

# CIS/2 Translator

**Version 1.0 - General Reference**

Last Revised 9/11/2014



26632 Towne Centre Drive, Suite 210

Foothill Ranch, California 92610

(949) 951-5815

(949) 951-5848 (FAX)

[www.risa.com](http://www.risa.com)

Copyright 2014 by RISA Technologies, LLC All rights reserved. No portion of the contents of this publication may be reproduced or transmitted in any means without the express written permission of RISA Technologies, LLC.

We have done our best to insure that the material found in this publication is both useful and accurate. However, please be aware that errors may exist in this publication, and that RISA Technologies, LLC makes no guarantees concerning accuracy of the information found here or in the use to which it may be put.

---

# Table of Contents

---

<b>Introduction</b> .....	<b>1</b>
Export mode .....	1
Import mode .....	1
<b>Cardinal Points</b> .....	<b>5</b>
Overview .....	5
Detailing Input and Modification: .....	6
Visualization of the Detailing Layer .....	9
File I/O .....	10
<b>What is Supported with this Translator?</b> .....	<b>12</b>
Elements .....	12
Loading conditions .....	12
Sections Sets .....	12
Physical Members and Member mapping .....	12
<b>Mapping Files</b> .....	<b>13</b>
<b>Overwrite and Update Section</b> .....	<b>14</b>
<b>Messages</b> .....	<b>15</b>
<b>Supported CIS/2 Entity List (Export)</b> .....	<b>17</b>
<b>Supported CIS/2 Entity List (Import)</b> .....	<b>20</b>
<b>Tekla Structures and RISA-3D Mapping</b> .....	<b>23</b>
How to Export from RISA-3D to Tekla Structures .....	23
How to Export from Tekla Structures to RISA-3D .....	26
Updating a Tekla Structures Model .....	27
Setting up Mapping Files .....	28
<b>Technical Support</b> .....	<b>29</b>



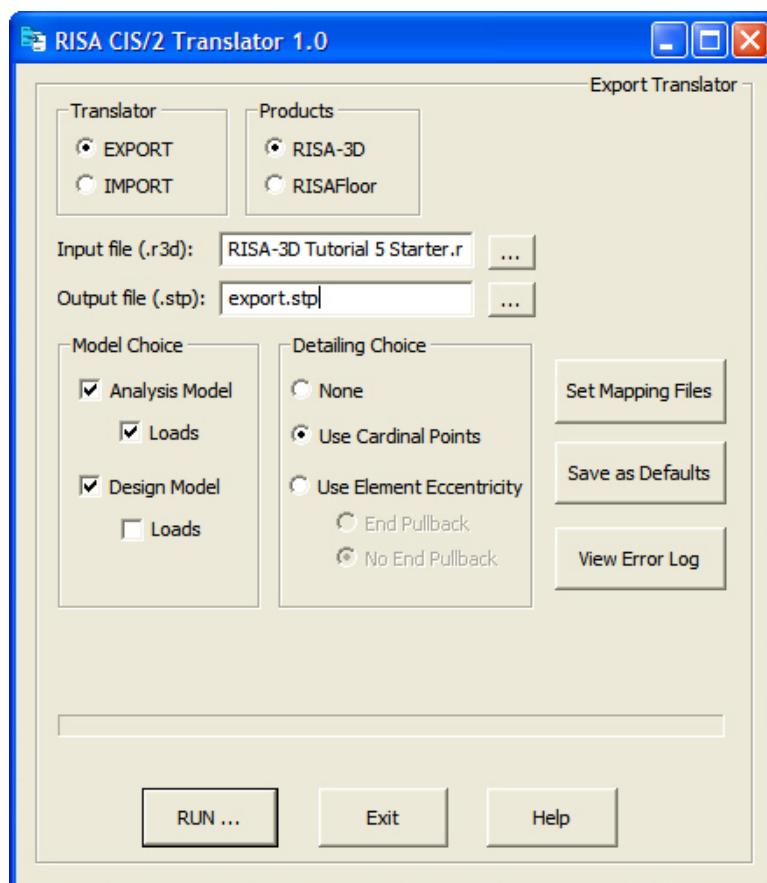
## Introduction

The RISA CIS/2 translator is a tool for transferring data between the RISA-3D program and the CIM steel part 21 file format. There are two basic modes for the CIS/2 translator: Export mode and Import mode.

When the user picks “EXPORT” option in the “Translator” group box, the main dialog box will be displayed as export mode. When the user picks the “IMPORT” option in the “Translator” group box, the main dialog box will be displayed as import mode.

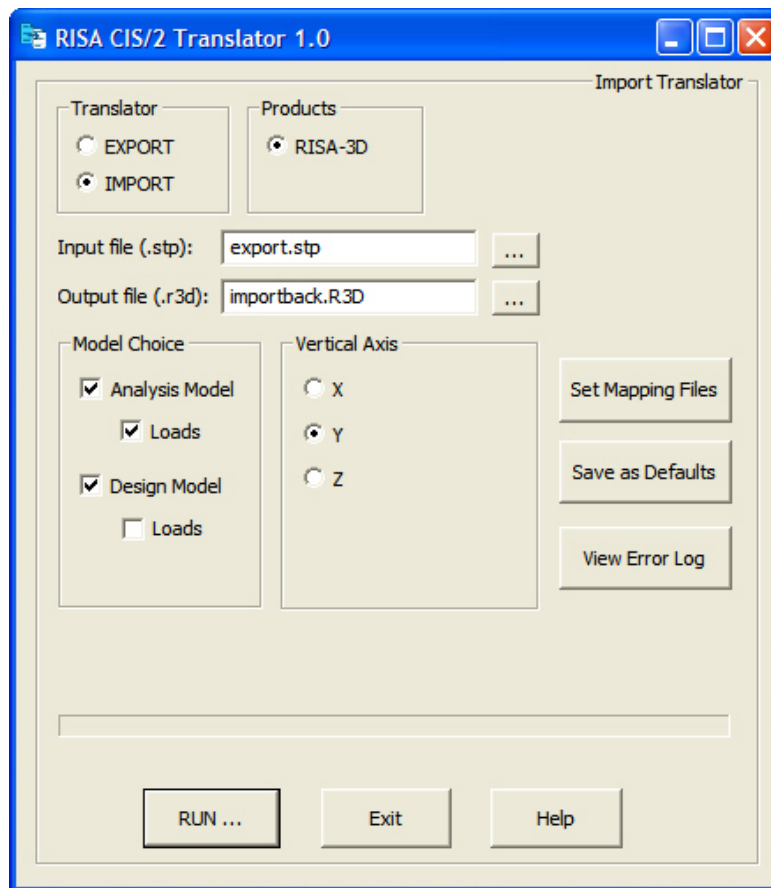
### Export mode

Files will be translated from RISA input file format to CIS/2 part 21 file. Export is available for both RISA-3D input files and RISAFloor input files.



### Import mode


Files will be translated from CIS/2 part 21 files to RISA-3D input format. (Note: RISAFloor format is not currently available for import).




### ***Products Group Box (available both Export and Import mode)***

The product group box defines the format of the input file. Eventually the user will be able to choose between a RISA-3D format or a RISAFloor format as the input file. However, RISAFloor import is not supported for the current version of the CIS/2 link. Therefore, only the RISA-3D radio button will appear in the “products” group box in import mode.

### ***Input File (available both Export and Import mode)***

An input file can be selected by clicking the  button. You can also type the directory and file name in the input file edit box. An input file is required for translation.

### ***Output File (available both Export and Import mode)***

To overwrite/update an existing output file, select the file using the  button or by typing the directory and file name in the input file edit box. To generate a new output file, you can type the directory and file name in the edit box, type in the file name only, or leave the edit box empty. If the edit box is blank, the translator will automatically generate a file name and place it in the same directory as the input file. If the edit box has a file name but no directory in it, the translator will place the output file in the same directory as the input file.

### ***Model Choice Group box (available in both Export and Import mode)***

There are three types of models in CIS/2: the analysis model, the design model and the manufacture model. RISA is capable of handling the analysis model and the design model. The manufacture model is not in the scope of RISA software and will not be processed by our translator. A typical analysis model contains elements, nodes, loading conditions, boundary conditions and end releases. A typical design model, on the other hand, contains the

information of member shapes, design parts, etc. (Please refer to the entity list section for details). By clicking on the checkbox “design model” and/or “analysis model”, you choose whether to export a design model, an analysis model, or both. For a complete export for a RISA model, it is recommended to have both the “design model” and “analysis model” selected. Both the CIS/2 design model and the analysis model contain loads. In most cases, exporting loads in either design or analysis model should be enough. If you choose to export loads in both design and analysis model when doing the round-trip import from CIS/2 model back to the RISA model, the load will appear doubled. Therefore, it is recommended to write the loads in the analysis model only.

By default, both analysis and design model will be exported, and load will only be exported to the analysis model.

### ***Detailing Choice Group box (available in Export mode only)***

The Detailing Choice group box controls the export of member detailing information. There are three options here: None, Use Cardinal Points and Use Element Eccentricity

#### **None**

When “none” is selected, no detailing information will be translated in the exported CIS/2 model.

#### **Use Cardinal Points**

If the “Use Cardinal Point” option is picked and if the defined connection location falls on a cardinal point, the connection point will be translated as the “cardinal\_point” value in the “section\_profile” entity for that particular member. For the time being, only cardinal point 1 thru 10 is supported in the current version. The “Use Cardinal Point” option is the more popular way for describing the detailing information.

