Ingenieur-Software
Dlubal

Version
February 2010

Interface

RX-Tekla

Interfaces between RSTAB/ RFEM and Tekla Structures

Program Description

Am Zellweg 2 D-93464 Tiefenbach

Tel.: +49 (0) 9673 9203-0
Fax: +49 (0) 9673 1770
E-mail: info@dlubal.com
Web: www.dlubal.com

Contents

1. Introduction

1.1 Data Exchange Options

There are three options to exchange data between RSTAB/RFEM and Tekla Structures:

1. Direct export of the analytical model from Tekla Structures and reimport of cross-sections and internal forces

2. Direct export/import of the physical model to and from Tekla Structures

3. Data exchange of the physical model between RSTAB/RFEM and Tekla Structures by means of stp files (DSTV - German Steel Construction Association)

In the following, the functionality and extent of these options are described.

1.2 RX-Tekla Team

The following people were involved in the development of RX-Tekla:

Program coordination

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

Programming

Ing. Radek Brettschneider

Dis. Jiří Šmerák

Program supervision

Ing. Petr Míchal

Ing. Tomáš Ferencz

Manual, help system and translation

Dipl.-Ing. (FH) Alexandra Lazar

Ing. Ladislav Kábrt

Dipl.-Ü (Uni) Gundel Pietzcker

Technical support and quality management

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

Dipl.-Ing. (FH) Alexandra Lazar

2. Procedure

2.1 General Information

If you have installed more than one version of RSTAB/RFEM on your computer, you can select the relevant version by using the following commands when exchanging data.

Windows XP:

RSTAB6: C:\Program Files\Dlubal\RSTAB6\RSTAB.exe /regserver RSTAB6
RSTAB7: C:\Program Files\Dlubal\RSTAB7\RSTAB.exe /regserver RSTAB7
RFEM3: C:\Program Files\Dlubal\RFEM3\RFEM.exe /regserver RFEM3
RFEM4: C:\Program Files\Dlubal\RFEM4\RFEM.exe /regserver RFEM4

Windows Vista:

RSTAB6: C:\Program Files\Dlubal\RSTAB6\RSTAB.exe /regserver RSTAB6
RSTAB7: C:\Program Files\Dlubal\RSTAB7\RSTAB.exe /regserver RSTAB7
RFEM3: C:\Program Files\Dlubal\RFEM3\ RFEM.exe /regserver RFEM3
RFEM4: C:\Program Files\Dlubal\RFEM4\ RFEM.exe /regserver RFEM4

2.2 Direct Export from Tekla Structures

2.2.1 Functionality

This kind of data exchange only works in a 32-bit environment.

The interface is integrated in Tekla Structures and is accessed by means of the COM interface using the program library Dlubal_TS_link.DLL.

The interface allows for the import of the **analytical model** from Tekla Structures to RSTAB/RFEM.

To start the interface,

click "Analysis & Design Models" on the "Analysis" menu in the Tekla Structures menu bar.

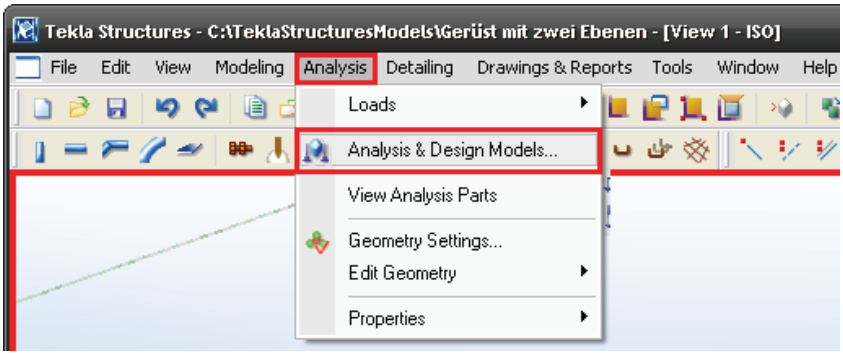


Figure 2.1: Starting the interface

The dialog box shown below opens. Select a structural model that has been previously created, and then click "Create model".

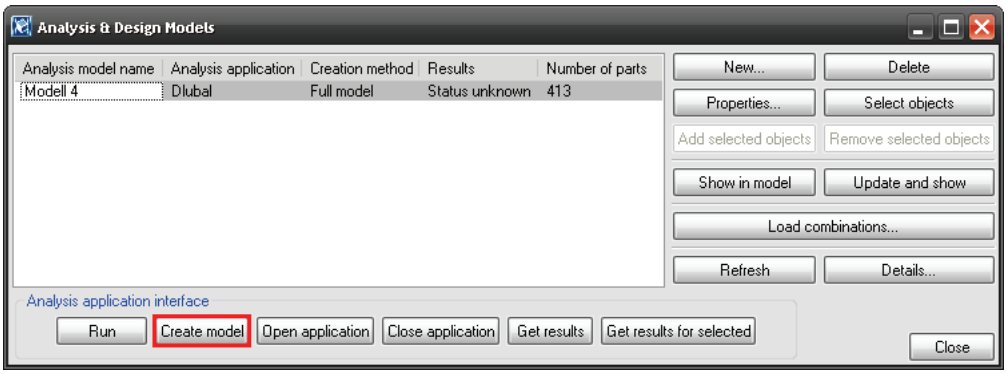


Figure 2.2: Starting the interface

Another dialog box opens where you can select the target program for the export.

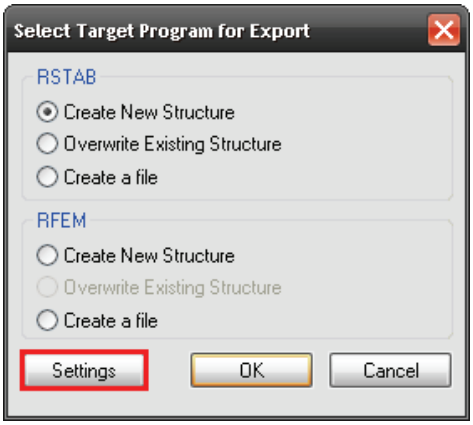


Figure 2.3: Export options

The dialog box shown above allows for different settings. You can either create a new RSTAB/RFEM structure or overwrite an active structure. It is also possible to create an RSTAB or RFEM file only. The last option, however, requires an installed full version of RSTAB/RFEM, but you do not need a hardlock for the third option.

Click "Settings" to access further settings, for example to define the orientation of the Z-axis or to mirror coordinates.

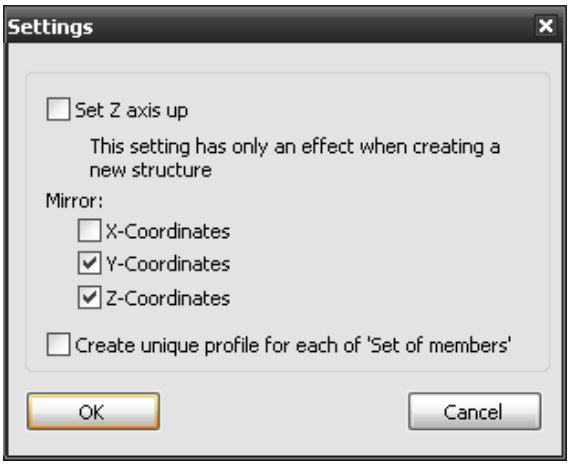


Figure 2.4: Settings for export

2.2.2 Export

In the following, the creation of an analysis model appropriate for RSTAB/RFEM is described.

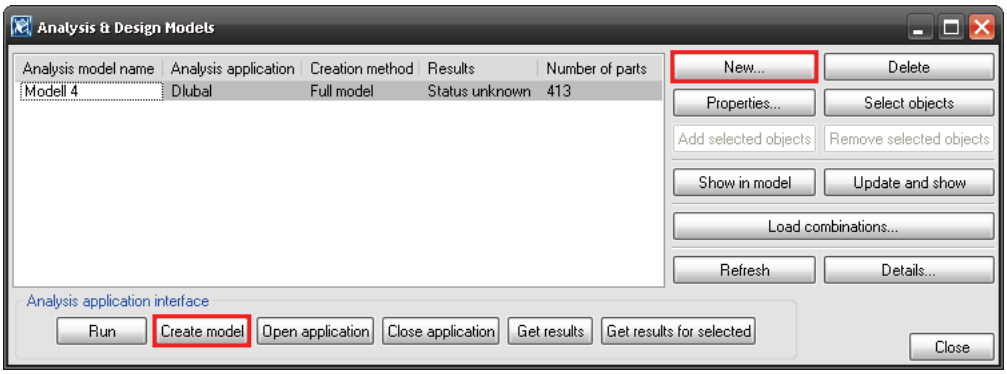


Figure 2.5: Creating an analysis model

Click "New" in the "Analysis & Design Models" dialog box.

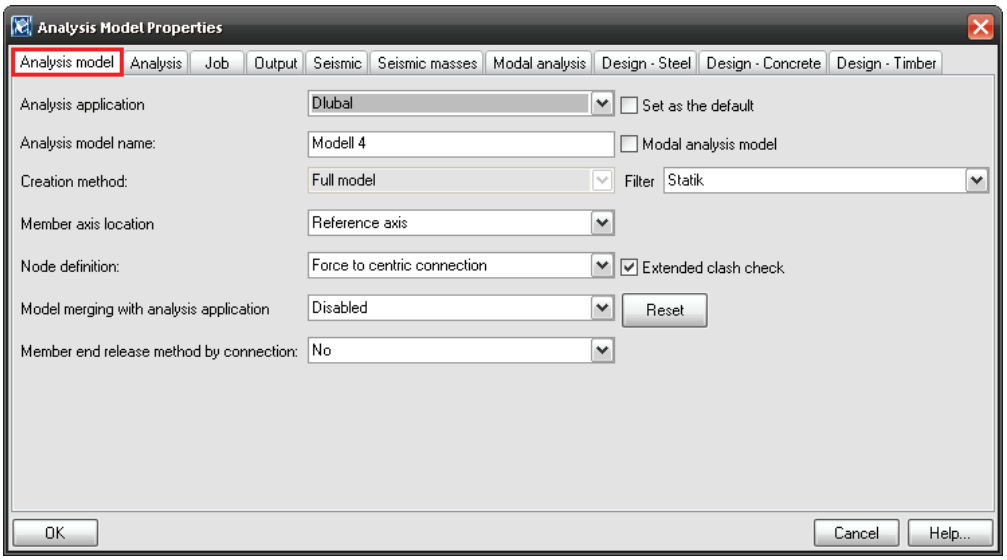
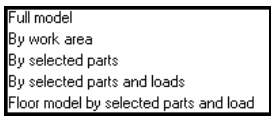


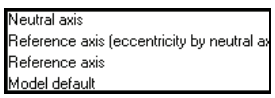
Figure 2.6: Creating an analysis model

In the dialog box shown above, only the tab "Analysis model" is of importance for the interface. Therefore, other tabs are not explained in this manual.

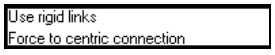
The "Dlubal" **Analysis application** is available in Tekla Structures when Dlubal applications have been previously installed. You can enter any description for the **Analysis model name**.



Concerning the **Creation method**, you should only be aware of the fact that the **Filter** for the settings "By selected parts", "By selected parts and loads" and "Floor model by selected parts and load" in the pull-down menu is only active and selectable when the properties of the analysis model are accessed again.



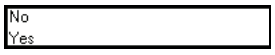
With the settings of the **Member axis location** it is possible, for example, to consider eccentricities.



Depending on the connection's definition in Tekla Structures, you can create a rigid-rigid coupling member between two nodes by using the setting "Use rigid links". It is also possi-

ble to summarize two nodes as one in a single plane by selecting the setting "Force to centric connection".

The option **Model merging with analysis application** should be "Disabled" as it is not important for the interface connection to RSTAB/RFEM.



If "No" is selected for **Member end release method by connection**, you transfer the release definition of the element (e.g. girder). If "Yes" is selected, the release definition of the connection will be transferred.

In the following, you find an overview with all available export options.

Structural data:

- Members
- Curved members divided in several segments
- Member types: beam, compression member, tension member, truss girder
- Surfaces
- Varying surface thicknesses
- Openings
- Member releases
- Nodal supports
- Eccentricities
- Rigid connections
- Sets of members → Members from Tekla Structures are created optionally as set of members with one cross-section for each set of members.
- Materials (see p. 18)
- Cross-sections (see p. 20)

Load data

- Load cases
- Load combinations
- Nodal loads: forces and moments
- Member loads with load distribution: uniform, trapezoidal and variable
- Area loads (divided in member loads for members) with load distribution: uniform and linear

It is also possible to export only selected elements of a structure.

2.2.3 Reimport

A direct import of structural data to Tekla Structures is **not** possible with this data exchange option. In this context, this means that possibly modified cross-sections can *only* be updated.

However, it is additionally possible to transfer result values of member ends to Tekla Structures, for example to design connections. In the course of this process, cross-sections are checked, too, and can be adjusted to the modified RSTAB/RFEM file.

The steps to reimport data are explained in the following.

Previously, an RSTAB/RFEM structure, whose results and possible cross-section modifications can be transferred to Tekla Structures, must be active and correspond to the model in Tekla Structures. If more than one structure is open, the transfer will always refer to the structure that was opened first. Therefore, it is recommended to have only one structure active.

First, click "Analysis & Design Models" on the "Analysis" menu in the Tekla Structures menu bar.

The dialog box shown below opens. Select a structural model that has been previously created, and then click "Get results".

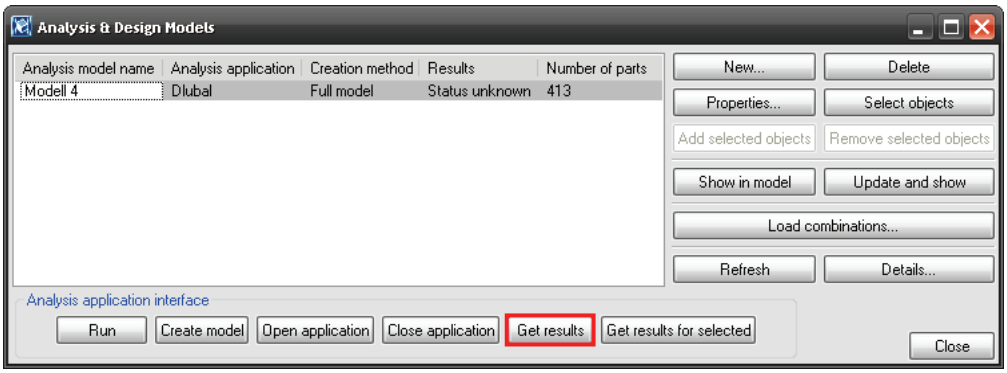


Figure 2.7: Get results

Another dialog box opens where you can select the program from which the results should be imported. Then, a dialog box appears where you can select the load case, load group and/or load combination from which the results will be transferred. A multiple selection is possible, but then the relevant results are summarized for one location.

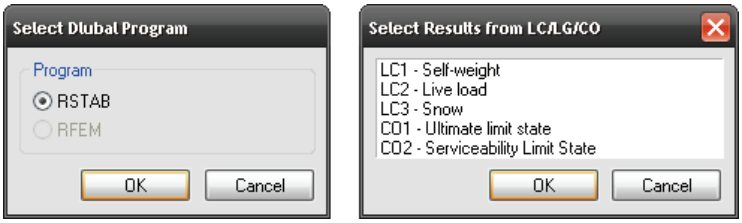


Figure 2.8: Selection of program and loads

Finally, after the data transfer, a dialog box opens displaying, amongst others, the cross-sections that have been modified. For a clear overview, it is also possible to display only the modifications.

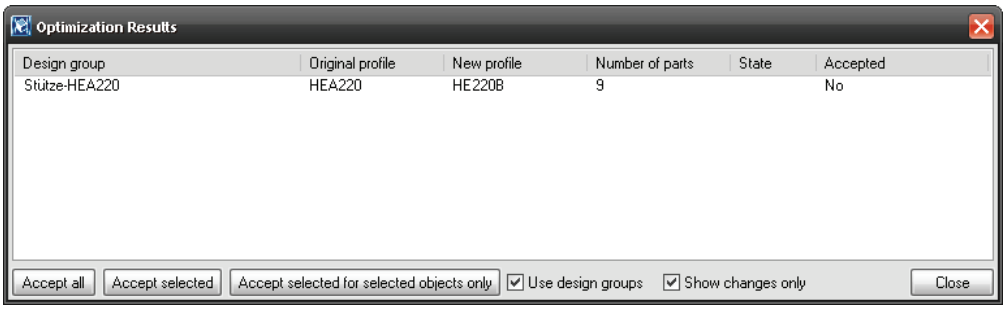


Figure 2.9: Modified cross-sections

When several cross-sections have been modified, you can decide if all modifications are accepted or if only single cross-sections are adjusted in the Tekla Structures model.

By double-clicking an element in Tekla Structures, you can access the settings of the element. Then click "User-defined attributes".

A dialog box opens. The tab "End codes" shows the results that have been imported from RSTAB/RFEM to Tekla Structures (see figure below).

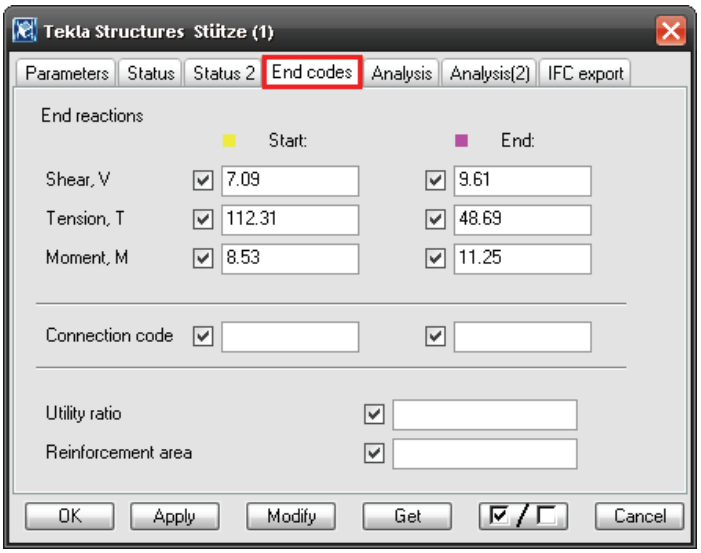


Figure 2.10: End codes

2.3 Direct Export/Import to/from RSTAB/RFEM

2.3.1 Functionality

This kind of data exchange works in 32-bit and 64-bit environments. Please note that RSTAB/RFEM and Tekla Structures are 32-bit versions. If you want to use the interface in an 64-bit environment, you have to install Tekla 32-bit.

The interface is integrated in RSTAB/RFEM. It works via .NET technology with RX-Common.NET (RSTAB7/RFEM4) or RX-Tekla.NET (RSTAB6/RFEM3).

The interface allows either for the export of the **physical model** from RSTAB/RFEM to Tekla Structures or for the import of the model from Tekla Structures to RSTAB/RFEM.

2.3.2 Export

To export a structure from RSTAB/RFEM to Tekla Structures, follow the instructions described below:

Click "Export" on the "File" menu in the RSTAB/RFEM menu bar as shown in the figure below.

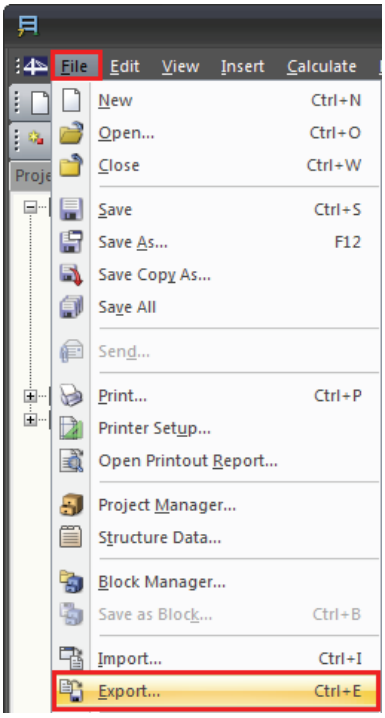


Figure 2.11: Starting the export

A dialog box opens where you can see a number of export options.

Select "Tekla Structures" in the **Direct Exports** dialog section. Further settings are available by clicking the "Details" button.

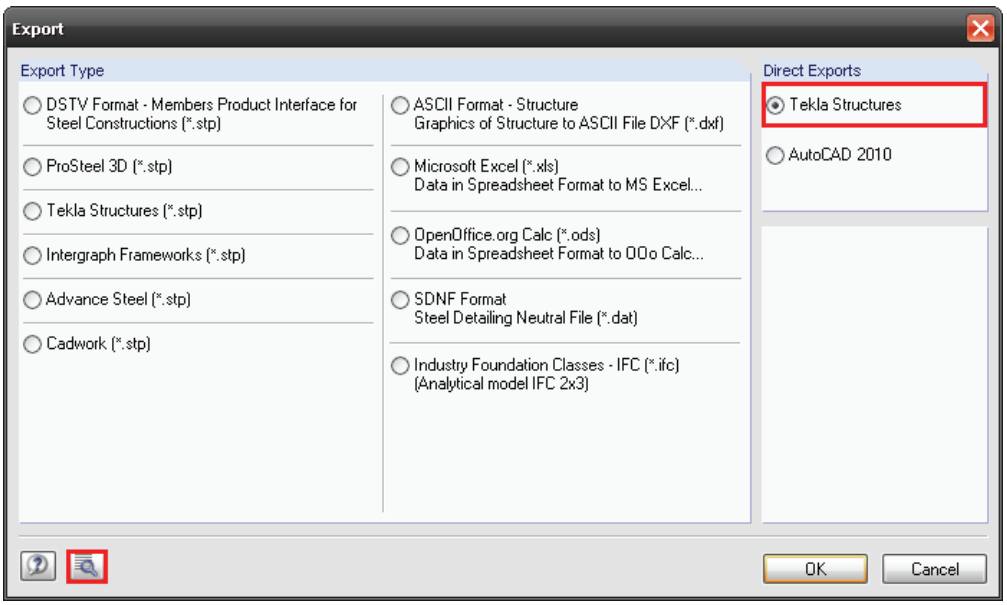


Figure 2.12: Export to Tekla Structures

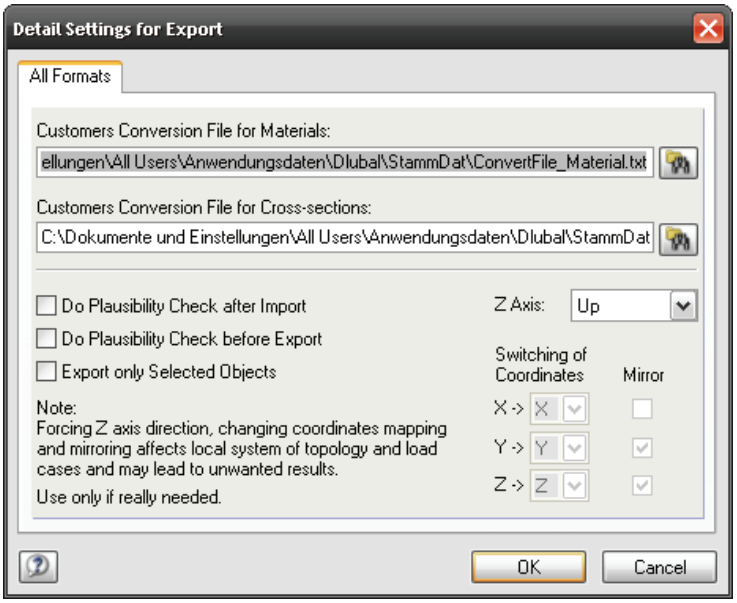
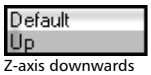
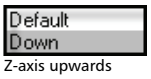


Figure 2.13: Detail settings

In this dialog box, you can define, for example, the orientation of the Z-axis. Please note that the Z-axis in Tekla Structures is always oriented upwards and negative Z-coordinates are not possible.



Z-axis downwards



Z-axis upwards

Depending on how the orientation of the Z-axis in the RSTAB/RFEM structure is determined, the settings "Default" and "Up" or "Default" and "Down" are available in the pull-down menu for the "Z Axis". If "Up" or "Down" are selected, the coordinates will be mirrored automatically. If you select "Default", the Z-axis will always be exported upwards. Furthermore, the "Default" setting allows for a free selection of coordinates and their mirroring.

With the help of this dialog box, you can also carry out a plausibility check by RSTAB/RFEM previous to the export.

Furthermore, it is possible to export only selected objects in case you have marked them previously.

When you have defined all possible settings, click "OK" in the "Export" dialog box. A dialog box opens with setting options specific to Tekla Structures.

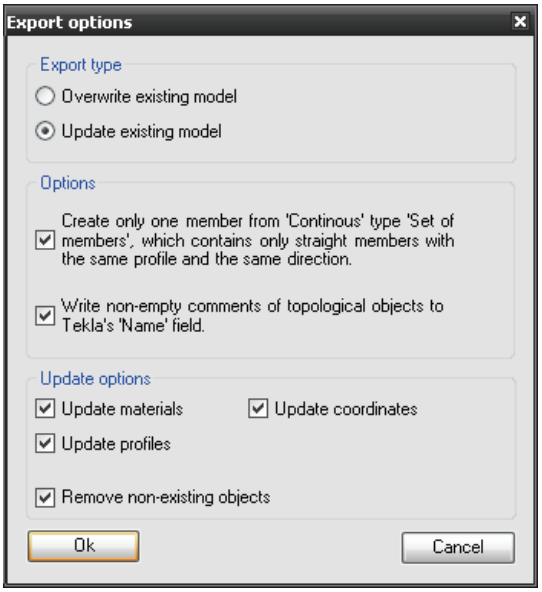


Figure 2.14: Export options

First, select the **Export type**. There are two options available:

- "Overwrite existing model" and
- "Update existing model".

In both cases, it is required that a model has been previously opened in Tekla Structures.

In the **Options** dialog section, you can decide if members that have been defined as continuous members in RSTAB/RFEM are transferred to Tekla Structures as *complete* columns/beams or as single members. In addition, it is possible to assign comments of members/sets of members as names of the corresponding elements in Tekla Structures.

The **Update options** are only available if you have selected "Update existing model" in the **Export type** dialog section. You can update materials, cross-sections and coordinates as well as remove objects that do not exist any longer.

In the following, you find an overview with all available export options.

Structural data:

- Members
- Curved members from RFEM
- Surfaces
- Varying surface thicknesses
- Openings
- Guidelines as grid
- Materials (see p. 18)
- Cross-sections (see p. 20)

Update of:

- Members
- Cross-sections
- Coordinates
- Materials
- Surface thicknesses
- Adding/Removing structural components

2.3.3 Import

To import a structure from Tekla Structures to RSTAB/RFEM, follow the instructions described below:

Click "Import" on the "File" menu in the RSTAB/RFEM menu bar.

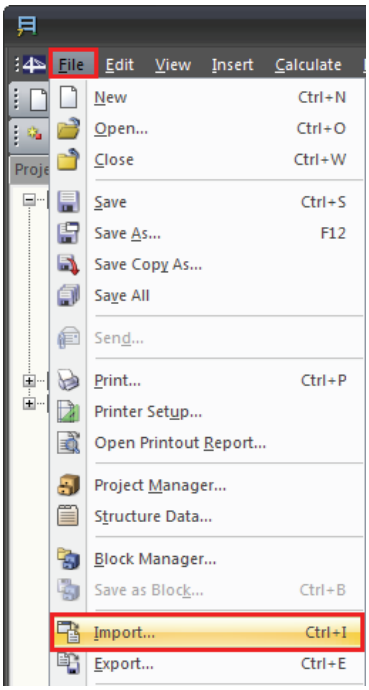


Figure 2.15: Starting the import

A dialog box opens where you can see a number of import options.

Select "Tekla Structures" in the **Direct Imports** dialog section. Further settings are available by clicking the "Details" button.

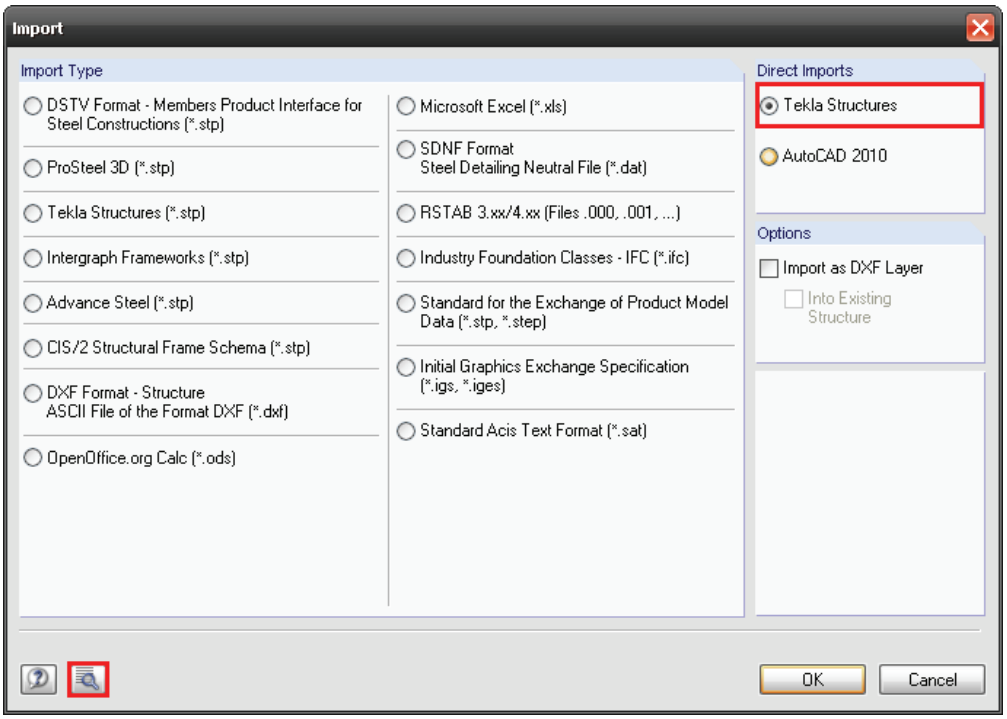


Figure 2.16: Import from Tekla Structures

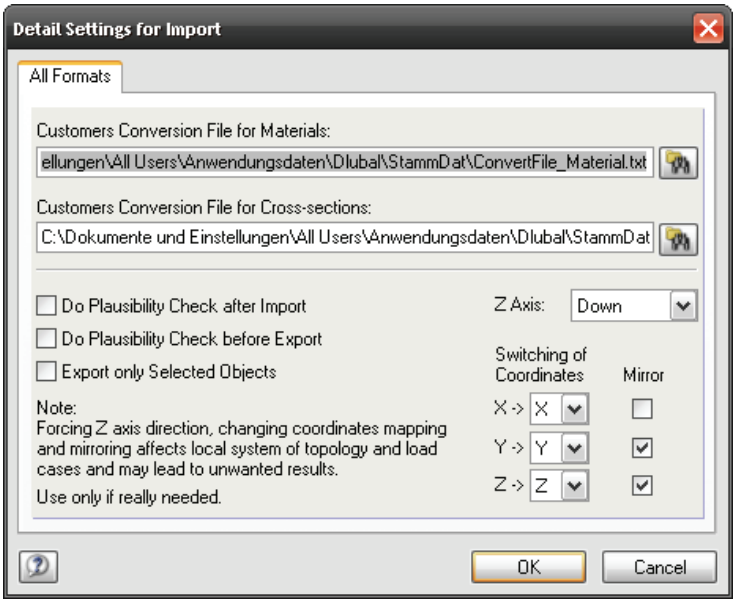
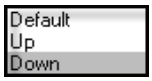


Figure 2.17: Detail settings

In this dialog box, you can define, for example, the orientation of the Z-axis.



Three options are available in the pull-down menu to define the orientation of the Z-axis in RSTAB/RFEM. You can select "Default", "Up" and "Down". If "Up" or "Down" are selected, the coordinates will be mirrored automatically. If you select "Default", the Z-axis will always be imported as it is defined in the source file. Furthermore, the "Default" setting allows for a free selection of coordinates and their mirroring.

With the help of this dialog box, you can also carry out a plausibility check by RSTAB/RFEM subsequent to the import.

Furthermore, it is possible to import only selected objects in case you have marked them previously.

When you have defined all possible settings, click "OK" in the "Import" dialog box to start the import. For this type of import, no further setting options are available. This means that an update is not possible in the import process of this kind of data exchange.

In the following, you find an overview with all available import options.

Structural data:

- Members
- Surfaces
- Varying surface thicknesses
- Openings
- Materials (see p. 18)
- Cross-sections (see p. 20)

2.4 Data Exchange with STP Files (DSTV)

This data exchange option can be used to transfer the **physical model** of a structure.

The following files can be exported to or imported from an stp file:

Structural data:

- Members
- Only selected structural components
- Materials (see p. 18)
- Cross-sections (see p. 20)

Furthermore, it is possible to control the orientation of the Z-axis in the import/export process.

3. Materials and Cross-sections

As a matter of principle, when exchanging data between Tekla Structures and RSTAB/RFEM, the data bases for materials and cross-sections from RSTAB/RFEM are used.

The respective Tekla Structure name is compared with the corresponding data base and, in case they are consistent, the relevant entry will be imported from the data base.

If they are not consistent, an error message will be displayed for materials in Tekla Structures and the Tekla Structures name will be imported for cross-sections. Even the cross-section properties will be transferred according to the cross-section data base of Tekla Structures.

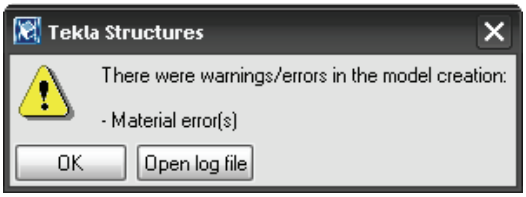
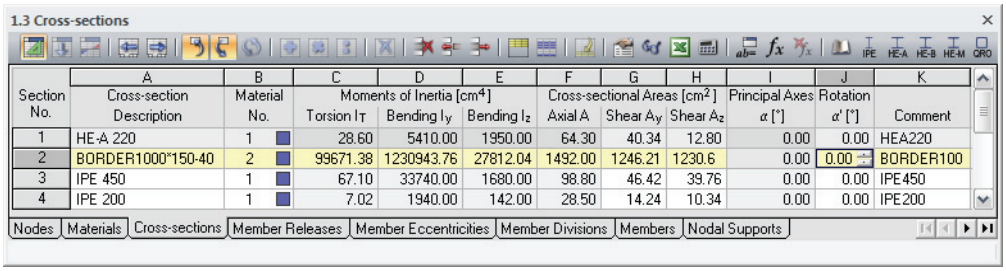


Figure 3.1: Material is not supported



Section No.	A Cross-section Description	B Material No.	C Moments of Inertia [cm⁴] Torsion I_T	D Bending I_y	E Bending I_z	F Cross-sectional Areas [cm²] Axial A	G Shear A_y	H Shear A_z	I Principal Axes α [°]	J Rotation α' [°]	K Comment
1	HE-A 220	1	28.60	5410.00	1950.00	64.30	40.34	12.80	0.00	0.00	HEA220
2	BORDER1000*150-40	2	99671.38	1230943.76	27812.04	1492.00	1246.21	1230.6	0.00	0.00	BORDER100
3	IPE 450	1	67.10	33740.00	1680.00	98.80	46.42	39.76	0.00	0.00	IPE450
4	IPE 200	1	7.02	1940.00	142.00	28.50	14.24	10.34	0.00	0.00	IPE200

Figure 3.2: RSTAB table 1.3 *Cross-sections*

3.1 Materials

User-defined materials from Tekla Structures can be imported to RSTAB/RFEM by means of a special material conversion. This data exchange applies also to the export.

In the following, you find a description on how to convert material.

First, find out where the material conversion file is stored. Select "Import" on the RSTAB/RFEM "File" menu to open the "Import" dialog box. Then click the "Details" button to open another dialog box. In the "All Formats" tab, the appropriate file directory is shown.

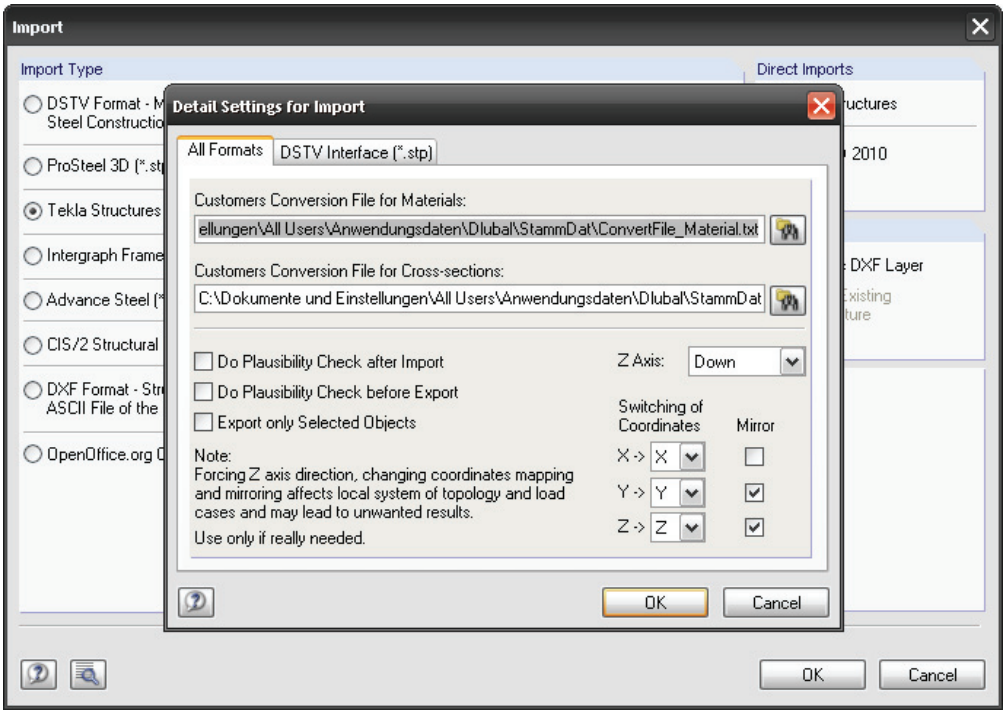


Figure 3.3: RFEM directory for material conversion file

Open the file "ConvertFile_Material.txt", for example by using the editor.

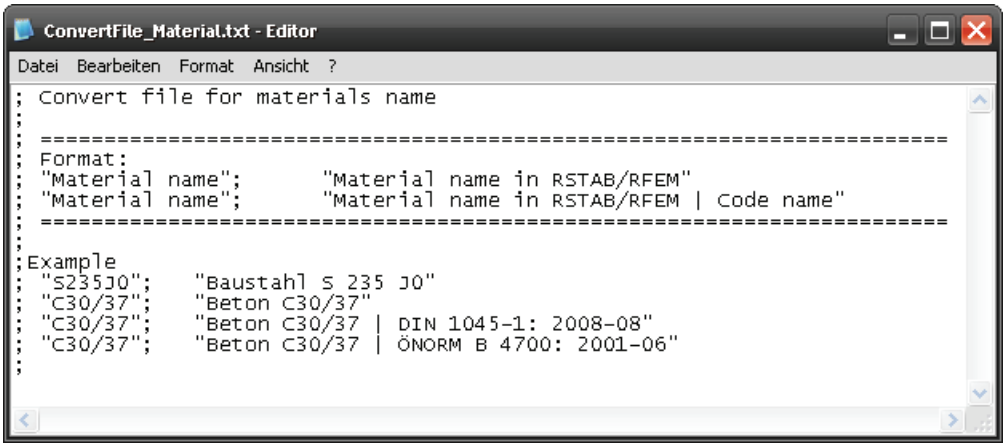


Figure 3.4: Material conversion file

Now you can define your own conversion settings by entering the accurate Tekla Structures name and the right name from RSTAB/RFEM as shown in the example above. Unfortunately, a standard cannot be assigned.

Please take care to avoid semicolons in front of the line containing the material name, as otherwise the complete line will be evaluated as a "comment" and therefore will lose its influence on the conversion.

Save the conversion file by clicking "Save as" on the "File" menu. In the next dialog box, select "UTF-8" in the coding field and overwrite it with the present file.

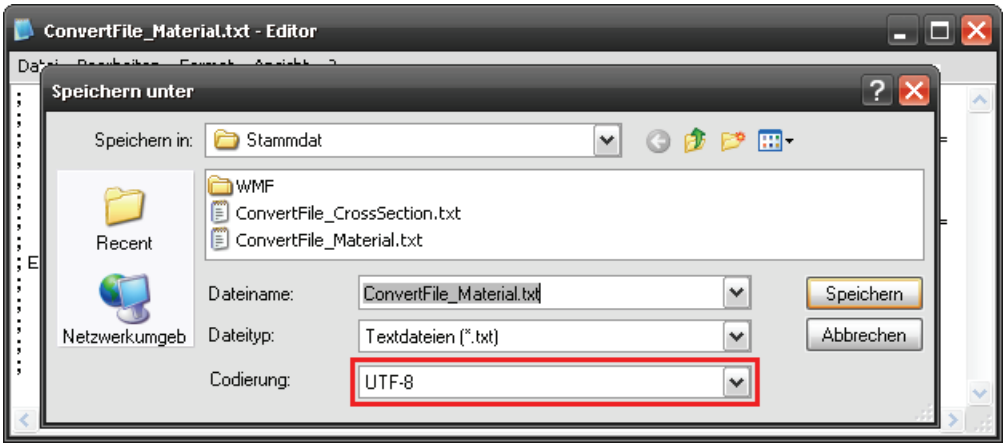


Figure 3.5: Material conversion file

3.2 Cross-sections

The conversion for cross-sections is similar to the conversion of materials. It is possible to create an appropriate cross-section conversion file, for example to recognize directly cross-sections created in the SHAPE add-on module when transferring data from Tekla Structures to RSTAB/RFEM.

In the following, you find a description on how to convert cross-sections.

First, find out where the cross-section conversion file is stored. Select "Import" on the RSTAB/RFEM "File" menu to open the "Import" dialog box. Then click the "Details" button to open another dialog box. In the "All Formats" tab, the appropriate file directory is shown.

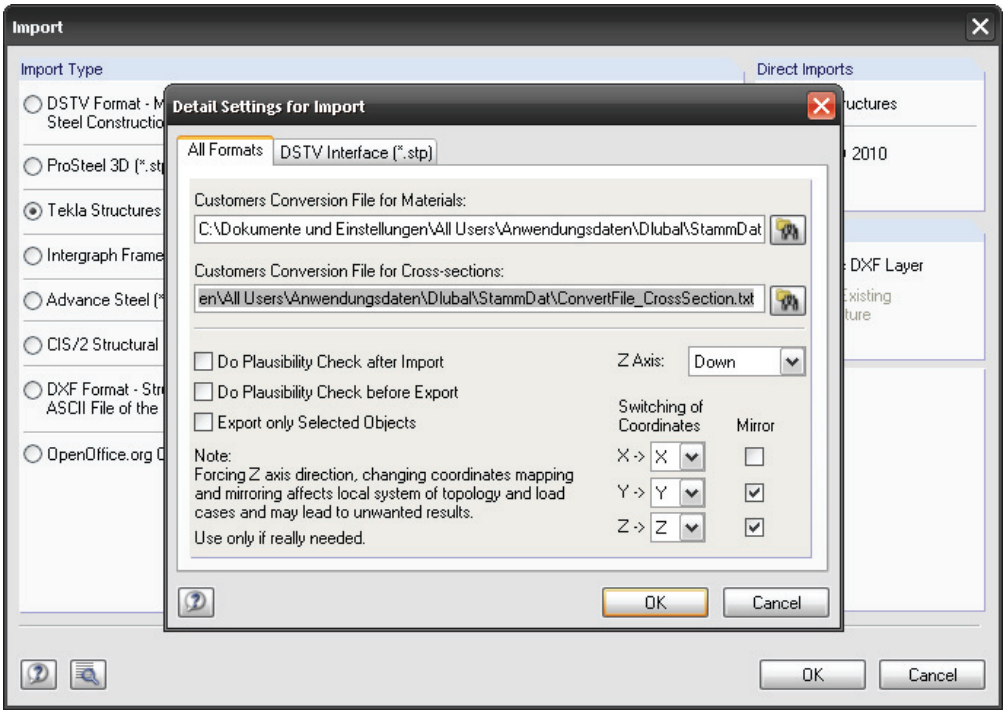


Figure 3.6: RFEM directory for cross-section conversion file

Open the file "ConvertFile_CrossSection.txt" by using the editor, for example.

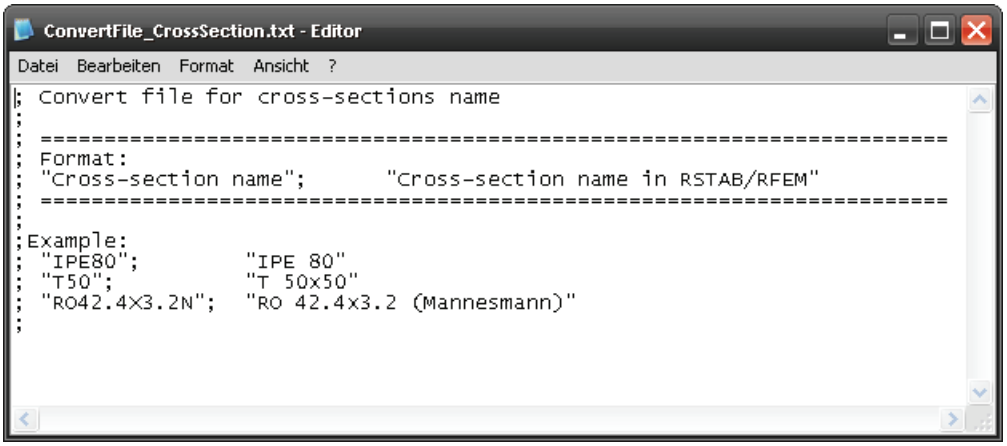


Figure 3.7: Cross-section conversion file

Now you can define your own conversion settings by entering the accurate Tekla Structures name and the right name from RSTAB/RFEM as shown in the example above.

Please take care to avoid semicolons in front of the line containing the cross-section name, as otherwise the complete line will be evaluated as a "comment" and therefore will lose its influence on the conversion.

Save the conversion file by clicking "Save as" on the "File" menu. In the next dialog box, select "UTF-8" in the coding field and overwrite it with the present file.

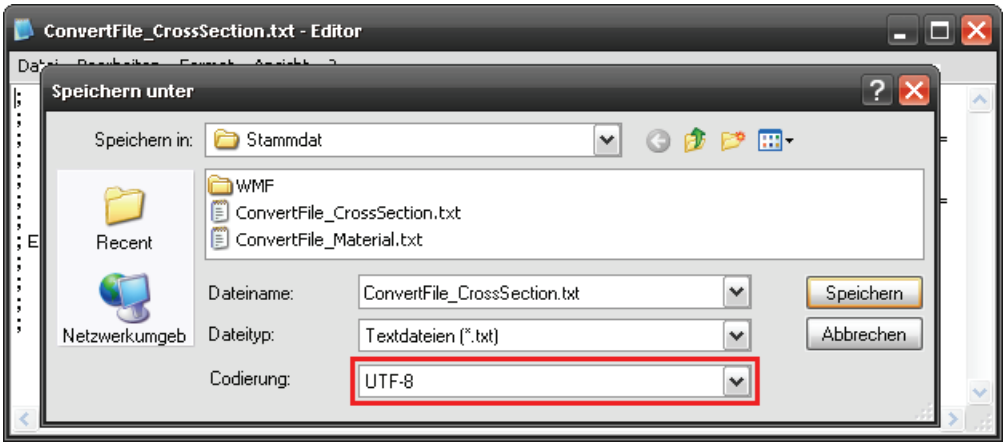


Figure 3.8: Cross-section conversion file

Parametric I-sections (IS, ICM) can also be transferred without explicit references within the cross-section conversion file in both directions.