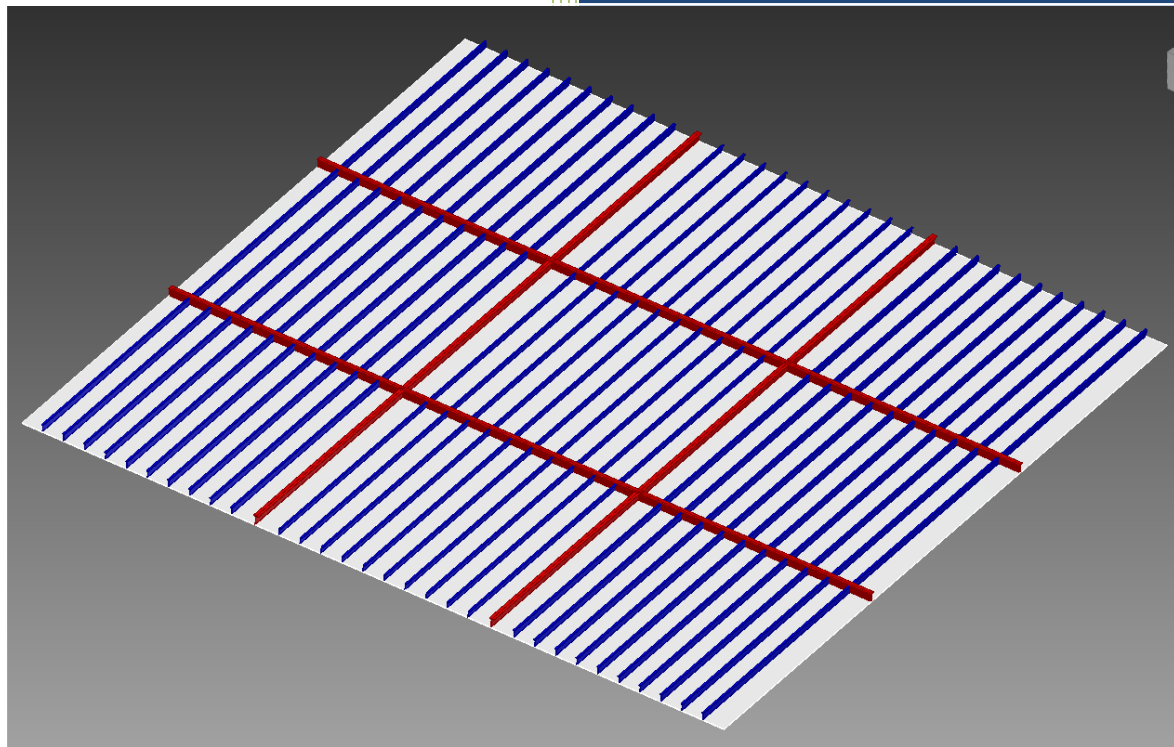


2013

# Guidance for CAD and Analysis Model in Simulation Based Design



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01.01.2013

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## 1 Expressions and abbreviations

<u>Abbreviations</u>	<u>Meaning</u>
ANSYS	Software for FEM analysis
CAD	Computer Aided Design
Constraints	Physical locking of drawings and parts in Inventor
FEM	Finite Element Method
Inventor	CAD tool for 3D models
Mesh	Applying elements to the model

## 2 Introduction

This guidance will help students taking the course Simulation Based Design in creating a 3D model using Inventor CAD software. The model will be simplified and idealized using ANSYS Design Modeler and then imported to ANSYS Mechanical to mesh and analyze the model.

Inventor will be used to make a realistic and correct 3D model of a stiffened plate. ANSYS Design Modeler will simplify the model. All volume parts will be converted to shell and plate elements. ANSYS Mechanical will mesh the idealized model and be used for FEM analysis.

## 3 Creating the 3D model using Inventor

### 3.1 Constraints

Constraints locks the sketch in the coordinate system. It is important that a sketch is fully constrained before you work further with it. Inventor offers several constrain options, but only a few will be introduced in this course. Examples of constrains are dimensional, horizontal, vertical and parallel. As an example the dimensional constrain means that an edge will be given a dimension which will not change.

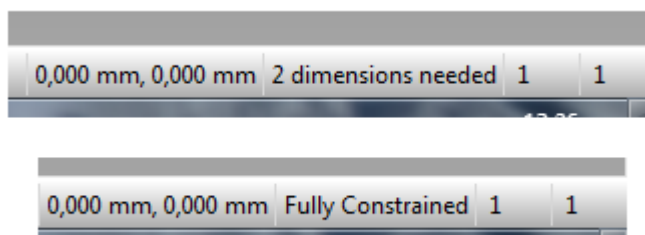


Figure 1 - Constraints

In the bottom right corner it will say how many constraints are needed.

## 3.2 Create a part

The first thing to do is to create a part in Inventor. Open Inventor which is installed on students computers. Select **PRO** → **NEW** → **PART**

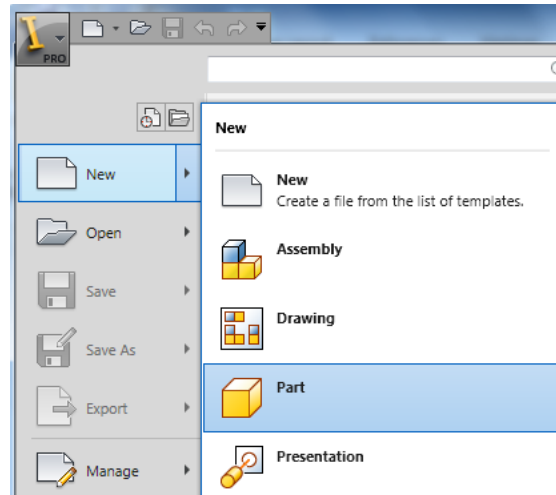


Figure 2 - New Part

## 3.3 Create a plate

### 3.3.1 Draw a 2D sketch

After creating a part you will automatically be guided to the sketch part of Inventor. Go to **Draw** → **Rectangle** and then create a rectangle. Start at origin which is marked by an orange dot. The dimension is not important at this stage.

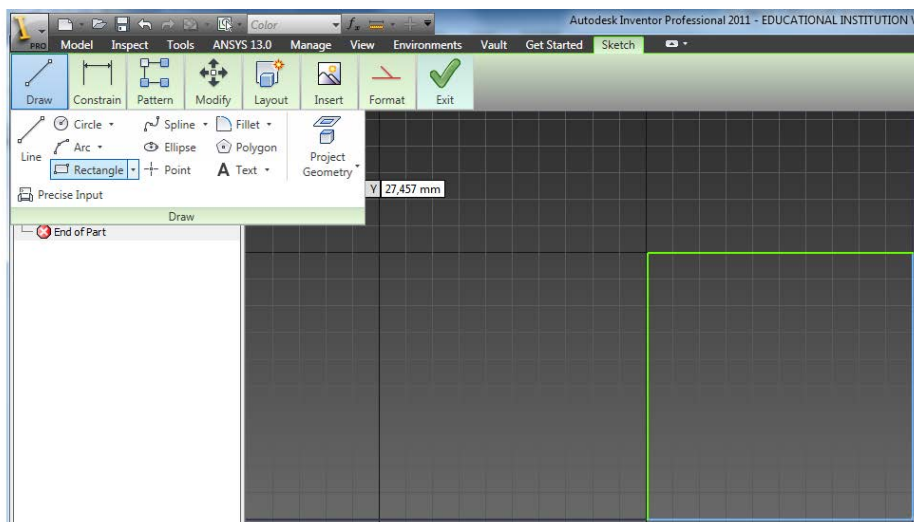


Figure 3 - Draw rectangle

### 3.3.2 Add dimensional constraints to the rectangle

When adding dimensions to a sketch, one will create dimensional constraints at the same time. Press the **Dimension** button and then press one of the lines. Write 4000 mm and press ok. Repeat for the same lines as in figure 4.

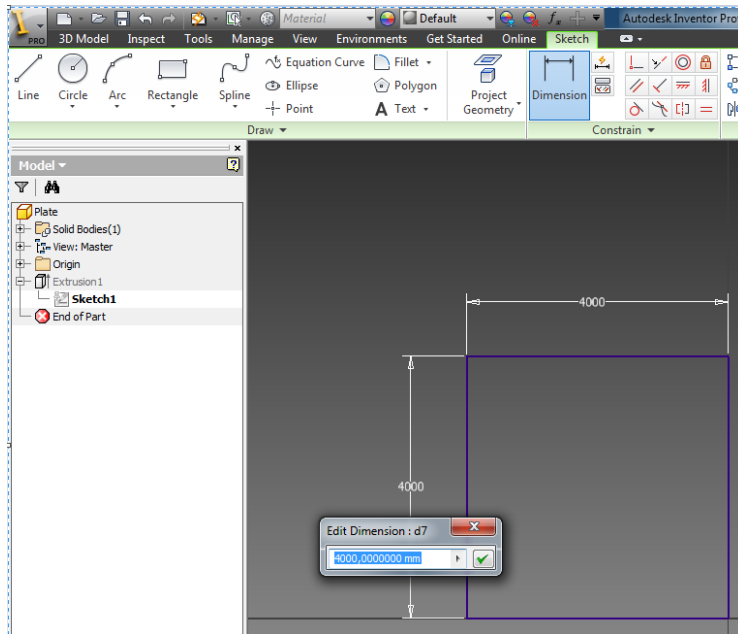


Figure 4 - Add Dimensions

Notice in the bottom right corner how the sketch has changed to fully constrained. When the sketch is fully constrained, right click in the sketching window and press **Finish 2D Sketch** or press **Finish Sketch** in the quick tool bar.

### 3.3.3 Extrude the sketch

After the sketch is finished, press **Extrude** and choose the rectangle sketch. Use default settings and change the value to 10 mm. The part is now a 3D volume part. Save the part before making a new part.

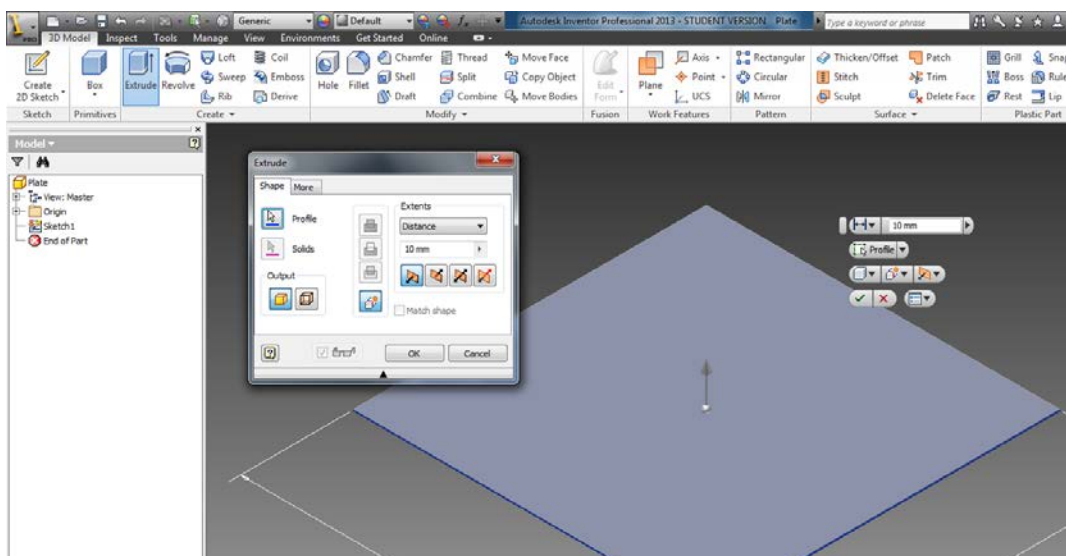


Figure 5 - Extrude

### 3.4 Create a T-Girder

Create a new part by following the steps in chapter 3.2. Select **PRO** → **NEW** → **PART**

#### 3.4.1 Draw a 2D sketch

Go to **Draw** --> **Line**. Start in origin marked by orange dot and make a sketch as shown in figure 6.

Notice how the lines will snap to 90/180 degrees angles and other lines in the sketch. The dimensions are not important yet.

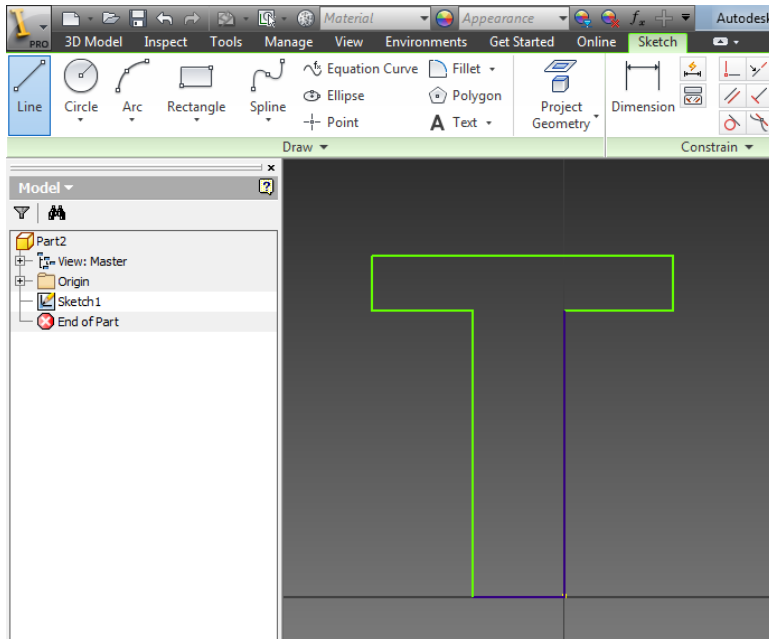


Figure 6 - Sketch of T-Girder

#### 3.4.2 Add dimensional constraints to the T-Girder

Press **Dimension** and add the dimensions which are shown in figure 7. The web is 120x7 mm and the flange is 8x70 mm.

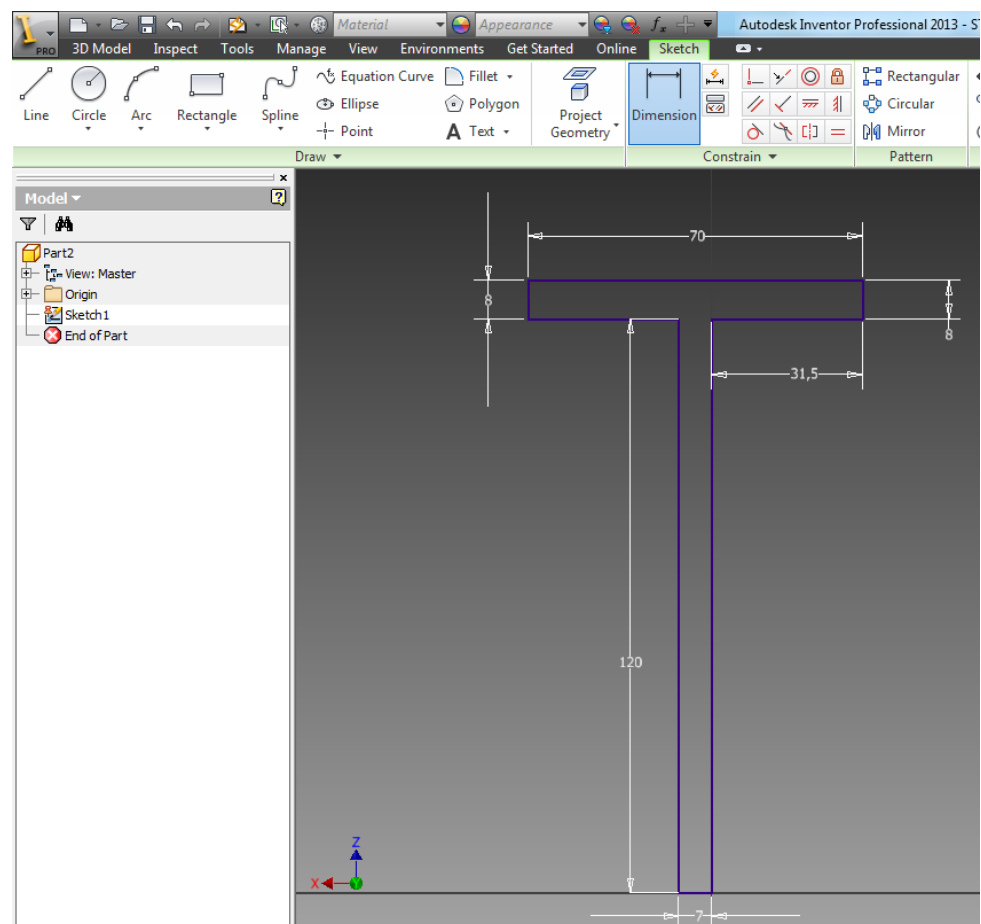


Figure 7 - Dimension T-Girder

### 3.4.3 Extrude the sketch

Extrude the sketch. Go to **3D Model** --> **Create** --> **Extrude**. Choose the sketch. Use default settings but change the distance to 4000 mm. Save the part before making the next part.

