New Features in

RSTAB 8
The Program for Spatial Frameworks

January 2013

Contents

1 General Features 2
2 Structural Input 10
3 Load Input and Design 14
4 Output of Results 17
5 Add-on Modules 18

Dlubal Engineering Software
• Am Zellweg 2 • D-93464 Tiefenbach • info@dlubal.com • www.dlubal.com
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. RSTAB distinguishes between
- Materials
- Cross-sections
- Member types
- Member releases
- User-defined visibilities

64-bit Version
RSTAB 8 is provided as complete 64-bit version. Thus, you can use all resources available in your computer and perform complex and extensive calculations.

Moving and Copying in Defined Coordinate System
Now, it is possible to move or copy objects in a user-defined system of coordinates.
**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.

**New Rendering Options**

- **Transparent display model**
  In addition to the wireframe display and the solid model, it is possible to represent the rendered structure as transparent model. The intensity of transparency 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)
- 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)
- 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 RSTAB 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 now for members, supports and loads. 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 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.
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.

Selection with Ellipse or Annulus
With this new option you can select objects by means of an ellipse or annulus.

MSI Installation
An MSI installation is used for RSTAB 8. 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.
2 Structural Input

New Member Types

The following new member types were implemented:
- Rigid member
- Stiffnesses
- Spring (in preparation)

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

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.
Furthermore, you can select a hybrid cross-section. In a separate dialog box you can assign different materials to individual cross-sections.

New Material Model: Temperature

Now, the nonlinear material model Temperature is available. Use the detail settings to define reference temperature and temperature curves. Data can be saved and re-imported later.
Non-linearities of Supports
The following support non-linearities were added:
- Friction
You have to define the corresponding friction values.
- Failure all, if $P$ positive
- Failure all, if $P$ negative

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.

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 member loads with load application width.

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. 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:
- Pipe content - full
- Pipe content - partial

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 loads from motions. Thus, braking and accelerating forces can be considered.
4 Output of Results

Mass Print
Model, loading and results can be printed in series. RSTAB 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.

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 4.9 shows the member coefficients $\varepsilon_y$ and $\varepsilon_z$ which depend on the member length, axial force and flexural resistance.
Table 4.10 lists the member slendernesses.
5 Add-on Modules

STEEL EC3

Now 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 $a_u$ and $a_o$ 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

CONCRETE

The following features are new:

- 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 provided reinforcement groups (already entered reinforcement properties can be taken over fast and easily).
- Improved calculation of crack widths for cross-sections with biaxial moment loading
- 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
- Calculation of required reinforcement for non-linear calculation in ultimate limit state
CONCRETE Columns

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

TIMBER Pro

TIMBER Pro offers the following new features:

- Design of result combinations
- 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
Get to know us

Would you like to know more about RSTAB? 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 RSTAB 8. 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 RSTAB.

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 suites you best and meets your individual needs.

Upgrades

You are already using RSTAB? Get the upgrade to RSTAB 8. 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 RSTAB 8 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: