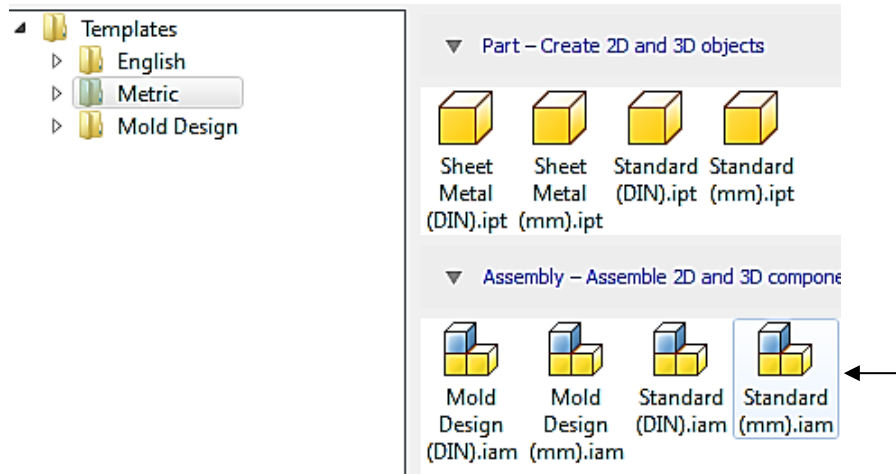
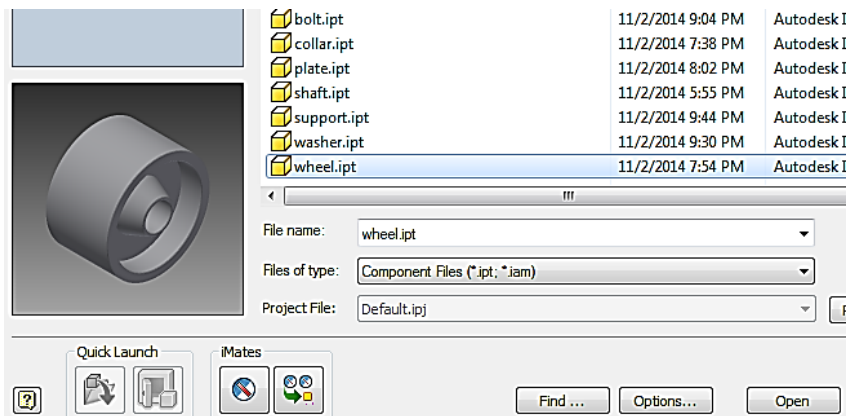
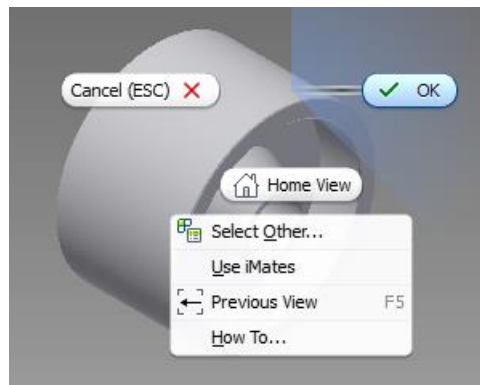
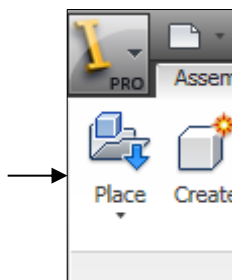


Part 2: Creation of the Belt System Assembly

The following section explains the basic necessary steps to create a static assembly in Autodesk Inventor. To begin, select **New > Metric > Standard(mm).iam**

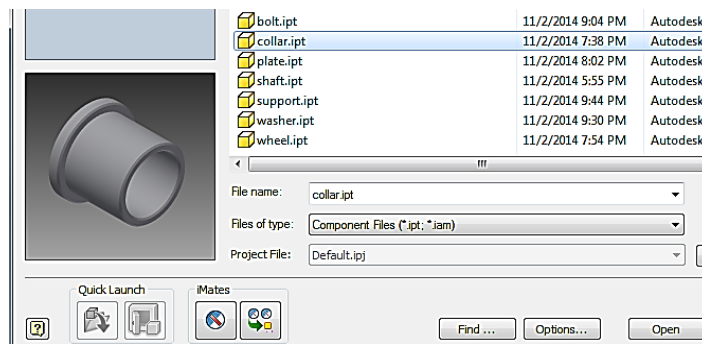


1. Select **Place** and right click to pick OK for inserting the Wheel component into the assembly.

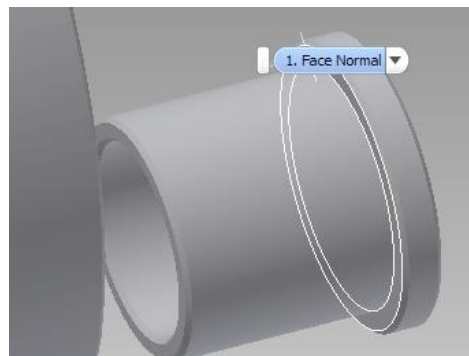
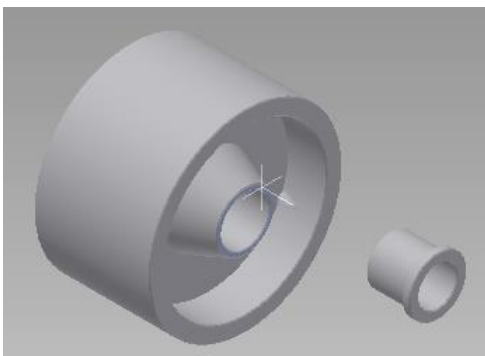
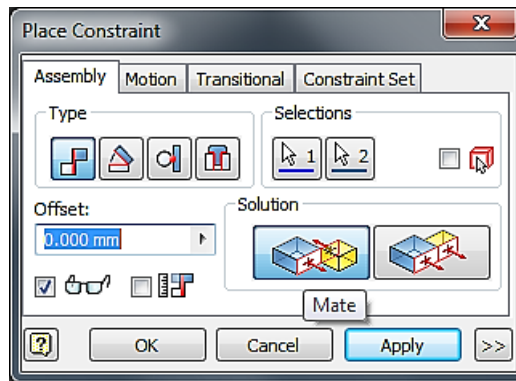
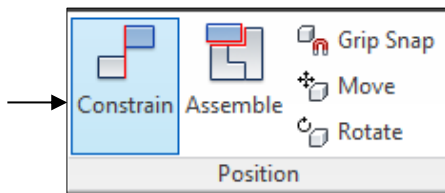


2. Next, insert the Sleeve component. Click **Place** > click on the Collar component > left click anywhere to place the component in the assembly. Click **Constraint**. Again, the Type is **Mate**. For Selection 1,

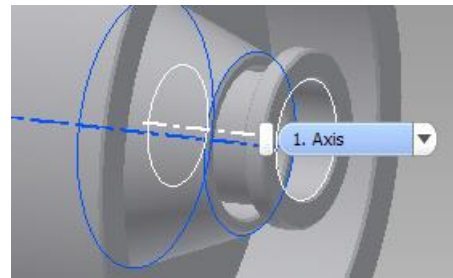
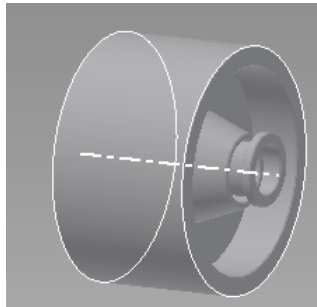
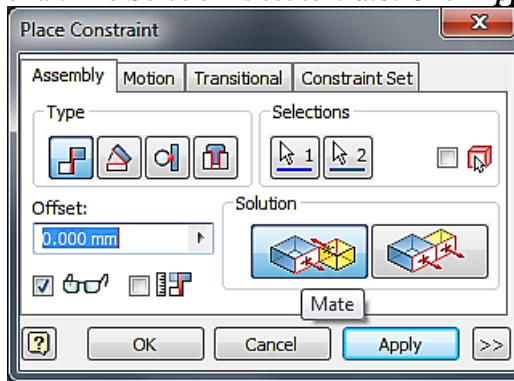
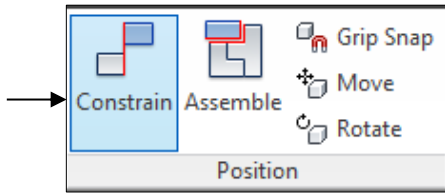
choose the right face of Wheel. For Selection 2, choose the left surface on Sleeve. The Solution is set to **Mate**. Click **Apply**.



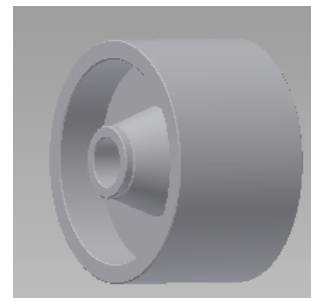
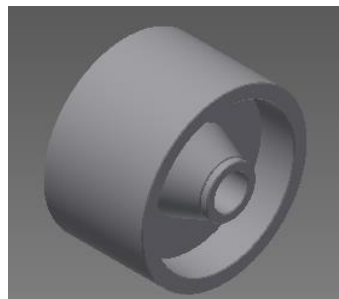
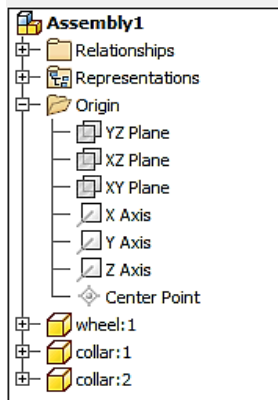
Click **Constraint**. Again, the Type is **Mate**. For Selection 1, choose the right face of Wheel. For Selection 2, choose the left surface on Collar. The Solution is set to **Mate**. Click **Apply**.



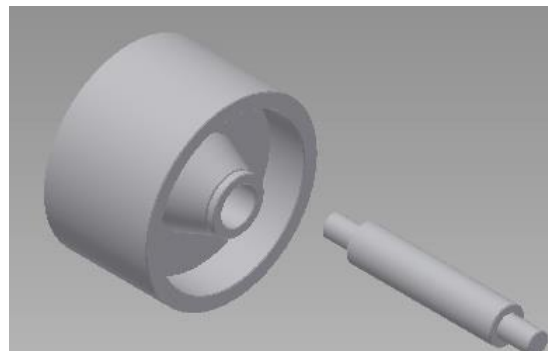
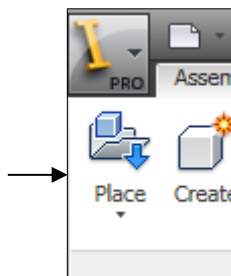
Constrain the axis of Wheel and the axis of Collar. The Solution is set to *Mate*. Click *Apply*.

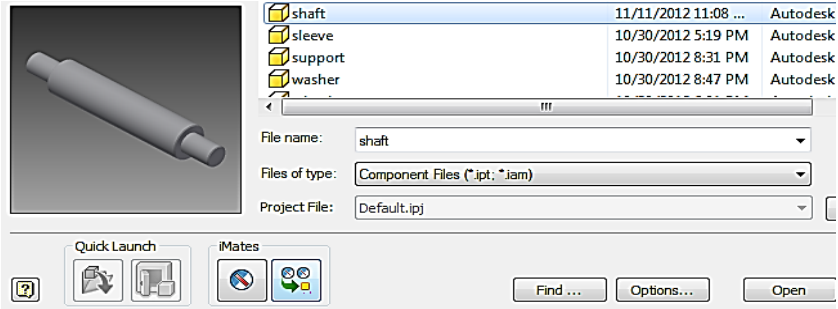


To insert the second Collar in the assembly, repeat the procedure described in the process of assembling the first Collar component.

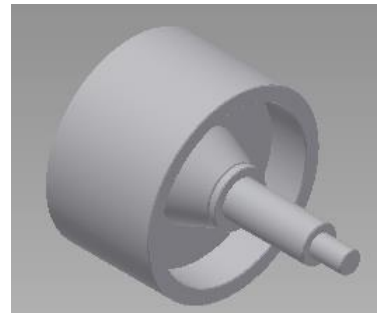
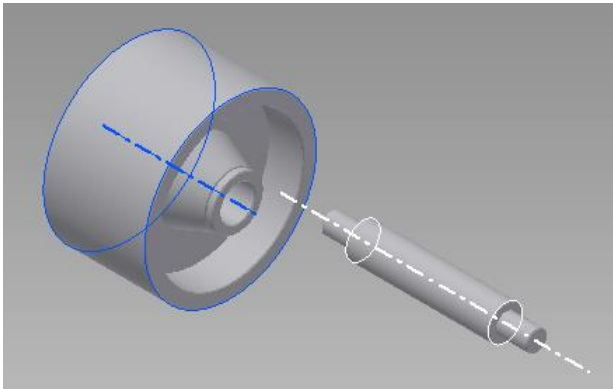
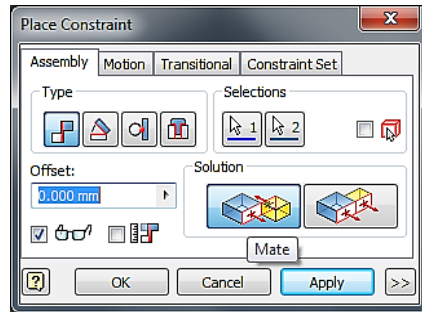
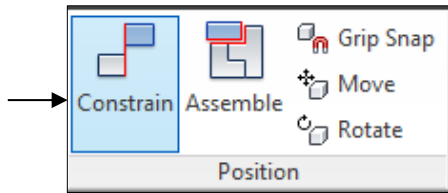


3. Click the icon of *Place*. Click on the Shaft component > left click anywhere to place the component in the assembly.

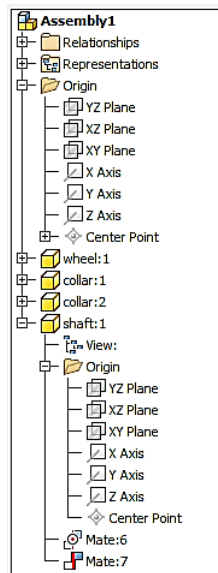
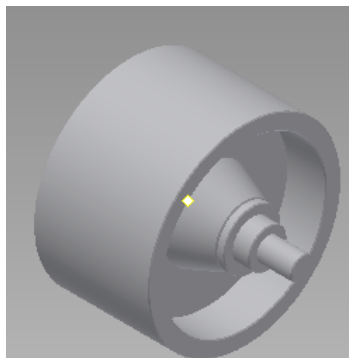
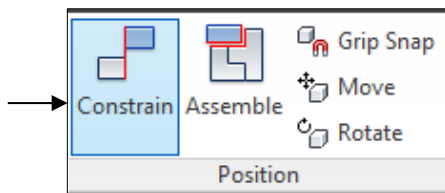




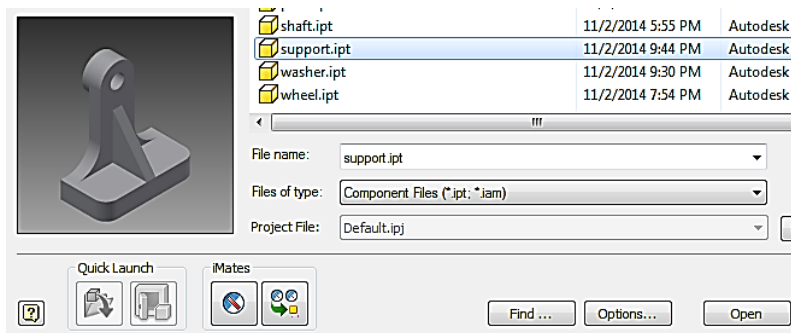
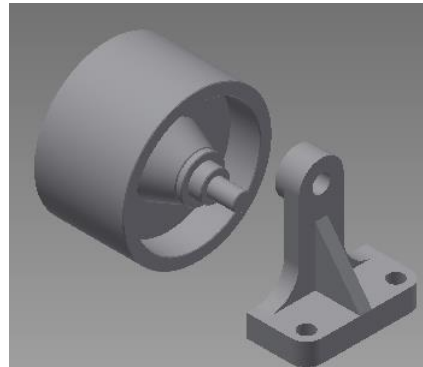
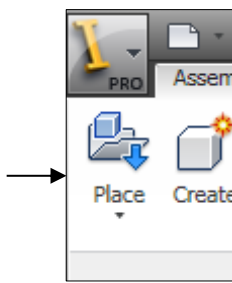
Click the icon of **Constraint**, Constrain the axis of Wheel and the axis of Shaft. The Solution is set to **Mate**. Click **Apply**.



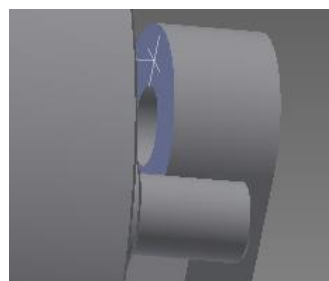
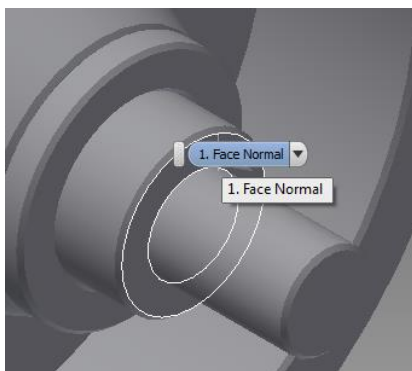
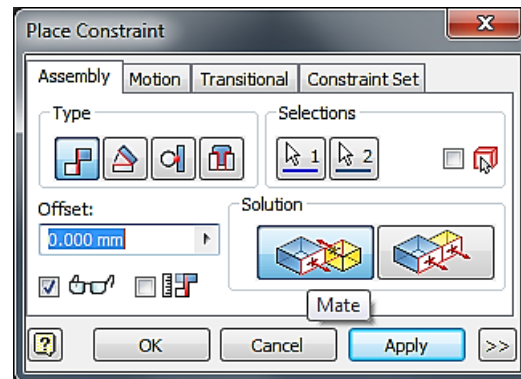
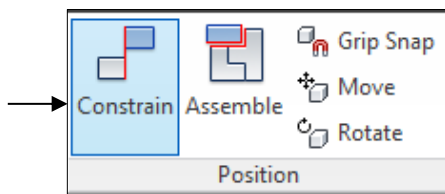
Constrain the Center Point (origin) of the assembly and the Center Point (origin) of Shaft. The Solution is set to **Mate**. Click **Apply**.



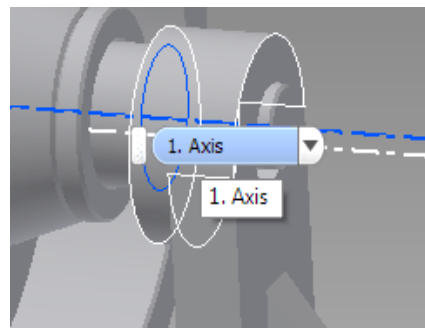
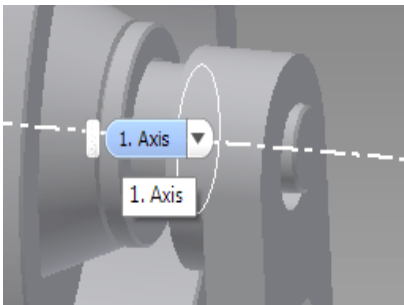
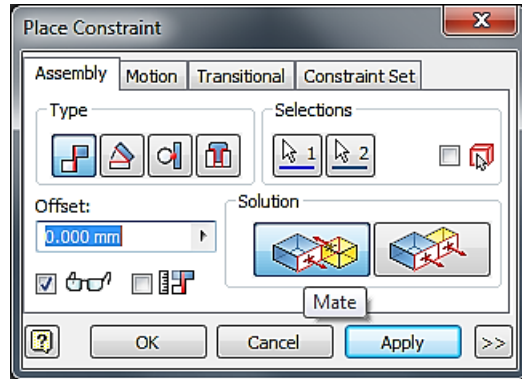
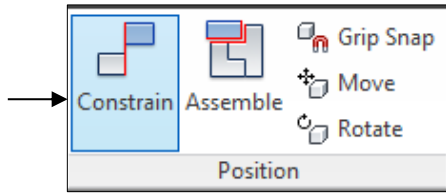
- Click the icon of **Place**. Click on the Support component > left click anywhere to place the component in the assembly.



