New Features in
RFEM 5
The Ultimate FEA Program

Contents

1 General Features 2
2 Structural Input 10
3 Load Input and Design 17
4 Output of Results 20
5 Add-on Modules 23

December 2012
1 General Features

Creating a New Model (Structure)

Now, when you create a new model (structure), you can select the design standard that you want to use. You can also decide if you want to create load or result combinations automatically.

In addition, you can use previously saved model templates.

The orientation of the global z-axis defined in the dialog box can be changed subsequently.

New Project Navigator - Views

A new tab was added to the Project Navigator. In the "Views" tab you can easily generate various views. It is also possible to save them for reuse.

New Program Languages

Six new program languages were implemented. Now it is possible to work with a German, English, Czech, Italian, Spanish, French, Portuguese, Polish and Russian user interface.
Display Properties
Now, you can assign different colors to different objects of the structure for a clear arrangement of the construction’s rendered presentation. RFEM distinguishes between:
- Materials
- Cross-sections
- Member types
- Member releases
- Surface types - geometry
- Surface types - stiffnesses
- Surface thickness
- Surface sides
- Solid types
- User-defined visibilities

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

Moving and Copying in Defined Coordinate System
Now, it is possible to move or copy objects in a user-defined system of coordinates.
64-bit Version
RFEM 5 is provided as complete 64-bit version.
Thus, you can use all resources available in your computer
and perform complex and extensive calculations.

New Rendering Options
• Transparent display model
  In addition to the wireframe display and the solid
  model, it is possible to represent the rendered struc-
  ture as transparent model. The intensity of transpar-
  ency can be set in the program options.
• Loading
  Loads can be displayed as filled transparent.
• Lighting
  You can activate six different lights and display light
  positions.

Repeat Recent Function
The recently used function can be easily repeated. Use
the [Enter] key on your keyboard, or select the Repeat
function in the context menu that you open by a right-
click into the work window.
**Preselection**
Now, the Display navigator offers the possibility to set object properties that you want to be displayed in the object’s pre-selection, for example you can set the member orientation, tension sides or member axis systems for members.

**Easy Deletion of Module Data**
Data of add-on modules can be deleted easily in the Data navigator.

**New Options for Work Plane and Grid/Snap**
The snap distance can be adjusted manually.
You can create a user-defined coordinate system directly in the dialog box available for work planes.
The object snap can be activated for background layers, line grids and guidelines.
Now, you can decide whether you want to display only the model or all included guidelines in the "Show Whole Model" view. For example, if you use guidelines running far beyond the structure, you can set this new option to display only the structure in the window-filling view.
New Work Planes
You can define new work planes:

- 3 points plane (free definition of a plane by clicking 3 points)
- With line in X (definition by line and x-direction)
- With line in Y (definition by line and y-direction)
- With line in Z (definition by line and z-direction)
- Axes xy of members (definition of a plane in the xy-axis of a member to be selected)
- Axes xz of members (definition of a plane in the xz-axis of a member to be selected)
- Axes xy of surfaces (definition of a plane in the xy-axis of a surface to be selected)
- Offset (offsetting the previously selected work plane by a certain value)

Comments in Planes
Comments can be entered in the planes X-Y, Y-Z, X-Z or the current work plane. Moreover, they can be rotated.

Conversion of Node Coordinates
You can enter a node with reference to a previously entered node and RFEM provides the option to convert the node coordinates automatically relating to the zero point.
**Templates**
You can save a model as template. When you create a new model, it is possible to import such a template.

**Clipping Plane**
Now you can define a virtual plane clipping the structure. You can decide if you want to see the cut-off area that is in front of or behind the clipping plane. The plane can be shifted gradually.

**New Program Options**
Different intensities of transparency can be defined for members, surfaces, solids, supports, loads and results of isosurfaces. Pictures can be compressed for printing. Furthermore, you can activate the automatic search for updates.
New Dimension: Height Level
Structures can be described by height indications.