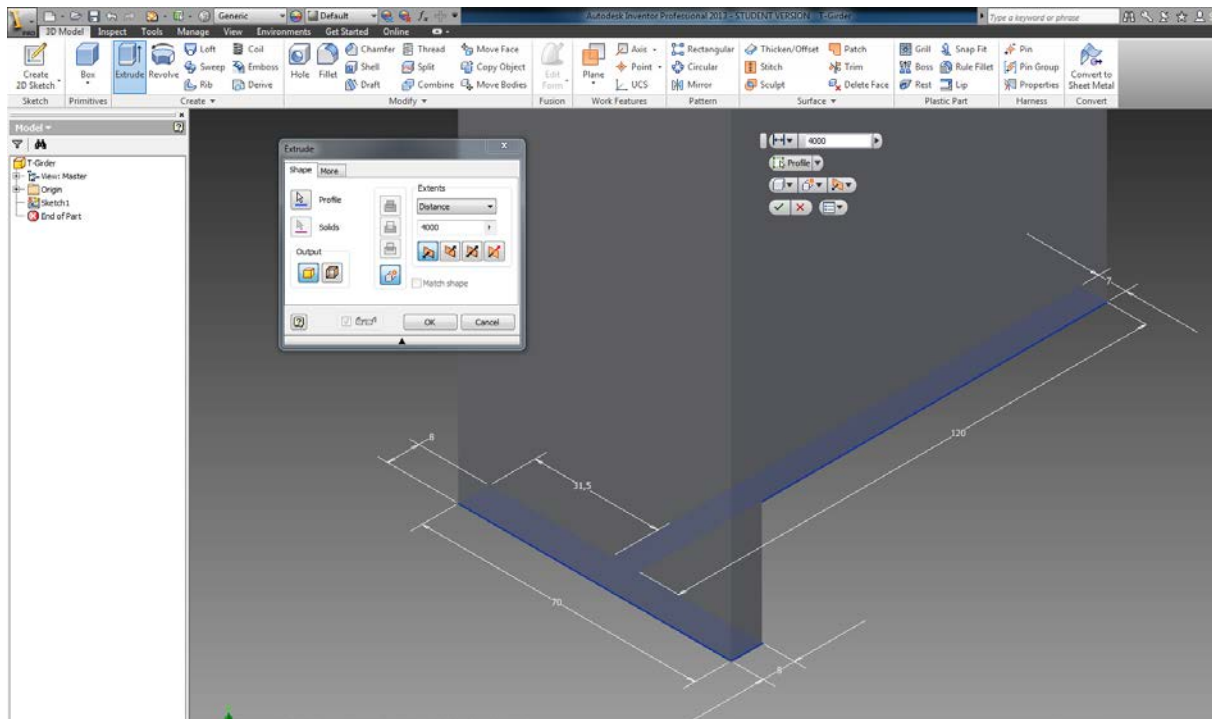


Figure 8 - Extrude T-Girder

## 3.5 Create a Bulb Flat Stringer

Create a new part by following the steps in chapter 3.2. Select **PRO** → **NEW** → **PART**

### 3.5.1 Draw a 2D sketch

Start by drawing the lines as shown in figure 9. The dimensions should be 100 mm, 80mm and 25 mm.

Insert an arc by pressing **Arc** --> **Center Point**

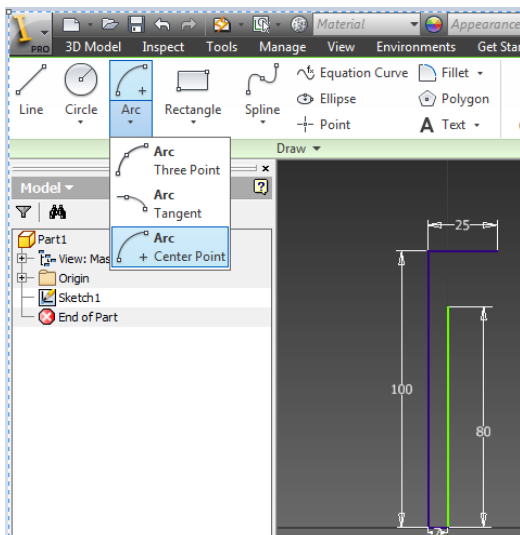


Figure 10 - Draw Stringer

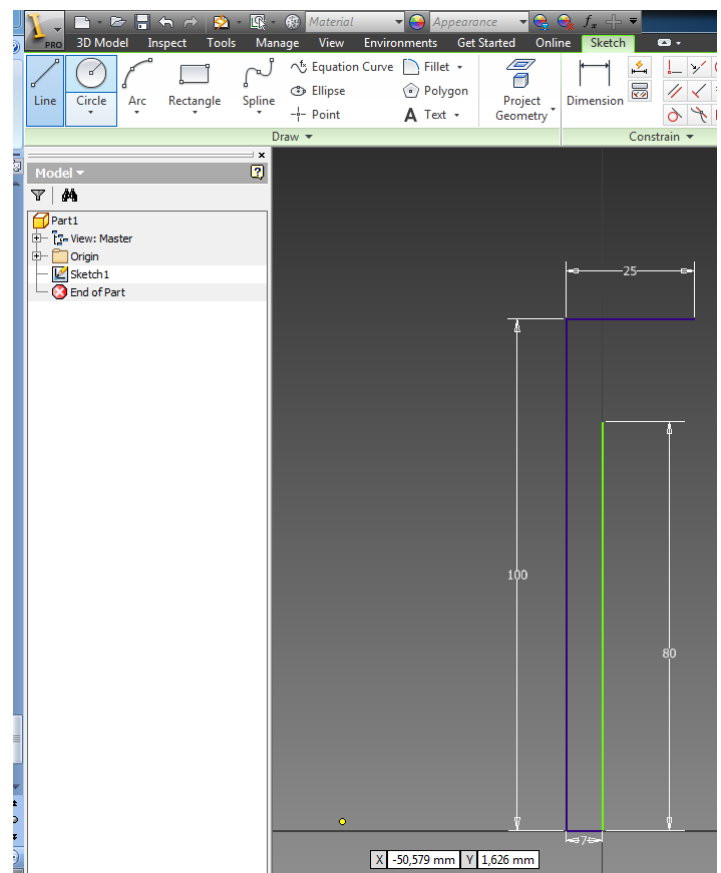


Figure 9 - Draw Stringer



Snap the mouse pointer to the green dot by moving the mouse to the end of the line shown in figure 11. Do not press the mouse buttons, but press TAB two times to change to the y-parameter. Give the parameter the value 100-4,5 and press ENTER.

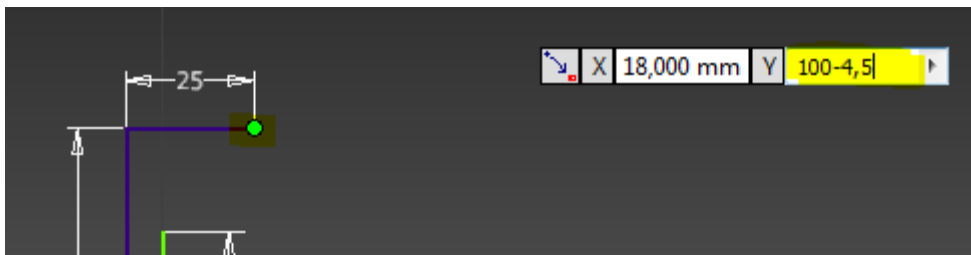


Figure 11 - Draw Stringer

The dot will now move down 4,5 millimeters. This is now the origin of the arc. By clicking at the end of the line as in figure 11 and press LEFT MOUSEBUTTON, you will have set the radius of the arc. Type in 150 degrees and press okay as shown in figure 12.

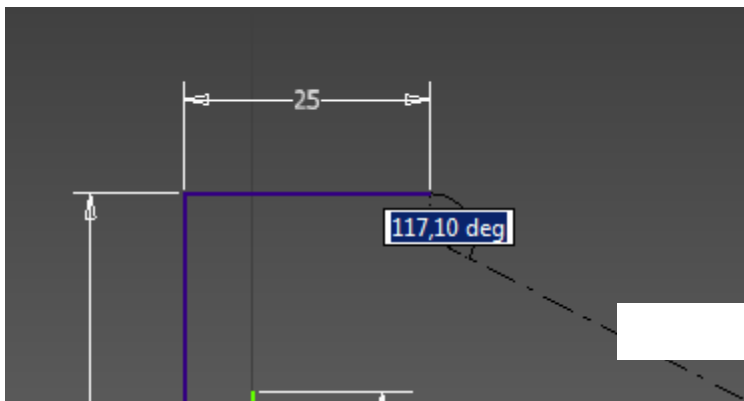


Figure 12 - Draw Arc

Draw a line which continues the 150 degree arc, and make sure it crosses the vertical line as shown in figure 13. Make the vertical line longer by clicking on the dimension for the line. Type in 90 as the new value.

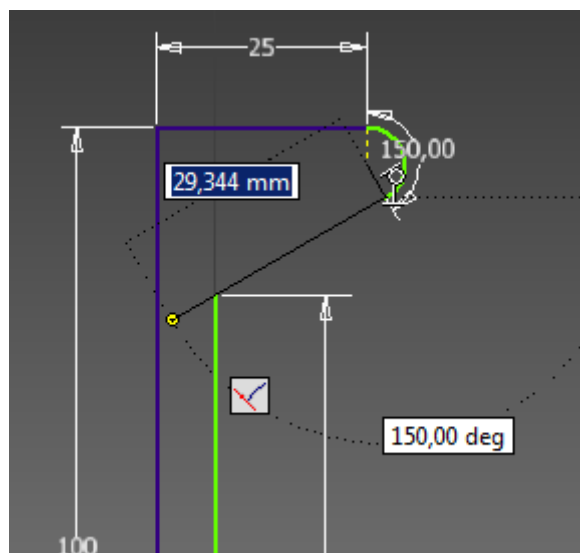
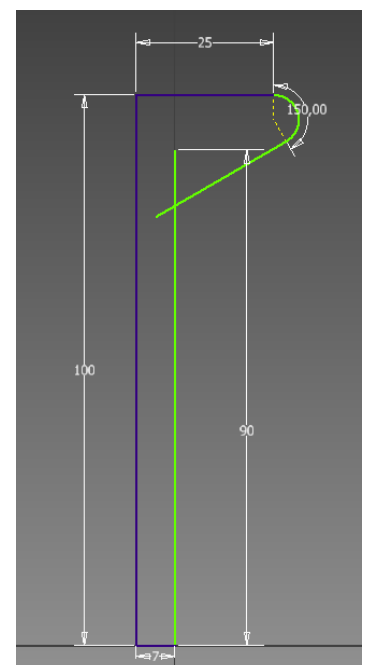


Figure 13 - Draw Arc



Before trimming the lines, the 90 mm dimensional constraint has to be removed. Try to trim with the constraint and see what happens. Trim the intersecting lines which are too long. Press **Trim** and the two lines as shown in figure 14.

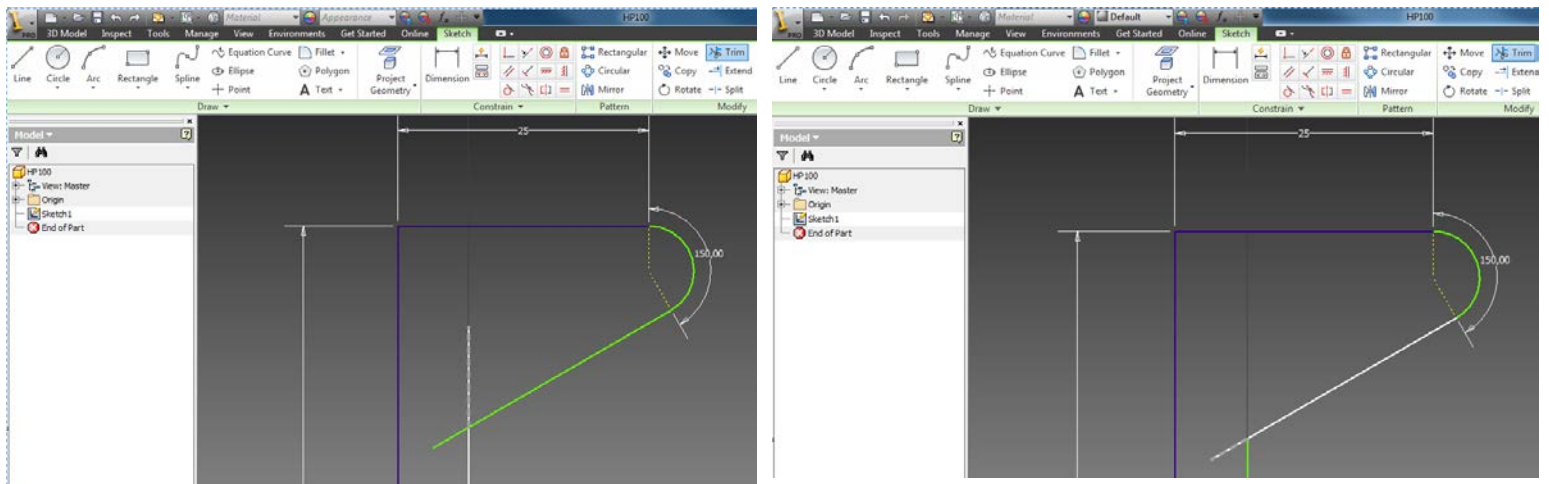


Figure 14 - Trim Lines

### 3.5.2 Add dimensional constraints

Add the dimensions which are shown in figure 15. The angular dimension is made by pressing **DIMENSION** and then the two lines where you want to add the angle. Press **Finnish Sketch**

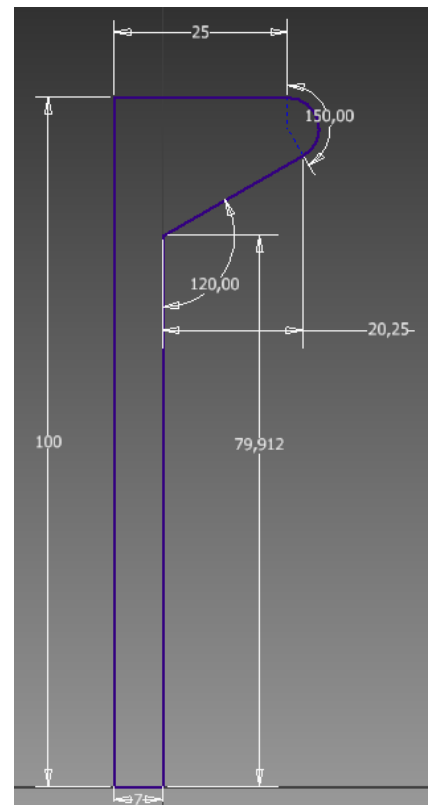


Figure 15 - Constraints

### 3.5.3 Extrude the sketch

Extrude the sketch to 4000 mm.

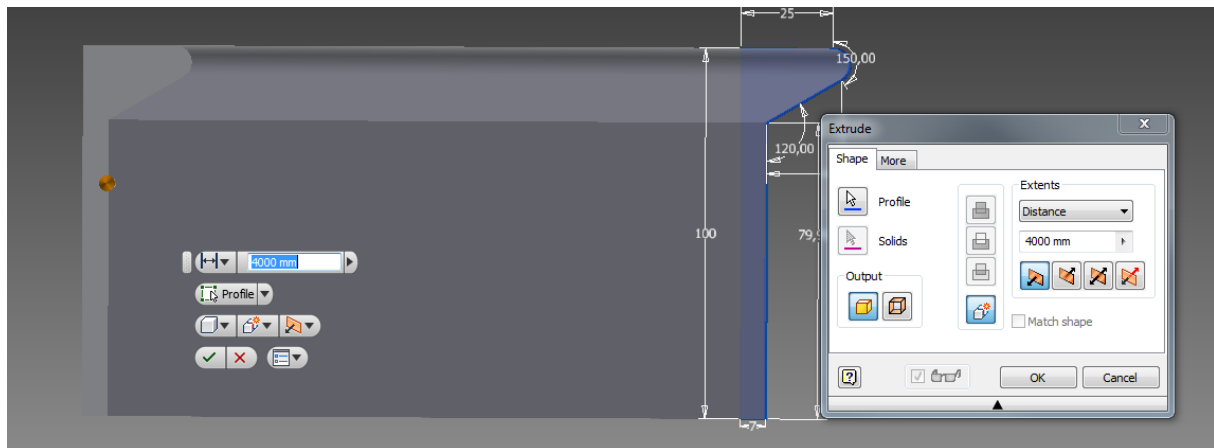


Figure 16 - Extrude Stringer

### 3.6 Make an assembly

To make a new assembly press **PRO** → **NEW** → **ASSEMBLY**

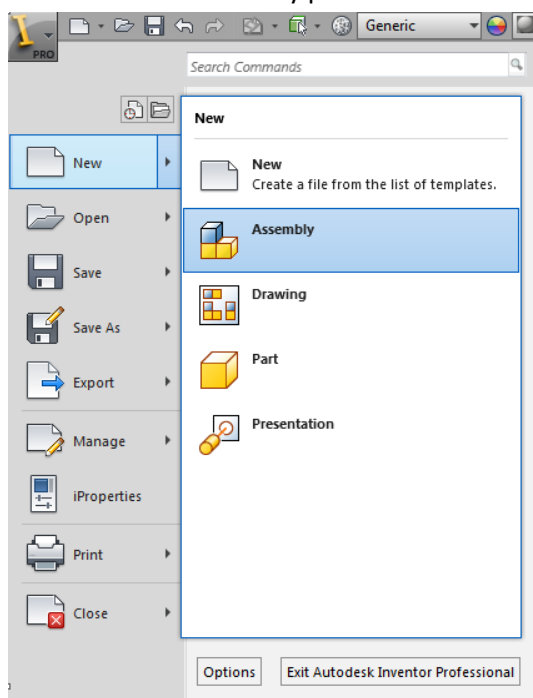


Figure 17 - New Assembly

### 3.6.1 Place Components

In an assembly one can place several parts and combine them to an assembly. Press the **Place** button and chose the plate. Press open to open the plate in the assembly. Do the same for the T-Girder. You will now have two parts which are not connected in any way.

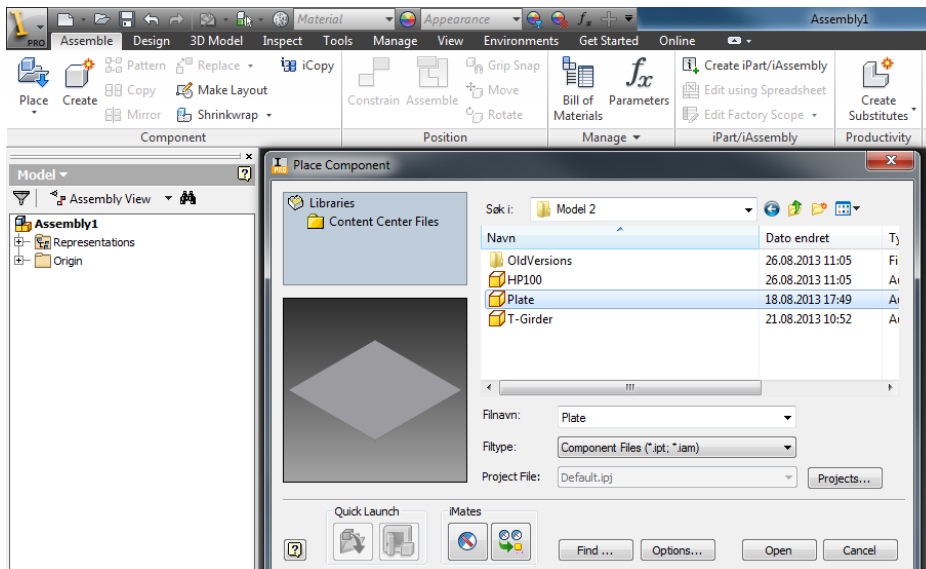
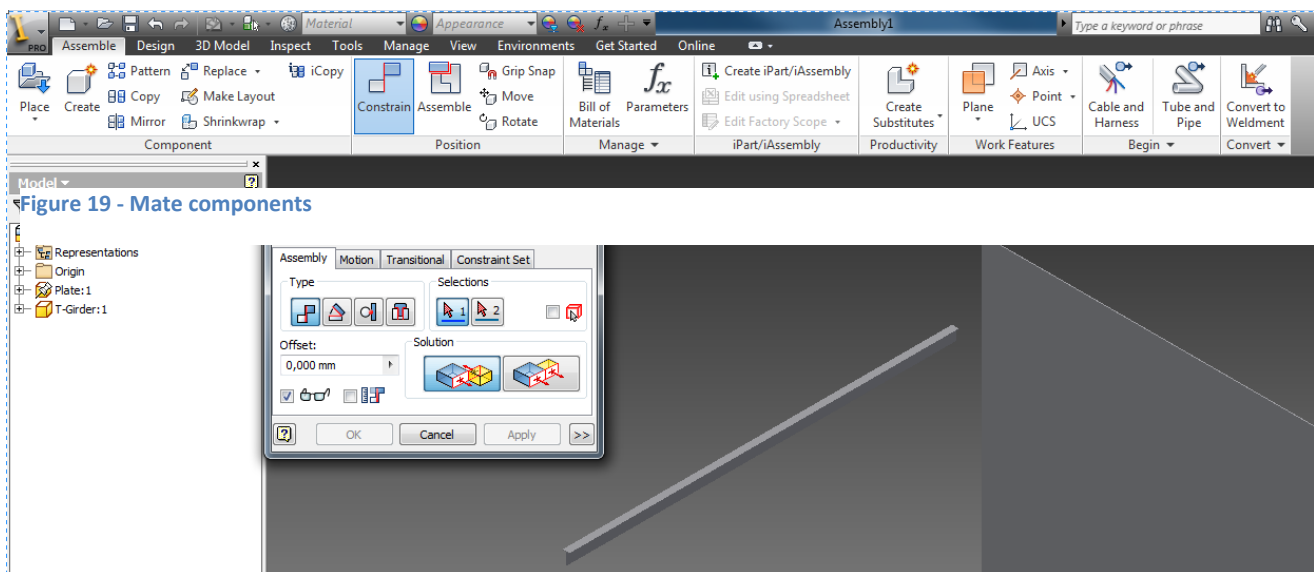


Figure 18 - Place components

### 3.6.2 Constrain parts in an assembly

There are several ways to constrain different parts together. Constraints can be used to lock parts together. There are possibilities to give angular constraints between parts and make parts tangent to each other. It is recommended to play a little with the constraints to see the behavior of the different constraints.

This guide will introduce the mate function. Press the **Constrain** button. Mate is the default option when choosing constrain.



First choose the underside of the web and then chose the plate. The T-Girder will now snap to the plate. See what happens if you type a value in the offset option. Set the offset to 0 before pressing

**Apply** to save this constraint. The bottom of the web is now mated with the plate and will stay this way unless the constraint is deleted or edited. See figure 20.

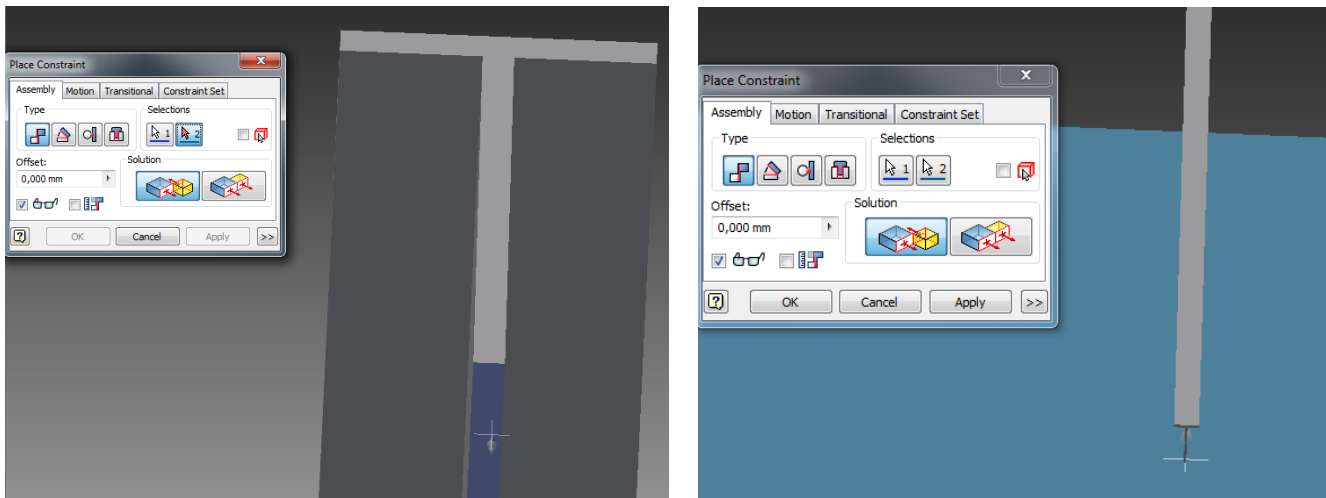
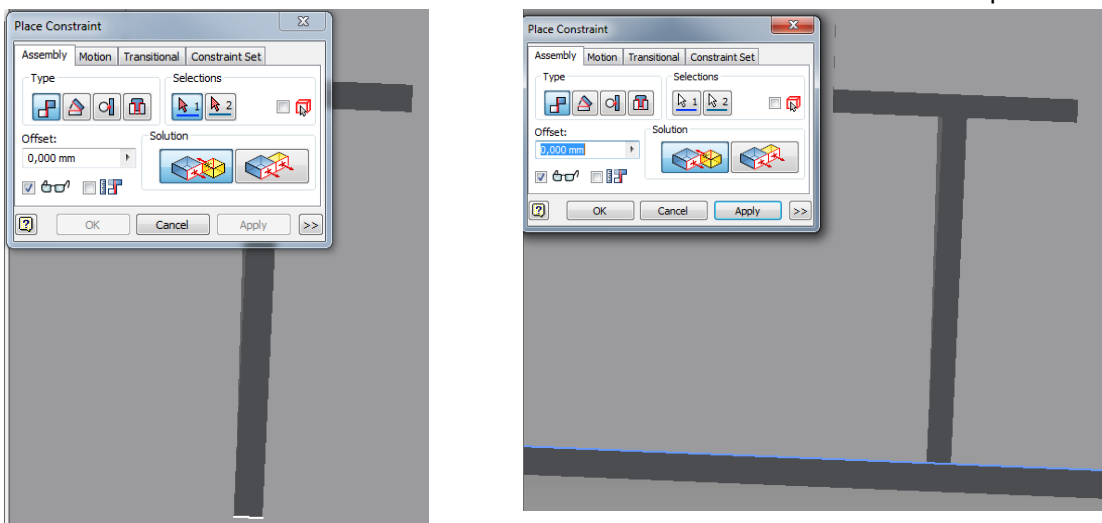


Figure 20 - Mate Components

The girder should now be placed at the correct position at the plate. It should be within the lines of the plate. Since the plate is a square, it does not matter which line we constrain the web against. Press the line at the bottom of the web and then one of the lines at the plate. See figure 21.



Figur 21 - Mate Components

The girder should be placed at the center of the plate. By mating the bottom length web line to the plate side and type 1996,5 mm in the offset box the girder will be at the center of the plate.

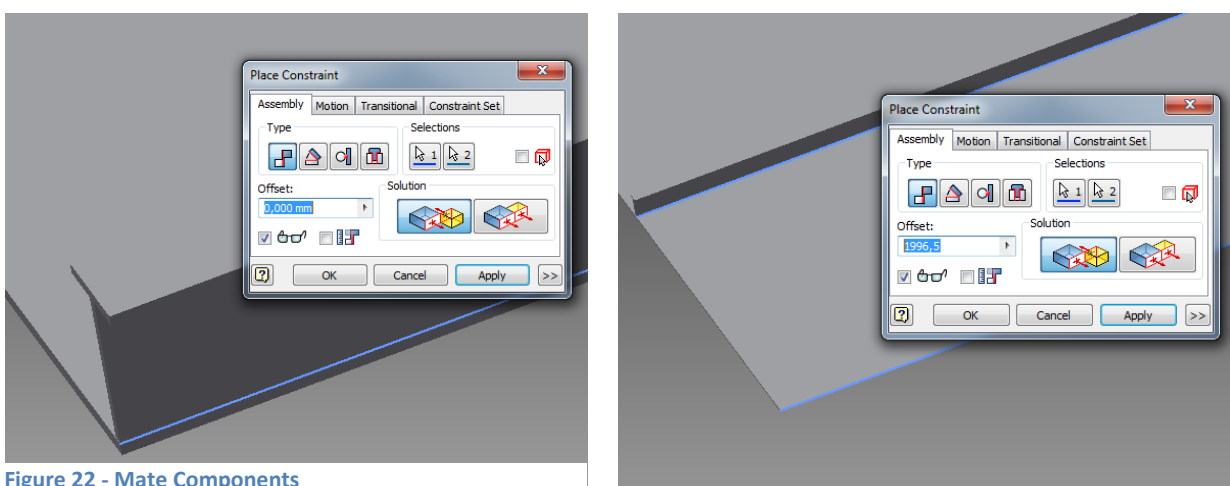


Figure 22 - Mate Components

### 3.6.3 Place the bulb flat stringer

The procedure for the Bulb-Flat stringer is the same as for the T-Girder. Follow the steps above when inserting the stringer. The difference is that the stringer should be orthogonal to the T-Girder. The model should now look like the model in figure 22.