#### **Note:**

- The connection point needs to fall exactly on a cardinal point location.
- A cardinal point is stored as a section property for each member. Thus, if you have a brace that is connected at a different cardinal point at either end, one offset will be lost. It is recommended to "Use Element Eccentricity" in these cases.

#### **Use Element Eccentricity**

If the “Use Element Eccentricity” option is chosen the connection point information is translated to the “element\_eccentricity” entity in the CIS/2 model and can later on be further processed by a detailing program like SDS/2. There are two options for “Use Element Eccentricity” selection, “End Pullback” and “No End Pullback”. When the “End Pullback” option is selected, the complete detailing offset will be exported. When the “No End Pullback” option is selected, the detailing offset local y and local z direction (perpendicular to the members) will be exported, the local x offset in the member direction will be 0. (SDS/2 will prefer this option for better handling of data)

The exported detailing information has been tested with SDS/2. By default, “No End Pullback” is selected.

### ***Vertical Axis Group box (available in Import mode only)***

In export mode the RISA CIS/2 translator will always export a model with Z as the vertical axis. In import mode, the RISA CIS/2 translator will always assume that the .stp file has the global Z as the vertical axis.

RISA-3D and RISAFloor accept the X, Y, or Z axis as the vertical axis, thus when the user is importing a model from the CIS/2 model to RISA format, it is required for the user to define the vertical axis.

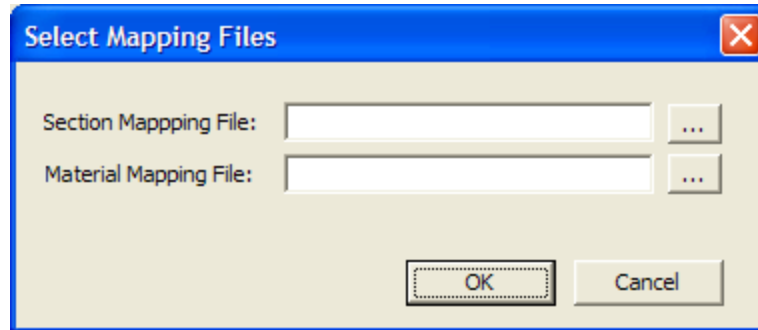
When the “X”, “Y”, and “Z” option is selected, the translator will define both the vertical axis of the import model as well as the member orientation option. When “X” is picked, the vertical axis will be in the global X direction and the default member orientation will Force member local z axis to be on the YZ plane. When “Y” is picked, the vertical axis will be in the global Y direction and the default member orientation will force member local z axis


to be on the XZ plane. When “Z” is picked, the vertical axis will be in the global Z direction and the default member orientation will Force member local z axis to be on the XY plane.

(For detailed information about vertical axis and member orientation, please refer to “RISA-3D reference manual”)

### ***Set Mapping Files Button (available in both Export and Import mode)***

When the Set Mapping File button is clicked, the following dialog box will show up.



You can pick a section/material mapping file by clicking the  button or by typing in the directory and file-name of the mapping file. If a mapping file is not desired, just leave the edit box empty. For details about sectional mapping and material mapping, please refer to the [“Mapping files”](#) section.

### ***Save As Defaults Button (both Export and Import mode)***

The RISA CIS/2 translator automatically remembers the previously used “translator mode (import/export)”, “file format (RISA3D/RISAFloor)” and file names. However, for the more advanced settings, such as “Model choices”, “Vertical axis”, and “Detail Choice”, the program will go back to the default setting every time the program is restarted. In order to save the current setting to be the default, the “Save As Defaults” button needs to be clicked. The “Save As Defaults” button only saves the options for the current translator mode. For example, when user is running in the export mode, only the options associated with export will be saved. The default settings are saved in three files in the program directory: cis2.def, cis2import.def and cis2export.def. The “cis2.def” file includes previous translator mode, file format and file names. The “cis2.def” file is saved automatically every time you hit the “RUN...” button. The “cis2import.def” file includes all the default advanced options for import mode. It is saved when you press the “Save As Defaults” button under import mode. The “cis2export.def” file includes all the default advanced options for export mode. It is saved when you press the “Save As Defaults” button under export model.

### ***View Error Log Button (both Export and Import mode)***

By clicking this button, the error log of the previous translation will show up. An **Error\_log.txt** file will be written out with each import or export to the output location. This is an overall log of what got translated and what did not.

### ***Run Button (both Import and Export mode)***

The program will start the translation if this button is clicked.

### ***Exit Button (both Import and Export mode)***

The program will exit if this button is clicked.

### ***Help Button (both Import and Export mode)***

The help menu will be launched if this button is clicked.



# Cardinal Points

## Overview

A detailing layer has been built for both 3D and Floor that lets the user set the "true" elevations and locations (start/end, top of steel, etc...) for all members. For each member (both column and beam in the case of Floor), a new data structure has been added to describe the connection point at member ends.

The areas in RISAFloor and RISA-3D where member detailing parameters can be added/viewed are:

- RISAFloor
  - Beams Data Entry Spreadsheet>Detailing tab
  - Columns Data Entry Spreadsheet>Detailing tab
  - Plot Options>Beams/Columns/Walls tab
- RISA-3D
  - Members Data Entry Spreadsheet>Detailing tab
  - Double-clicking Individual Member>Detailing tab
  - Graphic Member Drawing>Member Detailing tab
  - Plot Options>Members tab

For each end of the member, both cardinal point positions and decimal local offsets are used to describe the connection location. The location of the cardinal point is plotted as follows:

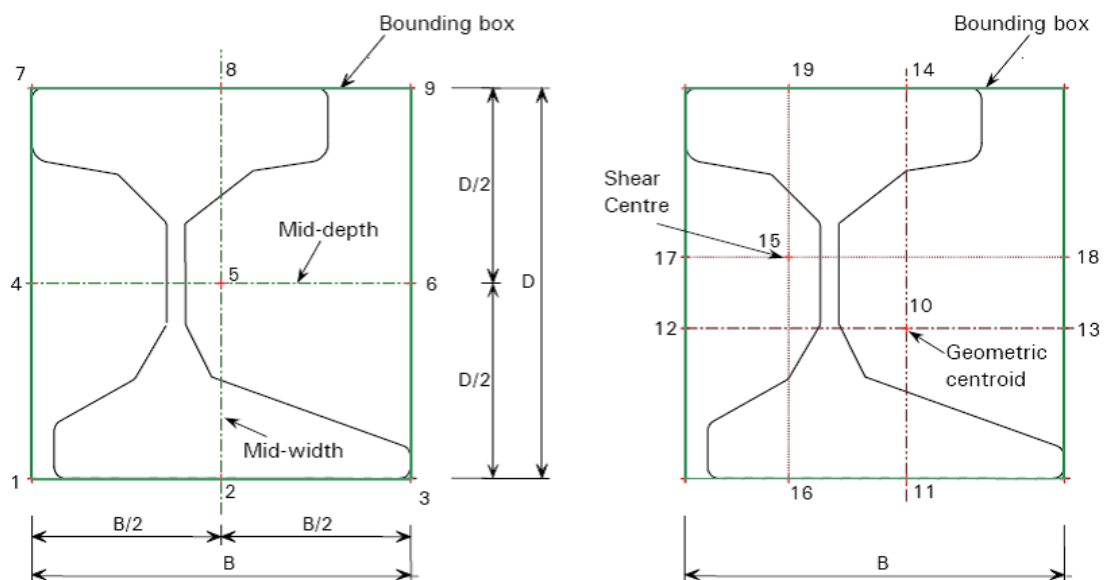


Figure 1 Cardinal point Location and Local Axis

- Cardinal point 1: lower left corner of the member section bounding box
- Cardinal point 2: lower center point in the member section bounding box
- Cardinal point 3: lower right corner of the member section bounding box
- Cardinal point 4: mid-depth left point in the member section bounding box
- Cardinal point 5: mid-depth center point in the member section bounding box
- Cardinal point 6: mid-depth right point in the member section bounding box

Cardinal point 7: upper left corner of the member section bounding box

Cardinal point 8: upper center point in the member section bounding box

Cardinal point 9: upper right corner of the member section bounding box

Cardinal point 10: Geometric centroid of the member section

**Note:**

- Only cardinal point 1-10 are supported by RISA. You may instead use an offset.

The x, y, and z offsets are based on the local axis of the member. The x local axis is defined along the member from I to J. It coincides with the geometric centroid of the member section. The standard cardinal point positions (1-10), as well as the decimal local offsets are both supported in current detailing definition. If the cardinal point is set, the y and z offset values will be automatically calculated and filled in the data structure. If there is a situation that doesn't match a cardinal point (angle brace as an example) you can also set the y and z offsets directly.

**Note:**

- The TOM and Rigid End Offset are completely different concepts from the detailing offset. The purpose of the detailing offsets is more realistic visualization and plotting of the model. The detailing offsets are not considered during the analysis. In the analysis, all members are still connected at their geometric centroid. TOM and Rigid End Offsets, on the other hand, affect the load distribution and design of the member. They are analysis parameters. In our program, the TOM / Rigid End Offset are completely separate data structure from the detailing offsets. When TOM is set for a model, for example, the detailing information will NOT be automatically updated to cardinal point 8 accordingly.

## Detailing Input and Modification:

There are three ways for setting and modifying the detailing layer information. User can use the graphical member modify, the double-click dialog and the member detailing spreadsheet to set the detailing information.

### **Default**

When a member is drawn, by default, x offsets are set to be 0 on both ends. If the member type is beam, by default the y and z detailing offsets will be on cardinal point 8 (top center) for both ends. If the member type is column, by default the detailing offset will be on cardinal point 10 for both ends.

### **Graphical member modify**

A **Member Detailing** tab has been added to the **Graphical member draw/modify** window. The user can choose to "set y, z offset using cardinal point" or "set y, z offsets directly". When a cardinal point is picked, the value in the "y Offset" and "z Offset" text box will automatically be updated.

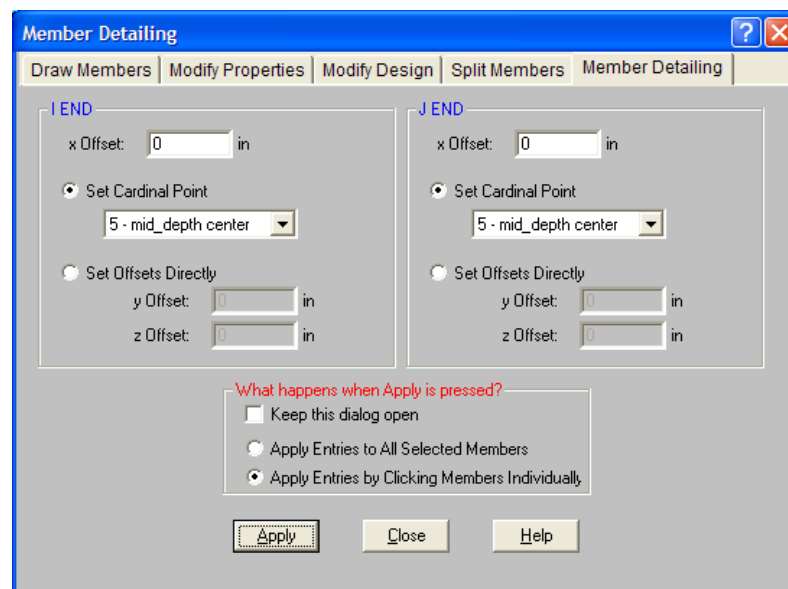


Figure 2.A Member Detailing Tab in Graphical Member Modify (RISA-3D)

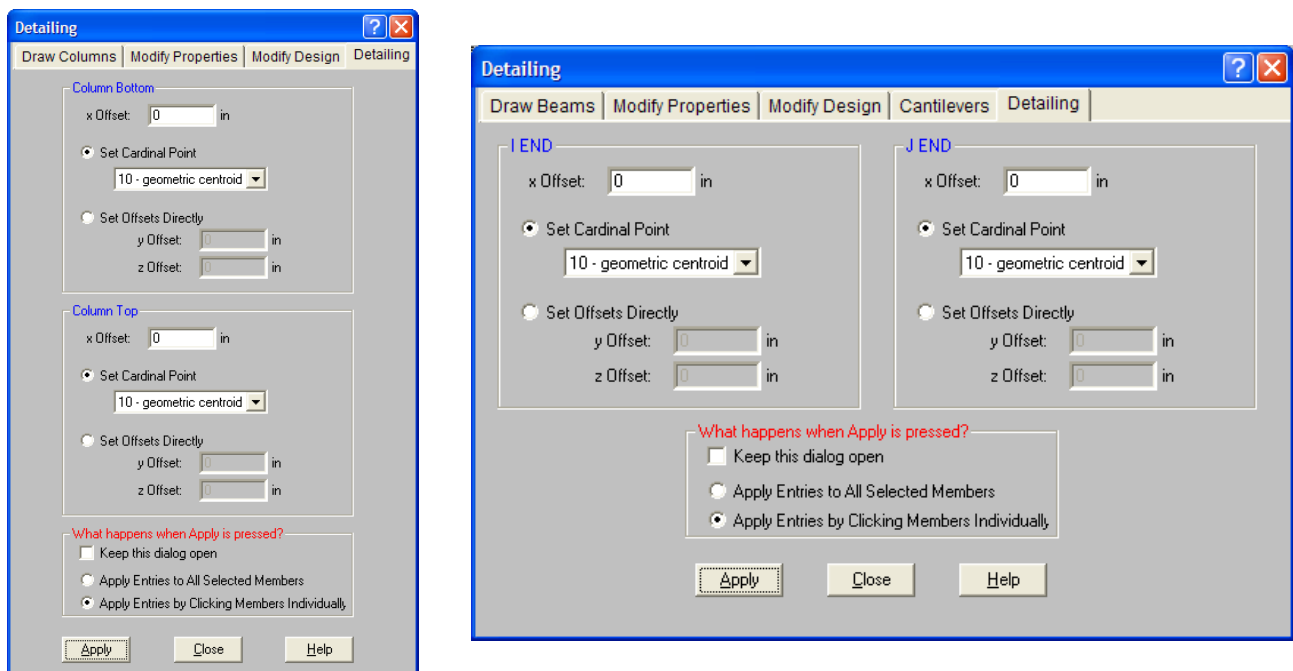


Figure 2.B Column/Beam Detailing Tab in Graphical Member Modify (RISAFloor)

### ***Double click dialog***

A **Detailing** tab has been added to the double click dialog for the displaying and modification of the detailing info. The user can choose to “set y, z offset using cardinal point” or “set y, z offsets directly”. When a cardinal point is picked, the value in the “y Offset” and “z Offset” text box will automatically be updated.

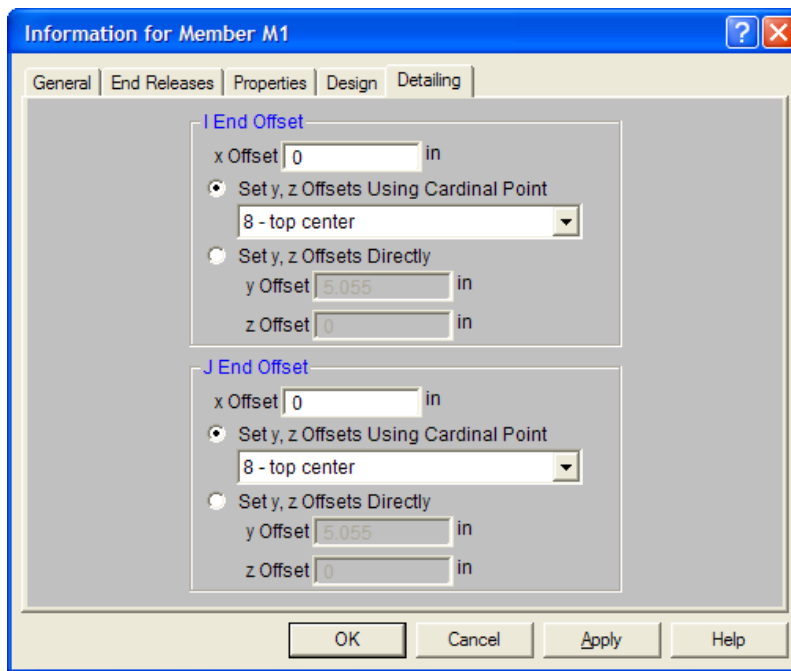


Figure 3 Detailing Tab in double click dialog

### Spreadsheets

A detailing tab has been added for the member spreadsheet (Column and Beam grid in the case of FLOOR). The spreadsheet shows a label and 8 values for each member. The “I cardinal point” and “J Cardinal Point” are pull down list with numbers and a description. When user picks a cardinal point, the y and z offset values will update automatically based on the current shape. When a new y and z offset are input by the user and it doesn’t fall on any cardinal point, the “Cardinal Point” will automatically change to “None”. When the “Member Detailing” grid is active, a toolbar button is added to recalculate values based on new shapes (Figure 5).

**Note:**

- When the design shape of the member is changed, the detailing offsets of the member is not automatically updated. The recalculate tool bar button needs to click to get new y, z offset with respect to the cardinal point picked)

Member Detailing Data									
Primary   Advanced   Hot Rolled   Cold Formed   Wood   Concrete Beam   Concrete Column   Detailing									
	Label	I Cardinal Point	I x Offset[in]	I y Offset[in]	I z Offset[in]	J Cardinal Point	J x Offset[in]	J y Offset[in]	J z Offset[in]
1	M1	8	0	5.055	0	8	0	5.055	0

Figure 4.A Detailing Tab in Member Spreadsheet(RISA-3D)

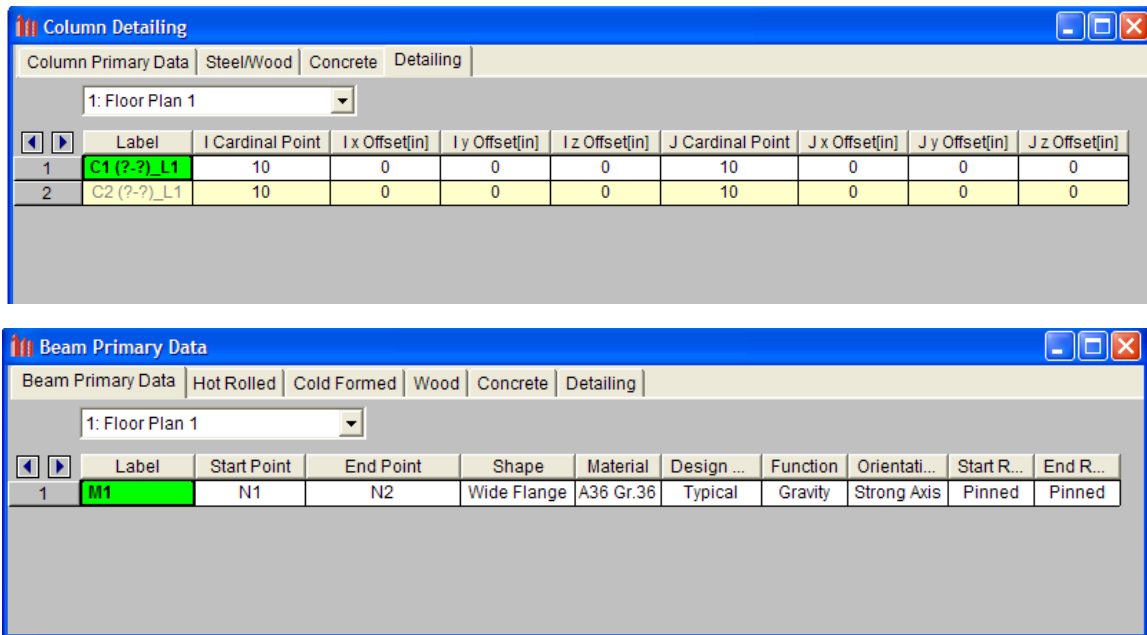


Figure 4.B Beam/Column Detailing Tab in Member Spreadsheet(RISAFloor)



Figure 5 Recalculate Detailing Offset button when Member Detailing Spreadsheet in Active

## Visualization of the Detailing Layer

To visualize the model with the detailing layer, go to the plot option dialog box and member tab. Check the "Detailing Info" check box (Figure 6).

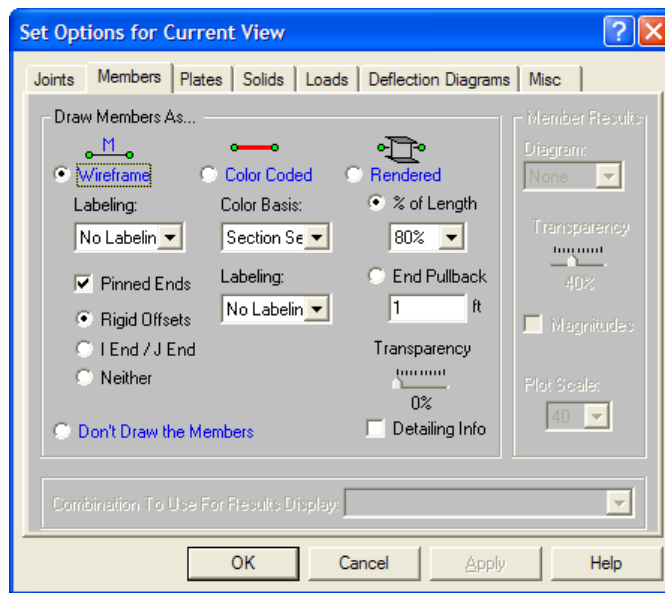


Figure 6.A Detailing Info checkbox in plot option window (RISA-3D)

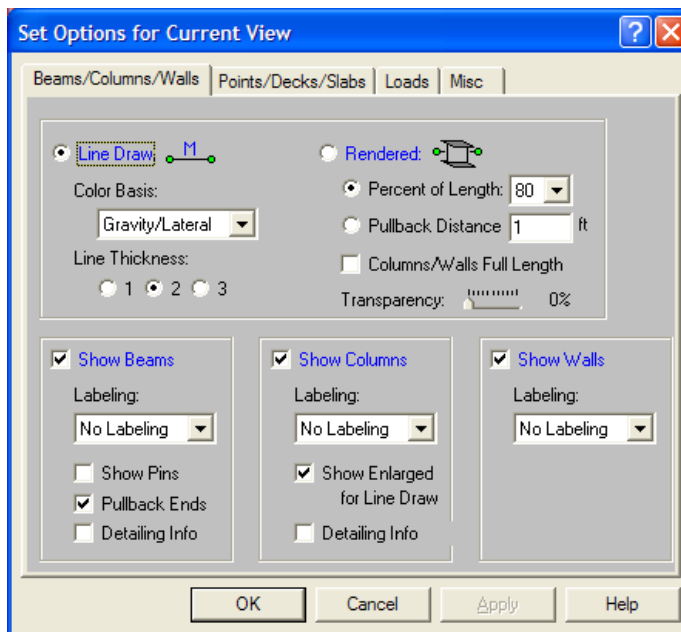


Figure 6.B Detailing Info checkbox in plot option window (RISAFloor)

**Note:**

- When the Detailing info box is checked. The members will be plotted solely using the x, y and z offsets. The “TOM” and “Rigid End Offset” will not be considered in the plotted here.

**File I/O**

[.MEMBERS\_DETAILING\_DATA]data section is added to the .R3D file to hold the member true location data. [..RF\_COLUMNS\_DETAILING\_DATA] and [..RF\_BEAMS\_DETAILING\_DATA] data section are added to the .RFL file to hold

member true location data. The CIS2 translator is updated to read/write this new data section.

# What is Supported with this Translator?

## Elements

Only member elements are supported with the CIS/2 translator. Plate and solid elements are not supported in the current version.

## Loading conditions

The current CIS/2 translator only supports joint loads, member concentrated loads and member distributed loads. Member area loads and plate surface loads are not supported. In the current version, only static loads will be considered in the translation. Dynamic loading in the model will be ignored. RISA joint loads can be defined in the form of mass, displacement and forces. During the translation, only the force type of load will be translated. The mass and displacement loads will be ignored.

## Sections Sets

CIS/2 does not understand the concept of section set. Therefore, in order to export the model correctly, members need to be explicitly defined with a shape name.

## Physical Members and Member mapping

In CIS/2, design members and analysis members are related by the entity “member\_mapping”. The “member\_mapping” entity maps one design assembly to one or multiple analysis elements. This concept is consistent with the concept of physical member in RISA-3D. The physical members are broken down to several FEM elements during the analysis, and considered to be one single piece during the design.

When exporting, if the design model is selected, the RISA CIS/2 translator will generate one “assembly\_design\_structural\_member\_linear” entity for each physical member in the design model. If the analysis model is selected, the RISA CIS/2 translator will generate multiple analysis entity according to the joints (or physical pieces) on the RISA physical member. If both the analysis model and design model is selected, the “assembly\_map” entity will be generated to store the logical relationship between the design entities and analysis entities

When importing, if the user chooses to import both the design model and analysis model, the program will search for the “assembly\_map” and map the design entity and analysis entity to be one physical member. If “assembly\_map” entity isn't present in the CIS/2 model, then design assembly and analysis entity will be translated as separated members into the RISA file. There will be no physical member generated. If only Analysis model or Design model is selected in the translator, the program will not import the “assembly\_map” entity. Even though it might be present in the model, no physical members will be generated.

Model Choice	assembly_design_structural_member_linear	element_curve_simple	assembly_map
Analysis model	No	Yes	No
Design model	Yes	No	No
Analysis & Design model	Yes	Yes	Yes



## Mapping Files

Different programs have their own naming convention for materials and shapes. For example, the angle section “LL8X8X18X0” in RISA, is named “2L8x8x1 1/8” in SDS2, and is named “2L8x8x1-1/8” in SmartPlant3D. Mapping files are used for this purpose. If a section mapping file is provided during export, the translator will search the mapping file for the corresponding name of the objective program, and use the new name in the CIS/2 part21 file. During import, the translator will search the mapping file for the RISA shape name and write the RISA file. If the mapping file is not provided by the user, the original section name will be written by the translator.

Sample section and material mapping files with SmartPlant3D, SDS/2, and Tekla Structures are provided with the RISA CIS/2 translator package. Both the section mapping and the material mapping file is of XML format. Using the provided mapping files as template, you can write your own mapping files if communication with a third party program is desired.

To generate a new mapping file, copy the provided section mapping file to a new file name. Open the file in Microsoft Excel and the section mapping will be of the following format:

<b>RISA Shape Name</b>	<b>Shape Name</b>
LL8X8X18X0	2L8x8x1 1/8
LL8X8X18X3	2L8x8x1 1/8x3/8
LL4X4X8X0	2L4x4x1/2
LL4X4X8X3	2L4x4x1/2x3/8
LL4X4X8X6	2L4x4x1/2x3/4
LL4X4X6X0	2L4x4x3/8
LL4X4X6X3	2L4x4x3/8x3/8
LL4X4X6X6	2L4x4x3/8x3/4

The first column is the RISA shape name. Write the corresponding shape in the second column and save the xml file.

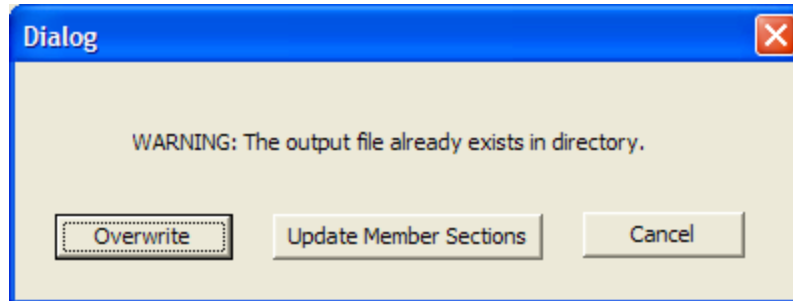
The format of the material mapping is:

<b>RISA Material Name</b>	<b>Material Name</b>	<b>Material Type</b>
A36 Gr.36	GRADE36	HOT_ROLLED
A572 Gr.50	GRADE50	HOT_ROLLED
Conc3000NW	Concrete:Fc 3000	CONCRETE
Conc4000NW	Concrete:Fc 4000	CONCRETE
Conc3000LW	Light Weight Concrete:Fc 3000	CONCRETE

The first column is the RISA material name. The third column is the material type, which could be one of the six types: GENERAL, HOT\_ROLLED, COLD\_FORMED, WOOD, CONCRETE, OTHER. Write the corresponding material name in the second column and write the correspond material in the third column. (NOTE: the material type name is case sensitive)

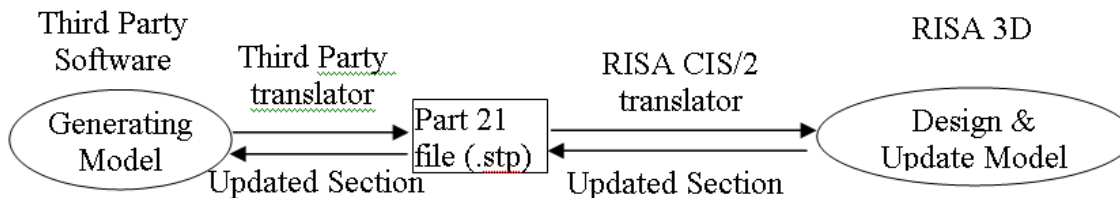
# Overwrite and Update Section

During export, when an existing output file (.stp file) is picked, a dialog box will show up:



If the “Overwrite” button is clicked, the existing “.stp” file will be overwritten. If the “Cancel” button is clicked, the translation will be canceled.