Object Selection in Data Navigator
Now, when objects like nodes, lines, cross-sections, members, surfaces etc. are marked in the Data navigator, they are also selected in the graphic.

Inserting Visual Objects
3D objects such as cars, people etc. can be inserted now. Use this option to illustrate for example the size of the structure in relation to those objects.

MSI Installation
An MSI installation is used for RFEM 5. MSI installations can be distributed more easily to client computers in a network with a Microsoft server.
Moreover, two separate installations are available, a 32-bit version and a 64-bit version.
**Grid Lines in Work Plane**

The active work plane can now be displayed with grid lines. The spacing of grid lines can be modified in both directions.

---

**Panel for Variable Surface Thicknesses**

Variable thicknesses of surfaces can be displayed as color gradient. The color scale can be modified in the panel.

---

**Selection with Ellipse or Annulus**

With this new option you can select objects by means of an ellipse or annulus.
New Member Types
The following new member types were implemented:

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

Line Grid
Now you can create a line grid quickly in the Cartesian coordinate system. Furthermore, it is possible to specify the grid with markings and dimensions. Moreover, you can create spherical or cylindrical grids. You can also rotate the grid about one or more axes. In addition, you can save settings for the line grid and reimport them later.

Relative Member Eccentricity
Now you can assign a relative and automatic member eccentricity to a member.

For example, it is possible to align the bottom edge of a member with the top edge of a reference member. The advantage is that the member eccentricity will be adjusted automatically when cross-sections change.
New Cross-section Library
The cross-section database was reorganized and extended by corresponding standards. Now the library is arranged more clearly. With the filter option it is possible to show beams of particular standards, sectional shapes or section types.
For example you can look at all I-sections of medium size shown in a table.
Materials can be defined optionally. In addition, you can summarize the most frequently used cross-sections in a favorites list.

Combined Timber Cross-sections
The cross-section database offers new profiles: combined timber cross-sections. A wide range of cross-section shapes is provided, for example U-beams, T-beams, I-beams and box-shaped beams.
Single elements are combined by rigid or semi-rigid connections.
Now it is possible to calculate elements produced by LIGNATUR and LIGNOTREND.
Furthermore, you can select a hybrid cross-section. In a separate dialog box you can assign different materials to individual cross-sections.

Displaying Components from Intersections
Now, individual components of intersections are displayed.
The display makes it easier to edit and modify data subsequently.
**Compound Solids**

Another important function was added for the creation of solid models: Now you can use compound solids. Two solids can be merged by means of Boolean operators. It is also possible to subtract a solid or to create the intersecting set from both volumes.

**New Material Models**

The following new material models are now available:

- Orthotropic elastic 3D
- Isotropic elastic-plastic 1D/2D/3D *
- Isotropic nonlinear elastic 2D *
- Orthotropic elastic-plastic 3D *
- Isotropic thermal-elastic *
- Isotropic masonry 2D *

* Add-on module RF-MAT NL is required

**Non-linearities of Supports**

The following support non-linearities were added:

- Failure all, if $P$ positive
- Failure all, if $P$ negative
- Partial activity...
- Diagram...

Use these options to consider for example slippage as well as tearing or yielding arising from a particular support force.

- Friction

You have to define the corresponding friction values.
**New Features**

**RFEM 5**

**2 Structural Input**

**Non-linearities of Releases**
The following release non-linearities were added:

- Failure, if force/moment positive
- Failure, if force/moment negative
- Partial activity...
- Diagram...

Use these options to limit for example the torsion of a release.

**Surface Support**
Now you can define surface supports. For example, when creating a symmetric or rotationally symmetric solid model, you can model only a part of it. In this way, calculation time can be reduced.

**Modifying the Axis System of Solids**
The axis systems of solids can now be aligned with surfaces, lines and points.
Moreover, axes can be aligned with a user-defined system of coordinates.
**Scissors Releases**

Now, RFEM allows for the arrangement of scissors releases. With scissors you can model a crossing of two members whose diagrams of moments, shear or axial force do not affect each other (helpful for example for rafter-purlin systems).