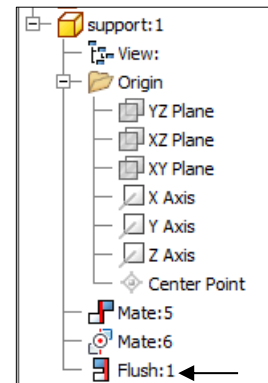
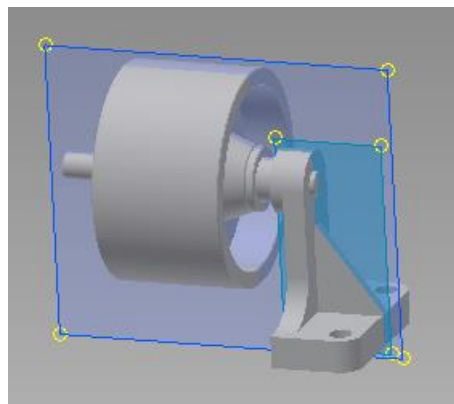
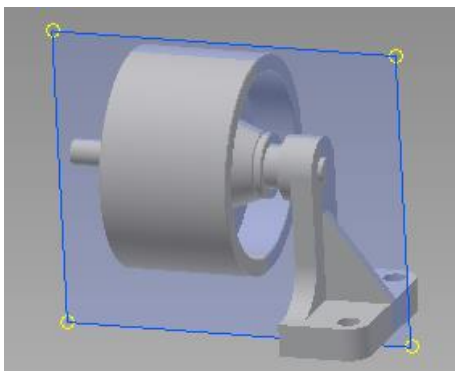
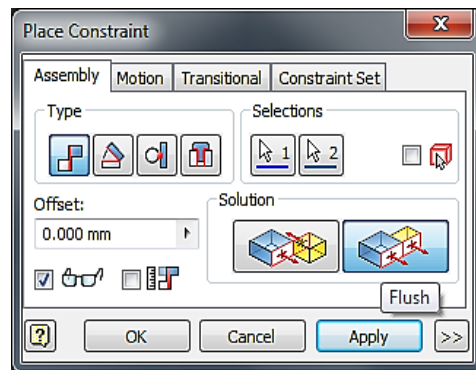
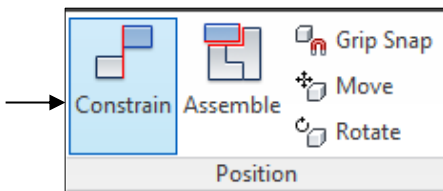
- Click **Constraint**. For Selection 1, choose the right face of Shaft. For Selection 2, choose the left surface on Support. The Solution is set to **Mate**. Click **Apply**.



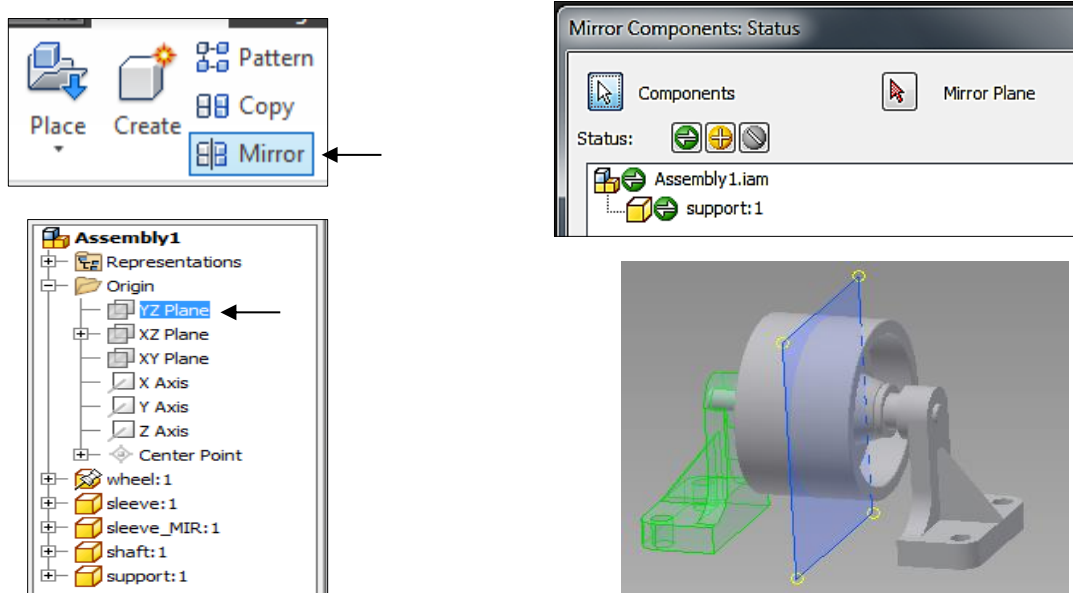
Click the icon of **Constraint**, Constrain the axis of the hole on Support and the axis of Shaft. The Solution is set to **Mate**. Click **Apply**.



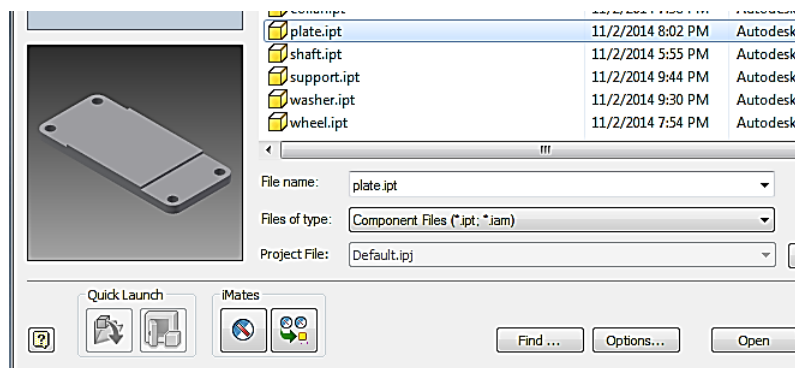
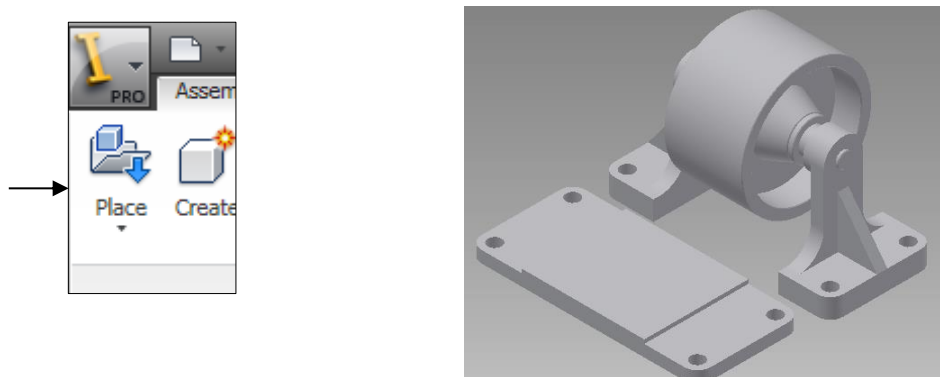
Constrain the (origin) of the assembly and the Center Point (origin) of Shaft. The Solution is set to **Mate**. Users may need to click **Flush** to align the normal vectors of the 2 surfaces. Click **Apply**.



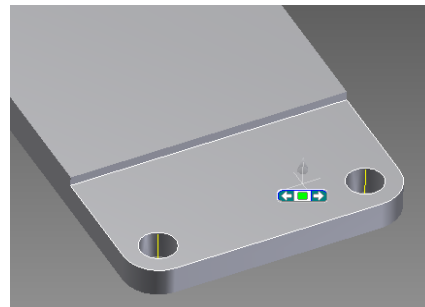
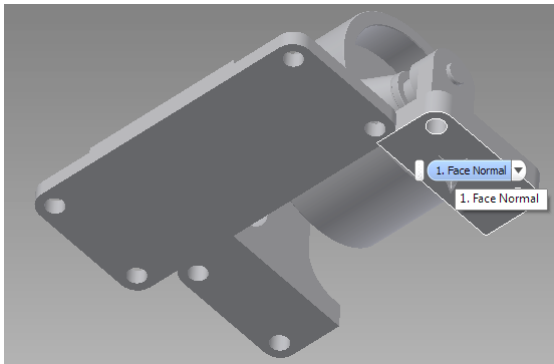
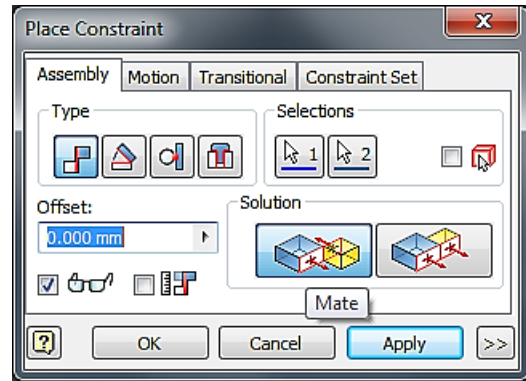
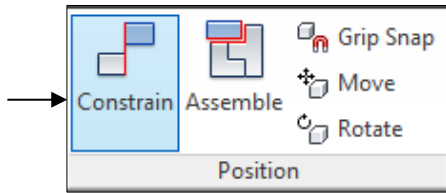
- To create the second Support in the assembly, click **Mirror** in the Component submenu. Select the support component as **Component** to mirror. Choose the YZ Plane of the assembly Origin as the **Mirror Plane**. Click **Next**. Everything in the Mirror Components: File Names window should be correct, so select **OK** to accept Support_MIR as the name of the mirrored component.



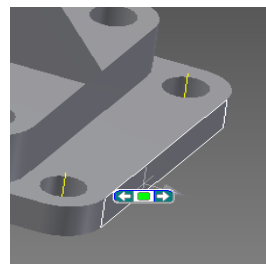
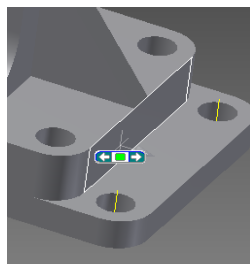
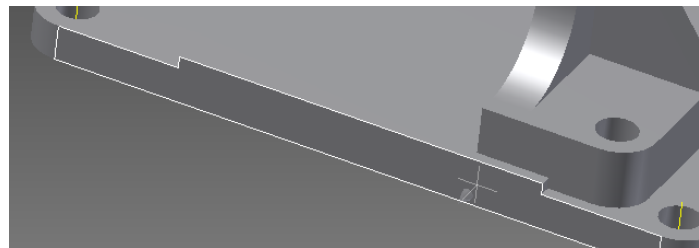
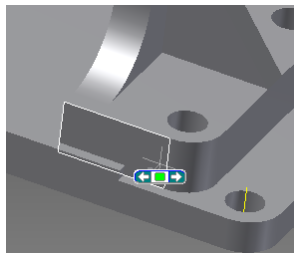
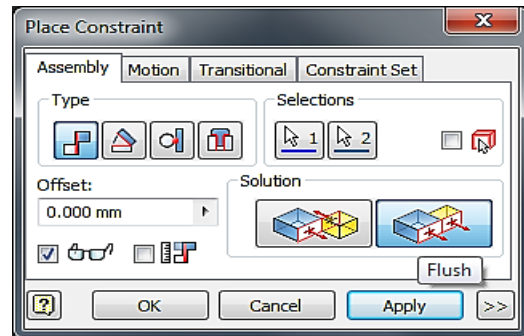
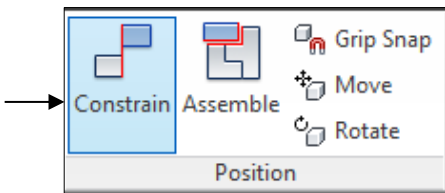
- Assemble the Plate component. Click **Place** > click on the Plate component > left click anywhere to place the component in the assembly.



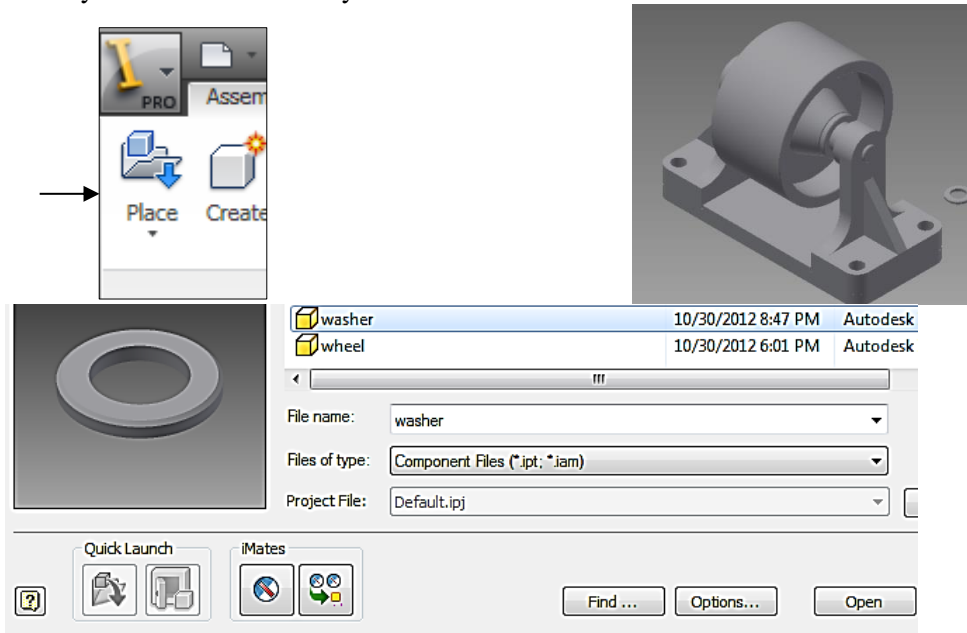
Click **Constraint**. For Selection 1, choose the bottom face of the support. For Selection 2, choose the indentation on the Plate component. The Solution is set to **Mate**. Click **Apply**.



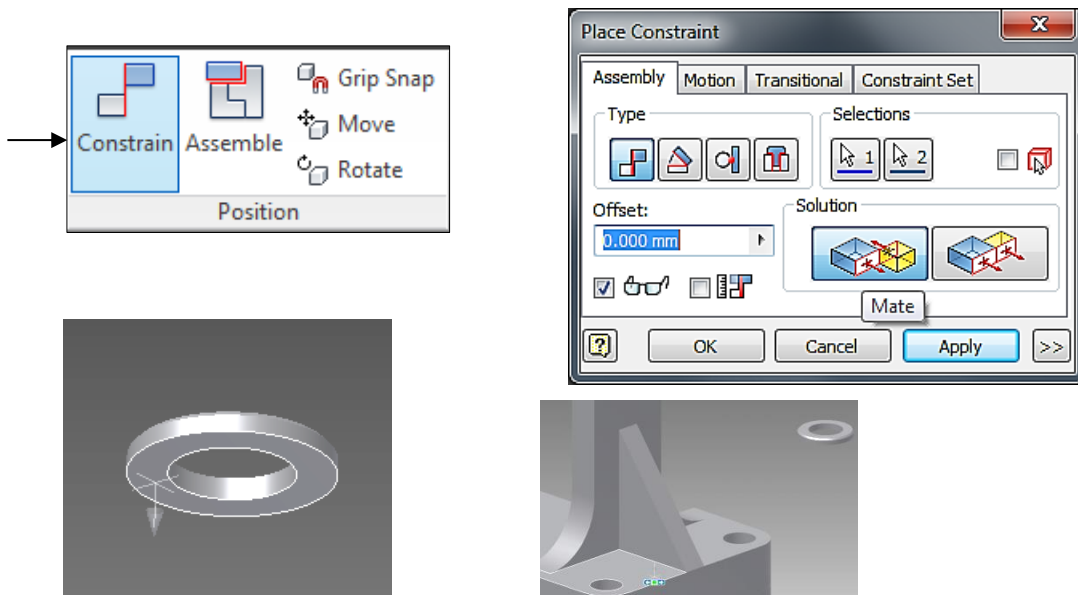
Constrain the left side of Support to the front side of Plate in the same fashion. The Solution is set to **Flush**. Do the same for the back of Support and the right of Plate, as shown in the figures. Click **OK** when done.



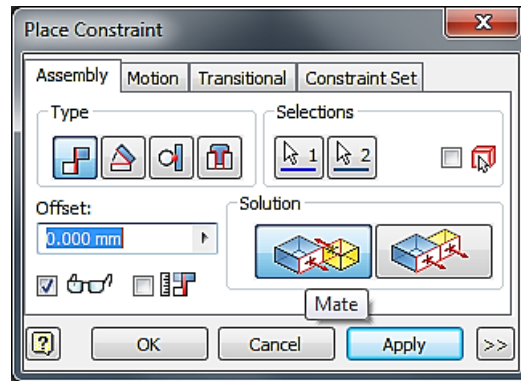
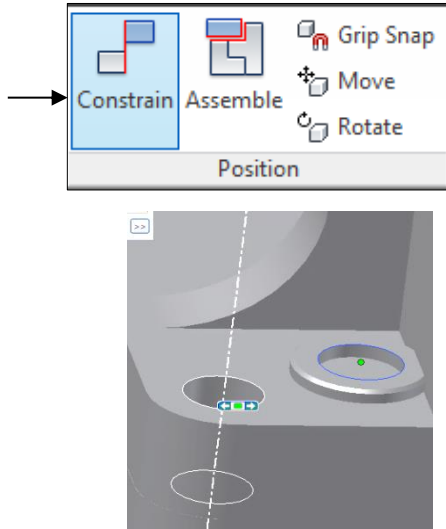
- The Washer component should be inserted next. Choose the washer using the Place command and place it anywhere in the assembly.



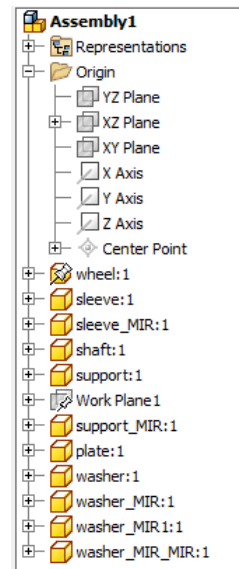
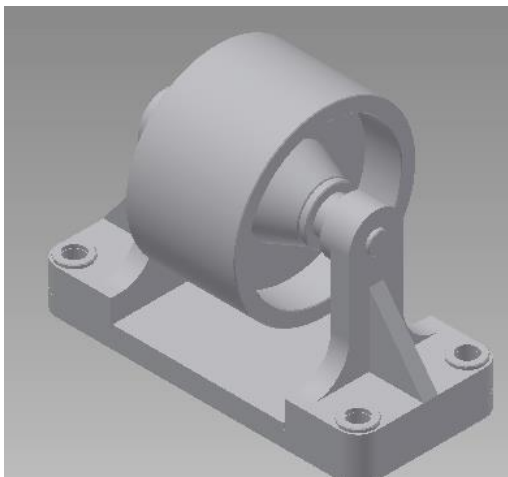
Click **Constraint**. For Selection 1, choose the bottom face of Washer. For Selection 2, choose the top surface on the Support component. The Solution is set to **Mate**. Click **Apply**.



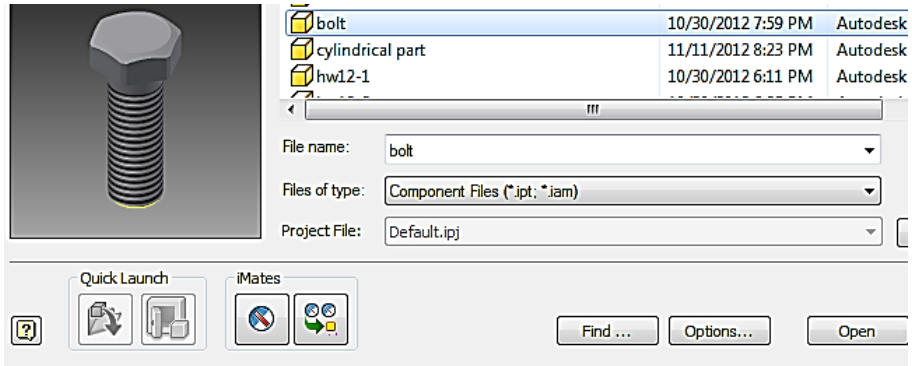
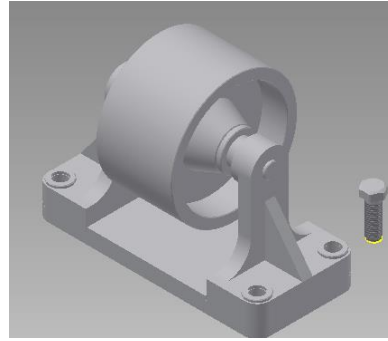
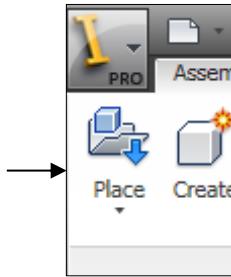
Next, constrain the inner surface of the Washer to be concentric to the Support securing hole. Click the inner surface of the washer for Selection 1, then click the inner surface of the Support hole for Selection 2. The type is **Mate** and the Solution is also **Mate**. Click Apply. Click **OK** when done.



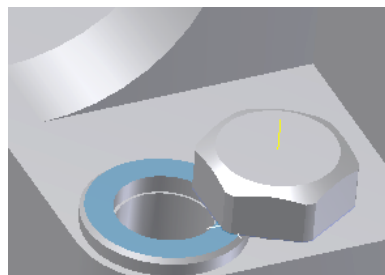
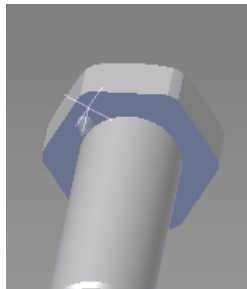
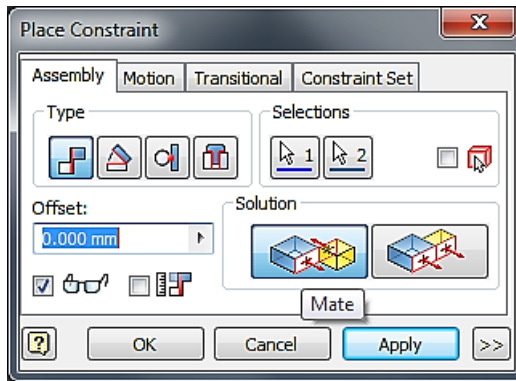
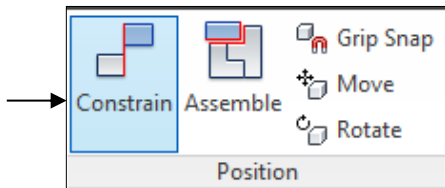
The Washers can be mirrored in the same manner as the Support components in order to place a washer at each hole.



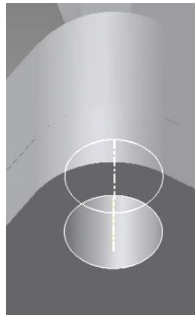
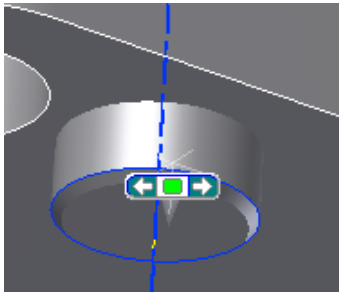
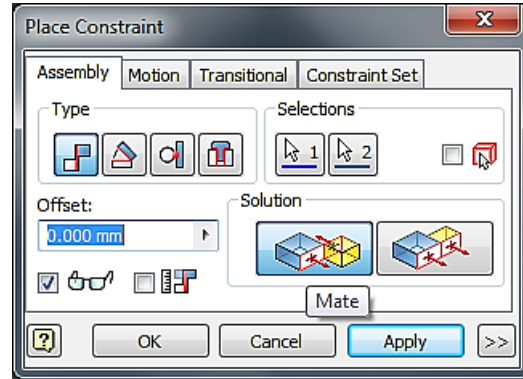
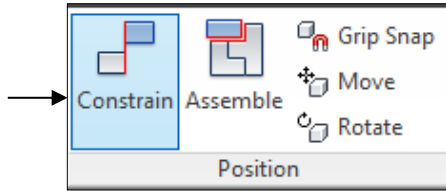
8. The Bolt component should be inserted next. Choose the Bolt component using the **Place** command and place it anywhere in the assembly.



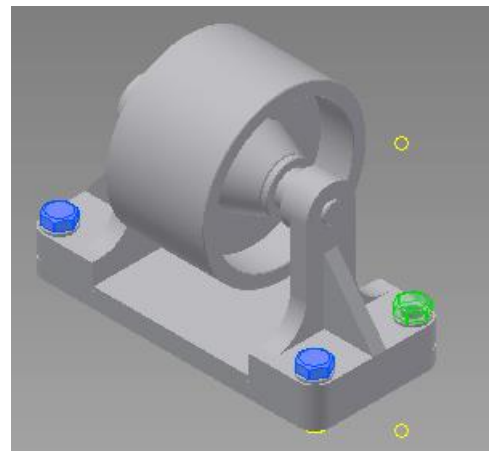
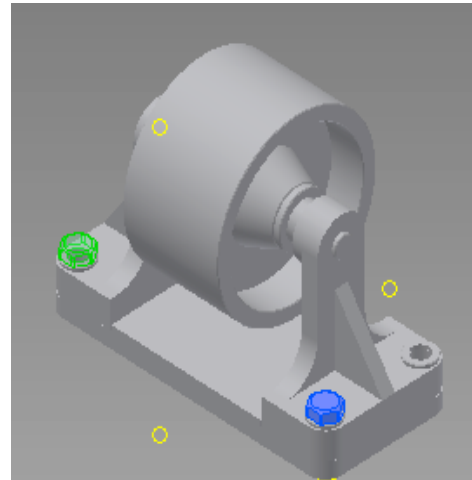
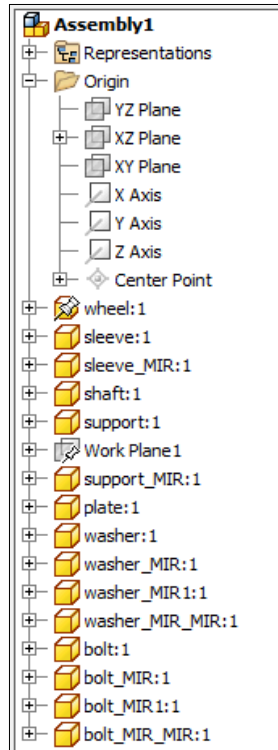
Click **Constraint**. For Selection 1, choose the underside of the Bolt head. For Selection 2, choose the top surface of the Washer. The type is **Mate** and the Solution is also **Mate**. Click Apply. Click **OK** when done.



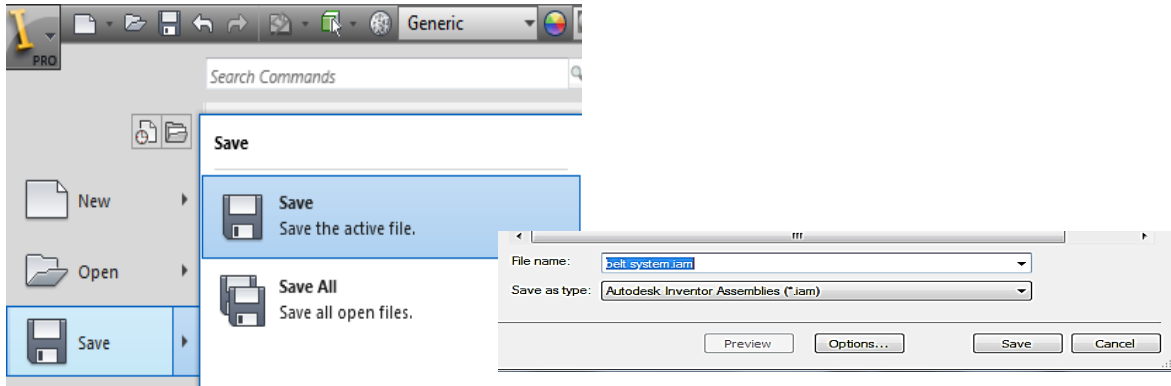
Next, constrain the outer surface of the Bolt shaft to the inner surface of the hole of the Support. For Selection 1, click the axis of the Bolt shaft. For Selection 2, click the axis of the hole of the Support. The type is **Mate** and the Solution is also **Mate**. Click Apply. Click **OK** when done.



9. After following the same steps as for the washer, four Bolts are now in the assembly through **Mirror**. This concludes all the necessary processes for creating the entire assembly.

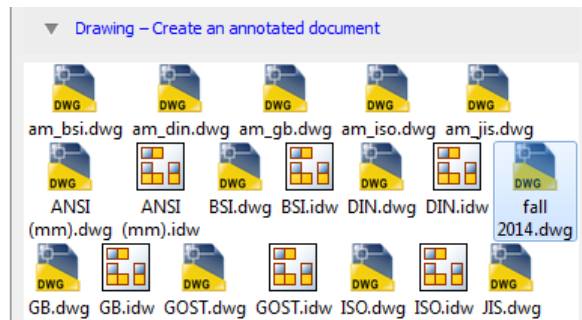
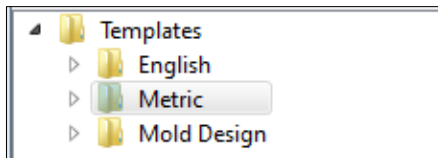
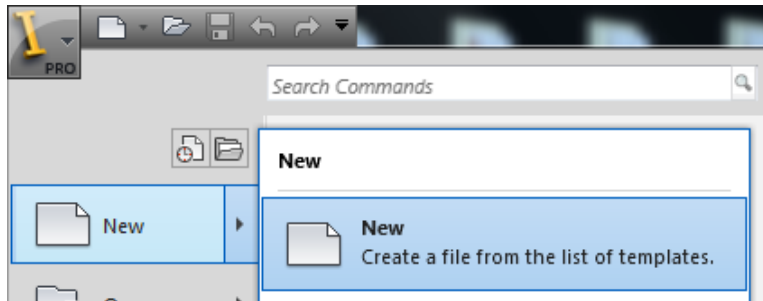


10. Save the 3D solid model of the Belt System. Click the icon of **Save**. Specify Belt System as the file name.

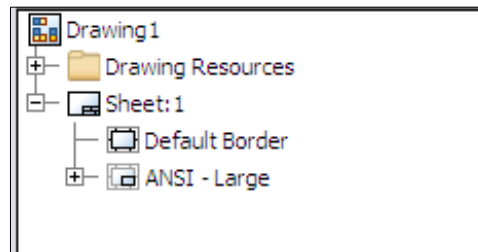


Part 3: Creation of the Assembly Drawing with a User-Defined BOM Table

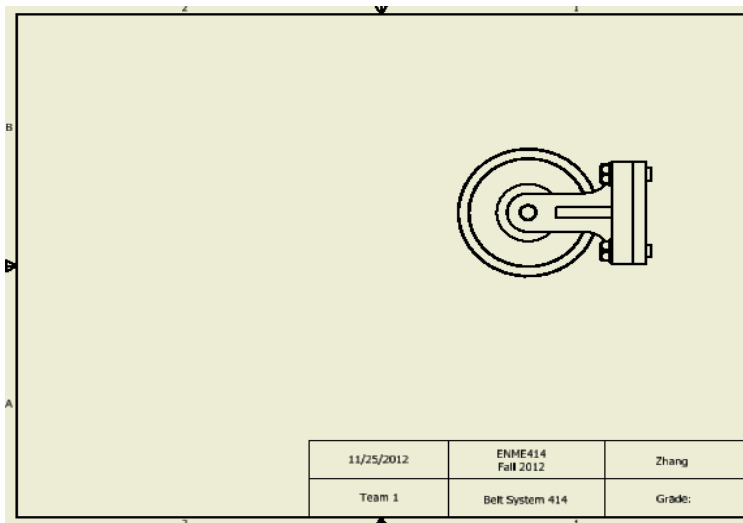
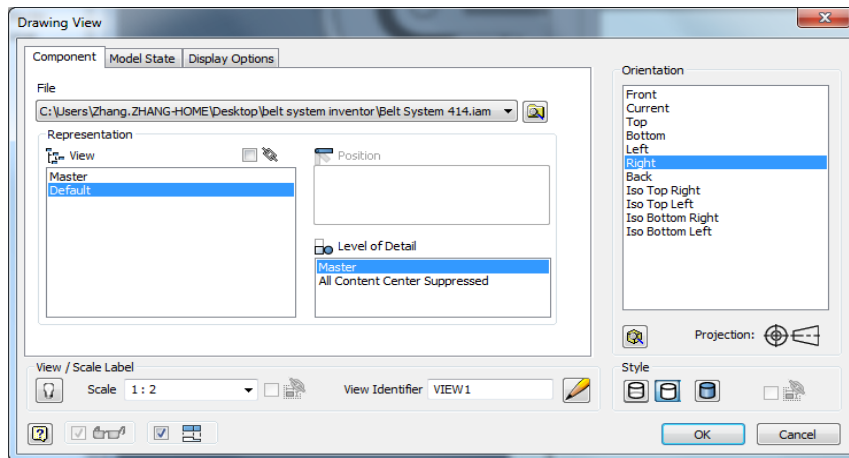
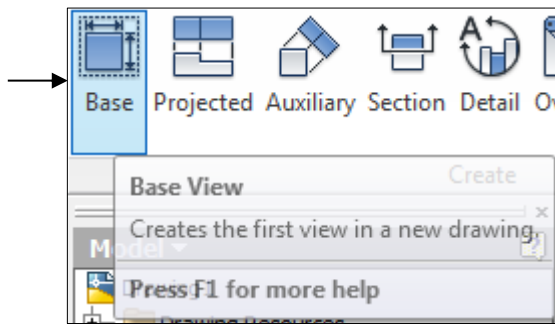
1. Create a new drawing by clicking *New > Metric > ANSI(mm).idw or Fall 2014.dwg > Create.*



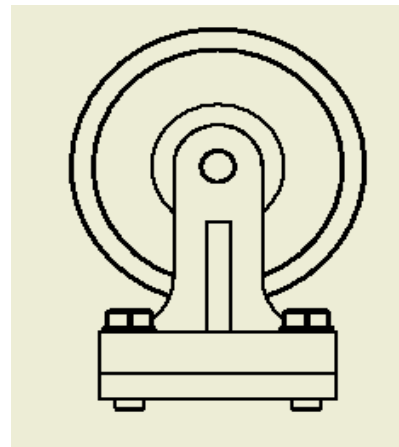
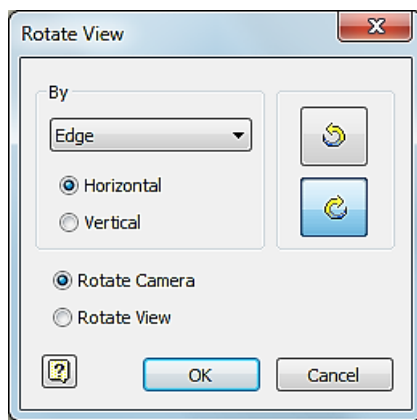
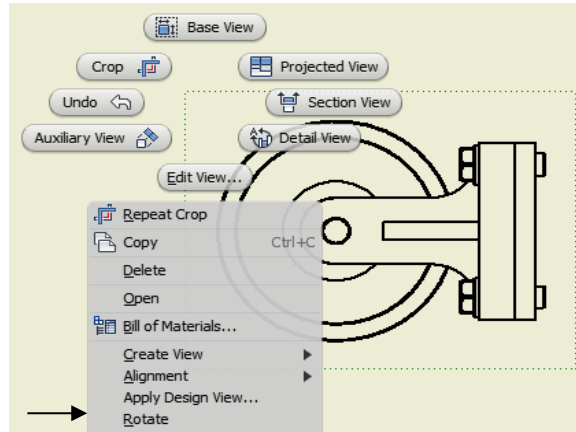
2. If *ANSI(mm).idw* is used, right click *Sheet* from the model tree > *Edit Sheet* > change the size to A.



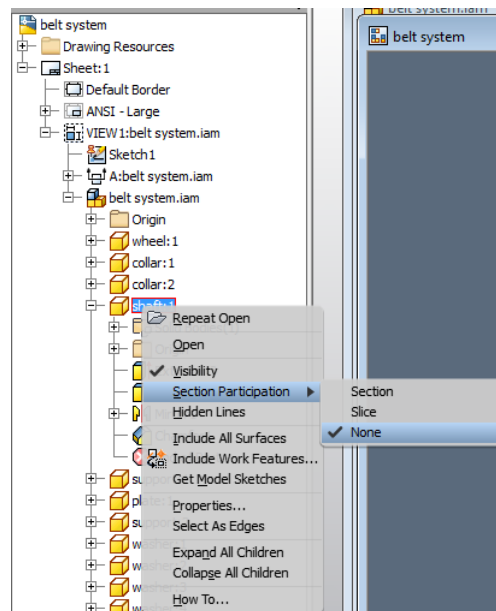
- Click the icon of **Base** to insert the Belt System Assembly. Choose **Right** as the Orientation and change the Scale to 1:2. Place this in the upper right section of the drawing.



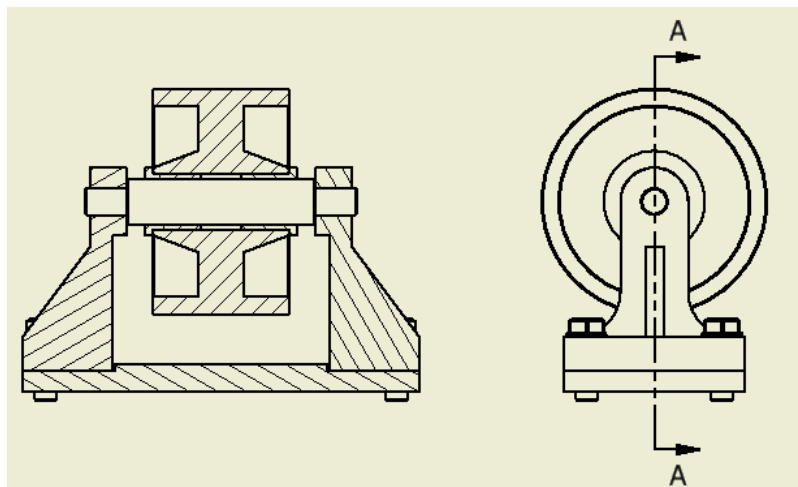
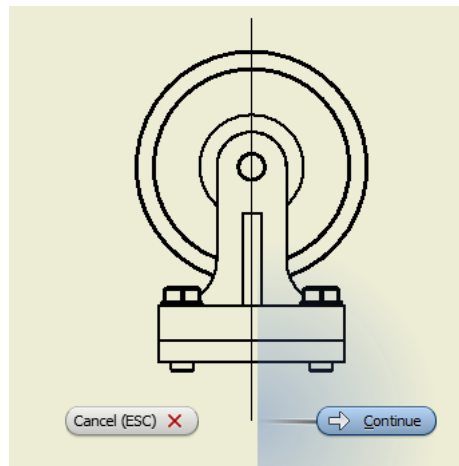
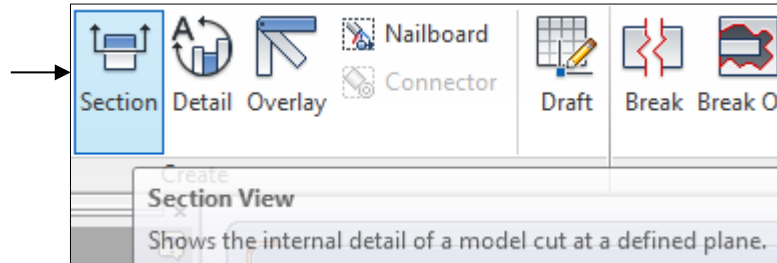
To rotate the right-side view, right-click the view and pick **Rotate**. In the **Rotate View** window, click the box of CW rotation and pick the edge representing the bottom surface of Support.



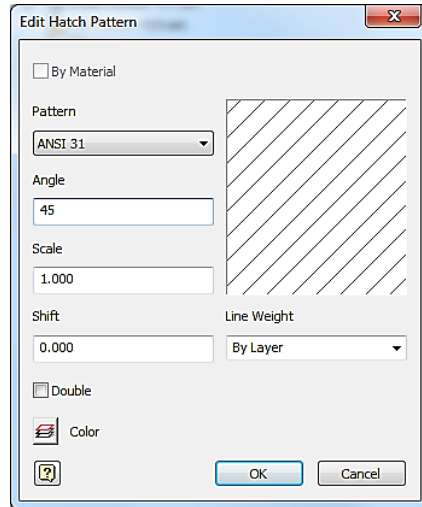
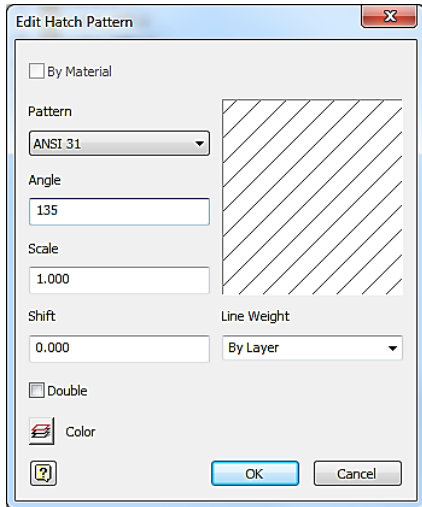
4. From the model tree, highlight Shaft and right-click to pick Section Participation. Expand it to pick **None**. In this way, there will be no hatched lines displayed on the shaft component.



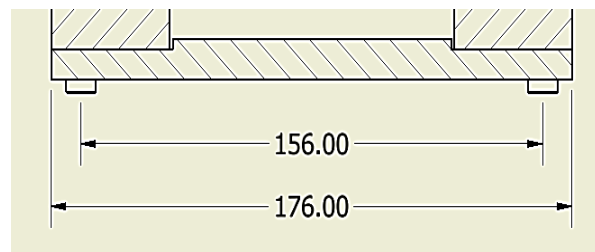
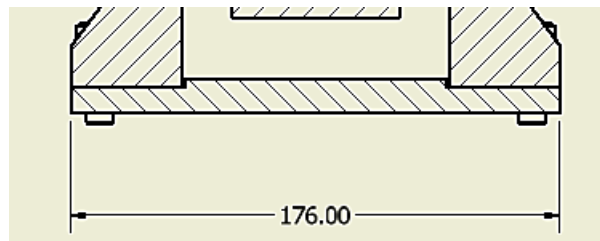
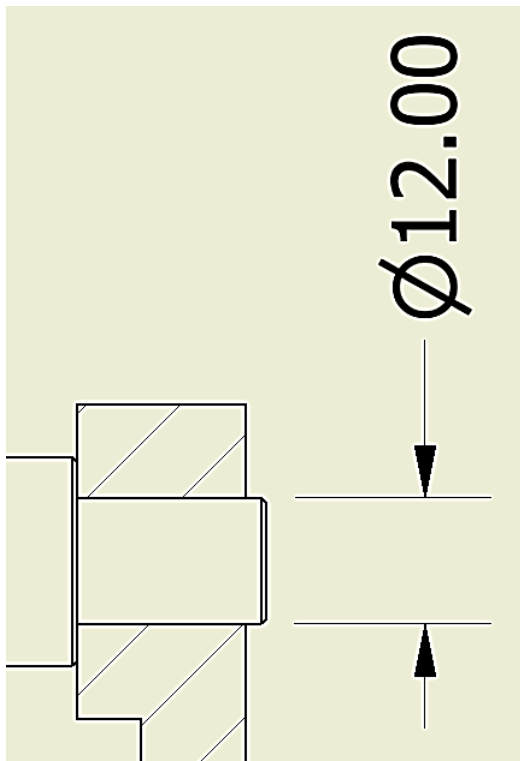
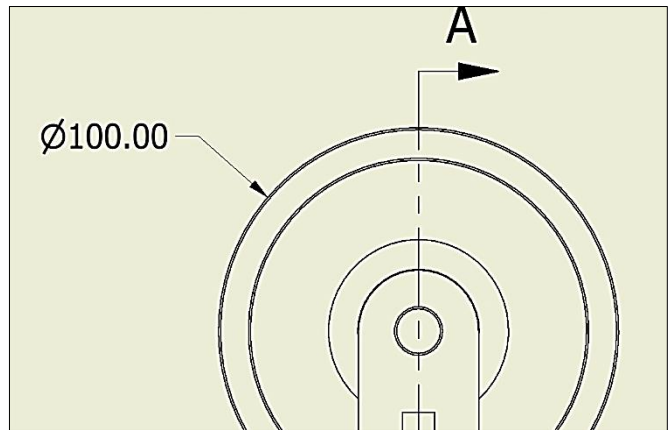
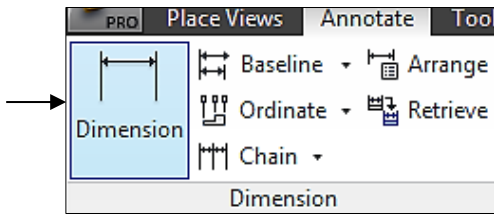
- To insert a section view, click the icon of **Section** in the Create menu. Click on the recently created Right view as the base view for the section. Place the cursor on the top center of the assembly until a green circle appears. Move the cursor straight up and a dotted line will appear. When sufficiently up, click the left mouse button. Bring the cursor straight down underneath the assembly view and click again. When done, right click and select **Continue**. Place the section view to the left.



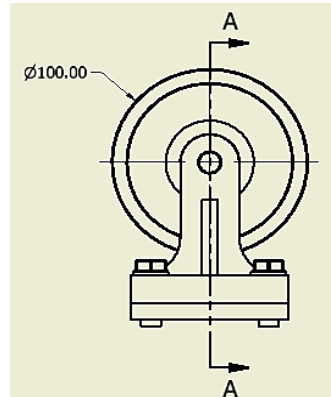
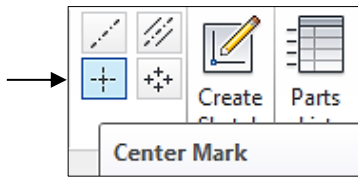
- Adjust the angel and spacing of the hatched lines for each of the components, except the shaft component, which is excluded in the section process. For example, click the hatched lines from the Plate component. In the Edit Hatch Pattern window, specify 135 and click **OK**. For the hatched lines of Support, specify 45 and click **OK**.



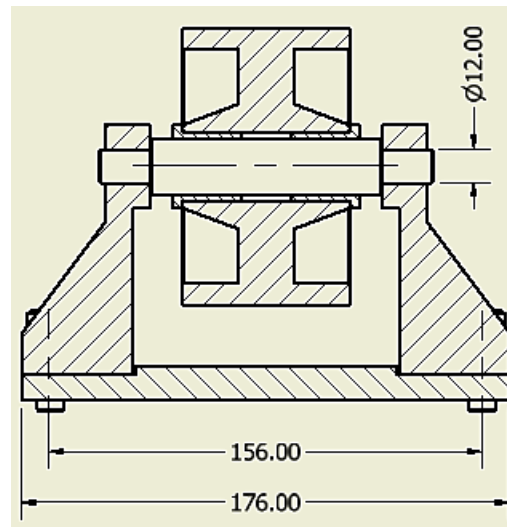
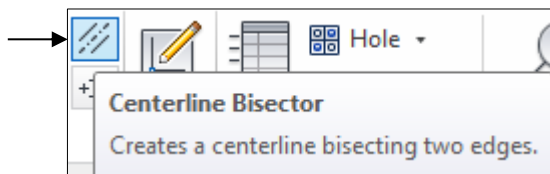
7. Annotate the views accordingly using the *Dimension* tool and centerlines tools. Click the icon of Dimensions. Click the outer circle and place the dimension of $\phi 100$.



To add centerlines, click the icon of **Center Mark** and click the outer circle to place the centerlines.

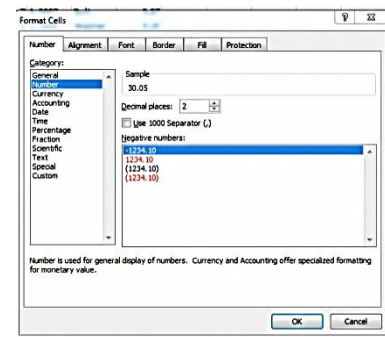


Click the icon of Bisector, and click 2 straight lines. A centerline will be displayed at the middle between the 2 straight lines.

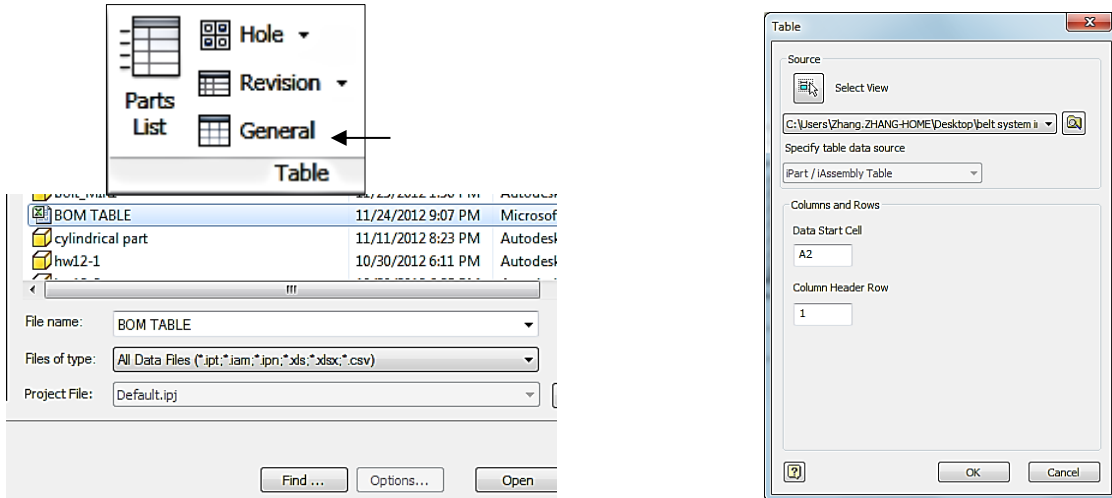


- A BOM table can be created in **Excel** and inserted into the drawing. First, create the BOM table in **Excel**. Open **Excel** and a new data sheet will be automatically created. Create the columns and enter values as shown in Figure E-7. Drag a box around the values in the **PRICE** column, right click and select **Format Cells**. In the Number tab, select Number, select 2 for Decimal places, and then click **OK**. When done, save the file as **BOM_Table.xlsx** or **BOM_Table.xls**.