If the “Update Member Sections” button is clicked, instead of writing a new “.stp” file, the program will read the member section in the RISA file, search the matching member in the existing .stp file and update the section. This feature is used when the user wants to build a model in a third party software, such as SMARTPLANT3D, export through the CIS/2 link to RISA, design it in RISA, and update the designed section back to CIS/2 file. You would then translate back to the third party file.

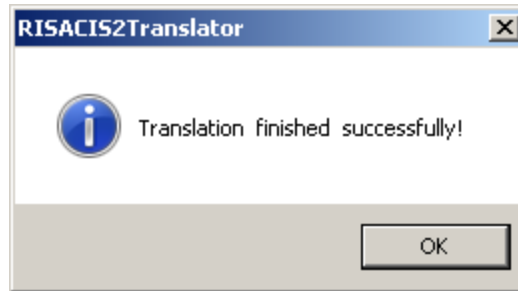


Note:

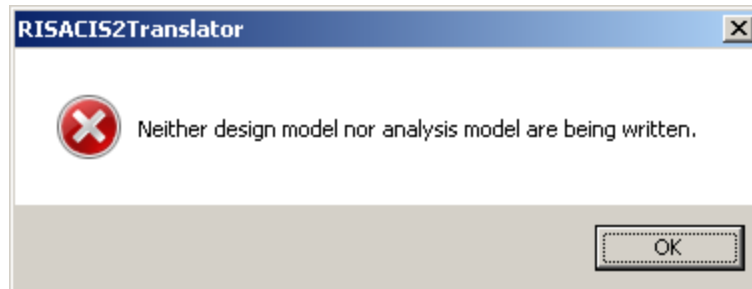
- The update member feature is not available in RISAFloor, because importing to RISAFloor is not available in this version.
- In order for the update member feature to work correctly, the RISA-3D input file will have to have been previously imported from the existing part 21 file. Also, the model choice (Analysis model / Design model/ Both) option will need to remain the same during the entire import/export process.
- It is recommended that all the changes to the model are being performed using the Third Party software. The user can delete/add/split members in RISA, but be aware that the deleted/added/splitted member will possibly not be recognized during the update member process.

# Messages

If the translation is finished normally, the following message box will show up:



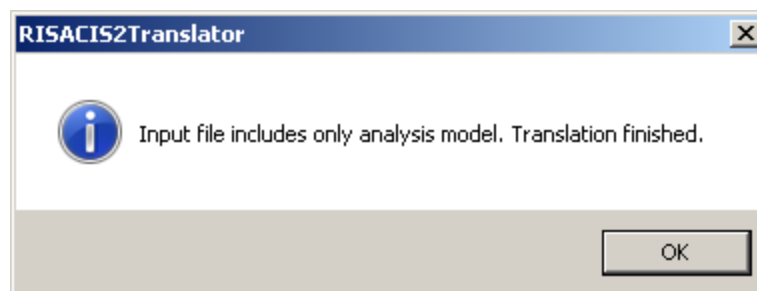
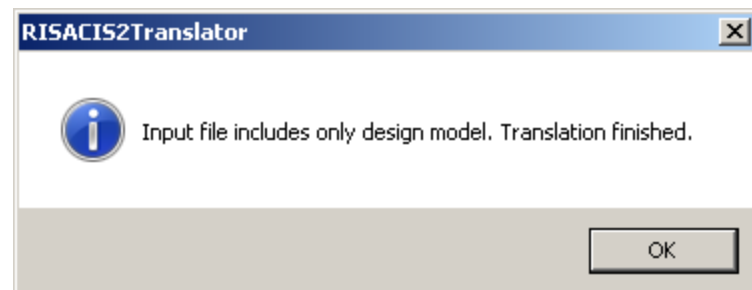
If the import model doesn't contain the model type chose by user, the following message will show up:



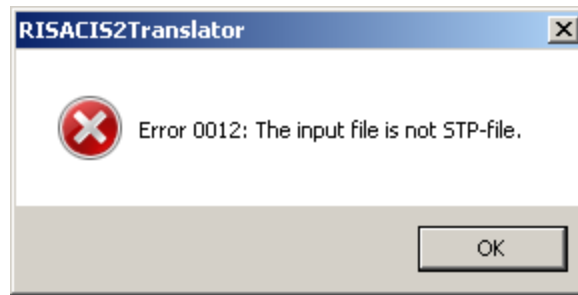
**Note:**

- In some cases the translator will simply stop working if it can not find any analysis or design elements in the file. Please view the [Supported CIS/2 Entity List Import](#) for the sections of the .stp file we are looking for. If you don't have these portions in your file then no information will be translated.

If the import model contains only a part of the information the user required the following message box will show up. For example, the user chose to export both "analysis model" and "design model", but the input file contains only "analysis model"



If the file is not a .stp file format then you may get this message:



## Supported CIS/2 Entity List (Export)

The supported CIS/2 entity list is as follows:

### **Analysis:**

ANALYSIS\_MODEL\_3D  
ANALYSIS\_MODEL\_LOCATED  
APPLIED\_LOAD\_STATIC\_FORCE  
ASSEMBLY\_DESIGN\_STRUCTURAL\_MEMBER\_LINEAR  
ASSEMBLY\_MAP  
AXIS2\_PLACEMENT\_3D  
BOUNDARY\_CONDITION\_LOGICAL  
CARTESIAN\_POINT  
CONTEXT\_DEPENDENT\_UNIT  
COORD\_SYSTEM\_CARTESIAN\_3D  
DERIVED\_UNIT  
DERIVED\_UNIT\_ELEMENT  
DIMENSIONAL\_EXPONENTS  
DIRECTION  
ELEMENT\_CURVE\_SIMPLE  
ELEMENT\_ECCENTRICITY  
ELEMENT\_NODE\_CONNECTIVITY  
ELEMENT\_WITH\_MATERIAL  
FORCE\_MEASURE\_WITH\_UNIT  
FORCE\_MEASURE  
GEOMETRIC\_REPRESENTATION\_CONTEXT  
GLOBAL\_UNIT\_ASSIGNED\_CONTEXT  
ITEM\_REF\_SOURCE\_STANDARD  
ITEM\_REFERENCE\_ASSIGNED  
ITEM\_REFERENCE\_STANDARD  
LENGTH\_UNIT  
LINE  
LOAD\_COMBINATION\_OCCURRENCE  
LOADING\_COMBINATION\_LOAD\_NODE  
LOAD\_ELEMENT\_CONCENTRATED  
LOAD\_ELEMENT\_DISTRIBUTED\_CURVE\_LINE  
LOAD\_MEMBER\_DISTRIBUTED\_CURVE\_LINE  
LOAD\_MEMBER\_CONCENTRATED  
LOAD\_CONNECTION  
LOAD\_CASE  
MANAGED\_DATA\_ITEM  
MATERIAL\_ELASTICITY  
MATERIAL\_ISOTROPIC  
MATERIAL\_MASS\_DENSITY  
MATERIAL\_PROPERTY\_CONTEXT\_DIMENSIONAL  
MATERIAL\_REPRESENTATION

MATERIAL\_STRENGTH  
MATERIAL\_THERMAL\_EXPANSION  
NODE  
ORGANIZATION  
PHYSICAL\_ACTION\_PERMANENT  
PLANE\_ANGLE\_MEASURE  
PLANE\_ANGLE\_MEASURE\_WITH\_UNIT  
PLANE\_ANGLE\_UNIT  
PRESSURE\_UNIT  
RELEASE\_LOGICAL  
REPRESENTATION  
SECTION\_PROFILE  
SI\_UNIT  
VECTOR