**New Solid Types**

New solid types were added to the already existing types Contact and Solid:

- Gas
- Gas between two surfaces
- Null

**Tangents to Circles or Arcs**

You can apply tangents to two circles or arcs. Moreover, you have the option to trim existing arcs.
New Material Database

The material library was extended by several filter options and lots of new materials in accordance with various standards. In addition, the material database was optimized so that the library opens faster.

New Options for Openings

A variety of new openings can be easily created, for example a rectangle with fillets, a parallelogram, triangle, circular segment, polygon etc.

In addition, you can create openings by using lines, arcs and ellipses for modeling. Then, select the objects with a selection window.

New Options for Solid Input

Now you can create solids with a special function. Model the respective base area and extrude the surface to a parallel plane or point.
New FE Mesh Generators for Solids
RFEM 5 uses a new generator (T3D) for creating the mesh on solids. In many cases, a better mesh is created.

Set Nodes Between two Nodes
It is possible to set a new node between two existing nodes. The distance to adjacent points can be selected freely. Relevant points such as nodes, snap points and grid points may lie in one plane or anywhere in the workspace.

Modifying Cross-section Properties and Stiffnesses
Cross-section properties can be modified by means of a factor. In addition, it is possible to modify stiffnesses of members.
3 Load Input and Design

Automatic Creation of Combinations

Now load cases and actions are entered in a new table. It is possible to create load and result combinations automatically after you have selected the relevant combination rules. The table data is clearly-arranged and you can copy, add or renumber load cases. In addition, it is possible to control load cases and combinations in tables 2.1 to 2.6.

Loads from Multilayer Structure

Now you have the possibility to define and save multilayer structures such as ceilings and floor structures which can be applied later as surface or member loads (with load application width).

Surface Loads on Openings

You can apply surface loads to openings. RFEM will generate automatically the corresponding line loads on the boundary lines of the opening.
Input of Inclination and Precamber in Absolute Values

In addition to the relative input of imperfections, you can now enter imperfections with absolute values.

Additional Calculation Parameters

Some new options were added to the calculation parameters. For example, when the model is calculated according to second-order analysis, you can relate the internal forces to the deformed structure.

For the calculation according to large deformation analysis the following approaches are now additionally available.

- Newton-Raphson combined with Picard
- Picard
- Newton-Raphson with constant stiffness matrix

Moreover, it is possible to modify stiffnesses of materials, cross-sections, supports and releases by load case or load combination.

New Member Loads

The following new load types were added:

- Initial prestress
- End prestress
- Imposed displacement
- Imposed rotation
- Pipe content - full
- Pipe content - partial
- Rotary motion
**New Features**

**RFEM 5**

**3 Load Input and Design**

**New Surface and Solid Loads**

The following new surface load was added:
- Rotary motion

The following new solid loads were added:
- Force
- Buoyancy
- Rotary motion

**Generated Area Loads Variable in Direction x and y**

When generating area loads, you can apply variable loads not only in direction z but also in x and y.

**Generation of Loads from Motions**

It is possible to generate member, surface or solid loads from motions. Thus, braking and accelerating forces can be considered. Loads can be produced from straight-line and rotary motions.
4 Output of Results

**Mass Print**

Model, loading and results can be printed in series. RFEM creates several graphics viewed from different directions to be specified.

For example, it is possible to print all internal forces in isometric view by a single mouse-click.

---

**Section through Solid**

Now, diagrams of deformations and stresses are displayed in the intersection plane when you cut through solids.

---

**Smoothing Singularities**

By defining smooth ranges it is possible to smooth singularities in results.
Printout Report
The printout report opens already during the generation of all pages. Thus, it is quickly available for display. In addition, it is possible to print the report directly in a PDF file.

