

Tekla StruXML Export

User's Guide to Tekla Structures - FEM-Design Integration

StruSoft AB

www.strusoft.com

Version: September 25th, 2017

Copyright

Copyright© 2017 by StruSoft. All rights reserved.

Content of this publication may not be reproduced or transmitted in any means without the written permission of StruSoft AB.

Trademarks

 $\mathsf{FEM}\text{-}\mathsf{Design}\ensuremath{\mathbb{R}}$ is a registered trademark of StruSoft.

Tekla® Structures is a registered trademark of Tekla.

Disclaimer

The Tekla StruXML Export is a tool that enables a link between Tekla Structures and FEM-Design. Substantial amount of time and effort have gone into development and testing Tekla StruXML Export tool. We did our best to ensure the reliability of the software and the accuracy of this document. However, the user must accept that no warranty is given by the developers concerning accuracy of this software or information found in this document.

Anyone that has doubts concerning the accuracy of the Tekla StruXML Export, or has suggestions regarding development of the Tekla StruXML Export, is welcome to contact us at: <u>iwona.budny@strusoft.com</u>.

For support, please use: <u>support.femdesign@strusoft.com</u>. When sending support question, please remember to always attach an original Tekla Structures model, FEM-Design model and a struxml file.

Current link versions

- Tekla StruXML Export 1.1.007

Compatibility

- Tekla Structures: version 18-21.1 and 2016, 2016i, 2017, 2017i
- FEM-Design 15.01 and later

Download

- FEM-Design Download Center
- StruSoft Installer

Additional materials

- Tekla FEM-Design Integration
- <u>Forum</u>
- StruSoft Official YouTube Channel

Table of Contents

1.	INTF	RODUCTION	. 7
1	L.1.	MANUAL SCOPE	7
1	L.2.	INSTALLATION	7
1	L.3.	CONCEPT	7
1	L.4.	WORKFLOW	8
1	L.5.	TRANSFERRED DATA	8
2.	ΤΕΚΙ	LA ANALYSIS MODEL	. 9
2	2.1.	CREATING ANALYSIS MODEL	. 9
2	2.2.	MODEL DISPLAY OPTIONS	12
	2.2.1	1. Node and bar numbers visibility	12
	2.2.2	2. Physical objects visibility	12
2	2.3.	ANALYSIS MODEL CHECK	13
	2.3.1	1. Analysis nodes colors	13
	2.3.2	2. Checking warnings	13
2	2.4.	Analysis model adjustment	15
	2.4.1	1. Analysis part properties	15
	2.4.2	2. Manual adjustment	22
3.	EXPO	ORTING TEKLA MODEL TO FEM-DESIGN	26
3	3.1.	CREATE AN ANALYSIS MODEL	26
3	3.2.	CHECK AND ADJUST AN ANALYSIS MODEL	28
3	3.3.	TEKLA STRUXML EXPORT	32
3	3.4.	OPEN STRUXML FILE IN FEM-DESIGN.	36
3	8.5.	Errors at exporting an analysis model	37
	3.5.1	1. Lack of material definition	37
	3.5.2	2. Lack of material / section mapping	37
	3.5.3	3. Invalid mapping	38
	3.5.4	4. Bad surface geometry	38
	3.5.5	5. How to find a certain element in Tekla?	38
	3.5.6	6. How to find a certain element in FEM-Design?	42

NEW FEATURES, FIXES AND CHANGES COMPARED TO TEKLA STRUXML EXPORT 1.1.006

ADDED:

1. Compatibility with Tekla Structures 2017i.

FIXED:

2. Sections rotation was not exported correctly at some cases.

NEW FEATURES, FIXES AND CHANGES COMPARED TO TEKLA STRUXML EXPORT 1.1.005

ADDED:

- 1. Compatibility with Tekla Structures 2017.
- 2. FEM-Design accuracy is used to check the quality of the analysis parts exported from Tekla.

Now also walls are checked against FEM-Design accuracy requirements. If a surface object does not meet FEM-Design geometry requirements, it will be stopped at the export process. More about bad geometry errors can be found in Chapter 3.5.4.

3. Possibility to copy object's GUID by right clicking on processed object.

It is now possible to right click on the log line with processed object and copy its GUID that can be used in order to search for this object in Tekla, struxml file, or in FEM-Design. More can be found in Chapters 3.5.5 and 3.5.6.

CHANGED:

4. Guid is used as identification parameter, instead of Tekla ID.

<u>Tekla 2017 no longer supports object ID</u> (it still exists in Tekla, but is not a constant value for a given object anymore, and cannot be used in Selection Filter). Therefore, we introduced following changes:

- objects are no longer exported with Tekla ID as Name (FEM-Design default names are used instead, e.g. C, B, P, W),

- in the log area, a GUID of processed element's parent object (physical object) is listed, instead of Tekla ID,

Tekla GUID is now exported to struxml along with an element, instead of random GUID.

FIXED:

5. More clear error message in case a section cannot be found at the export.

1. Introduction

1.1. Manual scope

This document describes the concept behind the link between Tekla Structures and FEM-Design, and explains how to exchange data between those two programs using the Tekla StruXML Export tool.

1.2. Installation

Download the latest version of Tekla StruXML Export from StruSoft Installer or from <u>FEM-Design</u> <u>Download Center</u> and run the installation file.

When the installation process is completed, Tekla StruXML Export tool will appear under StruSoft folder in the Start menu, as shown in Figure 1-1.

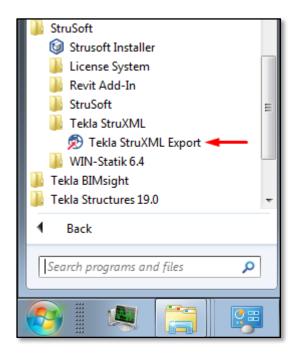


Figure 1-1

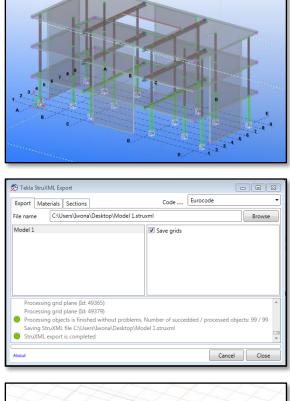
1.3. Concept

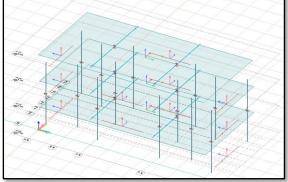
Tekla StruXML Tool enables direct communication between FEM-Design and Tekla Structures.

The direct data transfer is solely based on Tekla Analysis Model export. The exact geometry of selected analysis model can be exported to an external file in xml format (struxml), which can be opened in FEM-Design.

1.4. Workflow

The workflow to export a Tekla Structures model to StruSoft FEM-Design is shown in the Figure 1-2:





Open a Tekla Structures model:

- Create Tekla Analysis Model.
- Check and if necessary, adjust the Tekla Analysis model.



Start the Tekla StruXml Export tool:

- Select the analysis model to export.
- Map the materials and sections.
- Export the model to a struxml file.



Open FEM-Design 3D Structure:

- Open the struxml file in FEM-Design.
- Add loads and supports.
- Modify members end conditions, if necessary.
- Start analysis and design.



1.5. Transferred data

The Tekla StruXml Export tool transfers the geometry of analysis model together with some fundamental parameters. The list of transferred data is as follows:

- geometry of analytical model,
- mapped cross-section of linear member (cross-section in start node),
- mapped material,
- releases of linear members (semi rigid releases are exported as fixed),
- rigid links (exported as fictitious bar)
- grids system.

2. Tekla Analysis Model

Tekla Analysis model is a structural representation of a Tekla physical model. Analysis model is used for structural analysis and design, and can be exported into number of FEM-programs, including StruSoft FEM-Design.

Tekla analysis model is generated automatically based on existing physical model geometry and according to some predefined settings. In many cases the analysis model however, requires some adjustments before it can be successfully exported to a FEM-program. Since the exact geometry of analysis model is exported, it is extremely important to make sure that all analysis parts are connected with each other and that the model is consistent.

Therefore, in this chapter some basic information regarding Tekla Analysis Model is gathered. Areas such creating analysis model, model display options, model connectivity check, and model adjustments are covered.

Please note, that to get the best overview of Tekla analysis model, it is highly advised to get familiar with the official <u>Tekla Analysis Guide</u>.

2.1. Creating analysis model



More information: <u>What is Tekla analysis model?</u> <u>Creating Analysis model</u> <u>Analysis model properties</u>

To create an analytical model of an existing Tekla physical model, one shall go to *Analysis* tab and start *Analysis and Design Models*... A new dialog will pop out (Figure 2-1), where one can create and modify analysis models.

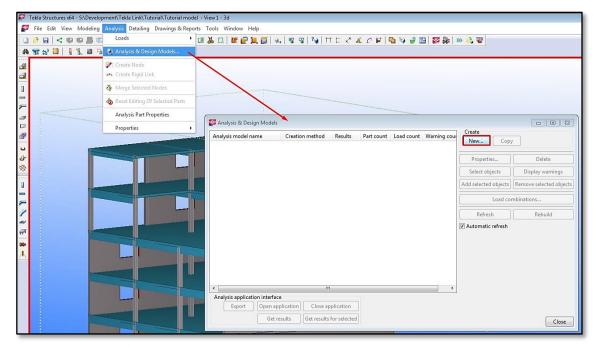


Figure 2-1

Press <u>New</u> to create an analysis model. An <u>Analysis Model Properties</u> dialog will pop out (Figure 2-2). Here you can:

- give an unique name to the analysis model,
- decide about the model creation method (full model, or selected parts),
- define which objects should be included in the analysis model, based on one of the selection filters,
- define secondary members,
- choose an analysis application note that Tekla StruXML Export tool is not listed here so this parameter is not relevant.

	Analysis & Design Models				
	Analysis model name Creation method Results	Part count Load cour	t Warning cou	Create New Copy	/
				Properties	Delete
				Select objects	Display warnings
Analysis Model Properties		X		Add selected objects	Remove selected objects
Analysis Model Properties				Load cor	mbinations
Save	Save as			Refresh	Rebuild
Analysis model Analysis Job Output Seismic Seis	mic masses Modal analysis Design - Steel Design - Concrete	e Design - Timber		Automatic refresh	
Analysis model name: Model 1	Browse for export folder	r			
Creation method: Full model	•				
Filter None	•				
Secondary member filter Auto-detec	t secondary members 👻				
Analysis application	Set as the default				
More settings					
					Close
ОК		Cancel Help	2220-s-sec. Q		

Figure 2-2

Press <u>More settings...</u> to see the other available settings (Figure 2-3).

Analysis application		 Set as the default
Less settings]	
Use rigid links	Disabled, with keep axis: Default	•
Default keep axis for secondary members	No	•
Analysis model rules	Analysis model rules	
Curved beams	Split into straight segments	•
Consider twin profiles	Disabled	•
Member axis location	Model default	•
Member end release method by connection:	No	•
Automatic update	Yes - Physical model changes are considered	•
Model merging with analysis application	Disabled	•
ОК		Cancel Help

Figure 2-3

Here, among other things, you can decide:

- if to include rigid links in the analysis model,
- location of the member analysis axis (neutral axis, reference axis, model default).

Note that 'Automatic update' and 'Model merging with analysis application' parameters are not relevant in case of integration with FEM-Design.

Press <u>OK</u>. Analysis model will be constructed and listed in the Analysis & Design Models dialog as shown in Figure 2-4.

533 144 1991 1991 199	91243	254	10001		
71,251	👺 Analysis & Design Models				
	Analysis model name	Creation method Results	Part count Load count W	Varning co	
1261361381401422181	Model 1	Full model	72 0 6	New	Сору
248 102 68				Properties	Delete
12950				Select objects	Display warnings
100 167 107293				Add selected obje	cts Remove selected objects
15 5156158 160162233 (211 254				Load	I combinations
110/22330 201				Refresh	Rebuild
287 290 22 81				Automatic refre	sh
80 172					
40 31 49					
42044 046 04015-					
4144 146 148 ¹ 50 151 7 47 50		III		+	
46 8 13	Analysis application interface				
B	Get re	esults Get results for selected	J		Close
Y	69		11111-1-1-1-1-1-1-1-1-1-1-1-1-1-1-1-1-		

Figure 2-4

In Tekla it is possible to create many analysis models independently. Each of the analysis models can be built upon different settings, or can include different parts of the model.

Analysis model name	Creation method	Results	Part count	Load count	Warning (Create	
Model 1	Full model		72	0	6	New Copy	r
Model 2 - Top floor	By selected parts and loads		20	0	12		
Model 3 - Ground floor	By selected parts and loads		18	0	8	Properties	Delete
						Select objects	Display warnings
						Add selected objects	Remove selected object
						Load co	mbinations
						Refresh	Rebuild
						✓ Automatic refresh	
					Þ		
Analysis application interfa Export Open a	ce pplication Close application						
Get	results Get results for selected						Close

Figure 2-5

2.2. Model display options

2.2.1. Node and bar numbers visibility

If you do not wish to see an analysis bar, member, or node number in analysis model view, go to *Tools* -> *Options* -> *Advanced Options* and set the following parameters to FALSE:

- XS_AD_MEMBER_NUMBER_VISUALIZATION
- XS_AD_NODE_NUMBER_VISUALIZATION
- XS_AD_NODE_NUMBER_BY_Z

Close and reopen an analysis model in order to apply the changes.

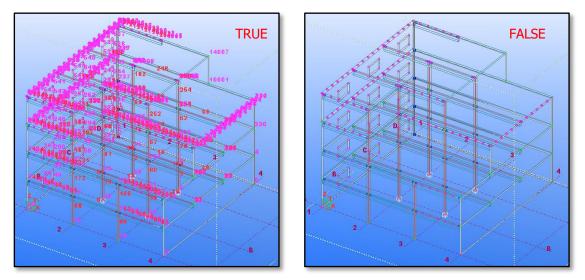
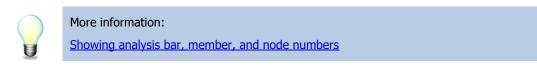


Figure 2-6



2.2.2. Physical objects visibility

If you wish to hide a physical model and display only analysis parts, double click on the screen to start the *View Properties*. Go to <u>Display</u> and uncheck the visibility of 'Parts' in model (Figure 2-7).

Siew Properties			X	Settings Advanced				×
Save Load standard		 Save as standard 		Settings Advanced	Visibility		Representation	
View Name:	3d				In model I	in components		
🗸 Angle:	🗊 3D		-33.50	All				
Projection:	🗇 Orthogonal	Rotation around X:	31.75	Points				
Representation				Parts			Exact	•
Color and transparency in	all views: standard	•	Representation				Wire frame	
							whendhe	
Visibility View depth: View Up:	25000.00						In components: Shaded wire fram	e •
Visibility View depth: VD: VDown:	25000.00 2000.00			Bolts	 V			e •
View depth: View Up:				Bolts Holes		V	In components: Shaded wire fram	•
View depth: 📝 Up:	2000.00	▼ [Object group]					In components: Shaded wire fram	

Figure 2-7

2.3. Analysis model check

2.3.1. Analysis nodes colors

In Tekla, each analysis part has node at its end or corner. These analysis nodes can have one of two colors: magenta or green, depending on its connectivity status (Figure 2-8 (source)). Magenta color means that a node is disconnected, and the green color means that a node is connected. Checking nodes colors is therefore, a good way to visually verify the model connectivity.

Note that not all disconnected nodes are bad. Some of the nodes e.g. at the corner of unsupported slab or at the corners of openings will always be marked as disconnected.



More information:

Analy	/sis	nodes	co	lors

Node color	Connectivity status	Selection	Example
Magenta	Disconnected	Selected	and a second
Magenta	Disconnected	Not selected	and a
Green	Connected	Selected	E.
Green	Connected	Not selected	

Figure 2-8 (<u>source</u>)

2.3.2. Checking warnings

Tekla can check an analysis model against some criteria. If there are some recognized problems with the analytical model, one shall go to <u>Display Warnings</u> in *Analysis & Design Models* dialog and press <u>Details</u> to see a full list of warnings / errors (Figure 2-9).



More information: Checking warnings about analysis model

Indel 1 Full model 72 0 6 Properties Delete Select objects Display warning Analysis element warning(s) Add selected objects Remove selected objects OK Details East 53 is not fully connected or supported, part Id: 15810 WARNING: Bar 52 is not fully connected or supported, part Id: 15825 WARNING: Bar 52 is not fully connected or supported, part Id: 15902 WARNING: Bar 59 is not fully connected or supported, part Id: 26431 WARNING: Bar 55 is not fully connected or supported, part Id: 26431 WARNING: Bar 55 is not fully connected or supported, part Id: 26431 WARNING: Bar 55 is not fully connected or supported, part Id: 26430 WARNING: Bar 55 is not fully connected or supported, part Id: 26400 Mart Id: 26400	nalysis model name	Creation me	thod	Results	5	Part coun	t Load cou	nt	Warning co	Create				
Texts structures Zs Image: Content of the model creation: Select objects Display warning Analysis element warning(s) Add selected objects Remove selected objects OK Details Eist Image: Content of the model creation of the model	lodel 1	Full model				72	0		6	INEW		Cop	y	
Analysis element warning(s) OK Details Add selected objects Remove selected objects Report WARNING: Bar 53 is not fully connected or supported, part Id: 15810 WARNING: Bar 52 is not fully connected or supported, part Id: 15825 WARNING: Bar 50 is not fully connected or supported, part Id: 15902 WARNING: Bar 59 is not fully connected or supported, part Id: 26431 WARNING: Bar 55 is not fully connected or supported, part Id: 26358 WARNING: Bar 55 is not fully connected or supported, part Id: 26400	Structures			Σ	3					Pro	pertie	·s		Delete
OK Details WARNING: Bar 53 is not fully connected or supported, part Id: 15810 WARNING: Bar 52 is not fully connected or supported, part Id: 15825 WARNING: Bar 50 is not fully connected or supported, part Id: 15902 WARNING: Bar 59 is not fully connected or supported, part Id: 26431 WARNING: Bar 56 is not fully connected or supported, part Id: 26358 WARNING: Bar 55 is not fully connected or supported, part Id: 26400 MARNING: Bar 55 is not fully connected or supported, part Id: 26400	There were warni	ings/errors in t	ne model	creatio	on:					Sele	ct obj	ects	Λ	Display warnii
Ist Report WARNING: Bar 53 is not fully connected or supported, part Id: 15810 WARNING: Bar 52 is not fully connected or supported, part Id: 15825 WARNING: Bar 50 is not fully connected or supported, part Id: 15902 WARNING: Bar 59 is not fully connected or supported, part Id: 26431 WARNING: Bar 56 is not fully connected or supported, part Id: 26358 WARNING: Bar 55 is not fully connected or supported, part Id: 26400	- Analysis elemer	nt warning(s)						-		Add sel	ected	objects	Rem	nove selected obj
Report WARNING: Bar 53 is not fully connected or supported, part Id: 15810 WARNING: Bar 52 is not fully connected or supported, part Id: 15825 WARNING: Bar 50 is not fully connected or supported, part Id: 15902 WARNING: Bar 59 is not fully connected or supported, part Id: 26431 WARNING: Bar 56 is not fully connected or supported, part Id: 26358 WARNING: Bar 55 is not fully connected or supported, part Id: 26400 MARNING: Bar 55 is not fully connected or supported, part Id: 26400	OK Details													e
WARNING: Bar 53 is not fully connected or supported, part Id: 15810 WARNING: Bar 52 is not fully connected or supported, part Id: 15825 WARNING: Bar 50 is not fully connected or supported, part Id: 15902 WARNING: Bar 59 is not fully connected or supported, part Id: 26431 WARNING: Bar 56 is not fully connected or supported, part Id: 26358 WARNING: Bar 55 is not fully connected or supported, part Id: 26400		🔯 List												
WARNING: Bar 52 is not fully connected or supported, part Id: 15825 WARNING: Bar 50 is not fully connected or supported, part Id: 15902 WARNING: Bar 59 is not fully connected or supported, part Id: 26431 WARNING: Bar 56 is not fully connected or supported, part Id: 26358 WARNING: Bar 55 is not fully connected or supported, part Id: 26400	`	Report												
WARNING: Bar 50 is not fully connected or supported, part Id: 15902 WARNING: Bar 59 is not fully connected or supported, part Id: 26431 WARNING: Bar 56 is not fully connected or supported, part Id: 26358 WARNING: Bar 55 is not fully connected or supported, part Id: 26400		WARNI	NG: Bar	53 i	s no	t fully	connected	or	supported	, part	Id:	15810		
WARNING: Bar 59 is not fully connected or supported, part Id: 26431 WARNING: Bar 56 is not fully connected or supported, part Id: 26358 WARNING: Bar 55 is not fully connected or supported, part Id: 26400						-				-				
WARNING: Bar 56 is not fully connected or supported, part Id: 26358 WARNING: Bar 55 is not fully connected or supported, part Id: 26400						-				-				
WARNING: Bar 55 is not fully connected or supported, part Id: 26400														
nalysis application interface						-								
		WARNI	NG: Bar	55 1	s no	t fully	connected	or	supported	, part	10:	26400		
	nalysis application interface													
Export Open applica														
	Export Open app	olica												

Figure 2-9

Press CTRL+5 (shortcut to show only selected objects) and select one of the warnings. The corresponding element will be highlighted in the model.

54170 ⁵⁴⁵ 422985543 646 646 646 646 646 646 646 646 646 6	5750.00 5560 08 248 526725926 57522249910 091 243	254 10001	
29. 464 74 251	S Analysis & Design Modes		
251 0 0 g	Analysis model name Creation method Model 1 Full model	Results Part count Load count 72 0	Warning ct 6 Create Copy
00 ² (43340 0 ¹⁰² 68	Tekla Structures	El creation:	Properties Delete Select objects Display warnings
106 167 29107 ²⁹³ 266	- Analysis element warning(s)		Add selected objects Remove selected objects
62233 (294 87 22 81	Z List Report	50 /	
0 0	WARNING: B	ar 53 is not fully connected or r 52 is not fully connected or	supported, part Id: 15825
30 172 49 31 49 30	WARNING: Ba WARNING: Ba	ar 50 is not fully connected or ar 59 is not fully connected or ar 56 is not fully connected or ar 55 is not fully connected or	supported, part Id: 26431 supported, part Id: 26358
0 151 7 47 60 ∞ C 8 13	Analysis application interface Export Open applica		
55 Contraction of the second s	Get result OK		

Figure 2-10

2.4. Analysis model adjustment

In many cases an automatically generated analysis model requires some smaller or bigger adjustments, in order to create the most accurate structural representation of a Tekla model.

Tekla Structures offers a few different possibilities on how to modify and adjust position of analysis members. Most of the adjustments possibilities are found in an individual analysis part properties dialog (Chapter 2.4.1) but some manual modifications are also possible (Chapter 2.4.2). Examples of both can be found in the following paragraphs.

2.4.1. Analysis part properties



More information:

<u>Analysis part properties</u> <u>Modifying the properties of an analysis part</u>

Double click on any analysis part to access its analysis properties (Figure 2-11). The settings available in the *Analysis Properties* dialog vary depending on the object type and its class.

😵 Concrete Slab Analysis Pro	operties - Model 1	x
Save Load standard	▼ Save as standard	
Analysis	- Spanning Loading Design Position - Area attributes	
Analysis member propertie	5	
Class	Slab - Shell 👻	
	Not part of built-up section	
Built-up section mode	Not part of built-up section	
📝 Design group		
Automatic update	Yes - Physical model changes are considered 💌	
ок	Apply Modify Get Cancel	

Figure 2-11

Note that properties found in other tabs than 'Analysis', 'Start / End releases' and 'Position' are not relevant in case of exporting the Tekla analysis model to FEM-Design.

2.4.1.1. Analysis member class

Analysis properties dialog -> 'Analysis' tab -> 'Class'

Here one can change a class of an analysis member. A 'Class' parameter defines how Tekla considers a certain analysis member in analysis.

Save Load standard Save as standard
Analysis Start releases End releases Composite - Loading Design Position Bar attributes -
Analysis member properties
Class Beam
☑ Built-up section mode Beam
Beam - Truss
Design group Beam - Truss-Compression only
Automatic update Beam - Truss-Tension only red 🔻
Beam - Ignore
Secondary Secondary
Secondary - Truss
Secondary - Truss-Compression only Secondary - Truss-Tension only
Secondary - Ignore
Slab - Shell
Slab - Plate
Slab - Shell
Slab - Blate
Slab - Membrane
Slab - Mat foundation
OK A Slab - Rigid diaphragm Get 🔽 / 🗖 Cancel
Slab - Ignore

Figure 2-12

One can for instance:

- Ignore a certain analysis member in analysis model (set it to 'Ignore'),
- Change an analysis member class to another one, e.g. change a hollow core member class from 'Beam' to 'Slab'. This way Tekla Structures will handle the hollow core as a slab member (Figure 2-13), and it will be exported as a plate object to a FEM-Design.

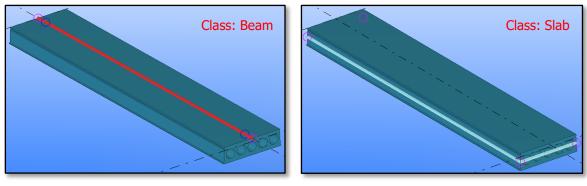


Figure 2-13

More information: Analysis class options 2.4.1.2. Analysis member axis position

Analysis properties dialog -> 'Position tab -> 'Axis

Here is where one can decide upon position of an analysis part in relation to the physical object.

	👺 Concrete Slab Analysis Prop	Concrete Slab Analysis Properties - Model 1			
0	Save Load standard	Save as standard			
	Analysis	- Spanning Loading Design Position - Area attribute	25		
	🖉 Axis 🔶	Middle plane			
	Keep axis position	Neutral axis Reference axis (eccentricity by neutral axis)			
	Connectivity	Reference axis Top left			
	Axis modifier X	Top center Top right			
	Axis modifier Y	Middle left Middle center			
	Axis modifier Z	Middle right Bottom left			
	Offset	Bottom center Bottom right			
	Longitudinal offset mode	Top plane Middle plane			
		Bottom plane			
		Left plane Right plane			
	ОК Ар	Middle plane (of left/right)	Cancel		

Figure 2-14

The drop-down list contains all available options however, some of them are used only for beam objects, or some of them only for plate objects.

Nota that Tekla Structures uses this position when constructing analysis mode if a 'Model default' is chosen from the 'Member axis location' list in the *Analysis Model Properties* dialog (Figure 2-3).



More information: Analysis axis options

Example:

The analysis model of the structure visible in Figure 2-15 (concrete floor placed on the top of a beam) is not constructed well. Both the individual objects have their analysis part placed in the neutral axis (default 'Axis' location for those objects was set to 'Neutral Axis'), so there is no connection between the two members.

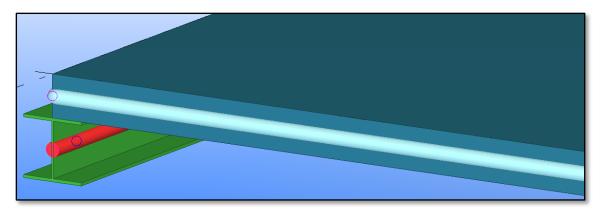


Figure 2-15

To adjust the model:

- Double click on the analysis slab to activate the *Concrete Slab Analysis Properties* and set the 'Axis' position to 'Bottom Plane',
- Double click on the analysis beam to activate the *Beam Analysis Properties* and set the 'Axis' position to 'Top center'.

		S Concrete Slab Analysis Properties - Model 1
		Save Load standard Save as standard
		Analysis Spanning Loading Design Position - Area
		V Axis Bottom plane
All and a second se	in the second	Keep axis position No
Communication	Beam Analysis Properties -	Model 1
	Save Load standard	Save as standard
	Analysis Start releases End	releases Composite - Loading Design Position Bar attributes -
	Axis	Top center 👻
	V Keep axis position	No
	Connectivity	Automatic 🔹

Figure 2-16

2.4.1.3. Analysis members connectivity

Analysis properties dialog -> 'Position tab -> 'Connectivity'

Here is where one can decide if an analysis part is allowed to snap, or connect with rigid links to other parts.

Sconcrete Slab Analysis Prop	erties - Model 1
Save Load standard	Save as standard
Analysis	- Spanning Loading Design Position - Area attributes
🔽 Axis	Middle plane 👻
Keep axis position	Yes - Keep end positions also
🔽 Connectivity 🛛 🗕	Automatic
Axis modifier X	Automatic Manual
Axis modifier Y	None • Y: 0.00

Figure 2-17

The available options are:

- 'Automatic': The analysis member snaps or connects with rigid links to other members.
- 'Manual': The analysis member does not snap or connect with rigid links to other members. Automatic connectivity to other members is created only if the member position matches the other member exactly.

Example:

a. The 'Connectivity' parameter for all the walls visible in the plan view in Figure 2-18 is set to 'Automatic' therefore, the objects are allowed to snap to each other to create a consistent analysis model.

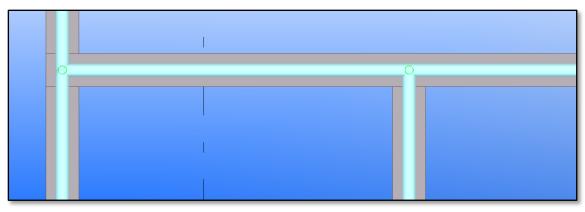
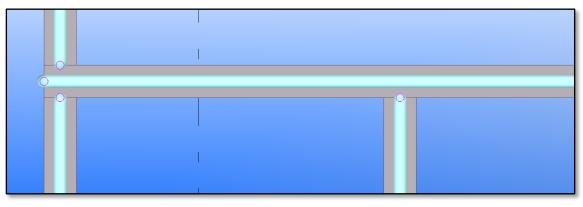


Figure 2-18

b. The 'Connectivity' parameter for all the walls visible in the plan view in Figure 2-19 is set to 'Manual' therefore, the objects are not allowed to snap to each other. The analysis model is not accurate.





The 'Manual' connectivity can be used in order to prevent an object from automatic snapping to an object that it should not be connected with.

2.4.1.4. Keep member axis position

Analysis properties dialog -> 'Position tab -> 'Keep axis position'

Here, one can decide whether the axis position of a part is kept or adjusted to the position of adjacent objects. There are five options:

- 'No': the axis is free to move when snapping end positions to nearby objects. Use this option for secondary members.
- 'Partial keep in major direction': the axis is free to move partially, but the member is not moved in the major (stronger) direction of the part profile.
- 'Partial keep in minor direction': the axis is free to move partially, but the member is not moved in the minor (weaker) direction of the part profile.

- 'Yes': the axis is not moved, but the end positions can move along the axis (thus extending or shortening the member).
- 'Yes keep end position also': The axis and the end positions of the member are not changed.

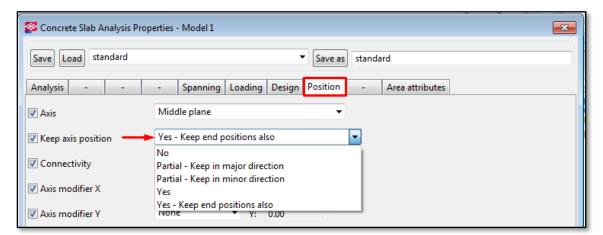


Figure 2-20

Example 1:

A concrete slab is placed on the top of concrete beam. The edge of the slab is aligned with the beam side as shown in Figure 2-20.

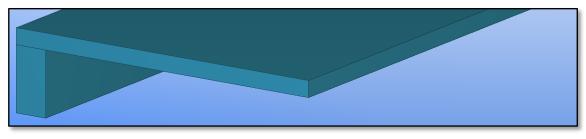


Figure 2-21

Here is how the analysis model of the example structure looks like, depending on the 'Keep axis position' settings applied to the slab:

a. 'Keep axis position': 'Yes – keep end position also'

Since the axis and the end position are restricted from changes, the analysis slab is aligned with the edges of physical slab and therefore, is not aligned with the analysis beam.

S Concrete Slab Analysis Pro	roperties - Model 1
 Save Load standard	▼ Save as standard
 Analysis	- Spanning Loading Design Position - Area attributes
🔽 Axis	Bottom plane 🔹
Keep axis position	Yes - Keep end positions also
Connectivity	Automatic •
Axis modifier X	None • X: 0.00
Axis modifier Y	None V: 0.00
Axis modifier Z Offset	None • Z: 0.00 X: 0.00 Y: 0.00 Z: 0.00
Conset Conset Conset Conset	
OK A	Apply Modify Get F/F Cancel

Figure 2-22

b. 'Keep axis position': 'Yes'

The slab axis is restricted from changes, but its ends are free to move. Therefore, Tekla Structure can adjust one of the slab edges to connect with the beam axis position.

S Concrete Slab Analysis Pro	perties - Model 1	
 Save Load standard	 Save as standard 	
 Analysis	- Spanning Loading Design Position - Area attributes	
🔽 Axis	Bottom plane 👻	
V Keep axis position	Yes 🗸	
Connectivity	Automatic 👻	
Axis modifier X	None 👻 X: 0.00	
Axis modifier Y	None Y: 0.00	
🕼 Axis modifier Z	None Z: 0.00	
☑ Offset	X: 0.00 Y: 0.00 Z: 0.00	
Congitudinal offset mode	Offsets are not considered 🔹	
ОК А	pply Modify Get Cancel	

Figure 2-23

Example 2:

Two beams are connected with each other as show in the figure below. Here is how the analysis model looks like, depending on the 'Keep axis position' settings applied to both the beams.

a. 'Keep axis position': 'Yes - keep end position also'

Since the axis and the end position are restricted from changes, the analysis beams are extended to the edges of physical beams and therefore, do not snap to each other.

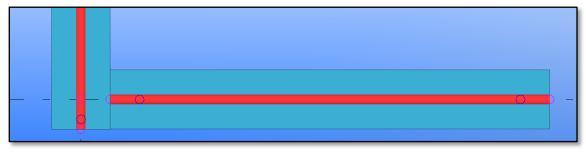


Figure 2-24

b. 'Keep axis position': 'Yes'

Beams axes are restricted from changes, but their ends are free to move. Therefore, Tekla Structure can adjust both the beams ends, so they can snap to each other.

	Properties - Model 1					
S Concrete Beam Analysis Properties - Model 1						
Save Load standard	Save as standard					
Analysis Start releases End	nd releases Composite - Loading Design Position Bar attributes -					
☑ Axis Reference axis ▼						
Keep axis position	Yes v					
Connectivity	Automatic 👻					
Axis modifier X	None 🔻 X: 0.00					
Axis modifier Y	None Y: 0.00					
Axis modifier Z	None Z: 0.00					
☑ Offset	X: 0.00 Y: 0.00 Z: 0.00					
Longitudinal offset mode	Offsets are not considered 🔹					

Figure 2-25

2.4.2. Manual adjustment

2.4.2.1. Move special

Analysis nodes and analysis parts can be moved, just as other Tekla objects. A particularly useful command to do so is 'Move Special' -> 'Linear'.

Note that any change applied to the analysis part does not influence the physical object.



Example 1: Moving an analysis part

A concrete slab is placed on the top of concrete beam. The analysis slab and analysis beam are not placed in the same plane.

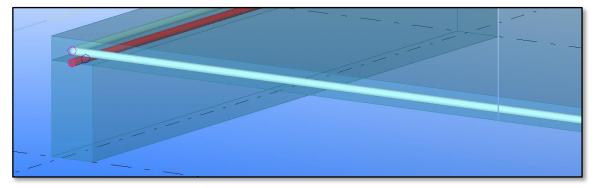


Figure 2-26

To adjust the model:

- Right click on the analysis slab, go to 'Move Special' -> 'Linear'

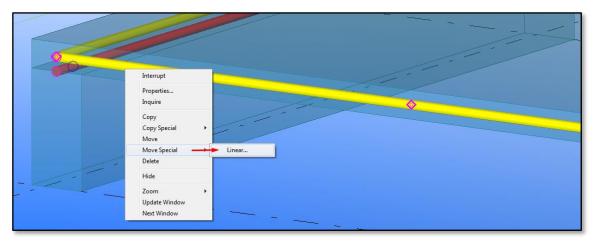


Figure 2-27

- If you know the distance between the elements, type it in the 'dZ' box (to move an element downwards, type the number with '-'). Otherwise, pick two nodes from the model to indicate the movement direction and calculate the distance, as shown in Figure 2-28.

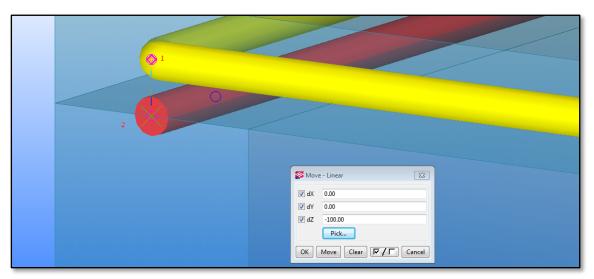


Figure 2-28

- Press <u>Move</u>. The analysis slab will be moved 100 mm in the Z-direction.

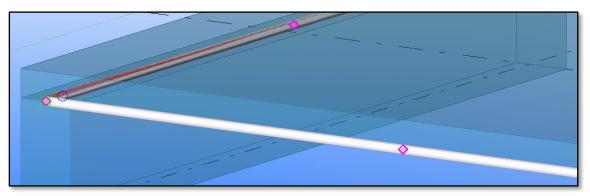
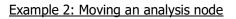


Figure 2-29



Two analysis beams shown in Figure 2-30 are not connected with each other.

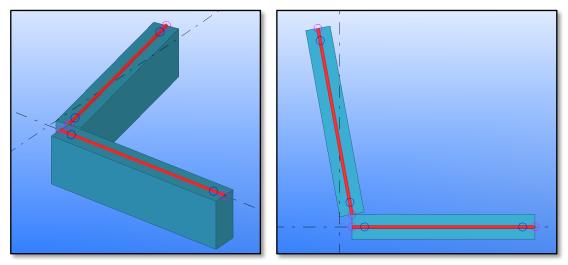


Figure 2-30

To adjust the model:

- Right click on the node that you wish to move, go to 'Move Special' -> 'Linear'

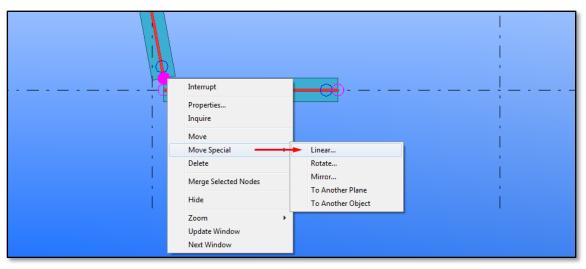


Figure 2-31

- Chose Pick, and select the two nodes to indicate the movement direction, and press Move.

Move - Linear V dX 50.00 V - 200.00 V dZ 0.00 Pick OK Move Clear V Cancel	

Figure 2-32

2.4.2.2. Merging analysis nodes

It is possible to merge two analytical nodes into one single node.



More information: Merging analysis nodes

Example:

- Select the two columns ends.
- Go to Analysis -> Merge Selected Nodes.
- Select the target node in the upper column.

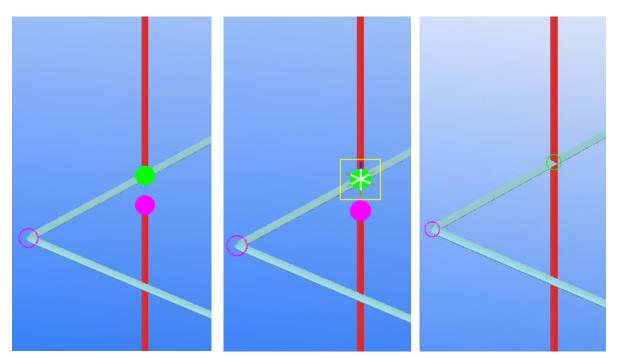


Figure 2-33

3. Exporting Tekla model to FEM-Design

This chapter contains a step by step guide on how to export a Tekla Structures model to StruSoft FEM-Design. A sample model is used to demonstrate the process.

3.1. Create an analysis model

a. Open a Tekla Structures model. The example model is a concrete structure composed of panels, slabs, beams and columns.

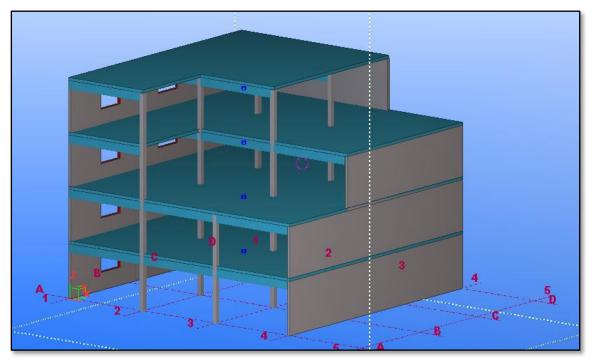


Figure 3-1

- b. Go to Analysis -> Analysis and Design Models ...
- c. In *Analysis & Design Models* dialog press <u>New</u> to create an analysis model.

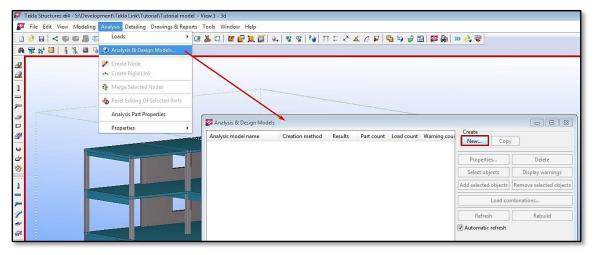


Figure 3-2

d. Define analysis model properties.

In this example, a full analysis model is defined with no rigid links allowed, and the axis location is set to model default, as shown in Figure 3-3.

S Analysis Model Properties		
Save Load	▼ Save as	
Analysis model Analysis Job Output S	eismic Seismic masses Modal analysis Desi	ign - Steel Design - Concrete Design - Timber
Analysis model name:	Model 1	Browse for export folder
Creation method:	Full model	•
Filter	None	•
Secondary member filter	Auto-detect secondary members	•
Analysis application		 Set as the default
Less settings		
Use rigid links	Disabled, with keep axis: Default	•
Default keep axis for secondary members	No	•
Analysis model rules	Analysis model rules	
Curved beams	Split into straight segments	•
Consider twin profiles	Disabled	•
Member axis location	Model default	•
Member end release method by connection:	No	•
Automatic update	•	
Model merging with analysis application	•	
ОК		Cancel Help

Figure 3-3

e. Press <u>OK</u>. An analysis model will be constructed and listed in the *Analysis & Design Models* dialog as shown in Figure 3-4.

510 114 1150	91243	2	54	10001		
71 251	Analysis & Design Models				Create	
106136136140142218	Analysis model name Model 1	Creation method Re Full model	sults Part count 72	Load count Warning co	New Copy	
	Model 1	Full model	12	U O		
245 248 302 68					Properties	Delete
105 167					Select objects	Display warnings
29207293					Add selected objects	emove selected objects
15 7156159 160162233 1 201					Load comb	inations
107271022330201					Refresh	Rebuild
287 298 22 81					Automatic refresh	
80 172						
42144 046 048 60 151 7 47 50						
8	Analysis application interface			4		
43 46 13	Export Open ap		tion			
B	Get re	sults Get results for s	elected			Close
82 1	89 11					

Figure 3-4

- 3.2. Check and adjust an analysis model.
- a. In the first (optional) step turn off the visibility of members and nodes numbers.

Go to Tools -> Options -> Advanced Options, and set the following parameters to FALSE:

- XS_AD_MEMBER_NUMBER_VISUALIZATION
- XS_AD_NODE_NUMBER_VISUALIZATION
- XS_AD_NODE_NUMBER_BY_Z
- b. In *Analysis and Design Models* dialog press <u>Display warnings</u> and <u>Details</u>, to see the list of warnings.

The report says that there are several bar members in the example Tekla model that are not fully supported.

Analysis & Design Models								
Analysis model name	Creation	method	Results	Part cou	int Load cou	nt Warning co	Create New Copy	
Model 1	Full mod	lel		72	0	6	Сору	
Structures			X				Properties	Delete
There were warning	ngs/errors	in the model	creation:				Select objects	Display warnings
- Analysis elemen	t warning((s)					Add selected objects	Remove selected objects
OK Details								1
	- 🔯 Li							
	Rej	port						
							d, part Id: 15810	
							d, part Id: 15825	
					-		d, part Id: 15902	
							d, part Id: 26431	
							d, part Id: 26358	
	WA	RNING: Bar	55 18	not full	y connected	or supporte	d, part Id: 26400	

Figure 3-5

c. Localize the members listed in the warnings list.

Press CTRL+5 (shortcut to show only selected objects) and select one (or all) of the warnings. The corresponding element(s) will be highlighted in the model.

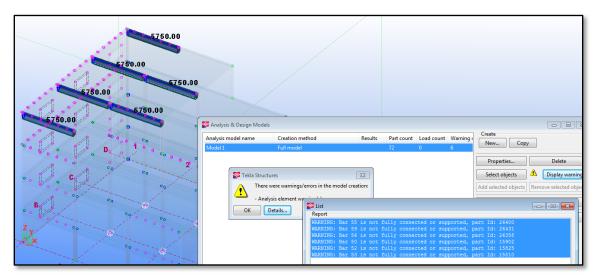


Figure 3-6

d. Zoom into each of the members to check the exact problem.

In this example the problem is that analysis axis of each of the listed beams do not connect with the adjacent objects (walls, columns), as shown in Figure 3-7.

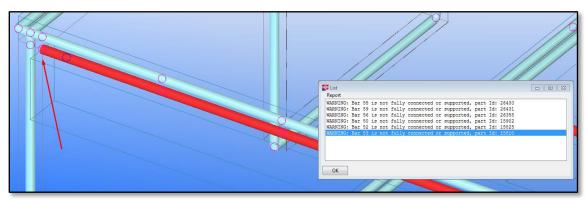
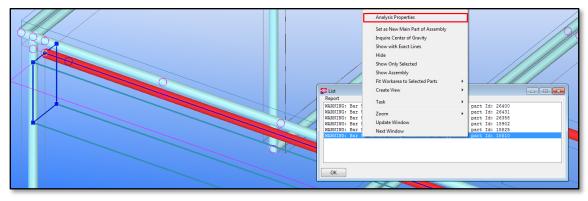


Figure 3-7

e. Check the analysis properties of the listed members.

To do so, double click on one of the analysis beams in the model, or right click on the warning message and choose *Analysis Properties*.





f. Change 'Keep Axis position' parameter from 'Yes – Keep end position also' to 'Yes'. Press Modify.

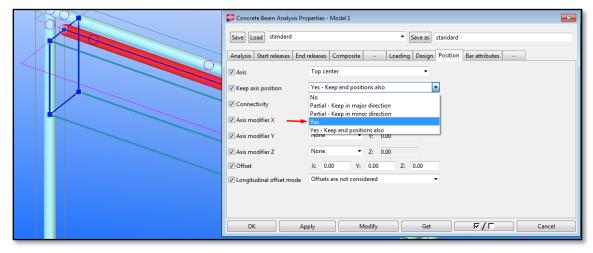


Figure 3-9

- g. Select next unsupported element (in the model or in the warning list) and press <u>Modify</u> to apply the new settings. Apply the properties to all listed beams.
- h. There are no more unsupported elements in the model, but not all the model inconsistencies are listed in the <u>Warnings</u> list. Therefore, check the model visually and verify the nodes connectivity.

Visual checking the example model shows that there are some disconnected nodes (in magenta color) on the top two floors. The nodes are located along the floor perimeter as shown in Figure 3-10.

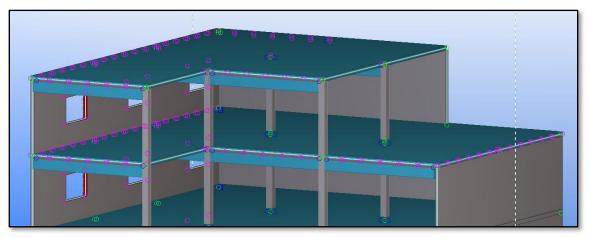


Figure 3-10

i. Zoom into the disconnected nodes area. It looks like the analysis slabs are not connected with beams underneath, and that they are not well connected to the walls, as shown in Figure 3-11.

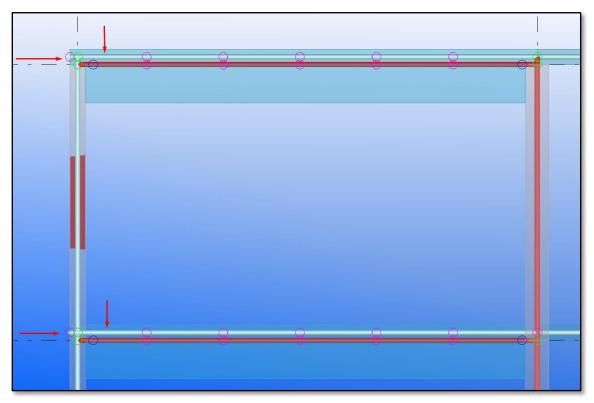


Figure 3-11

j. Double click on one of the analysis slabs to open the analysis properties dialog.

Figure 3-12 shows how the default properties for this slab look like:

- 'Axis' is set to 'Middle Plane'. That is why the analysis slab is not aligned with the analysis beam underneath.
- 'Keep axis position' is set to 'Yes Keep End position also'. That is why the slab edge is not aligned with the wall in the horizontal direction.

	•	0
Concrete Slab Analysis Pro		E
Save Load standard	Spanning Loar ing Design Position - Area attributes	
Axis Keep axis position	Middle plane Yes - Keep end positions also	
Connectivity Axis modifier X	Automatic	
 ✓ Axis modifier Y ✓ Axis modifier Z 	None ▼ 0.00 None Z: 0.00	

Figure 3-12

- k. Change the slab properties to the following, and press Modify.
- 'Axis' -> 'Bottom Plane'

The analysis slab position is now moved to the bottom of the physical slab and therefore, is aligned with the analysis beam.

- 'Keep axis position' -> 'Yes'

The edges of the slab are now adjusted to the position of the walls, and the wall height is adjusted to the new (vertical) position of the slab.

Save Load standard		e as standard	×
Analysis	- Spanning Loading Design Position	on - Area attributes	
🕼 Axis	Bottom plane	•	
V Keep axis position	Yes	•	

Figure 3-13

I. Select the other analysis slab and click <u>Modify</u>, to apply the same properties to it. The model is now fully connected.

3.3. Tekla StruXML Export

Keep your Tekla model open. Go to Start Manu, then to StruSoft Folder. Find Tekla StruXML folder and run Tekla StruXML Export.

a. In the first step, select the design code so the correct libraries can be loaded in the mapping dialogs.

🔊 Tekla StruXML Export				
Export Materials Section File name	Select code Code independent Eurocode Eurocode (NA: British) Eurocode (NA: Danish) Eurocode (NA: Finnish) Eurocode (NA: German) Eurocode (NA: German) Eurocode (NA: Norwegian) Eurocode (NA: Norwegian) Eurocode (NA: Swedish) Eurocode (NA: Polish)	Code	Eurocode	Browse

Figure 3-14

b. The left upper window shows all analysis models that are created in the current Tekla Structures model. Select the model you wish to export. Now the program will collect information about all the materials and sections used in that model.

🔊 Tekla StruX	ML Export	- • •
Export Mat	erials Sections Code Eurocode	•
File name	S:\Development\Tekla Link\Tutorial\Tutorial model - 23-07-2015\Analysis	s\M Browse
Model 1	Save grids	
Connecte	d to Tekla Structures 21	
	Collecting current materials and sections	
Model 1:	Materials and sections collected	

Figure 3-15

c. Decide whether to export grid system or not. Check / uncheck the Save grids box in the upper right window, as shown in Figure 3-15.

d. Go to *Materials* tab. Here one shall map all the materials used in the Tekla Structures model to corresponding materials from FEM-Design library.

Note that if you have a custom material library in FEM-Design, you shall update the material library in the Tekla StruXML Export tool to the current FEM-Design material library before you start the mapping process.

To do so go to FEM-Design, start Beam/Column command -> Default Settings ->Materials tab, -> Export and save the library as a .struxml file type, as shown in Figure 3-16. Use the <u>Load</u> <u>material library</u> to import the new library into Tekla StruXML Export tool.



Figure 3-16

e. Map the materials

There are two concrete materials used in the example model and both have to be mapped (as they are marked in red), as shown in Figure 3-17.

🔊 Tekla StruXML Export	
Export Materials Sections	Code Eurocode
Show current materials Apply	Load material library Reset
C25/30 : C30/37 :	 Concrete Steel Timber General
Connected to Tekla Structures 21	

Figure 3-17

Select a material in the left dialog, then select a corresponding material in the right dialog (FEM-Design library) and press the < button. Mapped material will turn black. Press <u>Apply</u>.

🔊 Tekla StruXML Export	
Export Materials Sections	Code Eurocode
Show current materials Apply	Load material library Reset
	 ▲ Concrete C12/15 C16/20 C20/25 C25/30 C30/37 C35/45 C40/50 C45/55 C50/60
Connected to Tekla Structures 21	

Figure 3-18

f. Go to *Sections* tab and map all the sections used in the model to corresponding sections from FEM-Design library.

Note that if you have a custom sections library in FEM-Design, you shall update the sections library in the Tekla StruXML Export tool to the current FEM-Design sections library before you start the mapping process.

g. Map the sections.

This time not all the sections have to be mapped. One of the concrete sections (500x300, as shown in Figure 3-19) has been already mapped before in another project and the program remembers the mapping choice.

🔊 Tekla StruXML Export	
Export Materials Sections	Code Eurocode
Show current sections Apply	Load section library Reset
300*300 : 500*300 : Rectangle300x500	 Concrete sections Steel sections Timber sections Hollow Core
Connected to Tekla Structures 21	

Figure 3-19

Map the remaining section. The procedure is the same as in case of materials mapping. Press <u>Apply</u> when you are done.

nt Frekla StruXML Export		
Export Materials Sections	Code Eurocode	•
Show current sections Apply	Load section library	Reset
300*300 : Square300 500*300 : Rectangle300x500	 ▲ Concrete sections ▶ Rectangle ▲ Square 150 200 250 300 350 400 450 	* <u>II</u>
Connected to Tekla Structures 21		

Figure 3-20

h. Go back to *Export* tab.

Press Browse to find a new name and location for the export file (if necessary) and press Export.

i. Check the export status and if there were any problems with exporting the model.

Exporting the example model was successful. The report says that 114 out of 114 objects in the analysis model (including grids system) have been exported to the struxml file.

🔊 Tekla StruXML Export	
Export Materials Sections	Code Eurocode 🔻
File name C:\Users\Iwona\Desktop\Tekla Export Tutori	al\Model 1.struxml Browse
Model 1	✓ Save grids
Processing grid plane (Id: 135) Processing grid plane (Id: 121) Processing grid plane (Id: 107) Processing grid plane (Id: 93) Processing grid plane (Id: 79) Processing grid plane (Id: 52) Processing objects is finished without problems. Numbe Saving StruXML file C:\Users\Iwona\Desktop\Tekla Expo StruXML export is completed	
About	Cancel Close

Figure 3-21

- 3.4. Open struxml file in FEM-Design.
- a. Start FEM-Design 3D Structure.
- b. Go to *File* \rightarrow *Open* and browse for the struxml file type, as shown in Figure 3-22.

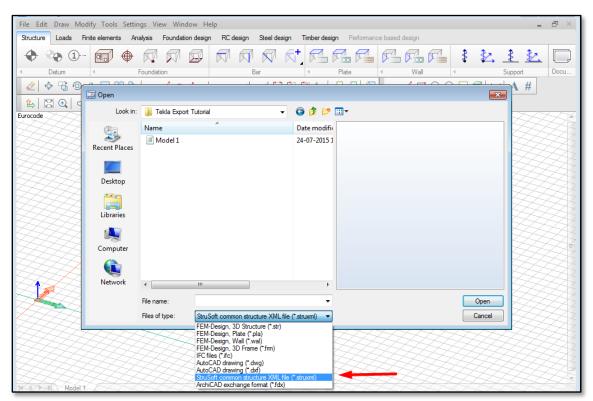


Figure 3-22

The exported Tekla analysis model is now imported to FEM-Design.

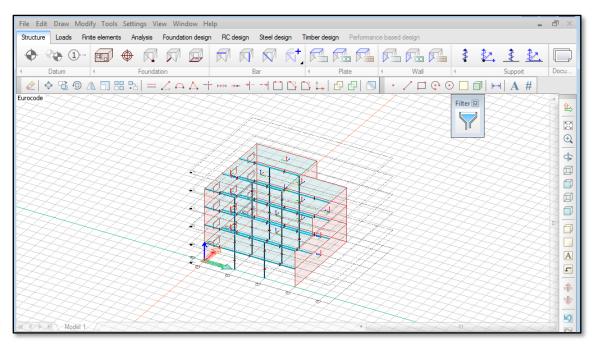


Figure 3-23

3.5. Errors at exporting an analysis model.

There are a few possible situations when the export of a Tekla Structures analysis model may not be successful. All the recognized issues are described in the following paragraphs.

3.5.1. Lack of material definition

For the successful export it is required that each of the analysis part has a valid material assigned to its physical representation. It is because it is not possible to have an element in FEM-Design that does not have a structural material.

Figure 3-24 shows an example, where a concrete column in Tekla model does not have any material assigned to it. Such element will not be exported and following error message will be sent:

"Cannot find matching material defined, the object will not be saved"

Save Load standard Kave as standard Attributes Position Cast unit Deforming	2	Tekla StruXML Export Code Eurocode File name C:\TeklaStructuresModels\New model 44\Analysis\Column - No material\Col Browse Column - No material Save nride
Image: Name COLUMN Image: Profile 400°400 Image: Select Select Image: Material Select Image: Select Select		Connected to Tekla Structures 21 Column - No material: Collecting current materials and sections Column - No material: Materials and sections collected Column - No material: StruXML export is started

Figure 3-24

The same error message will be shown if the material assigned to the element in not present in the Tekla material catalog.

3.5.2. Lack of material / section mapping

If one forgets to perform material or section mapping, following error messages will be thrown and the objects that were not mapped will not be exported.

"Cannot find matching material (section) defined, the object will not be saved"

🔊 Tekla StruXML Export	– – ×
Export Materials Sections	Code Eurocode
Show current materials ~ Apply	Load material library Reset
C25/30 : S235JR : S 235	 ▷ Concrete ▷ Steel ▷ Timber ▷ General
 Processing analysis bar (Physical object GUID: 9EF5CE4A-90F2-44 Processing analysis bar (Physical object GUID: 70DB166C-8224-44 Cannot find matching material defined, the object will not be sav Processing analysis bar (Physical object GUID: 50A2F971-E359-49 Cannot find matching material defined, the object will not be sav 	2D4-993F-B189A31F4BAB) ed (Tekla material: 'C25/30') 19-8DCA-66A486D12843)

Figure 3-25

3.5.3. Invalid mapping

If one accidentally maps a material to a wrong type of section, or the opposite, a following error messages will be thrown:

"Invalid mapped material and section combination. The bar will not be saved ()"

Export N	aterials	Sections				Code	Eurocode	Ý
Show curr	ent secti	ons	v	Apply		Load sectio	on library	Reset
400*400 800*400 HEA300 HEA400	: HE-A1 HE-A3	100 300			<	 Concrete sections Steel sections Timber sections Hollow Core 		
Proces Proces Invalid	sing ana sing ana mappeo	ilysis bar (P ilysis bar (P d material a	Physical object GUID Physical object GUID and section combine): 50A2F971-E): 952BAD47-A ation! The bar	359-4 ABC4- r will n	2D4-993F-B189A31F4BA8 919-8DCA-66A486D12843 4471-87D2-6DD5E4839C6 10t be saved! ('C25/30' + 'I 4727-A581-2846435455C	3) 58) HE-A100') -	-



3.5.4. Bad surface geometry

In FEM-Design it is required that surface elements (plates and walls) are defined in one plane only, with a certain accuracy. In Tekla Structure however, it is possible that an analysis slab or analysis wall has nodes that are out of plane. In this case, one of the following error messages will be thrown and such element will not be exported:

- "The analysis area has out of plane point(s)! The analysis area will not be saved!"
- "The wall vertically does not meet FEM-Design requirements! The analysis area will not be saved!"
- "Improper region! The analysis area will not be saved!"

In each case, one can search for the bad object using the method given in Chapter 3.5.5, and try to fix the analysis part of the object (using methods given in Chapter 2.4.

3.5.5. How to find a certain element in Tekla?

When exporting a model to struxml, Tekla StruXML Export shows a GUID of a parent object (physical member) for each processed object. The message in the log dialog looks like this:

lodel 1	Save grids
Connected to Tekla Structures 2017	
Model 1: Collecting current materials and section Model 1: Materials and sections collected Model 1: StruXML export is started	5
Processing analysis bar (Physical object GUID: 498 Processing analysis bar (Physical object GUID: 33	

Figure 3-27

This is the same GUID that one can see in Tekla, when using *Inquire objects*:

	B1F8-05AEBE805CC Typ	e: 2	Assembly phase:	1 Part phase: 1	
Name Profile	Material	Grids	Part position	-	
COLUMN HEA400	S235JR	2/B	c/0(?)	C/0(?)	
Total 4 Parts: 3.45 T, 2					
art	GUID: ID49BEDAA7		B1F8-05AEBE805C		
art 	GUID: ID49BEDAA7				
Part Slobal coordinates: Start point	GUID: ID49BEDAA7 : X= 6000.0) mm		Z= 0.0 mm	
Part Nobal coordinates: Start point End point	GUID: ID49BEDAA7 : X= 6000.0 : X= 6000.0) mm) mm	Y= 6000.0 mm	Z= 0.0 mm Z= 7200.0 mm	
Part Slobal coordinates: Start point End point Center of gravity	GUID: ID49BEDAA7 : X= 6000.0 : X= 6000.0) mm) mm	Y= 6000.0 mm Y= 6000.0 mm	Z= 0.0 mm Z= 7200.0 mm	
Part Slobal coordinates: Start point	GUID: ID49BEDAA : X= 6000.0 : X= 6000.0 : X= 6000.0) mm) mm	Y= 6000.0 mm Y= 6000.0 mm	Z= 0.0 mm Z= 7200.0 mm	

Figure 3-28

If there is a problem with a certain object, an error message will be thrown just under the processed object log (as shown in figures in previous paragraphs).

In order to find and correct the object (or group of objects) that failed the export, one can use the *Selection Filter* in Tekla Structures.

Example:

There are a few objects that have not been exported due to lack of valid material definition. One can see the error messages (**always displayed under the processed object**) in Figure 3-29.

🔊 Tekl	la StruXML E	xport								-		×	
Export	Materials	Sections				Code .		Eurocod	le			Ŷ	
File nam	ne C:\Te	eklaStructu	iresModels\	New mo	odel 23V	Analysis∖N	/lode	l 1\Mod	el 1.st	ruxml	Bro	owse]
	cessing anal	-										^	•
Pro	cessing anal	lysis bar (P	hysical obje	ct GUID	: 70DB1	66C-8224	-42D	4-993F-	31894				
😑 Car	nnot find ma	tching ma	terial define	d, the o	bject wi	ll not be s	aved	(Tekla n	1	Сор	y Guid f	for FEM-	-Design
Pro	cessing anal	lysis bar (P	hysical obje	ct GUID	: 50A2F9	971-E359-	4919	-8DCA-	5	Cop	y Guid f	for Tekla	
😑 Car	not find ma	tching ma	terial define	d, the o	bject wi	ll not be s	aved	(Tekla n	aterra			_	
Pro	cessing anal	lysis bar (P	hysical obje	ct GUID	: 952BA	D47-ABC4	-447	1-87D2-	6DD5	iE4839	C68)		
🖨 Car	not find ma	tching ma	terial define	d, the o	bject wi	ll not be s	aved	(Tekla n	nateria	al: 'C25	5/30')		
Pro	cessing anal	lysis area (Physical obj	ect GUII	D: 8DFA	A13E-4084	4-472	27-A581	2846	43545	5C5)		
Car	not find ma	tching ma	terial define	d. the o	biect wi	ll not be s	aved	(Tekla n	nateria	al: 'C25	5/30')		
-	cessing anal	-											
-	not find ma	-											
-	cessing grid			,	aject m			(read a ri	rare tell fi		,,		
	cessing grid	1 C C	-										
FIU	· · · ·		4030/									~	,
About N	fanual								Ca	ancel	(Close	

Figure 3-29

To find a specific object in Tekla model:

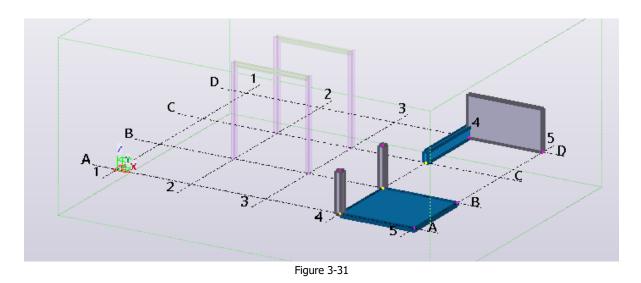
- a. In Tekla StruXML Tool right click on the log entry that refers to the object you want to search for here, it is always displayed above the error message, and press Copy GUID for Tekla.
- b. In Tekla Structures start the *Selection Filter* (Ctrl+G).
- c. Create filter definition according to the following rule (shown in Figure 3-30):
- Category: Object
- Property: GUID
- Condition: Equals
- Value: here paste the copied GUID (e.g. use Ctrl+C)
- d. You can add several rows into the filter, by repeating steps a-c.
- e. Save the filter and click <u>Apply</u> and <u>OK</u>.

As you can see in Figure 3-30, when choosing *Copy Guid for Tekla*, the GUID is copied with the ID prefix (thus the division; FEM-Design requires GUID without the prefix).

tandaı	rd			✓ Save standard	Si	ave as	>)
jects v	with matching	properties car	be selected				
(Category	Property	Condition	Value)	And/	Add rov
2 -	Object	Guid	Equals	ID70DB166C-8224-42D4-993F-B189A31F4BAB	-	Or	Delete ro
-	Object	Guid	Equals	ID50A2F971-E359-4919-8DCA-66A486D12843	-	Or	Deretern
-	Object	Guid	Equals	ID952BAD47-ABC4-4471-87D2-6DD5E4839C68	-	Or	
-	Object	Guid	Equals	ID8DFAA13E-4084-4727-A581-2846435455C5	-	Or	Moveu
-	Object	Guid	Equals	ID0E69ABFE-8FA5-4D8A-BE6E-86259B142982	-	Or	Move do
							Wove do
							New filt

Figure 3-30

f. Use CTRL+5 and select the whole model. Only the elements of the given GUIDS will be selected.



Also, the Tekla GUID is now assigned to the exported object in struxml file (in previous version of the Export Tool, the struxml GUID was assigned randomly and was not compatible with Tekla).

If you want to search for a specific object in the struxml file, I recommend using the *Copy Guid for FEM-Design* option (it copies the GUID without the ID prefix).

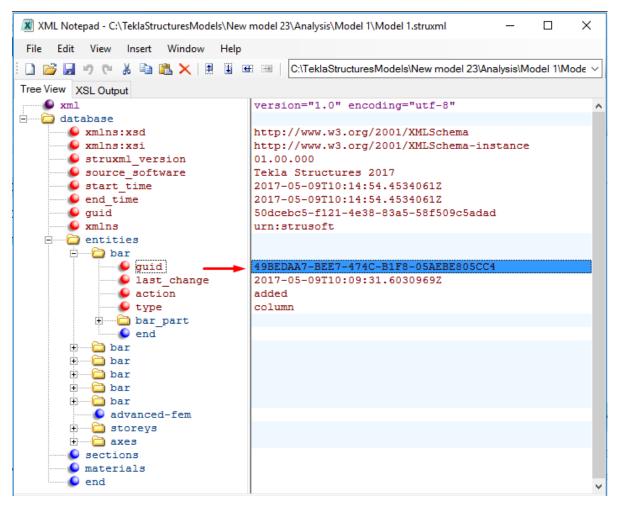


Figure 3-32

3.5.6. How to find a certain element in FEM-Design?

To find a specific object in FEM-Design model that was created from a struxml file exported from Tekla:

- g. In Tekla StruXML Tool right click on the log entry that refers to the object you want to search for, and press Copy GUID for FEM-Design (Figure 3-29).
- h. In FEM-Design, start Find tool (Tools -- > Find).
- i. In Find tool:
- select Guid,
- paste the copied GUID,
- press Enter.

The object will be selected in red, as shown in Figure 3-33.