**Design:**

ASSEMBLY\_DESIGN\_STRUCTURAL\_CONNECTION\_INTERNAL  
ASSEMBLY\_DESIGN\_STRUCTURAL\_MEMBER\_LINEAR\_BEAM  
ASSEMBLY\_DESIGN\_STRUCTURAL\_MEMBER\_LINEAR\_BRACE  
ASSEMBLY\_DESIGN\_STRUCTURAL\_MEMBER\_LINEAR\_COLUMN  
AXIS2\_PLACEMENT\_3D  
CARTESIAN\_POINT  
CONTEXT\_DEPENDENT\_UNIT  
COORD\_SYSTEM\_CARTESIAN\_3D  
DERIVED\_UNIT  
DERIVED\_UNIT\_ELEMENT  
DESIGN\_PART  
DIMENSIONAL\_EXPONENTS  
DIRECTION  
GEOMETRIC\_REPRESENTATION\_CONTEXT  
GLOBAL\_UNIT\_ASSIGNED\_CONTEXT  
ITEM\_REF\_SOURCE\_STANDARD  
ITEM\_REFERENCE\_ASSIGNED  
ITEM\_REFERENCE\_STANDARD  
LENGTH\_UNIT  
LOCATED\_ASSEMBLY  
MANAGED\_DATA\_ITEM  
MATERIAL\_ELASTICITY  
MATERIAL\_ISOTROPIC  
MATERIAL\_MASS\_DENSITY  
MATERIAL\_PROPERTY\_CONTEXT\_DIMENSIONAL  
MATERIAL\_REPRESENTATION  
MATERIAL\_STRENGTH  
MATERIAL\_THERMAL\_EXPANSION

ORGANIZATION  
PART\_PRISMATIC\_SIMPLE  
POSITIVE\_LENGTH\_MEASURE  
POSITIVE\_LENGTH\_MEASURE\_WITH\_UNIT  
PRESSURE\_UNIT  
REPRESENTATION  
SECTION\_PROFILE  
SI\_UNIT  
STRUCTURAL\_FRAME\_PRODUCT\_WITH\_MATERIAL  
STRUCTURE

## Supported CIS/2 Entity List (Import)

The supported CIS/2 entity list is as follows:

### **Analysis:**

ANALYSIS\_MODEL\_3D  
ANALYSIS\_MODEL\_LOCATED  
APPLIED\_LOAD\_STATIC\_FORCE  
ASSEMBLY\_DESIGN\_STRUCTURAL\_MEMBER\_LINEAR  
ASSEMBLY\_MAP  
AXIS2\_PLACEMENT\_3D  
BOUNDARY\_CONDITION\_LOGICAL  
CARTESIAN\_POINT  
CONTEXT\_DEPENDENT\_UNIT  
COORD\_SYSTEM\_CARTESIAN\_3D  
DERIVED\_UNIT  
DERIVED\_UNIT\_ELEMENT  
DIMENSIONAL\_EXPONENTS  
DIRECTION  
ELEMENT\_CURVE\_SIMPLE  
ELEMENT\_ECCENTRICITY  
ELEMENT\_NODE\_CONNECTIVITY  
ELEMENT\_WITH\_MATERIAL  
FORCE\_MEASURE\_WITH\_UNIT  
FORCE\_MEASURE  
GEOMETRIC\_REPRESENTATION\_CONTEXT  
GLOBAL\_UNIT\_ASSIGNED\_CONTEXT  
ITEM\_REF\_SOURCE\_STANDARD  
ITEM\_REFERENCE\_ASSIGNED  
ITEM\_REFERENCE\_STANDARD  
LENGTH\_UNIT  
LOAD\_COMBINATION\_OCCURRENCE  
LOADING\_COMBINATION\_LOAD\_NODE  
LOAD\_ELEMENT\_CONCENTRATED  
LOAD\_ELEMENT\_DISTRIBUTED\_CURVE\_LINE  
LOAD\_MEMBER\_DISTRIBUTED\_CURVE\_LINE  
LOAD\_MEMBER\_CONCENTRATED  
LOAD\_CONNECTION  
LOAD\_CASE  
LINE  
MATERIAL\_ELASTICITY  
MATERIAL\_ISOTROPIC  
MATERIAL\_MASS\_DENSITY  
MATERIAL\_PROPERTY\_CONTEXT\_DIMENSIONAL  
MATERIAL\_REPRESENTATION  
MATERIAL\_STRENGTH



MATERIAL\_THERMAL\_EXPANSION  
NODE  
ORGANIZATION  
PHYSICAL\_ACTION\_PERMANENT  
PLANE\_ANGLE\_MEASURE  
PLANE\_ANGLE\_MEASURE\_WITH\_UNIT  
PLANE\_ANGLE\_UNIT  
PRESSURE\_UNIT  
RELEASE\_LOGICAL  
REPRESENTATION  
SECTION\_PROFILE  
SI\_UNIT  
VECTOR

**Design:**

ASSEMBLY\_DESIGN\_STRUCTURAL\_CONNECTION\_INTERNAL  
ASSEMBLY\_DESIGN\_STRUCTURAL\_MEMBER\_LINEAR\_BEAM  
ASSEMBLY\_DESIGN\_STRUCTURAL\_MEMBER\_LINEAR\_BRACE  
ASSEMBLY\_DESIGN\_STRUCTURAL\_MEMBER\_LINEAR\_COLUMN  
AXIS2\_PLACEMENT\_3D  
CARTESIAN\_POINT  
CONTEXT\_DEPENDENT\_UNIT  
COORD\_SYSTEM\_CARTESIAN\_3D  
DERIVED\_UNIT  
DERIVED\_UNIT\_ELEMENT  
DESIGN\_PART  
DIMENSIONAL\_EXPONENTS  
DIRECTION  
GEOMETRIC\_REPRESENTATION\_CONTEXT  
GLOBAL\_UNIT\_ASSIGNED\_CONTEXT  
ITEM\_REF\_SOURCE\_STANDARD  
ITEM\_REFERENCE\_ASSIGNED  
ITEM\_REFERENCE\_STANDARD  
LENGTH\_UNIT  
LOCATED\_ASSEMBLY  
MATERIAL\_ELASTICITY  
MATERIAL\_ISOTROPIC  
MATERIAL\_MASS\_DENSITY  
MATERIAL\_PROPERTY\_CONTEXT\_DIMENSIONAL  
MATERIAL\_REPRESENTATION  
MATERIAL\_STRENGTH  
MATERIAL\_THERMAL\_EXPANSION  
ORGANIZATION  
PART\_PRISMATIC\_SIMPLE

## Supported CIS/2 Entity List (Import)

---

POSITIVE\_LENGTH\_MEASURE

POSITIVE\_LENGTH\_MEASURE\_WITH\_UNIT

PRESSURE\_UNIT

REPRESENTATION

SECTION\_PROFILE

SI\_UNIT

STRUCTURAL\_FRAME\_PRODUCT\_WITH\_MATERIAL

STRUCTURE

# Tekla Structures and RISA-3D Mapping

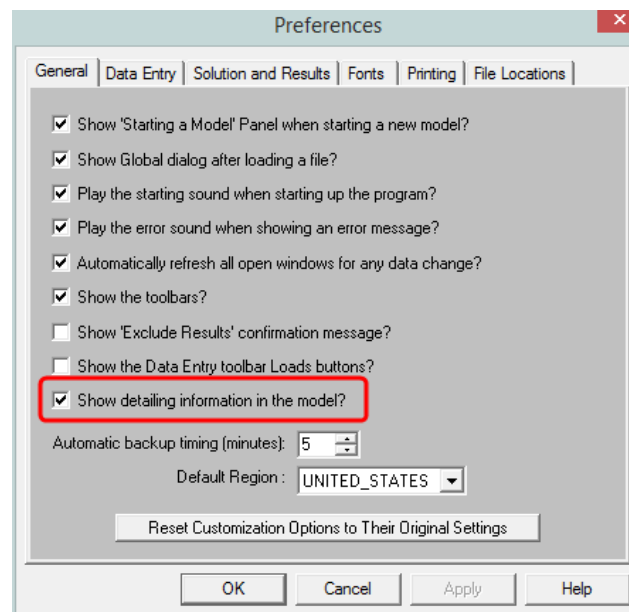
The CIS/2 Translator can be used to transfer information either direction between RISA-3D and Tekla Structures. This topic will give an overview on how this behavior works. The transfer of section properties, material properties and member orientation information are the primary uses of this translator.

## How to Export from RISA-3D to Tekla Structures

After creating a model in RISA-3D you can transfer the analytical model to Tekla Structures for the purpose of detailing the structure. Here is a basic outline of this process.

### Setup in RISA-3D before Export

RISA-3D has detailing information that can be defined for each member in the model. For this to be visible the **Show detailing information in the model?** checkbox must be checked in **Tools - Preferences**.



From here you can define cardinal point information. See the [Cardinal Points](#) topic for more information.

If the beam offsets are defined in the RISA-3D model as **8 Top - Center** as shown, the **Position** setting in Tekla Structures for beam objects will automatically be set as **Behind**.

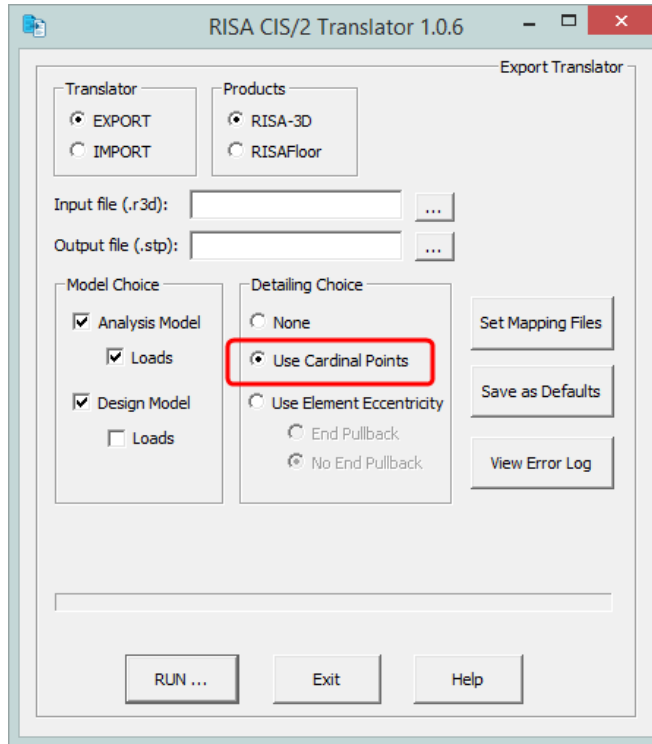
Member Detailing Data										
	Label	I Cardinal Point	I x Offset[in]	I y Offset[in]	I z Offset[in]	J Cardinal Point	J x Offset[in]	J y Offset[in]	J z Offset[in]	
1	M1	8 - top center	0	3.945	0	8	0	3.945	0	
2	M2	8	0	3.945	0	8	0	3.945	0	
3	M3	8	0	3.945	0	8	0	3.945	0	
4	M4	8	0	3.945	0	8	0	3.945	0	
5	M5	8	0	3.945	0	8	0	3.945	0	

### Using the RISA CIS/2 Translator

See the [Introduction](#) topic for the basics of the CIS/2 Translator.

A couple of notes on using the translator:

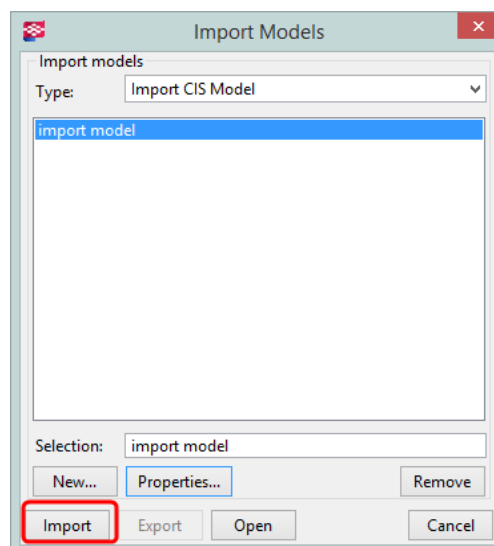
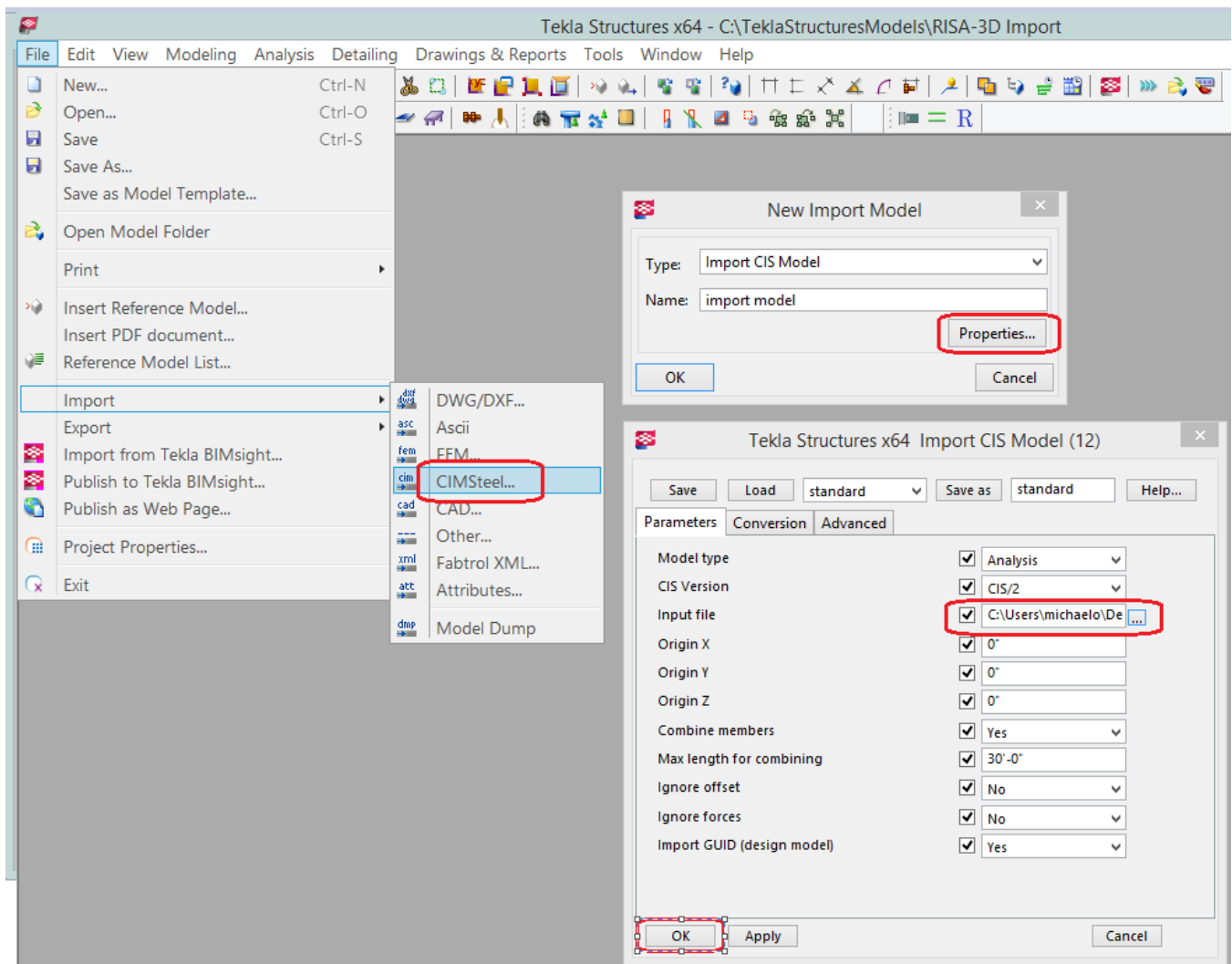
- Tekla Structures in general prefers to receive a CIS/2 analysis model from RISA.
- Mapping files for section shapes and material properties may be needed to be set up on either the export from RISA-3D or the import into Tekla Structures. For information on RISA-3D mapping see the [Mapping Files](#) topic.
- Use the Cardinal Point export option to utilize the cardinal point definitions for beam elements.



### **Import to Tekla Structures**

Once the CIS/2 file is created (with a suffix of \*.stp), the file can be imported into Tekla Structures. The steps for importing the CIS/2 file are shown in the figure below and outlined here:

1. Choose **File>Import>CIMSteel...**
2. On the **New Import Model** dialog, make sure that **Import CIS Model** is selected and press the **Properties...** button.
3. With the **Import CIS Model** dialog open, go to the **Input File** box and search for the file. Note that the file path length should be verified such that the entire .stp file path fits in the file path.
4. Select **OK**.
5. Press the **Import** button on the **Import Models** dialog and the model will then import.

**Note:**

- Once the model is imported, go to the View Properties and change the View Depth to be large enough to include the model. Also right click on the View and select Fit by Parts.

- If there is a problem with CIS/2 that aren't importing look at the heading of the CIS/2 file. If the Logic Product Model is anything other than LPM6 then it may not be possible to import it.

## How to Export from Tekla Structures to RISA-3D

After creating a model in Tekla Structures you can transfer the analytical model to RISA-3D for the design of the structure. RISA-3D also has the capability of transferring the model to RISAConnection, where individual connections can be designed. Here is a basic outline of how this works.

### Tekla Structures CIS/2 Export

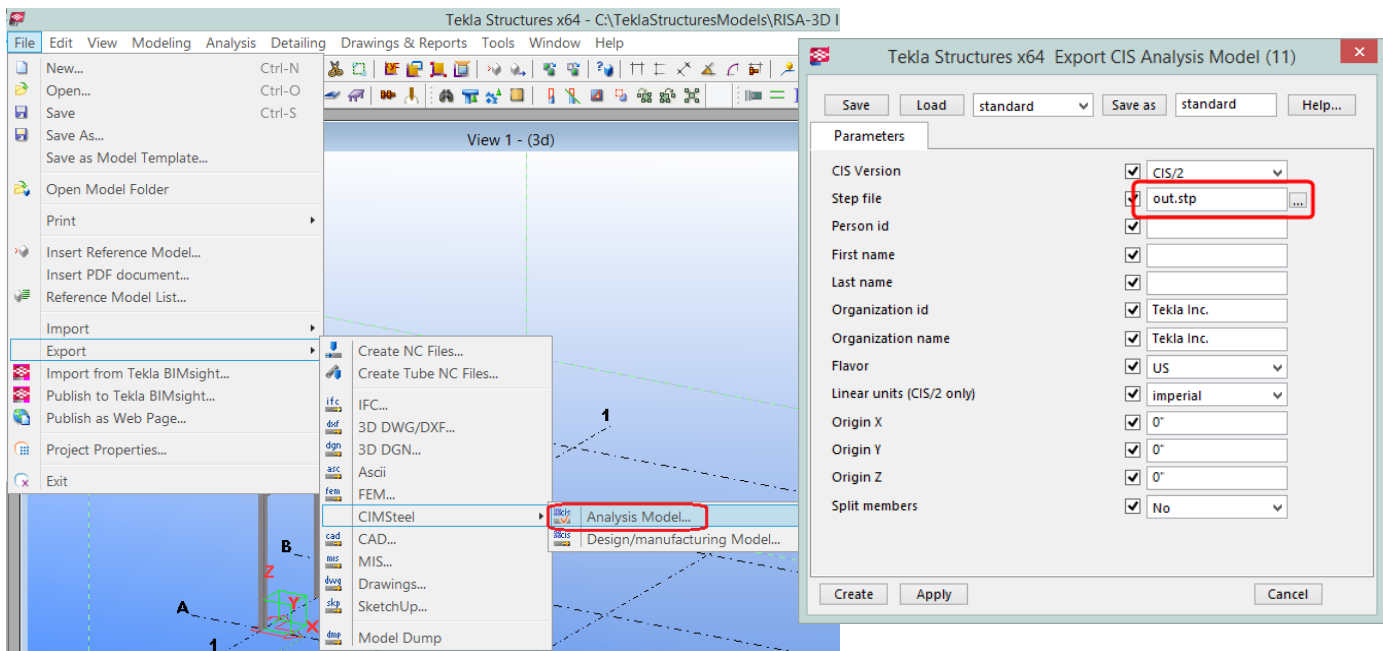
1. Open the model in Tekla Structures
2. Open a view showing the parts you would like to export. Make sure to load the View settings for exporting via CIS/2.
3. Make sure the Select Objects in Components icon is selected. If no connections are to be included in this export, only select the parts icon as well.



Note that if the

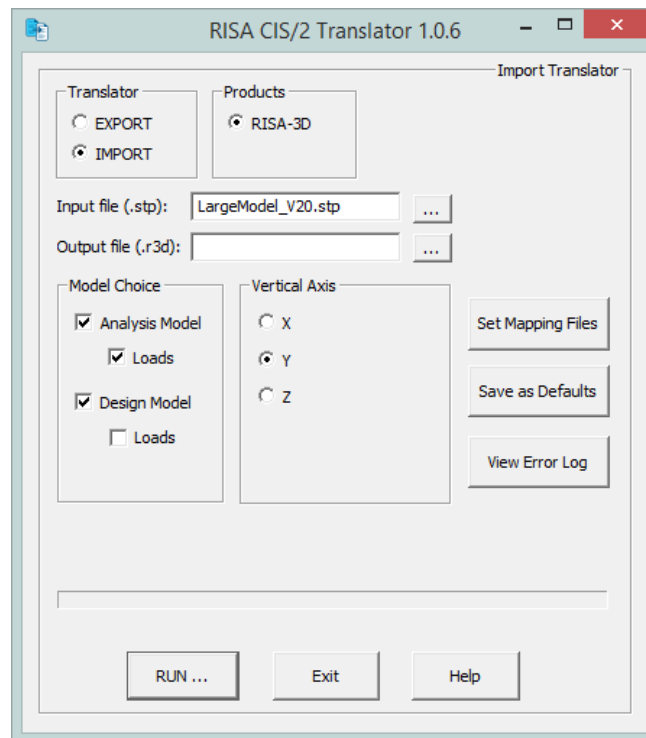
Select Components or the Select Assembly icons are selected instead, the parts themselves will not export, even though they show up as selected in the view.

4. Select entire model.
5. Choose **File>Export>CIMSteel>Analysis Model...**
6. Type in name of file ending in .stp. Note that once the user sets up their standard settings, they should save away an attribute setting in the upper right corner of the dialog box. This value can be used in the future by loading it from the Load icon.
7. With settings defined, select **Apply**, select **Create**. The CIS/2 model will run. A DOS window may appear in the background. That is a normal routine in the export. A note should appear in the lower left corner of the Tekla Structures window that states "parts were successfully exported."



### RISA-3D Import

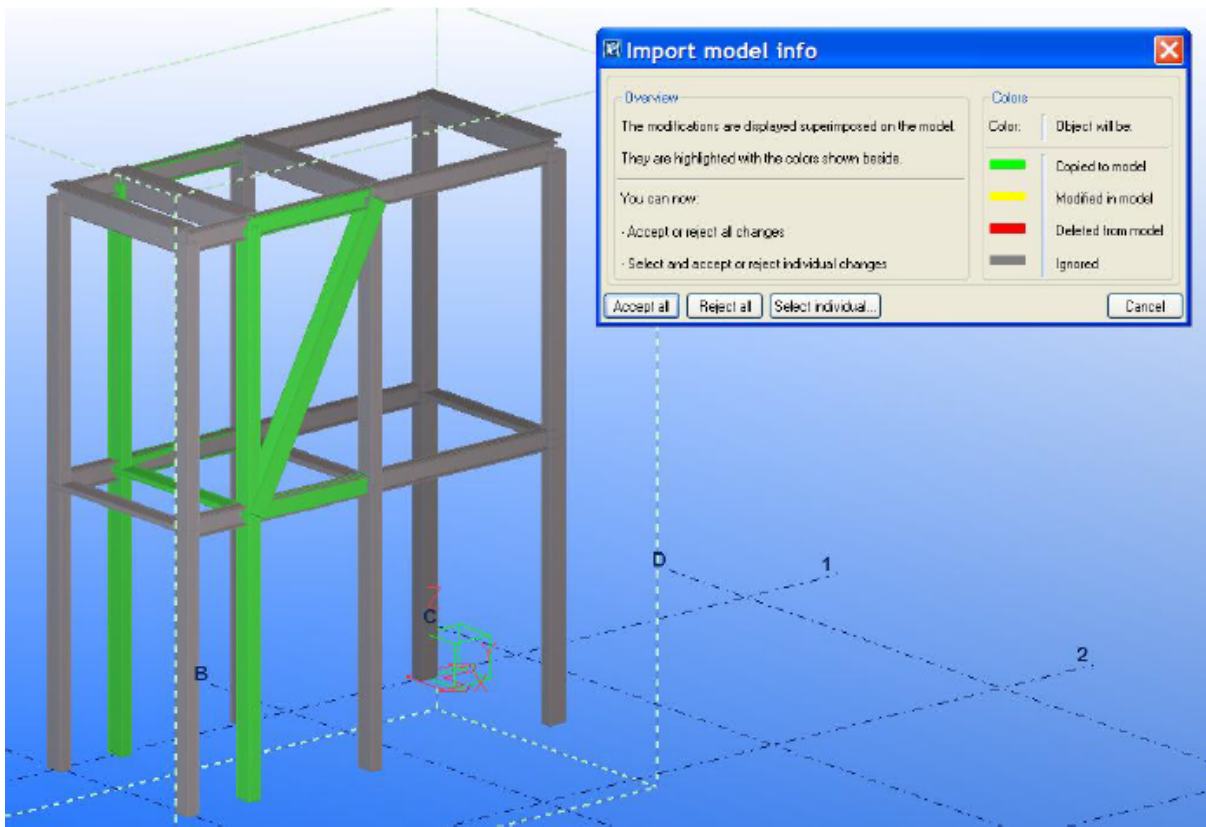
Using the CIS/2 Translator of RISA-3D, the user specifies a .stp file created from Tekla Structures and imports it as an analysis model into RISA-3D.



## Updating a Tekla Structures Model

It is also possible to receive updated models from RISA-3D and merge those changes into your current Tekla Structures model. Here are the basic steps to this process.

1. Import the original model as described previously. Choose to save the model for future imports.
2. After making changes in RISA-3D, import the model again into the same Tekla Structures model. This model will have its own import model name.
3. A change management dialog box will show up as shown stating what objects are added (green), modified (yellow) or deleted (red). Note that this change management of modified members is of section size, not moved or stretched members coming from RISA-3D.



## Setting up Mapping Files

### *Tekla Structures Mapping Files*

Setting up mapping files in Tekla Structures is as simple as editing a 2 column text file suffixed as a \*.cnv file. Editing these files is described further in the Tekla Help section of Tekla Structures. Some example cnv files are located in each Tekla environment folder. Here is folder containing several cnv mapping files for the US market.

C:\ProgramData\Tekla Structures\20.0\Environments\usimp\profil

The **prfexp\_\*** files are section property mapping files while the **matexp\_\*** files are material property mapping files.

### *RISA-3D Mapping Files*

See the [Mapping Files](#) topic for more information.



---

## Technical Support

Technical support is very important to the staff at RISA Technologies. We want our users to be able to reach us when they are having difficulties with the program.

**Hours:** 6AM to 5PM Pacific Standard Time, Monday through Friday

Before contacting technical support, you should typically do the following:

1. **Please search the Help File or General Reference Manual.** Many common questions asked about CIS/2 translator can be found in the Help File or General Reference Manual. Use the table of contents or index to find specific topics and appropriate sections.
2. Make sure you understand the problem, and make sure your question is related to the program or structural modeling.
3. Take a few minutes to experiment with the problem to try to understand and solve it.

**Email:** [support@risa.com](mailto:support@risa.com): Make sure you tell us your name, company name, serial number or Key ID, phone number, and give a decent problem description.

**Phone Support:(949) 951-5815:** Feel free to call, especially if you need a quick answer and your question is not model specific and therefore doesn't require us to look at your file.



---

# Index

---

## C

Cardinal Point Locations 5

Cardinal Points 5

## D

Default 6

Detailing Input 6

detailing layer 5

Detailing Layer 5

Detailing Modification 6

Double click dialog 7

## E

Element 12

Element Eccentricity Group box 3

Exit 4

Export Mode 1

## F

File I/O 10

## G

Graphical member modify 6

## H

Help Button 4

## I

Import mode 1

Input File 2

Introduction 1

## L

Loading Conditions 12

## M

Mapping Files 13

Messages 15

Model Choice Group box 2

## O

Output File 2

Overview 5

Overwrite 14

## P

Physical Member 12

Products Group Box 2

## R

Run 4

## S

Save As Defaults 4

Sections Sets 12

Set Mapping Files 4

Supported CIS/2 Entity List (Import) 17, 20

## T

Technical Support 29

## U

Update Section 14

---

**V**

Vertical Axis Group box 3

View Error Log 4