Output of Member Coefficients and Member Slendernesses
Now, table 4.8 shows the member coefficients \( \varepsilon_y \) and \( \varepsilon_z \) which depend on the member length, axial force and flexural resistance.
Table 4.9 lists the member slendernesses.

Animation of Surface Stresses and Internal Forces
It is possible to animate internal forces and stresses of surfaces, illustrating the changing distribution of internal forces and stresses for load increments.
Convergence Diagram

The convergence curve is shown in a table and represented as diagram. So you can see how many iteration steps were needed until convergence was reached.
5 Add-on Modules

RF-STEEL EC3
Now RF-STEEL EC3 is able to design result combinations (formerly: load combinations). In addition, the memory management was optimized which allows for a faster calculation. The cross-section classification for the respective load case is now shown by cross-section, by member and by x-location.

CRANEWAY
The following new features were implemented:
- Calculation according to EN 1993-6 (Eurocode 3) with national annexes (NAs) for Germany, Czech Republic, Finland, Cyprus and Italy
- Available designs according to:
  - EN 1993-1-5 (plate buckling analysis of unstiffened plates)
  - EN 1993-1-8 (design of welds)
  - EN 1993-1-9 (fatigue design)
- Design of overhead cranes acc. to EC 3 as single- or multi-span beam by second-order analysis for torsional buckling
- Design of non-continuous welds
- Separate design of welds for $a_1$ and $a_2$ for welded cross-sections
- Calculation of deformation according to EC 3
- Load case table for clear representation of load combinations with information about loading, dynamic coefficients and partial safety factors for the design situations: ultimate limit state, fatigue and deformation of support forces.
- New 3D rendering
- Detailed output of single load cases and combinations for the respective design situation

PLATE-BUCKLING
PLATE-BUCKLING provides the following new features:
- Design according to EN 1993-1-5, chapter 10, with NAs of Germany, Czech Republic, Finland, Cyprus and Italy by method of reduced stresses
- Available restraints (for example adjacent structural components) are taken into account in the form of a spring stiffness.
- Optional designs for all eigenmodes
RF-CONCRETE Surfaces

The following features are new:

- Analysis core for calculation of required reinforcement in ultimate limit state
- Unification of definition of reinforcement sides (top/bottom) for both possible directions of global coordinate system
- Description of reinforcement layer depending on local z-axis of surface
- Calculation of required reinforcement for different design situations in the ultimate limit state in one calculation run
- Definition of calculation method for result combination or for several load combinations selected for design. It is possible to calculate loads either individually one after the other or by means of an envelope.
- Design with smoothed internal forces in predefined smooth ranges (for example singularities above columns)
- Efficient layout of required longitudinal reinforcement for crack width analysis according to EN 1992-1-1
- Definition of different maximum allowable crack widths at top and bottom side
- Copy function for existing reinforcement groups
- Option to specify manually the direction of principal compression for walls when calculating the minimum longitudinal reinforcement
- Calculation of minimum longitudinal reinforcement for ductile behavior of structural components acc. to 9.3.1, EN 1992-1-1 or 13.1.1, DIN 1045-1 as well as calculation of minimum shear reinforcement acc. to 9.32, EN 1992-1-1 or 13.3.3, DIN 1045-1 possible
- User-defined control of partial safety factors of materials used for different design situations or limit states
- Optimized graphical results output (display of reinforcement applied for SLS design, of ratio $V_{Ed}/V_{Rd,c}$, of result values in reinforcement direction)
- Improved output of design details for each design point in the ultimate limit state
**RF-CONCRETE Members**

The following new features were implemented:

- Calculation of required reinforcement for different design situations in ultimate limit state in one calculation run
- Description of reinforcement layer depending on local z-axis of members
- Definition of different maximum allowable crack widths at top and bottom side
- Copy function for existing reinforcement groups
- Optional calculation of minimum longitudinal reinforcement for ductile behavior of structural components acc. to 9.3.1, EN 1992-1-1 or 13.1.1, DIN 1045-1 as well as calculation of minimum shear reinforcement acc. to 9.32, EN 1992-1-1 or 13.3.3, DIN 1045-1
- User-defined control of partial safety factors of materials used for different design situations or limit states
- Optimized reinforcement output for circulating reinforcement layout and for reinforcement in corners
- Results output in tables of total reinforcement amount available in cross-section by x-location
- Improved output of design details for stress conditions of provided reinforcement in the ultimate limit state design
- Extension of envelope to be designed for result combinations. In addition to the extreme values of internal forces, the program takes into account the extreme values of cross-section stresses (important for example for double bending).

**RF-CONCRETE Columns**

RF-CONCRETE Columns offers the following new features:

- Calculation of required reinforcement for different design situations in ultimate limit state in one calculation run
- Copy function for existing reinforcement groups
- Optional calculation of minimum longitudinal reinforcement for ductile behavior of structural components acc. to 9.3.1, EN 1992-1-1 or 13.1.1, DIN 1045-1 as well as calculation of minimum shear reinforcement acc. to 9.32, EN 1992-1-1 or 13.3.3, DIN 1045-1
- User-defined control of partial safety factors of materials used for different design situations
**RF-CONCRETE NL**

The following items are new:

- Considering influence from shrinkage on non-linear calculation
- Calculation of crack width taking into account tension stiffening of concrete applied between cracks
- Additional options to consider the concrete’s tension behavior for calculations
- Optional export of stiffnesses from non-linear calculation for further use in RFEM
- Graphical output of nonlinear support reactions
- Graphical output of enveloping results for calculation of several loads
- Non-linear calculation of spatial frameworks in the limit states for load bearing capacity and serviceability

**RF-PUNCH**

New features in RF-PUNCH:

- Calculation of required reinforcement for different design situations in ultimate limit state in one calculation run
- Input of properties for nodes of punching shear for several common punching shear points
- Transfer of already defined properties for nodes of punching shear to other punching shear nodes
- Choice between user-defined specification and automatic determination of axial force acting on the perimeter
- User-defined control of partial safety factors of materials used for different design situations
**RF-TIMBER Pro**

RF-TIMBER Pro offers the following new features:

- Radius of curvature is considered and transversal tension stresses are calculated for the design of curved members
- Calculation of result combinations is possible
- Optional calculation and graphical display of stress points
- Possibility to reduce stiffness due to creep effects in service classes 2 and 3
- Stability analysis can be deactivated globally
- Design of built-up cross-sections (including multi-part cross-sections such as horizontal beams for edges), with option for yield

---

**RF-LAMINATE**

The following new features were added:

- Optional output of strains $\varepsilon_x$ and $\varepsilon_y$ as well as of distortions $\gamma_{xy}$ in individual layers
- Output of stresses of selected points in printout report
- Optional output of graphical stress diagrams in printout report
Get to know us

Would you like to know more about RFEM?
Request more information and a free trial version. Or simply download the free trial version at www.dlubal.com.

Use the trial version with fully functional features to test RFEM 5. Get familiar with the program handling, take all the time you need to explore all program details and see yourself how easy it is to work with RFEM.

Find further information on our website www.dlubal.com, for example videos helping you to get started with our software. If you are interested in technical details, read the program manuals available for download in PDF file format. Or browse the FAQ page where you may find some solutions for everyday problems occurring in many engineering offices.

Of course, we would also be happy to speak with you directly by phone or video call. Our qualified engineers can assist you fast and personally. Modern technologies such as desktop sharing tools allow us to support you in no time and anywhere in the world where Internet is available.

If you are not sure which modules you need, we help you to create a software package that suits you best and meets your individual needs.

Upgrades

You are already using RFEM?
Get the upgrade to RFEM 5.
Please contact us, or order online at www.dlubal.com.

Service contracts

Top 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 you get the upgrade to RFEM 5 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 email 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 Engineering Software
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: