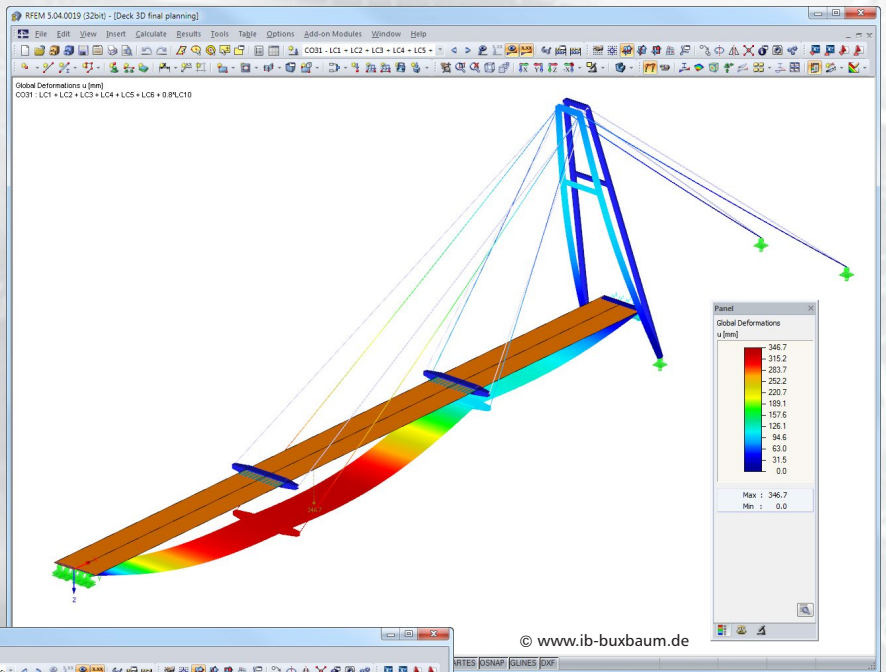
© www.wiehag.com

RFEM - the FEM tool for steel, reinforced concrete, glass and timber structures, plant construction, dynamics, aluminum, design according to Eurocodes / International Standards

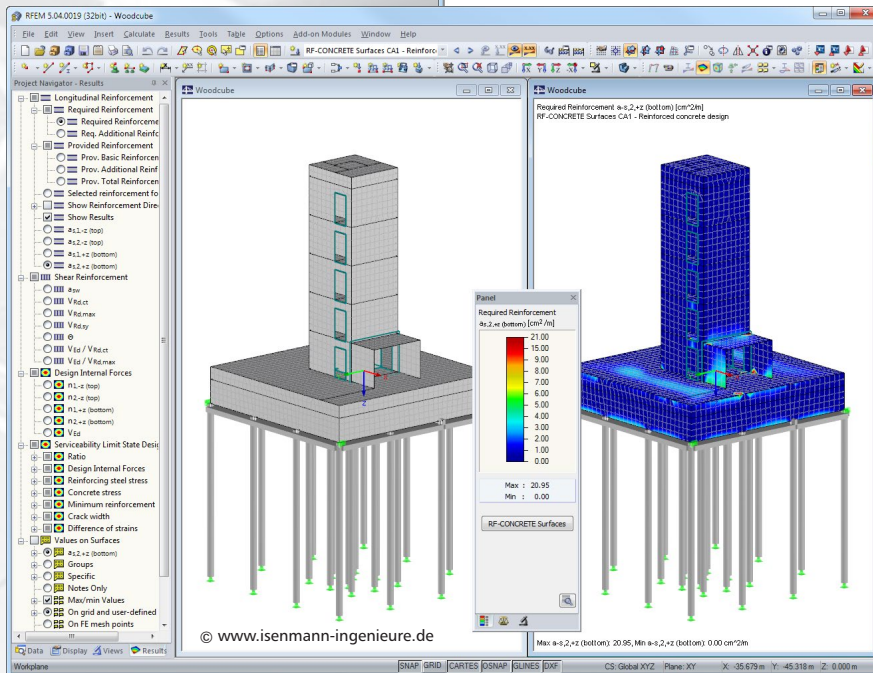


RFEM – the calculation program for universal use

- Calculation of supporting structures consisting of steel, concrete, timber, aluminum, glass and mixed systems
- Structural and dynamic analysis
- Member, surface and solid elements
- Integration of various international standards, continuous updating
- High-quality visualization of structure and loading
- Fire protection design according to Eurocode for steel, concrete and timber
- Continuous maintenance and program development
- Quick and qualified hotline support



