



Assignment 9

Inventor

Assembly Drawings

Objectives

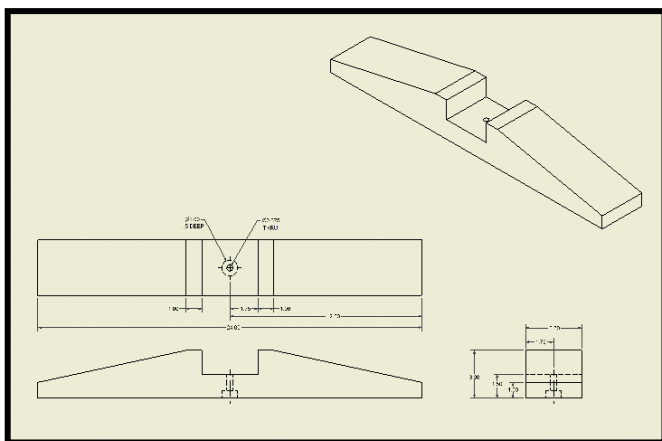
When you complete this assignment you will:

1. sketch and create models using basic commands.
2. create a multi-part model using the assembly drawings.
3. sketch and dimension problems.
3. extrude models to remove and add surfaces.
4. view and rotate the three-dimensional models.

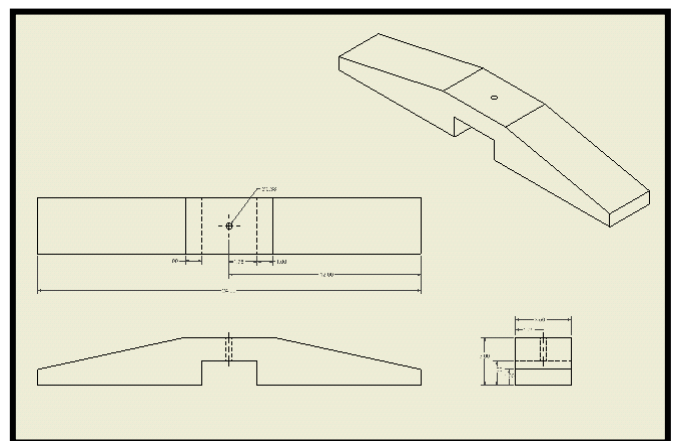
Getting Started

In this activity you will be introduced to the assembly feature of Inventor. This feature can get aggravating, but remember to be patient and if all else fails, click Undo!

1. Launch Inventor 5.3 and launch the Standard.ipt template file.
2. Create the following parts and save them as IN-31 and IN-32. The full size drawings can be found at the end of this section.



IN-31

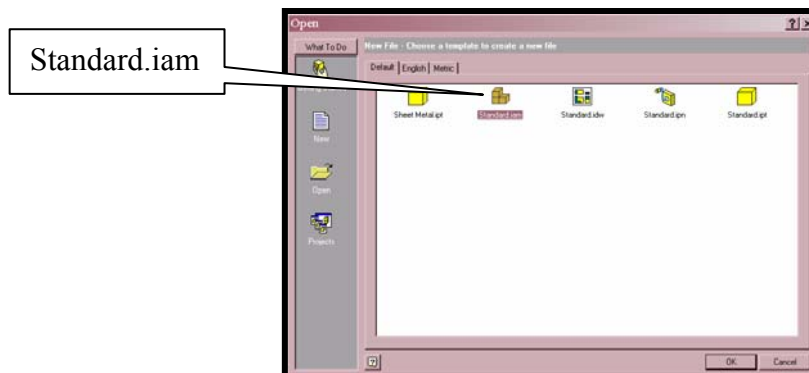


IN-32

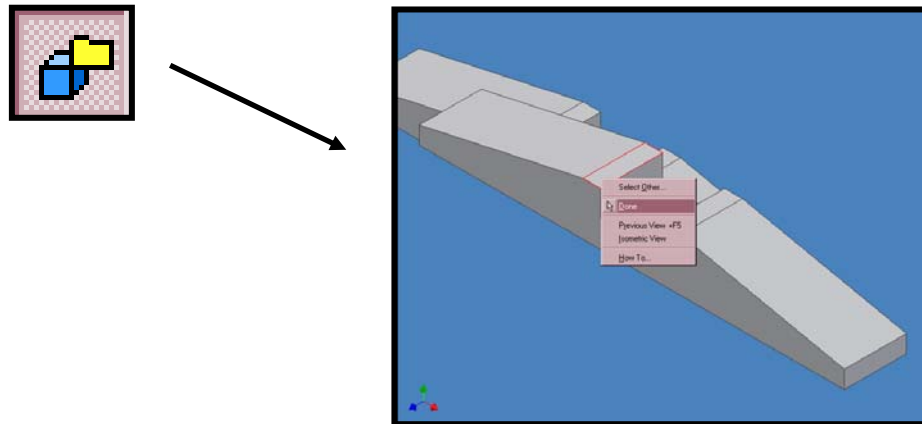


Placing Components

1. Once you have drawn the two parts as .ipt files, you can place them into an assembly drawing. Open a new drawing in Inventor using the **Standard.iam** template.

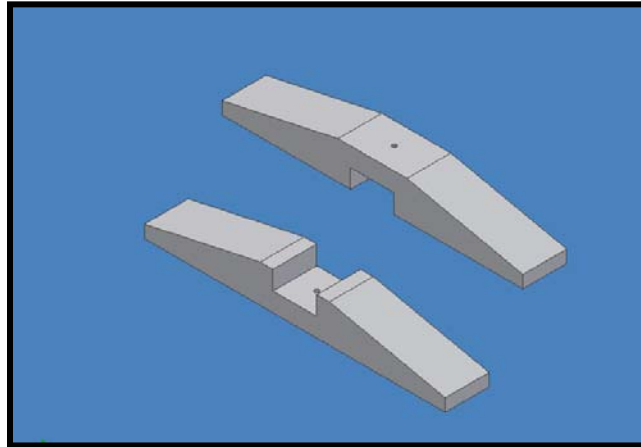


2. Select the **Place Component** tool from the Assembly toolbar. Browse to your network folder and select the IN-31 file you just created. Click open and notice that Inventor has imported that part into the assembly drawing you are working on now. **Right click** and select **done** to place the part.





- Repeat step 2, only this time place the **IN-32 file**. This time you will have to left **click** to **place the part's location**, then **right click** and select **done**. You will probably have to zoom out to make room for both parts.

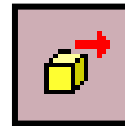


Adding Constraints

- Now that you have both parts ready to work with, it's time to **fit them together**. You may have to **rotate** and **move** the drawing and individual parts while adding **constraints**. The entire drawing can be moved the way you are used to, but individual parts have to be adjusted by using the **Rotate Component** and **Move Component** tools in the **Assembly toolbar**.

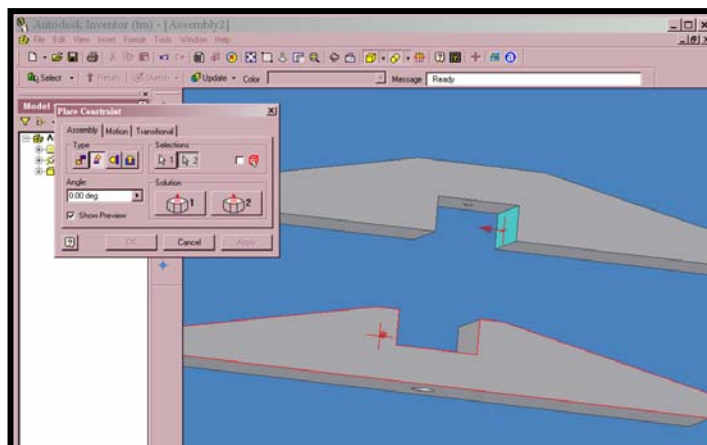


Rotate



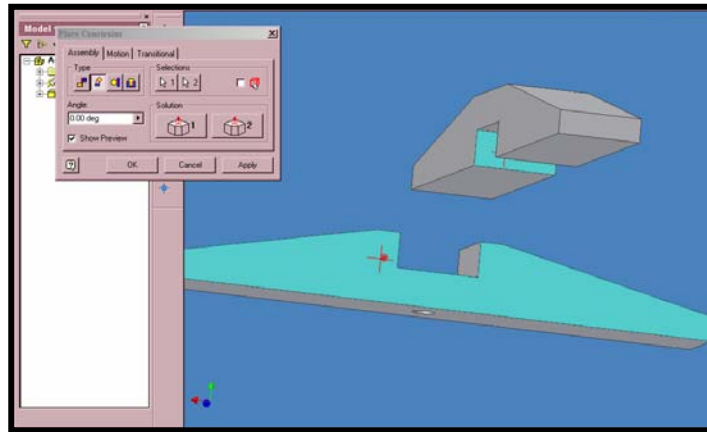
Move

- Rotate the entire drawing until you have a good view of the highlighted areas below. Click on the **Place Constraint** tool on the Assembly toolbar. Choose the **Angle** option and select the two highlighted faces from below.

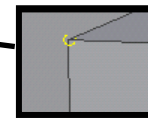
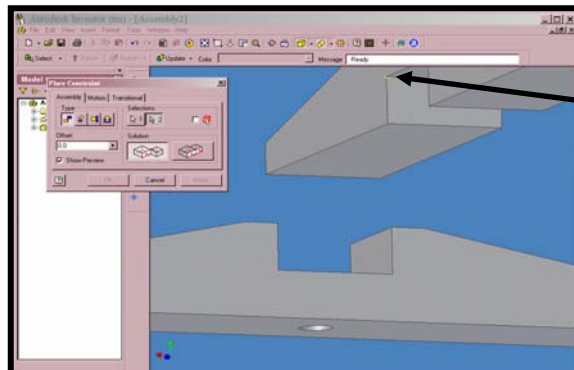




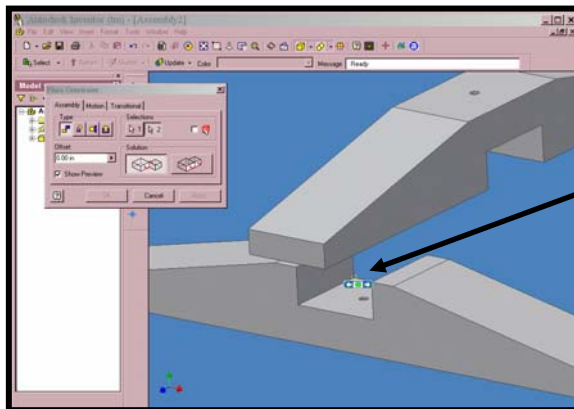
- Make sure the **angle** in the dialog box is set to **0.00 degrees** and click Apply. Notice the parts are positioned **perpendicular** to each other now.



- To finish constraining the two components, choose the **Mate** option in the Place Constraint dialog box and pick Point 1 below, then choose Point 2. You will probably have to **rotate** and **zoom** in to get the points accurately.



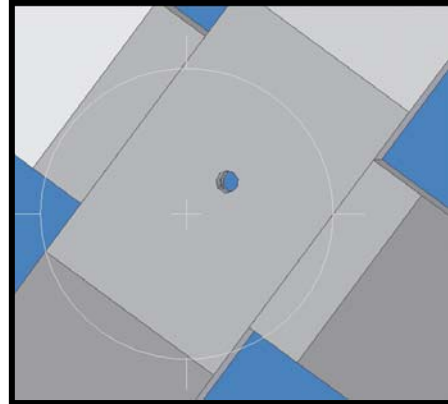
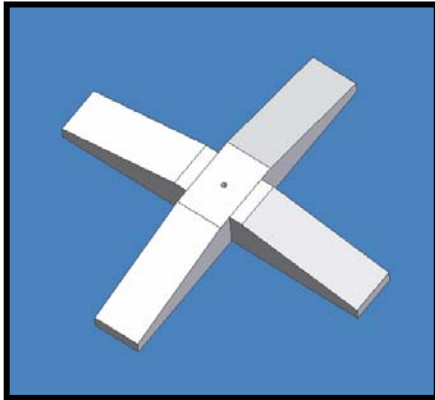
Point 1



Point 2



- Your final assembly should look like the one below. Rotate your model and make sure the surfaces mate correctly. Check to make sure the holes align as well.



Finishing Up

- Save the assembly in your server folder as IN-33.
- Notify your instructor that you have finished this assembly before you close it.

What Do You Know?

Load Assignment 9 of module 81.40 and answer the following questions:

- Which tool is used to insert a model into an assembly drawing?
 - Place Component
 - Place Part
 - Insert Part
 - Insert Component
- Which tool allows you to lock a part into place?
 - Lock
 - Place Constraint
 - Freeze
 - Place Component
- True or False? The Mate and Angle constraint tools perform the same function.
- The final assembly drawing should have been saved as:
 - IN-31
 - IN-32
 - IN-33
 - IN-34