

Version February 2010

Interface

RX-Tekla

Interfaces between RSTAB/ RFEM and Tekla Structures

Program Description

All rights, including those of translations, are reserved.

No portion of this book may be reproduced – mechanically, electronically, or by any other means, including photocopying – without written permission of Dlubal Engineering Software.

© Dlubal Engineering Software 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

	Contents	Page		Contents	Page
1.	Introduction	4	2.3	Direct Export/Import to/from RSTAB/RFEN	1 10
1.1	Data Exchange Options	4	2.3.1	Functionality	10
1.2	RX-Tekla Team	4	2.3.2	Export	10
2.	Procedure	5	2.3.3	Import	14
2.1	General Information	5	2.4	Data Exchange with STP Files (DSTV)	16
2.2	Direct Export from Tekla Structures	5	3.	Materials and Cross-sections	17
2.2.1	Functionality	5	3.1	Materials	18
2.2.2	Export	7	3.2	Cross-sections	20
2.2.3	Reimport	8			



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 crosssections 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 Dipl.-Ü (Uni) Gundel Pietzcker Ing. Ladislav Kábrt

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".

🕅 Analysis & Design	Models	_	_	_	_	_ 🗆 🔀
Analysis model name	Analysis application	Creation method	Results	Number of parts	New	Delete
Modell 4	Dlubal	Full model	Status unknown	413	Properties	Select objects
					Add selected objects	Remove selected objects
					Show in model	Update and show
					Load co	mbinations
					Refresh	Details
Analysis application i	nterface					
Run	Create model Open	application Close	application Ge	t results Get result	s for selected	Close

Figure 2.2: Starting the interface

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

Select Target Program for Export	×
RSTAB	
 Create New Structure 	
Overwrite Existing Structure	
◯ Create a file	
RFEM	
Create New Structure	
Overwrite Existing Structure	
◯ Create a file	
Settings OK Cancel	

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.

Settings	x
	_
Set Z axis up	
This setting has only an effect when creating a new structure	
Mirror:	
X-Coordinates	
✓ Y-Coordinates	
Z-Coordinates	
Create unique profile for each of 'Set of members'	
OK	

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.

🕅 Analysis & Design	🕅 Analysis & Design Models 🔹 🗖 🔼								
Analysis model name	Analysis application	Creation method	Results	Number of parts	New	Delete			
Modell 4	Dlubal	Full model	Status unknown	413	Properties	Select objects			
					Add selected objects	Remove selected objects			
					Show in model	Update and show			
					Load co	ombinations			
					Refresh	Details			
Analysis application i	nterface								
Run	Create model Open	application Close	e application Ge	t results Get resul	ts for selected	Close			

Figure 2.5: Creating an analysis model

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

🕅 Analysis Model Properties	
Analysis model Analysis Job Output	Seismic Seismic masses Modal analysis Design - Steel Design - Concrete Design - Timber
Analysis application	Dlubal
Analysis model name:	Modell 4 Modal analysis model
Creation method:	Full model V Filter Statik
Member axis location	Reference axis
Node definition:	Force to centric connection
Model merging with analysis application	Disabled V Reset
Member end release method by connection:	No
	Cancel Help

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-

Full model By work area By selected parts By selected parts and loads Floor model by selected parts and load

Neutral axis Reference axis (eccentricity by neutral a Reference axis Model default

Use rigid links Force to centric conr isabled) inabled



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".

	Analysis & Design	Models					_ 🗆 🔀
A	analysis model name	Analysis application	Creation method	Results	Number of parts	New	Delete
0	Modell 4	Dlubal	Full model	Status unknown	413	Properties	Select objects
						Add selected objects	Remove selected objects
						Show in model	Update and show
						Load co	mbinations
						Refresh	Details
	Analysis application in	nterface					
	Run	Create model Open	application Close	e application G	et results Get result	ts for selected	Close

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.

Select Dlubal Program	Select Results from LC/LG/CO
Program ⓒ RSTAB ◯ RFEM	LC1 - Self-weight LC2 - Live load LC3 - Snow C01 - Ultimate limit state C02 - Serviceability Limit State
OK Cancel	OK Cancel

Figure 2.8: Selection of program and loads

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

Optimization Resutts	_	_	_	_	_	×
Design group	Original profile	New profile	Number of parts	State	Accepted	
Stütze-HEA220	HEA220	HE2208	9		No	
Accept all Accept selected Acc	ept selected for selected o	objects only 🔽 Use	design groups 🛛 🗹 Show	changes on	y	Close

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).

🔀 Tekla Structures	Stütze (1)	$\overline{\mathbf{X}}$
Parameters Status	Status 2 End codes	Analysis Analysis(2) IFC export
End reactions		
	Start:	End:
Shear, V	7.09	9.61
Tension, T	112.31	48.69
Moment, M	8.53	✓ 11.25
Connection code	¥	
Utility ratio		
Reinforcement are	a	
ОК Арр	ly Modify	Get 🔽/Г Cancel

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.

Export		X
Export Type	1	Direct Exports
ODSTV Format - Members Product Interface for Steel Constructions (*.stp)	ASCII Format - Structure Graphics of Structure to ASCII File DXF (*.dxf)	Tekla Structures
O ProSteel 3D (*.stp)	Microsoft Excel (*.xls) Data in Spreadsheet Format to MS Excel	○ AutoCAD 2010
○ Tekla Structures (*.stp)	□ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □	
O Intergraph Frameworks (*.stp)	Data in Spreadsheet Format to OOo Calc	
○ Advance Steel (*.stp)	SDNF Format Steel Detailing Neutral File (*.dat)	
◯ Cadwork (*.stp)	O Industry Foundation Classes - IFC (*.ifc) (Analytical model IFC 2x3)	
		OK Cancel

Figure 2.12: Export to Tekla Structures



Detail Settings for Export	×
All Formats	
Customers Conversion File for Materials:	
ellungen\All Users\Anwendungsdaten\Dlubal\StammDa	at\ConvertFile_Material.txt 🐘
Customers Conversion File for Cross-sections:	
C:\Dokumente und Einstellungen\All Users\Anwendung	psdaten/Dlubal/StammDat 🌆
Do Plausibility Check after Import	Z Axis: Up 🗸
Do Plausibility Check before Export Export only Selected Objects	Switching of Coordinates Mirror
Note: Forcing Z axis direction, changing coordinates mapping and mirroring affects local system of topology and load	X → X ♥ Y → Y ♥
cases and may lead to unwanted results.	Z → Z 🔽 🔽
	OK Cancel

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.

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.

Default Up Z-axis downwards Default Down Z-axis upwards



Export options X
Export type
Overwrite existing model
 Update existing model
Options
Create only one member from 'Continous' type 'Set of members', which contains only straight members with the same profile and the same direction.
✓ Write non-empty comments of topological objects to Tekla's 'Name' field.
Update options
✓ Update materials ✓ Update coordinates
✓ Update profiles
Remove non-existing objects
Ok Cancel

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.

Import		×
Import Type		Direct Imports
DSTV Format - Members Product Interface for Steel Constructions (*.stp)	O Microsoft Excel (*.xls)	Tekla Structures
O ProSteel 3D (*.stp)	 SDNF Format Steel Detailing Neutral File (*.dat) 	O AutoCAD 2010
🔿 Tekla Structures (*.stp)	O RSTAB 3.xx/4.xx (Files .000, .001,)	Ontions
O Intergraph Frameworks (*.stp)	O Industry Foundation Classes - IFC (*.ifc)	Import as DXF Layer
O Advance Steel (*.stp)	Standard for the Exchange of Product Model Data (*.stp, *.step)	Into Existing Structure
◯ CIS/2 Structural Frame Schema (*.stp)	 Initial Graphics Exchange Specification 	
DXF Format - Structure ASCII File of the Format DXF (*.dxf)	(*.igs, *.iges)	
OpenOffice.org Calc (*.ods)	Standard Acts Text Format (".sat)	
(7) 3		

Figure 2.16: Import from Tekla Structures



Detail Settings for Import	
All Formats	
Customers Conversion File for Materials:	
ellungen\All Users\Anwendungsdaten\Dlubal\StammDa	at\ConvertFile_Material.txt 🐘
Customers Conversion File for Cross-sections:	
C:\Dokumente und Einstellungen\All Users\Anwendung	psdaten/Dlubal/StammDat 🐘
Do Plausibility Check after Import	Z Axis: 🛛 Down 🔽 🗸
Do Plausibility Check before Export	Switching of
Export only Selected Objects	Coordinates Mirror
Note:	X -> X 💌 🗖
and mirroring affects local system of topology and load	Y → Y 💌 🔽
cases and may lead to unwanted results.	Z→Z V V
Use only il really needed.	
	OK Cancel

Figure 2.17: Detail settings

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

Default Up. Dow

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 Crosssections

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 crosssection properties will be transferred according to the cross-section data base of Tekla Structures.



Figure 3.1: Material is not supported

	2) Cossistencins 2) [2] [2] [2] [2] [2] [2] [2] [2] [2] [2]												
	A	В		С	D	E	F	G	Н		J	K	1~
Section	Cross-section	Mater	ial	Mome	nts of Inertia [cm ⁴]	Cross-se	ctional Area	as [cm ²]	Principal Axes	Rotation		10
No.	Description	No.		Torsion IT	Bending ly	Bending Iz	Axial A	Shear Ay	Shear Az	α[*]	α' [*]	Comment	=
1	HE-A 220	1		28.60	5410.00	1950.00	64.30	40.34	12.80	0.00	0.00	HEA220	1
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	IPE 450	T
4	IPE 200	1		7.02	1940.00	142.00	28.50	14.24	10.34	0.00	0.00	IPE200	V
Nodes	Impediate Imped												

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.

nport			
Import Type		Direct Imp	oorts
ODSTV Format - M Steel Constructio	Detail Settings for Import	Đ	
O ProSteel 3D (*.st	All Formats DSTV Interface (*.stp)		2010
 Tekla Structures 	Customers Conversion File for Materials:		
🔿 Intergraph Frame	ellungen/All Users/Anwendungsdaten/Dlubal/StammDat	ConvertFile_Material.txt	: DXF Layer
🔿 Advance Steel (*	C:\Dokumente und Einstellungen\All Users\Anwendungs	adaten/Dlubal/StammDat	Existing ture
CIS/2 Structural	Do Plausibility Check after Import	Z Axis: Down	
DXF Format - Stri ASCII File of the	Do Plausibility Check before Export Export only Selected Objects	Switching of Coordinates Mirror	
OpenOffice.org C	Note: Forcing Z axis direction, changing coordinates mapping and mirroring affects local system of topology and load cases and may lead to unwanted results. Use only if really needed.	$\begin{array}{c} X \rightarrow X \checkmark \checkmark \\ Y \rightarrow Y \checkmark \checkmark \checkmark \\ Z \rightarrow Z \checkmark \checkmark \checkmark \checkmark \end{array}$	
	2	OK Cancel	
0		ОК	Cancel

Figure 3.3: RFEM directory for material conversion file

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

🖡 ConvertFile_Material.txt - Editor	\mathbf{X}
Datei Bearbeiten Format Ansicht ?	
; Convert file for materials name ;	^
Format: "Material name"; "Material name in RSTAB/RFEM" "Material name"; "Material name in RSTAB/RFEM Code name" Example "S235J0"; "Baustahl S 235 J0" "C30/37"; "Beton C30/37" "C30/37"; "Beton C30/37 DIN 1045-1: 2008-08" "C30/37"; "Beton C30/37 DIN 1045-1: 2001-06"	*

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.

	ConvertFile_Mater	rial.txt - Editor							×
Da		wardt Arniskt D	_	-	_	_	2 🗸		
E.	speicnern unter						🖸 🔼		^
Ľ	Speichern in:	🚞 Stammdat		~	G 🦻	ج 🔁		-	
; ; ;	D Recent	WMF ConvertFile_Cr ConvertFile_Ma	ossSection.txt aterial.txt					-	
]; [_]		Dateiname:	ConvertFile_Material.txt			~	Speichern		
ļ	Netzwerkumgeb	Dateityp:	Textdateien (*.txt)			~	Abbrechen		
;		Codierung:	UTF-8			~			~
<							.:	>	

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.

Import			×
Import Type		Direct Impo	orts
O DSTV Format - M Steel Constructio	Detail Settings for Import	×	uctures
O ProSteel 3D (*.st	All Formats DSTV Interface (*.stp)		2010
Tekla Structures	Customers Conversion File for Materials: C:\Dokumente und Einstellungen\All Users\Anwendung:	sdaten\Dlubal\StammDat	
O Intergraph Frame	Customers Conversion File for Cross-sections:		: DXF Layer
🔿 Advance Steel (*	en\All Users\Anwendungsdaten\Dlubal\StammDat\Com	vertFile_CrossSection.txt	Existing ture
CIS/2 Structural	Do Plausibility Check after Import	Z Axis: Down 🗸	
ODXF Format - Stri ASCII File of the	Do Plausibility Check before Export Export only Selected Objects	Switching of Coordinates Mirror	
OpenOffice.org C	Note: Forcing Z axis direction, changing coordinates mapping and mirroring affects local system of topology and load cases and may lead to unwanted results.	$\begin{array}{c c} X \Rightarrow X \checkmark & \blacksquare \\ Y \Rightarrow Y \checkmark & \checkmark \\ Z \Rightarrow Z \checkmark & \checkmark \end{array}$	
		OK Cancel	
2		OK	Cancel

Figure 3.6: RFEM directory for cross-section conversion file

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

ConvertFile_CrossSection.txt - Editor		$\mathbf{\times}$
Datei Bearbeiten Format Ansicht ?		
; Convert file for cross-sections name ;		^
Format: "Cross-section name"; "Cross-section name in RSTAB/RFEM" ;		
; Example: "IPE80"; "IPE 80" ; "T50"; "T 50x50" ; "R042.4x3.2N"; "R0 42.4x3.2 (Mannesmann)" ;		
<u><</u>	>	

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.

	🕞 ConvertFile_CrossSection.txt - Editor							
Dat ;	Speichern unter	unan Araisht 7		_	_	? 🔀	<u> </u>	
ļ	Speichern in:	🚞 Stammdat	[🕶 Ġ 🦻	۳ 📂		=	
; ; ; E	📁 Recent	🗀 WMF 赏 ConvertFile_CrossSection.txt ፪ ConvertFile_Material.txt				=		
ļ		Dateiname:	ConvertFile_CrossSection.txt		~	Speichern		
 [;]	Netzwerkumgeb	Dateityp:	Textdateien (*.txt)		~	Abbrechen		
		Codierung:	UTF-8		~		~	
							2	

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.