© www.ib-buxbaum.de



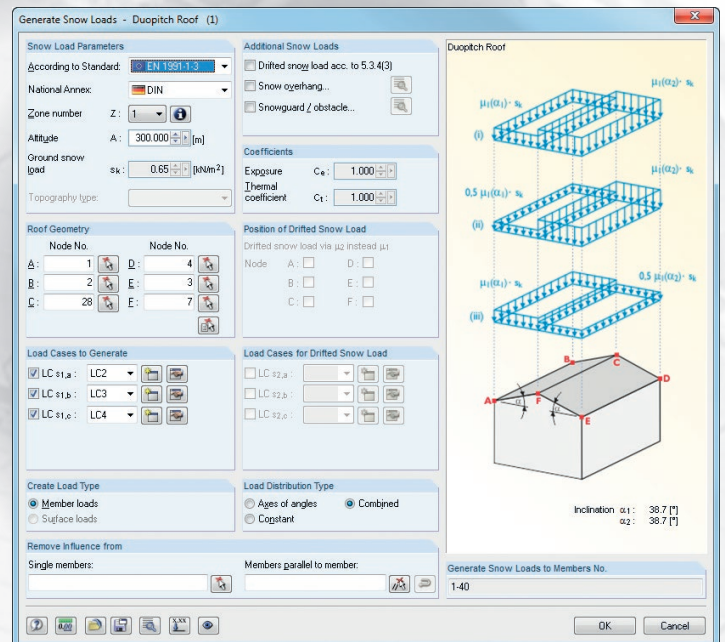
Graphical user interface

- Quick and easy program learning due to intuitive user interface
- Model input in table or directly in graphics
- Immediate structure check due to photo-realistic visualization in 3D rendering (either as solid model or with transparency)
- Free configuration of menus and toolbars
- Easy insertion of comments and dimensions
- Display control of objects in Display navigator
- Creation of structural and load data in nearly no time
- Multi-lingual program use



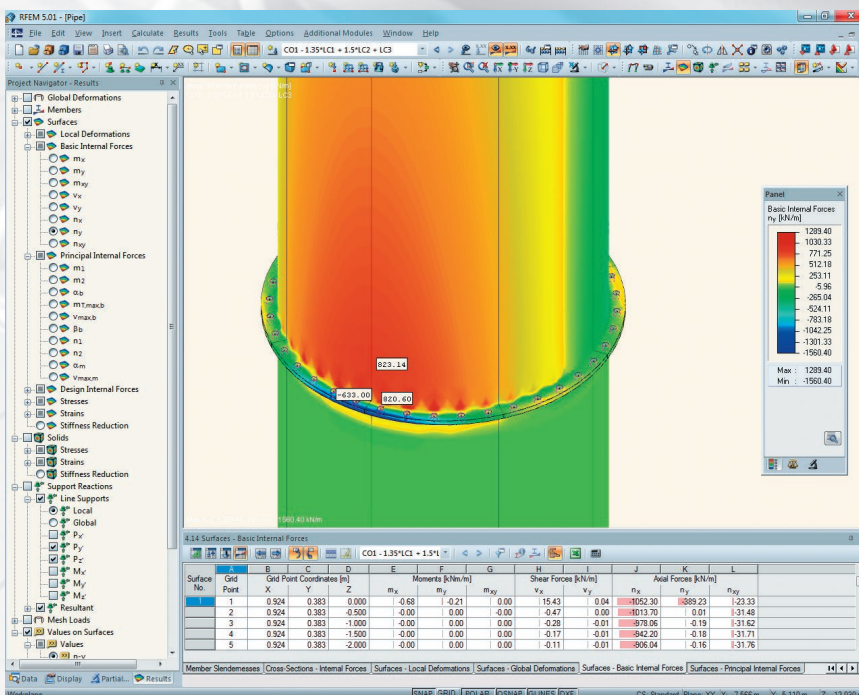
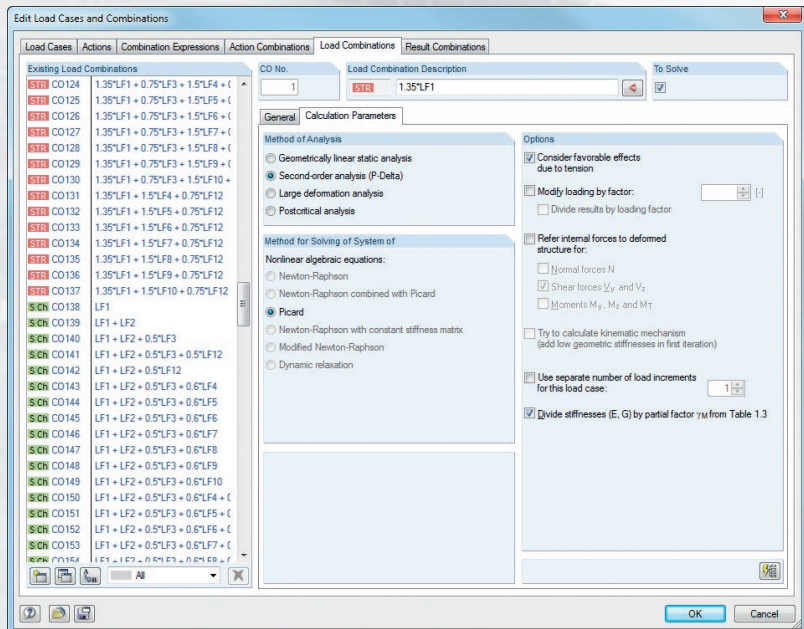
Modeling

- Useful tools for generating structures including loads
- Comprehensive cross-section and material library
- Design of special cross-sections created with SHAPE-THIN or SHAPE-MASSIVE
- Determination of imperfections according to Eurocode
- Wind and snow load generation according to Eurocode
- Easy property definition of grid and object snap
- Application of guidelines, line grids and background layers
- Efficient work using recurring structures by parameterized input
- Inserting and saving parametric structures as a block
- Input of member non-linearities, couplings and eccentricities for members and surfaces
- Large variety of member and surface types as well as material models
- Possibility to renumber structural objects automatically
- Detection and correction of input errors by structure check
- Various load types for member and surface loads



Calculation

- Linear calculation acc. to linear static analysis, or non-linear calculation acc. to second-order and large deformation analysis for all types of members and surfaces
- Automatic generation of action, load and result combinations according to selected combination rule, now directly integrated in RFEM
- Setting calculation parameters individually for load cases and combinations
- Analysis core with optimized multi-core processor technology and 64-bit technology
- Calculation of complex structures with quick and direct equation solver
- Option to consider non-linear effects such as failure of tension members, plastic hinges, slippage etc.



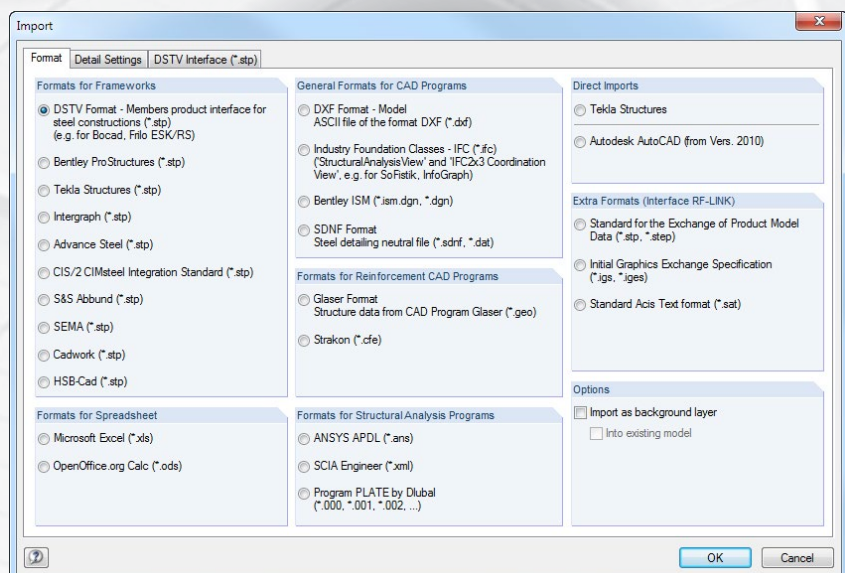
Results output and printout report

- Output of internal forces, deformations and support reactions in tables and graphics
- Colored results display on the rendered model, colors and range of values freely adjustable
- Results display for total structure or selected structural parts
- Animation of deformations, internal forces and stresses
- Manual or automatic printing of graphics in the printout report
- Multi-lingual printout report independent of input language
- Individual adjustment of printout extent
- Export of report in RTF or PDF file

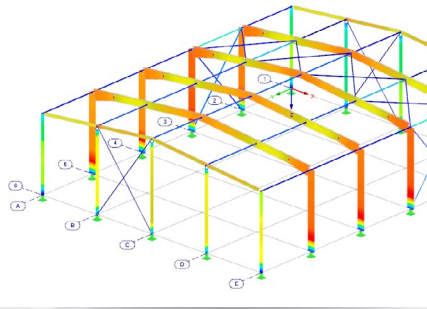


Building Information Modeling (BIM)

- Efficient teamwork for engineers planning design and details
- Working on 3D model with material and geometry information
- Direct interfaces with AutoCAD, Autocad Structural Detailing, Autodesk Revit Structure and Tekla Structures
- Keeping intelligence of objects during data transfer
- Easy transfer of model modifications
- Standard file formats for data exchange like DXF, IFC, DSTV, STP, SDFN etc.
- Interfaces with CAD reinforcement programs of Glaser, Strakon and Nemetschek



Steel



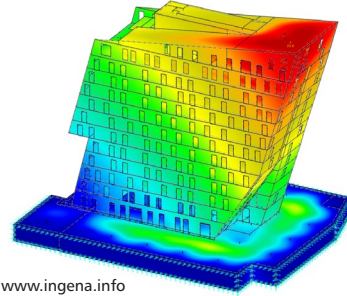
Stress analysis, designs (ultimate limit state, stability, deformation, flexural buckling, lateral-torsional buckling, safety against plate buckling, cross-sections acc. to elastic-plastic method), non-linear material laws to consider plastification of surface elements, buckling modes and shapes, critical load factors, effective lengths, imperfections, cross-section properties, standards: EC 3, AISC, SIA, IS, BS, GB, CSA, AS, NTC-DF, SP, SANS, NBR, layout of connections

With the RFEM add-on modules you always find the right solution to succeed in your professional life.

Design of surfaces, members and foundations, analysis (among others bearing capacity, punching, fire resistance, serviceability for surfaces in cracked sections (state II)), model column method or method based on nominal curvature, EC 2, SIA, ACI, GB

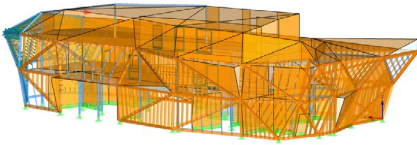


Concrete



© www.ingena.info

Timber

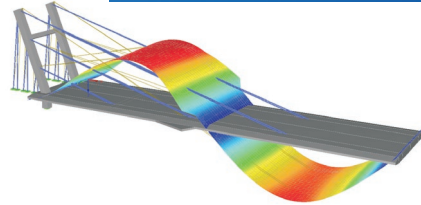


© www.lignaconsult.com

Eigenfrequencies and eigenmodes, forced vibration analysis, generation of equivalent lateral forces for earthquakes according to multi-modal response spectrum method



Dynamics



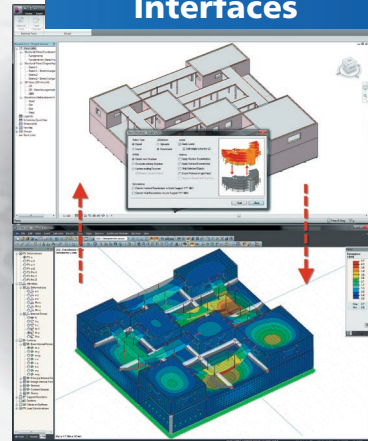
Member design acc. to EC 5, SIA, AWC, CSA, cross-laminated timber slabs, dowel connections with slotted sheets



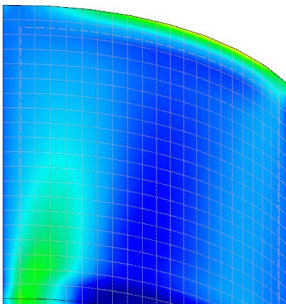
Product interface for steel construction (German DSTV), CIS/2 Structural Frame Schema, Microsoft Excel / OpenOffice.org Calc, DXF file format, IFC, SDNF, GLASER -isb cad-, DICAD STRAKON, Nemetschek Allplan, CADKON, Tekla Structures, integration in Autodesk AutoCAD and Revit Structure, reinforcement transfer to Autodesk Structural Detailing, background layers, Bautext, RTF, programmable COM interface, import of Standard Acis Text Format (*.sat), Initial Graphics Exchange Specification (*.iges) and Standard For The Exchange Of Product Model Data (*.stp) with RX-LINK



Interfaces



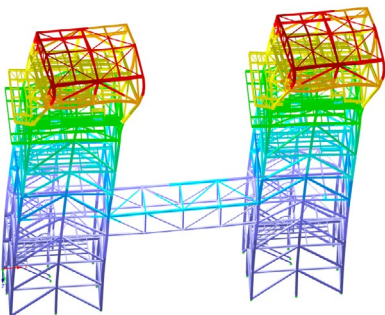
Glass



Calculation of glass panes and insulated glass



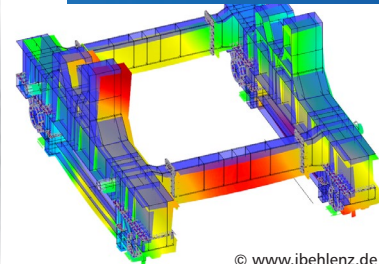
Plants



Frameworks and structures consisting of shells and solids, pressure vessels, plate buckling of shells, stability and contact problems, pipelines, machine dynamics, material handling, automotive engineering

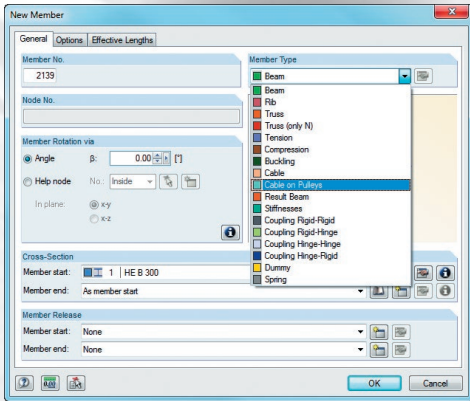
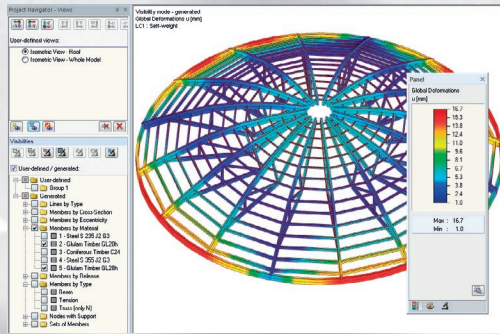


Engineering



© www.ibehlenz.de

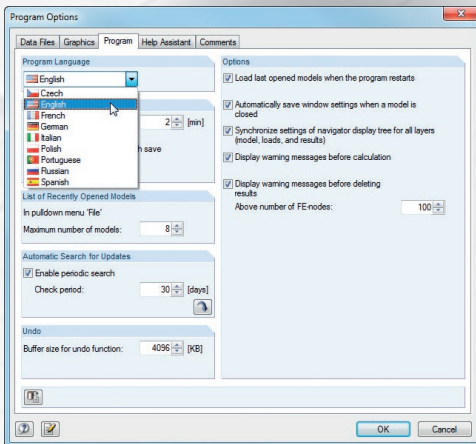
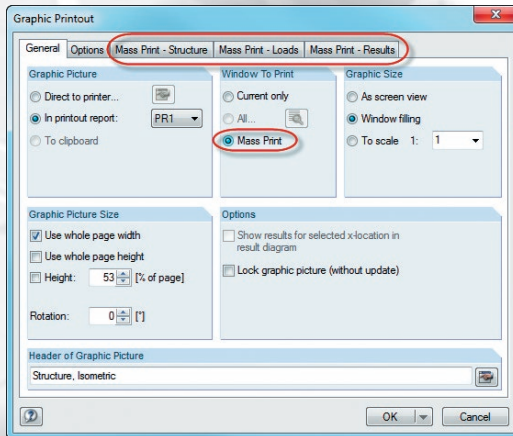
Easy working with generated and user-defined partial views in the new Project Navigator - Views



New member types:

- Cable on pulleys (sheave)
- Result beam (integrating stresses and internal forces)
- Stiffnesses
- Spring

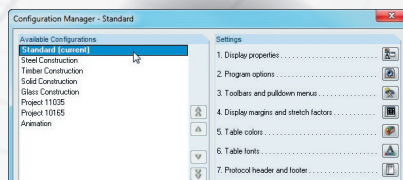
Direct mass print of structure, loads and results graphics in the printout report



Six new languages implemented

Now it is possible to work with a German, English, Czech, Italian, Spanish, French, Portuguese, Polish and Russian user interface.

With the new Configuration Manager you can specify user-defined settings for the display, program options, toolbars etc. and save them as separate configurations. Several configurations can be stored.



Selected Features for Structural Input in RFEM 5

- Creating and saving line grids easily
- Relative member eccentricity in relation to other member
- New material models: Isotropic elastic-plastic 1D/2D/3D, isotropic nonlinear elastic 2D, orthotropic elastic 3D, orthotropic elastic-plastic 3D, isotropic thermal-elastic, isotropic masonry 2D
- Favorites list for cross-section library
- Definition of semi-rigidly or rigidly coupled timber cross-sections and hybrid cross-sections
- New options for work plane (3 points plane, line + direction, member axis + plane, surface plane offset)
- Shifting and copying objects in user-defined coordinate system
- Uniting and subtracting solids by using Boolean operators
- Definition of object properties to be displayed in pre-selection
- "Slicing" objects by means of clipping plane
- Rendering for loads and transparent structural model
- Definition of surface supports
- Starting the recently used function by <Enter> or right-click
- Defining member start in relative distance of a member
- Nodal loads/nodal supports in direction of node, member, user-defined coordinate system

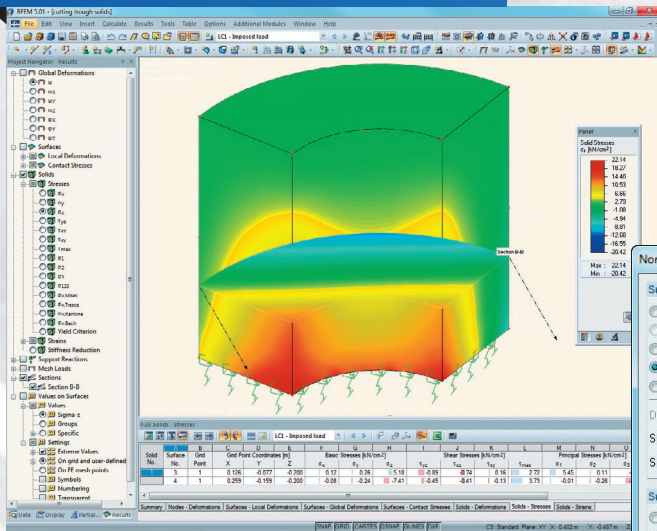
Selected Features for Load Input and Analysis