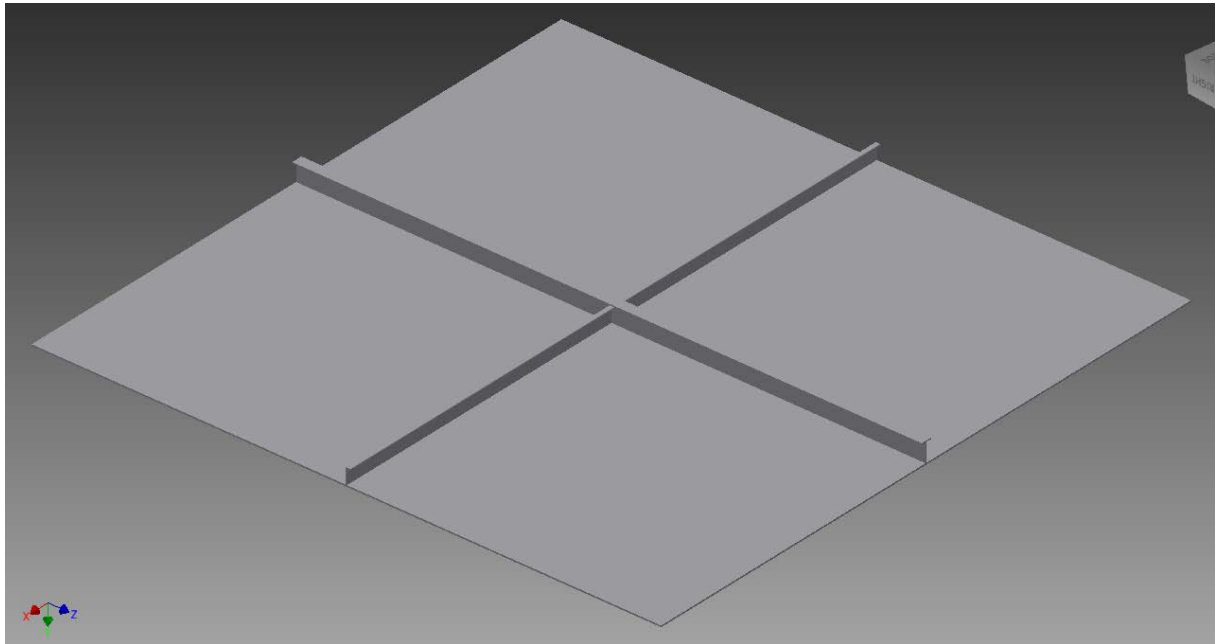


Figure 23 - 3D volume model.

## 4 Creating an analysis model

On the student computers at Tyholt there is installed an integrated Inventor and ANSYS software. This will make the process of converting the 3D model to a surface model much easier.

### 4.1 Exporting the model to Workbench

When the model is finished, it is possible to export the model directly from Inventor to ANSYS workbench. On the quick toolbar, locate **ANSYS → WORKBENCH**.

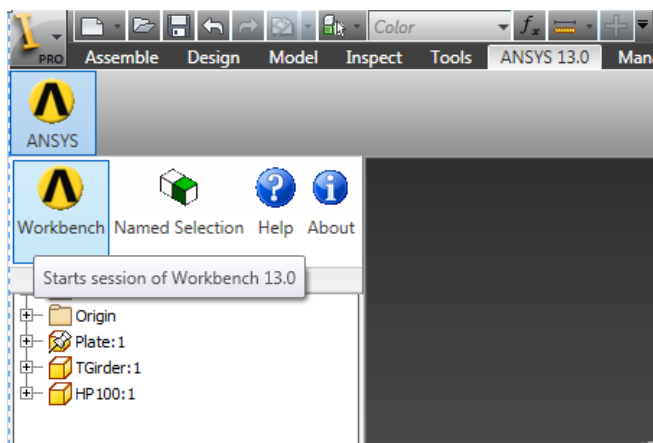


Figure 24 - Exporting to ANSYS

The data from the model will now be transferred to ANSYS Workbench. The window should look like figure 25.

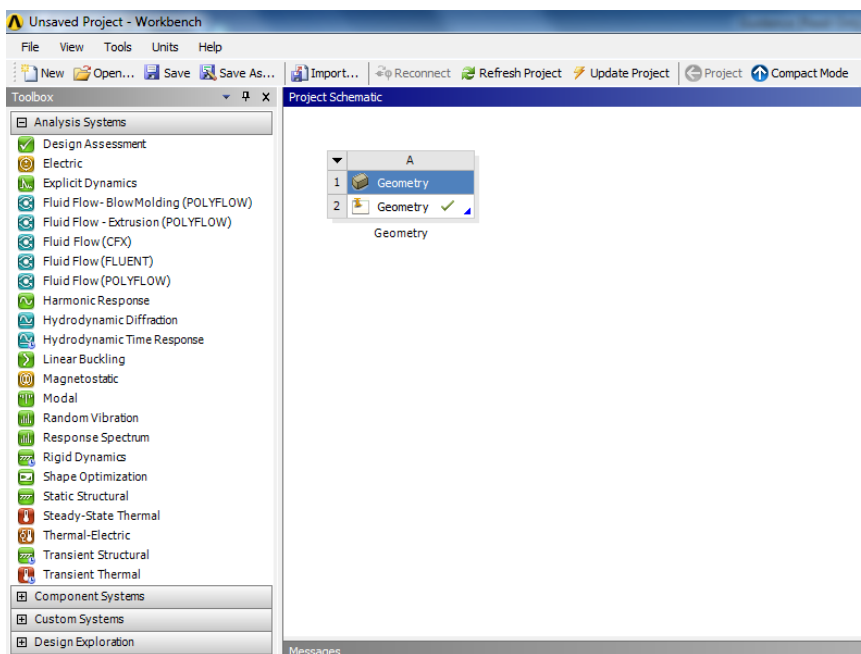


Figure 25 - Workbench

## 4.2 Export the model to DesignModeler

By double clicking on the geometry the model will be opened in DesignModeler where it is possible to alter the geometry of the figure. When the model is exported to DesignModeler, right click on **Attach** and press **Generate** like in figure 26.

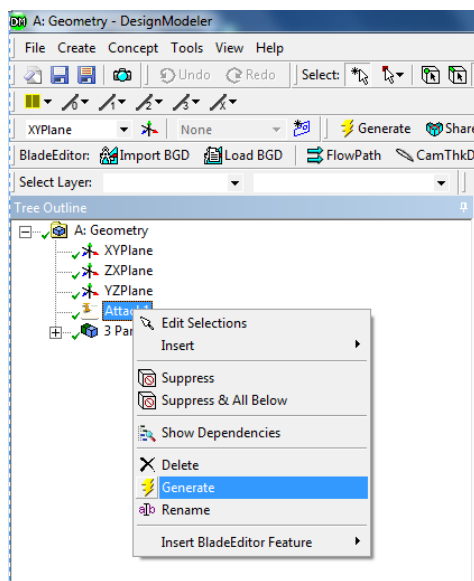


Figure 26 - Generate Model

## 4.3 Make a Mid-Surface Model

When the model is generated, it is possible to use the automatic function for generating the surface model. To analyze a model with plate theory using shell and plate elements, the model has to be converted from a volume model to a surface model. Press **Tools** → **Mid-Surfaces** which is shown in figure 20.

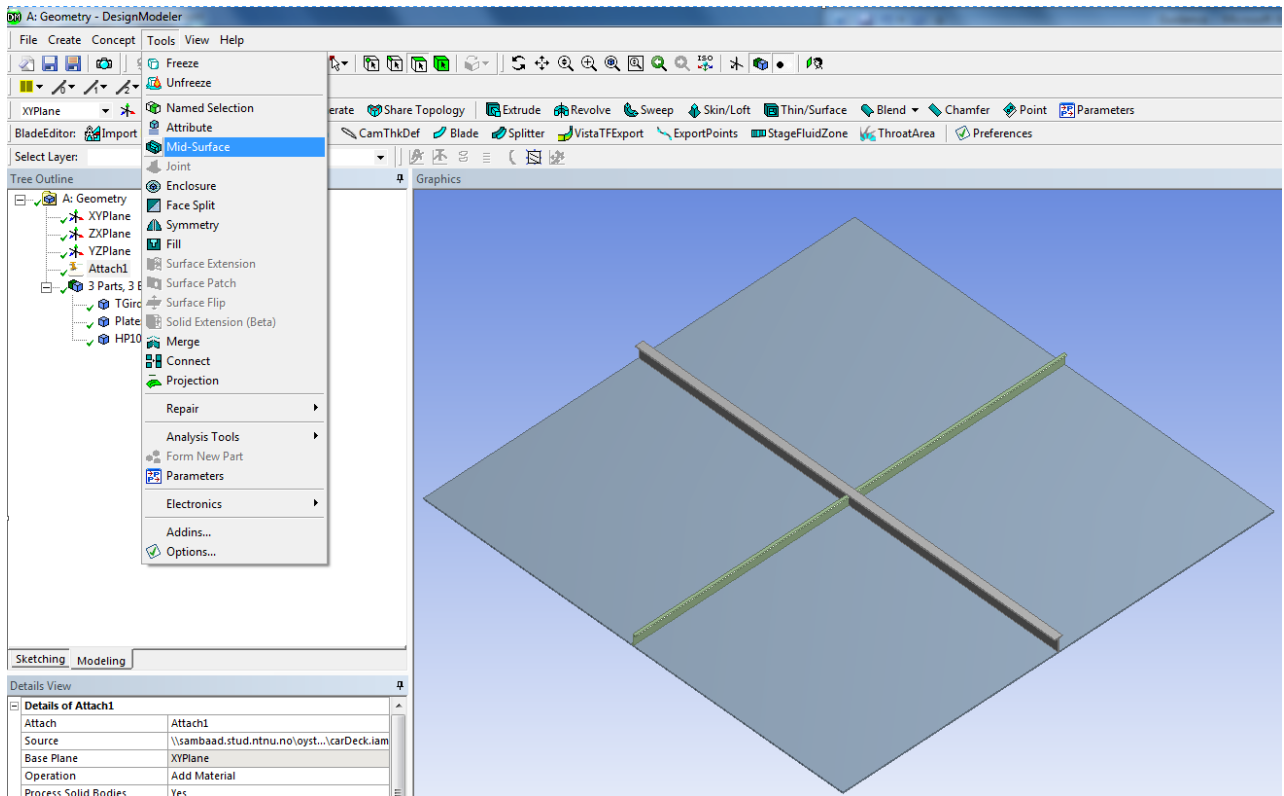


Figure 27 - Mid-Surface

In the lower left corner, a menu will appear for Mid-Surface. In the Selection Method, choose Automatic. Change the Minimum Threshold to 0,006 m and the Maximum Threshold to 0,01 m. Change the FD1, Thickness Tolerance to 0,006m. The minimum threshold defines the minimum distance between two pairing faces, while the maximum defines maximum distance between two pairing faces. The thickness tolerance defines maximum thickness. If there are any faces on a body with larger thickness, they will not be included. It can be dangerous to use the automatic selection method, but it will do well here. Try different parameters and see which faces are selected. After the threshold is set, choose **Yes – Replace Face Pairs**.

Details View		Details View	
<b>Details of MidSurf1</b>		<b>Details of MidSurf1</b>	
Mid-Surface	MidSurf1	Mid-Surface	MidSurf1
Face Pairs	0	Face Pairs	5
Selection Method	Manual	Selection Method	Automatic
<input type="checkbox"/> FD1, Thickness Tolerance (>=0)	Manual	Bodies To Search	Visible Bodies
<input type="checkbox"/> FD2, Sewing Tolerance (>=0)	Automatic	Minimum Threshold	0,006 m
Extra Trimming	Intersect Untrimmed with Body	Maximum Threshold	0,01 m
Preserve Bodies?	No	Find Face Pairs Now?	No
		<input type="checkbox"/> FD1, Thickness Tolerance (>=0)	No
		<input type="checkbox"/> FD2, Sewing Tolerance (>=0)	Yes - Add to Face Pairs
		Extra Trimming	Yes - Replace Face Pairs
		Preserve Bodies?	Intersect Untrimmed with Body
			No

Figure 28 - Automatic Mid-Surface



Generate the new Mid-Surface Model by right clicking **MidSurf1** → **Generate**

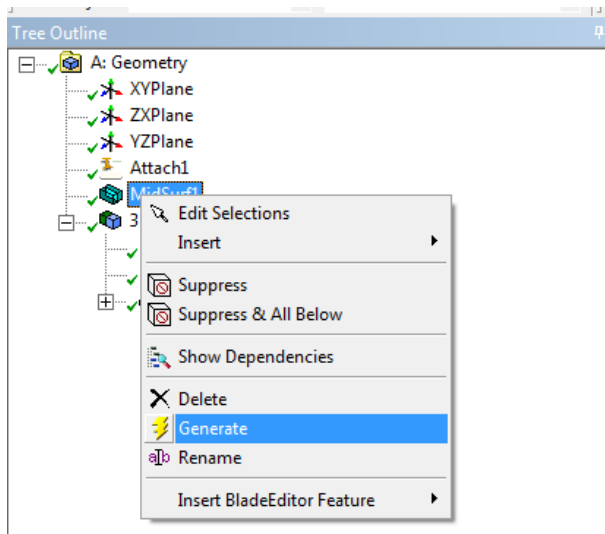


Figure 29 - Generate Mid-Surface Model

#### 4.4 Make joints in surface model

The surface model contains intersecting lines which are not marked. For the meshing to be correct, we need some more lines in the model. The red square in figure 30, marks where we need to insert a joint. Press **Tools** → **Joint**.

The reason for this joint is that the model needs lines to define areas. Without this joint line there are no lines to separate the area of the girder and the stringer. More adjustments need to be made in ANSYS Mechanical to have correct areas. This will be described later.

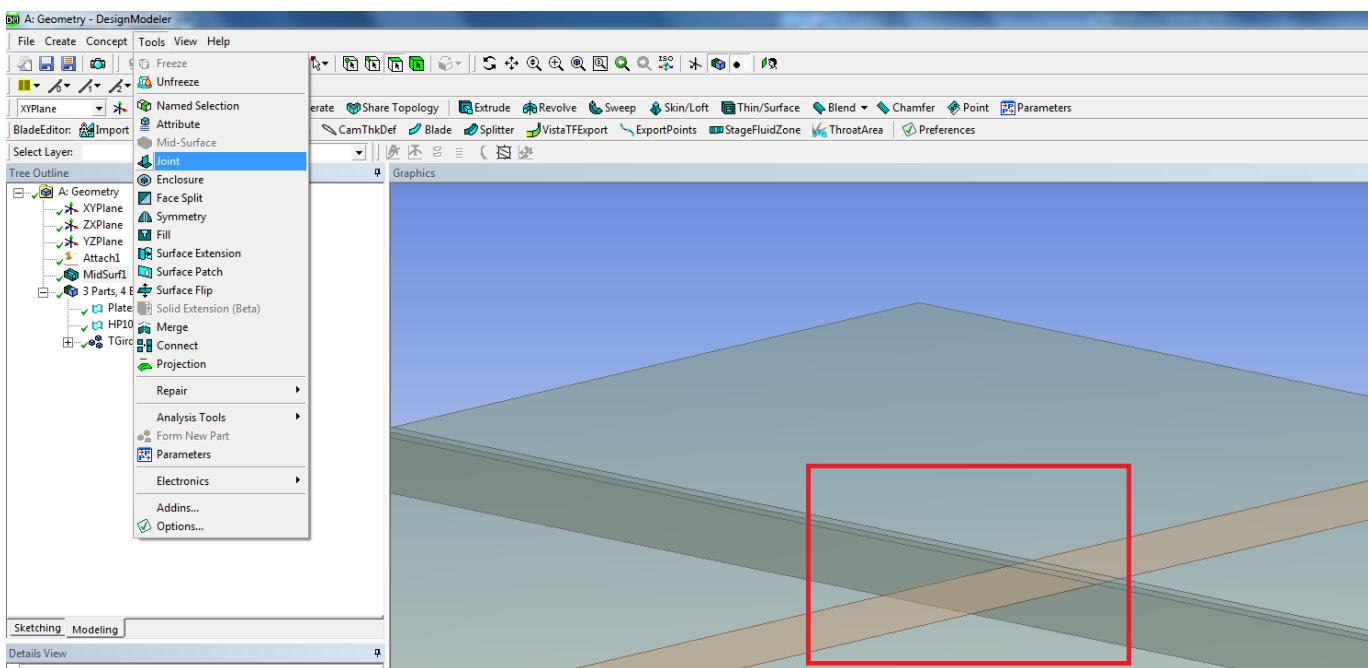


Figure 30 - Make Joint

Select the T-Girder and the Bulb-Flat Stringer and press **Apply** in **Target Geometry**.

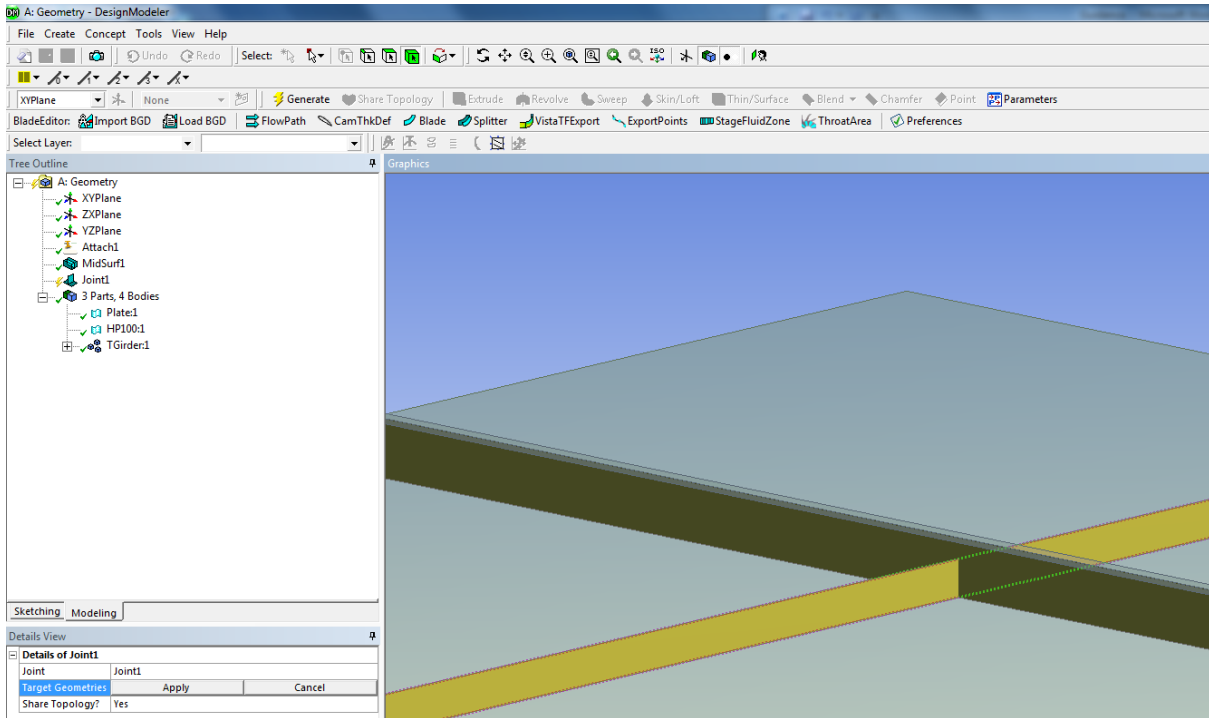
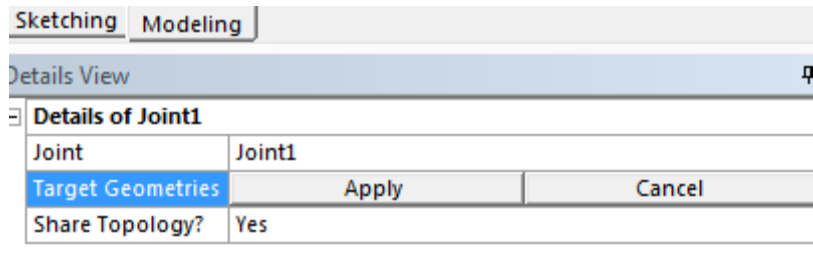


Figure 31 - Make Joint

The final step is to generate the model. Right click on **Joint1** → **Generate**. The joint line has now appeared in the intersection between the girder and the stringer. The model should now be a complete surface model which is ready to be imported to ANSYS Mechanical.

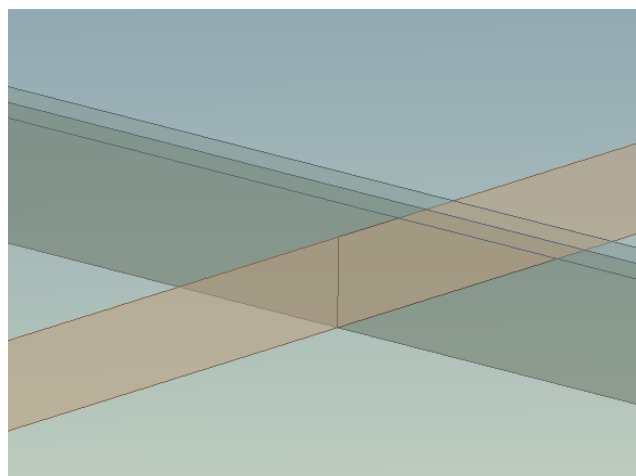
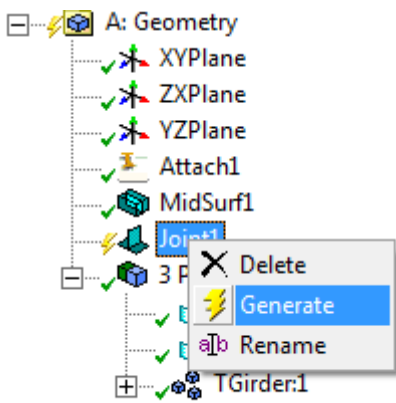


Figure 32 - Joint

## 4.4 Save the mode

Press **File** → **Export** to export the file.

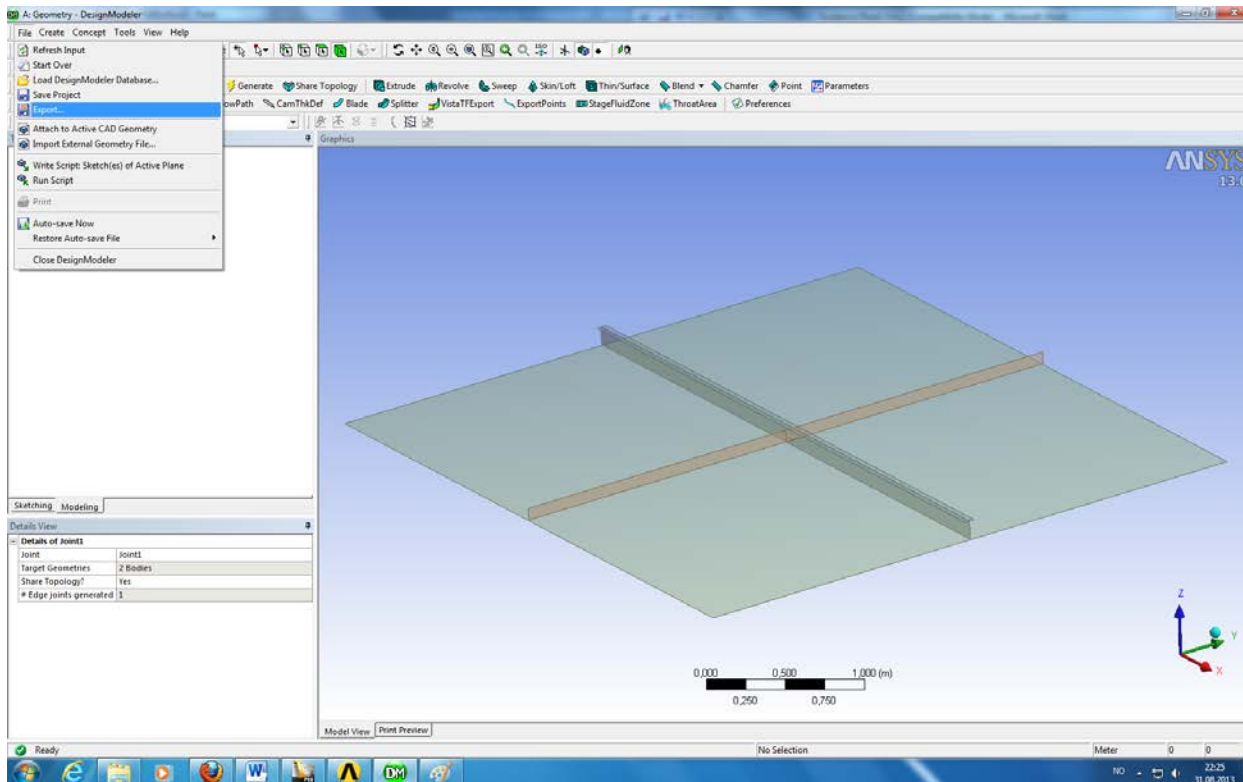


Figure 33 - Export model

The file should be saved as an IGES file. This format will include important information like areas, dimensions and lines in the model and will easily be imported to ANSYS Mechanical for meshing and analysis.

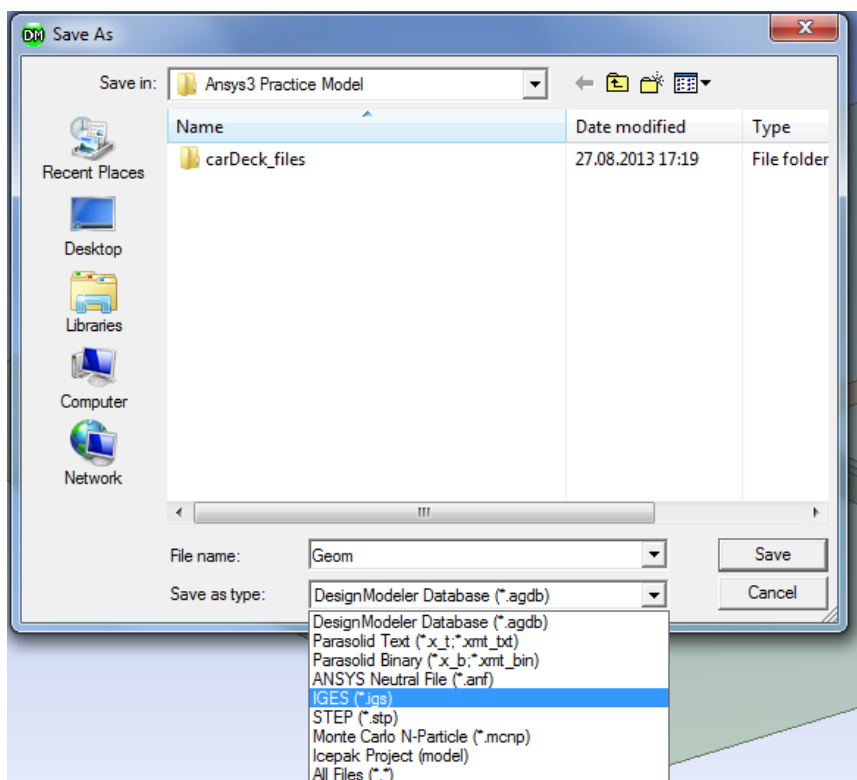


Figure 34 - Export Model