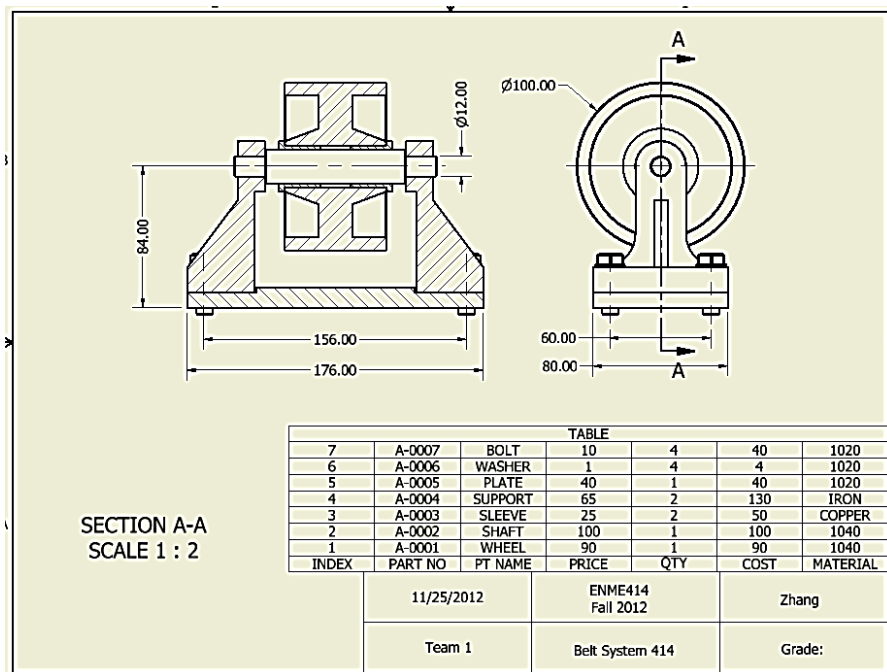
	A	B	C	D	E	F	G
1	7	A-0007	Bolt	9.87	4	39.48	1020
2	6	A-0006	Washer	0.25	4	1	1020
3	5	A-0005	ottom Plat	26.78	1	26.78	IRON
4	4	A-0004	Support	45.89	2	91.78	IRON
5	3	A-0003	Shaft	50.55	1	50.55	1020
6	2	A-0002	Sleeve	15.50	2	31	COPPER
7	1	A-0001	Wheel	30.05	1	30.05	1020
8	INDEX	PART NO	PT NAME	PRICE	QTY	COST	MATS



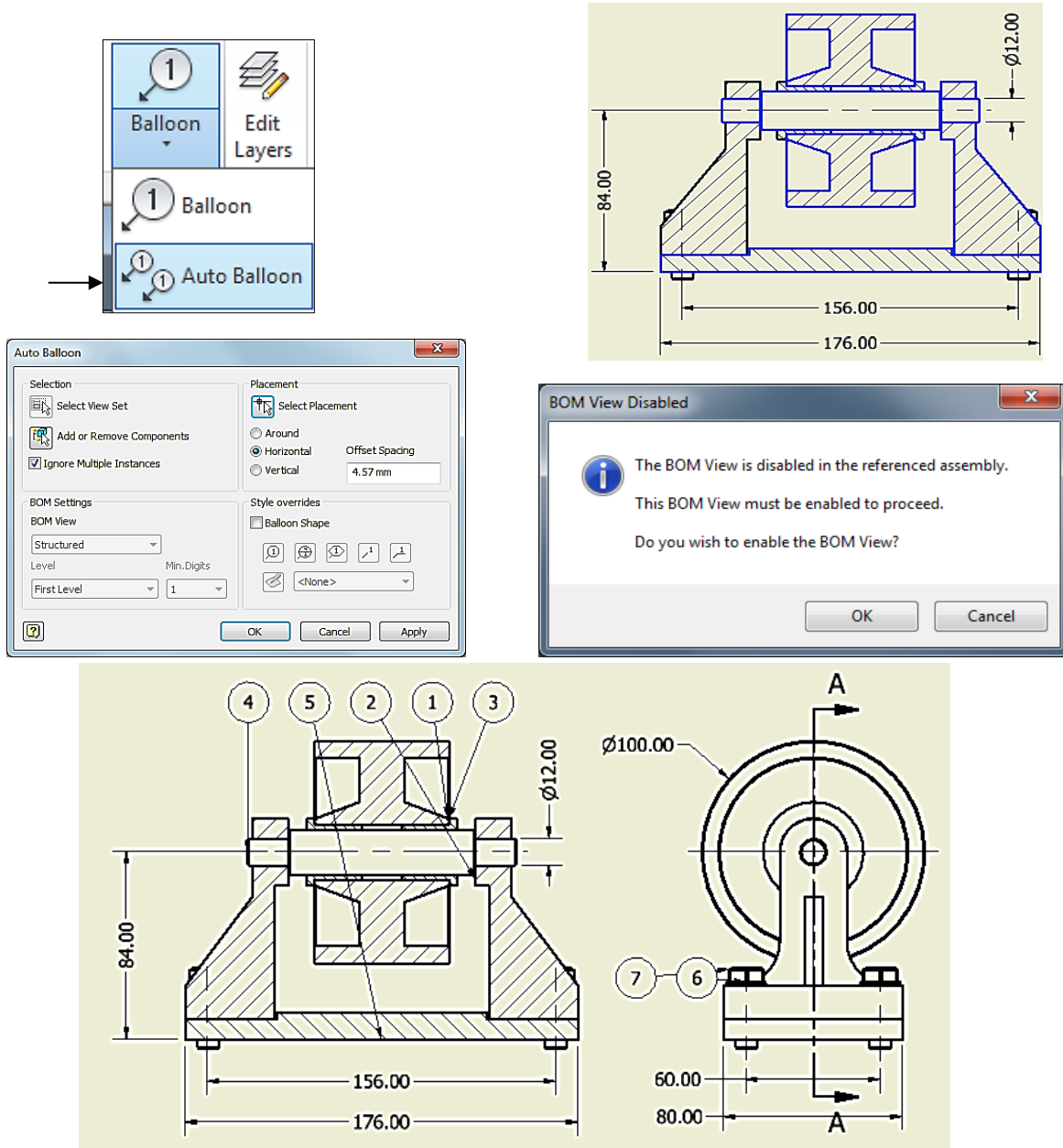
- Next, in the Annotate menu, click the icon of **General** in the Table sub-menu. Navigate to the saved Excel file and select it for insertion, then click **OK**.



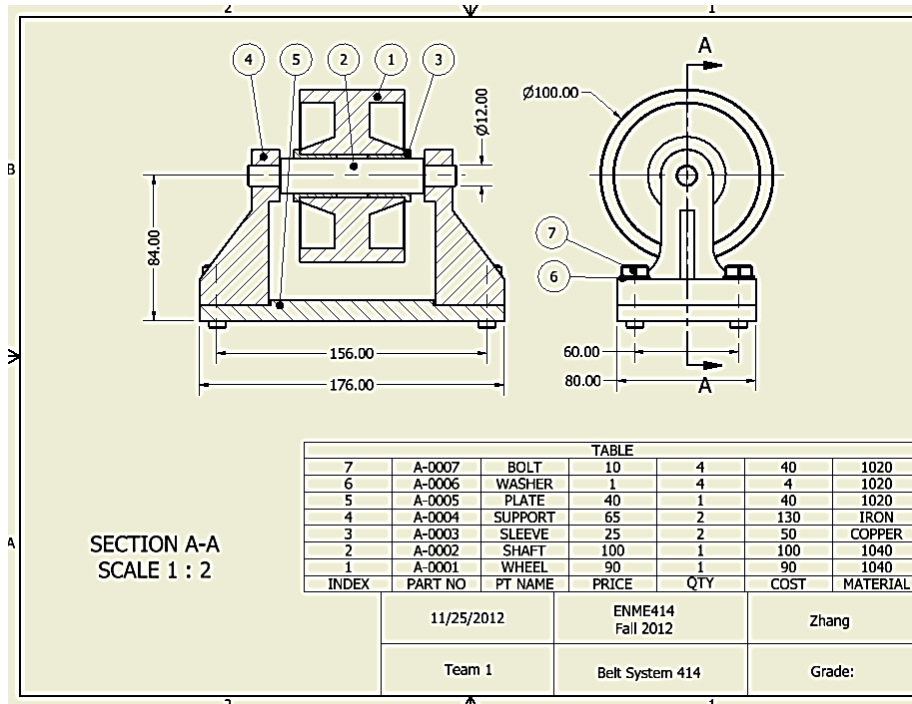
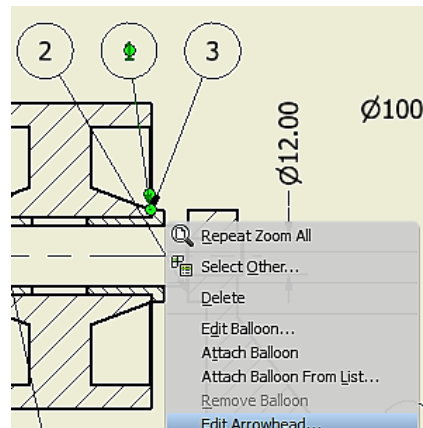
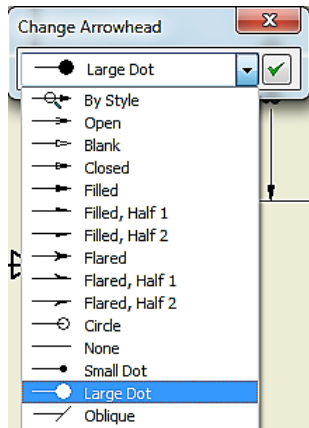
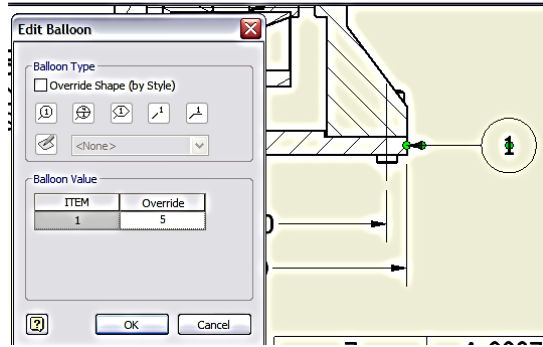
10. Place the created table above the pre-existing format table. The lower right corner of the BOM table will lock in place against the right border and the top of the existing table.



11. To insert BOM Balloons, click **Auto Balloon** in the Table sub-menu. Choose the Section view as the view to insert the balloons. Select the Support, Wheel, Shaft, Sleeve, and Plate as the parts to Balloon. Next, click **Select Placement** and place the balloons near the drawing. Move the balloons so that the drawings look presentable. Repeat the same process for the Bolt and Washer parts in the Right view. Balloons can also be placed individually by clicking Balloon and selecting individual components.



12. In order to modify the balloons so that they have the same number referenced in the INDEX column of the BOM table, right click the balloons needing changes and select **Edit Balloon**. Change the **Override** value to the value referenced in the BOM table. Repeat this for all of the balloons. The tips of the arrows of each balloon can also be dragged so that they change to dots and are on the surface of the part. This makes the balloons easier to follow. By right clicking the balloons and selecting **Edit Arrowhead**, the dots can be changed to whichever shape is best.



13. The drawing is now complete. Save the drawing.