- Application of loads from multi-layer structures like roof, ceiling or floor structures
- New load types: initial prestress, end prestress, imposed displacement, imposed rotation, pipe content - full/partial, rotary motion
- Definition of surface loads on openings
- New options for large deformation analysis (Newton-Raphson in combination with Picard, Picard or Newton-Raphson with constant stiffness matrix)
- Input of inclination and precamber in absolute values
- Displaying generated loads separately
- Parabolic and quadrilateral member loads

Selected Features for Results Output and General Functions

- Rendering colors adjustable according to object properties
- Easy deletion of module data directly in Data navigator
- Definition of average regions for smoothing singularities
- Animation of internal forces and stresses
- Output of filtered results (e.g. only designs with particular ratio)
- Model printed in 3D PDF (also with loads, results, etc.)

New Features in RFEM 5



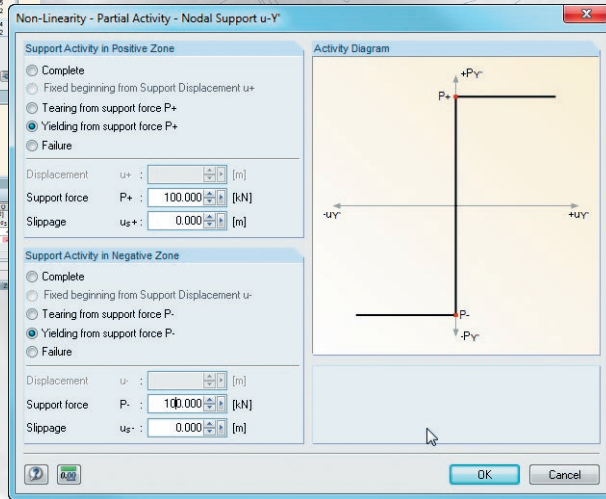
Display of deformation and stress diagrams in intersection plane when cutting through solids

New support and release non-linearities:

- Partial activity
- Diagram



For both you can define for example tearing or yielding from a particular support force or release moment.



Get to know us

Would you like to know more about **RFEM**? Ask for a **free trial version** without obligation or use the **download option at www.dlubal.com**.

With the trial version you can learn more about our programs, get acquainted with the handling and calculate structural systems.

See for yourself how easy it is to work with Dlubal software.

If you need help with the first steps with **RFEM**, go to www.dlubal.com/en/for-beginners.aspx where you can find introductory examples and tutorials, videos and webinars, manuals and information about our service contracts.

If you are looking for the answer to a particular question, browse the **FAQ** page, visit the Dlubal **blog** or follow us on **social networks** where you can find numerous tips and tricks as well as solutions for everyday problems occurring in many engineering offices. We would also be happy to advise you directly by phone or video call.

You also have the possibility to participate in our **free webinars**, where we give you insight into the operation of our software, show you new features and can discuss questions in detail.

In order to purchase the full version of the software, use our **Webshop on www.dlubal.com** or contact our **service team** which will be glad to assist you in putting together your individual program package.

Upgrades

You are already using **RFEM**?

Get the program upgrade.

Please contact us or order online at www.dlubal.com.

Service Contracts

Customer service is one of the main cornerstones of the **Dlubal company mission**. The interest in our customers does not end at the point of sale. We offer additional support if it is needed for your daily work.

With a service contract your questions will be taken care of with highest priority and you get upgrades at better rates.

For more information about our service contracts, contact us directly or visit our website at www.dlubal.com.

Technical Support

Our technical engineers are available to all customers whenever there is a question about Dlubal programs. Just send your question by **e-mail** or **fax**. The questions will be answered in the order received and only after enquiries of customers having a service contract have been completed. The extent and speed of response depend on the type of service contract you have purchased.

We welcome any feedback you may have on our products. Your comments and suggestions for improvements are important to us.

Further Information:

Dlubal Software GmbH
 Am Zellweg 2, D-93464 Tiefenbach
 Tel.: +49 9673 9203-0
 Fax: +49 9673 9203-51
www.dlubal.com
info@dlubal.com

Follow us on:

