AUTODESK **INVENTOR**

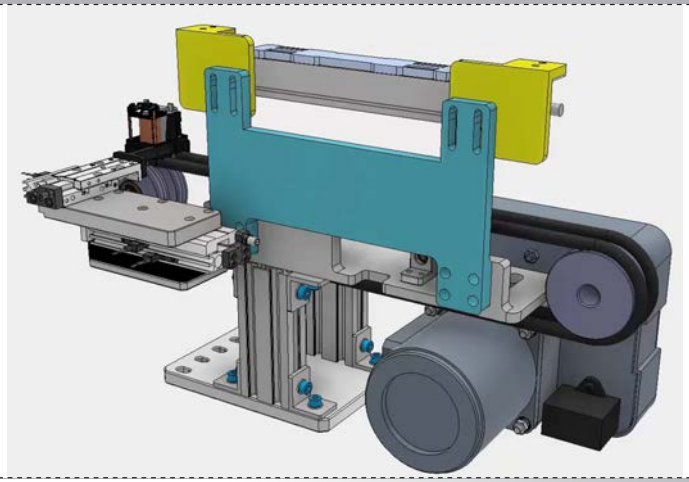
Trial Projects

Design Automation

Design a conveyor assembly

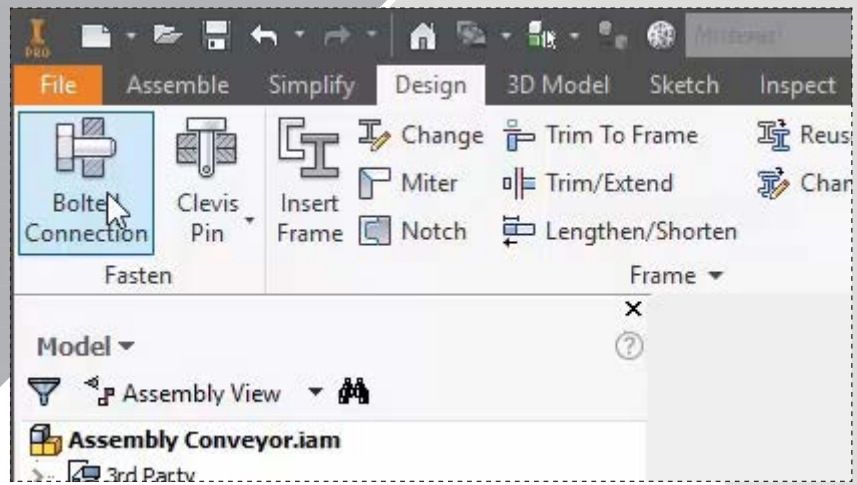
PART 1: DESIGN AUTOMATION

1.



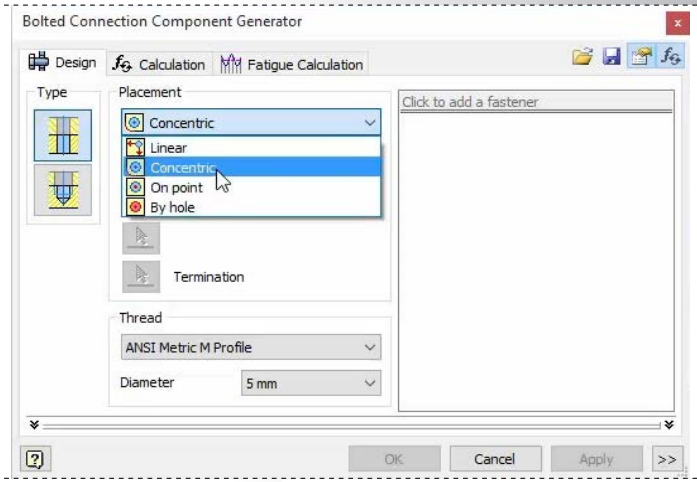
In Inventor, click the 'Projects' icon in the ribbon. Navigate to where you saved the project files and select ***Assembly Conveyor DA.ipj***. Then open ***Assembly Conveyor DA.iam***.

2.



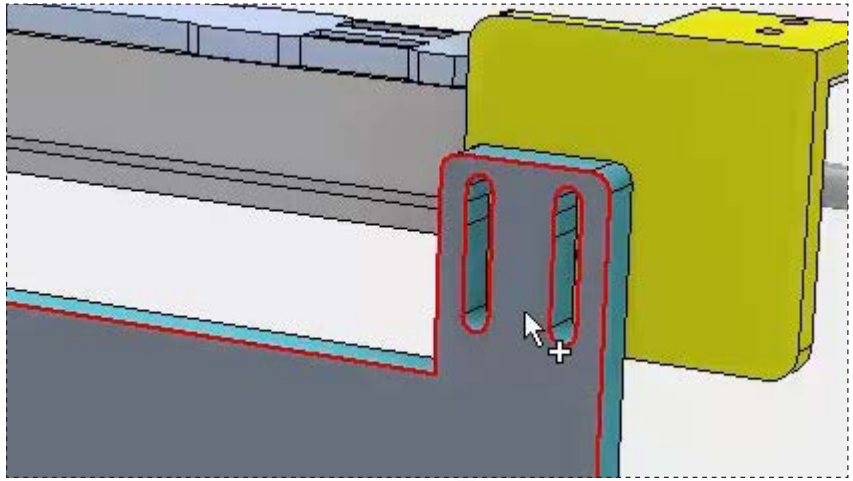
Access the 'Design' tab in the ribbon and click 'Bolted Connection', located in the 'Fasten' panel.

3.



Select 'Concentric' from the drop-down for the 'Placement'.

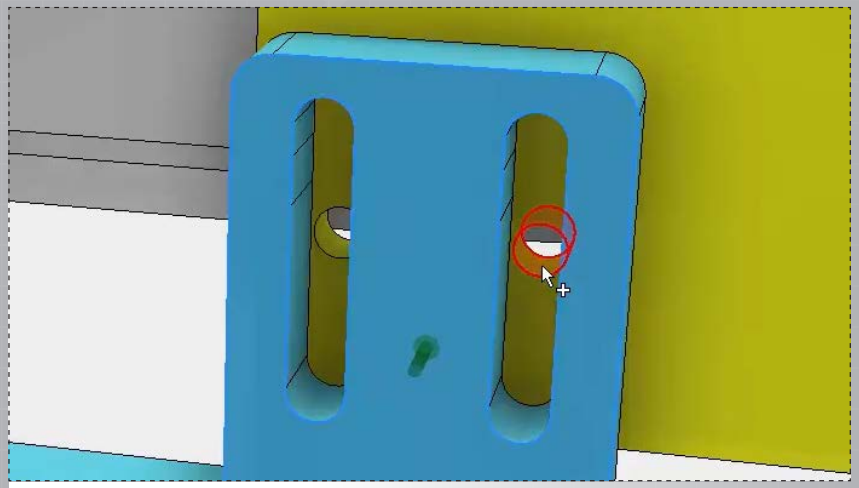
4.



Select the face shown for the 'Start Plane'.

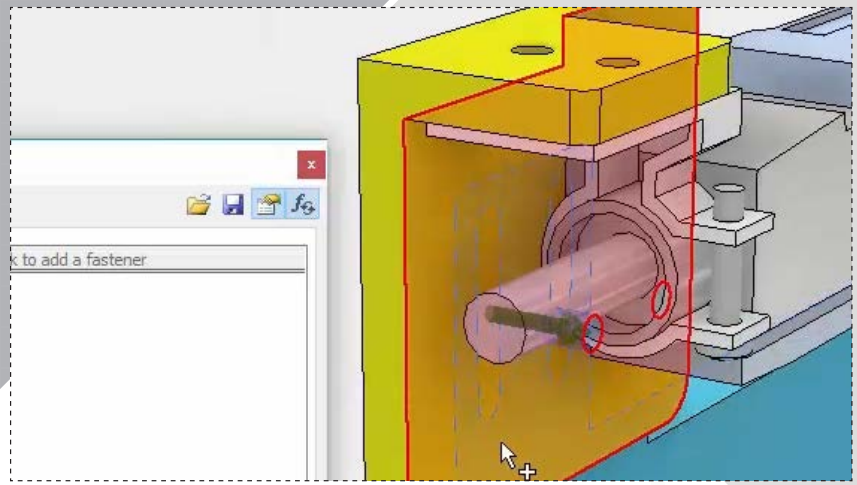
PART 1: DESIGN AUTOMATION

5.



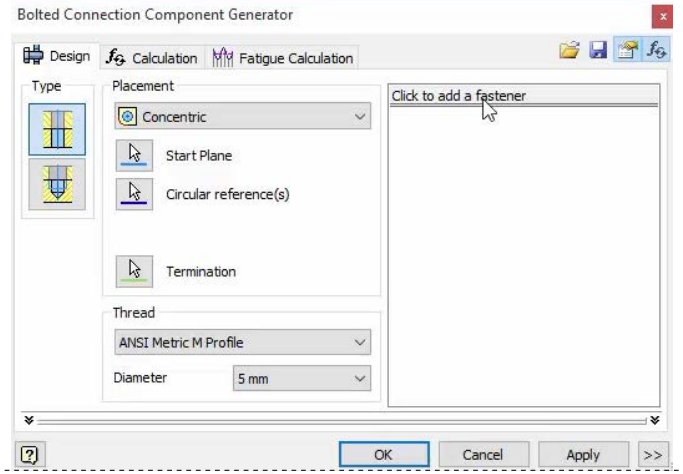
Select a ‘Circular’ reference on the yellow bracket for the location.

6.



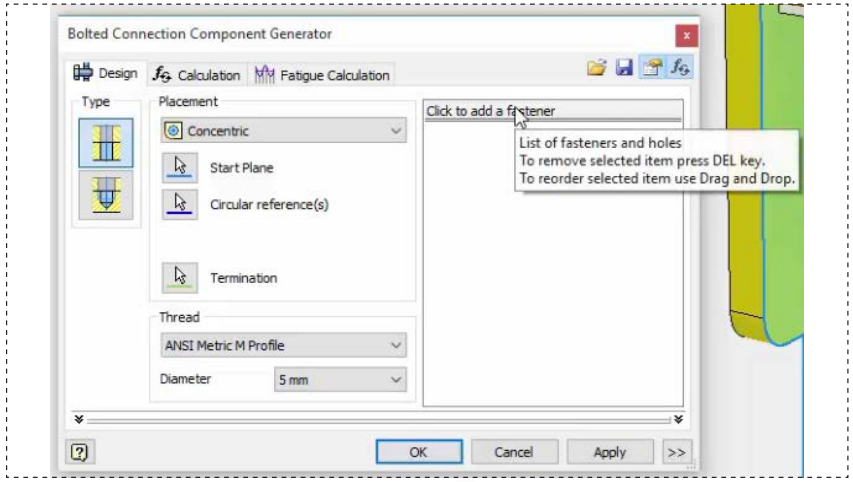
Choose the back face of the yellow bracket for the fastener ‘Termination’.

7.



Define the ‘Thread ‘using an ‘ANSI Metric M Profile’, and then set the ‘Diameter’ to 5mm.

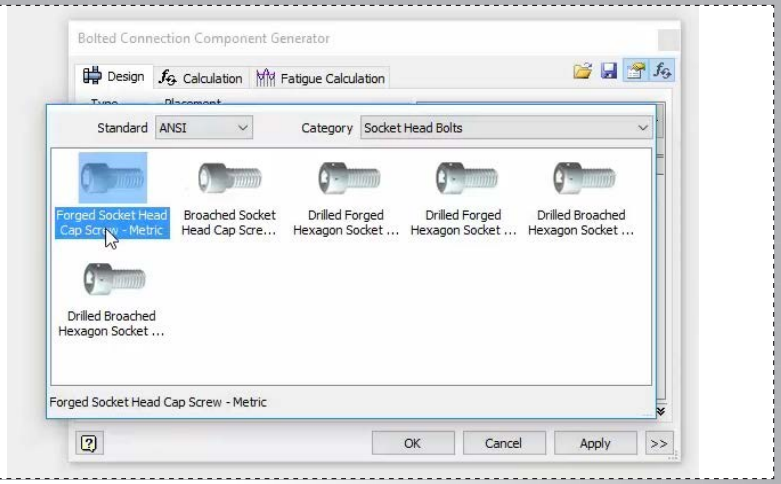
8.



Select ‘Click to add a fastener’ in the dialog to select a ‘Standard’ and ‘Category’.

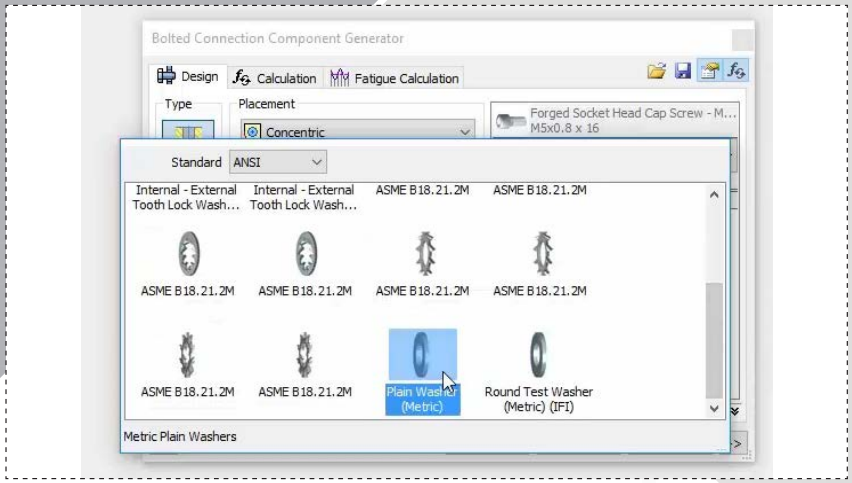
PART 1: DESIGN AUTOMATION

9.



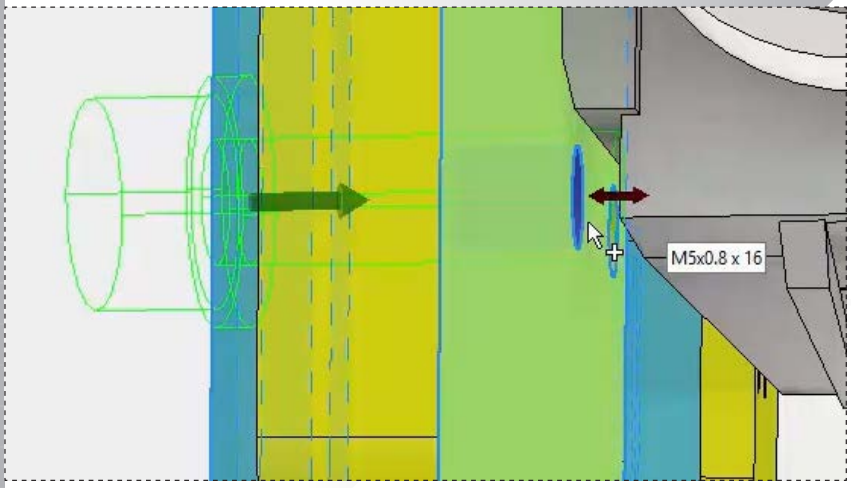
Filter to 'ANSI, Socket Head Bolts', and select the 'Forged Socket Head Cap Screw - Metric option'.

10.



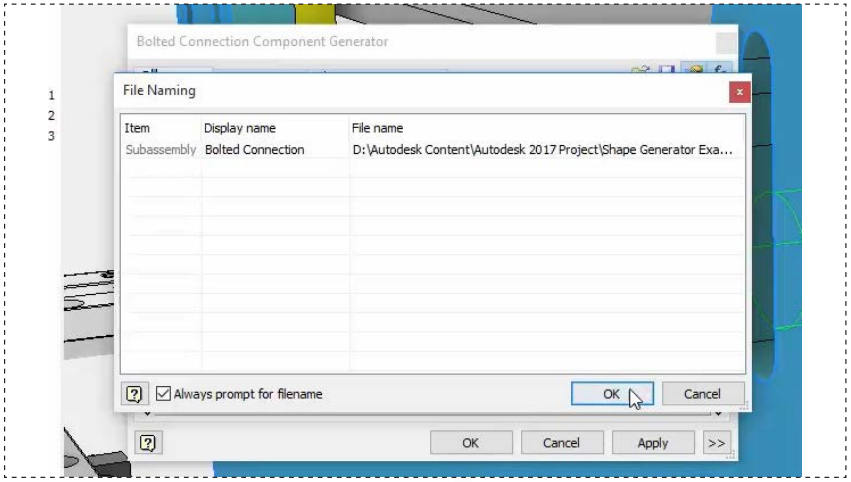
Select 'Click to add a fastener' below the screw you just inserted, and then insert the 'Plain Washer (Metric)' component.

11.



Drag the arrow at the end of the fastener preview to define the correct length. Click 'OK' on the dialog to create the bolted connection.

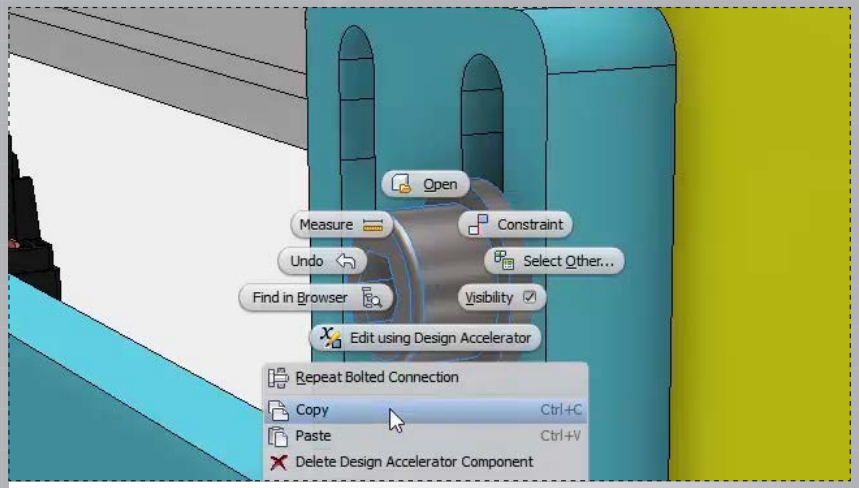
12.



Click 'OK' again the accept the default 'File Naming' for the new subassembly.

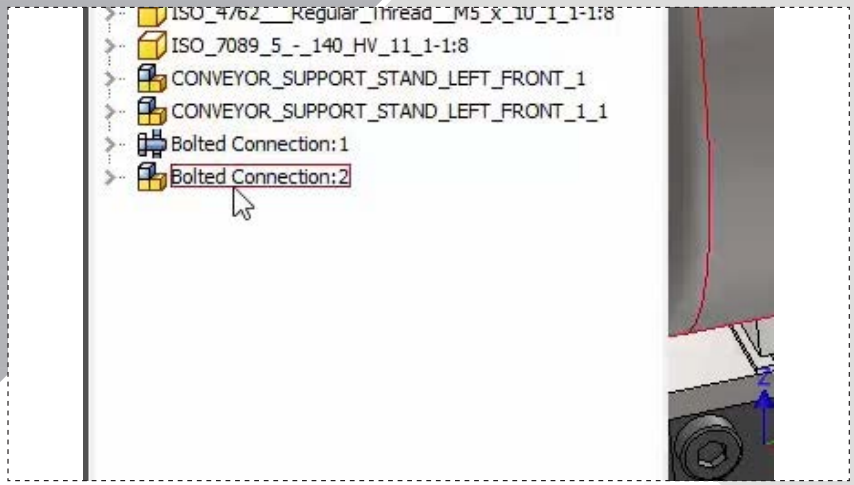
PART 1: DESIGN AUTOMATION

13.



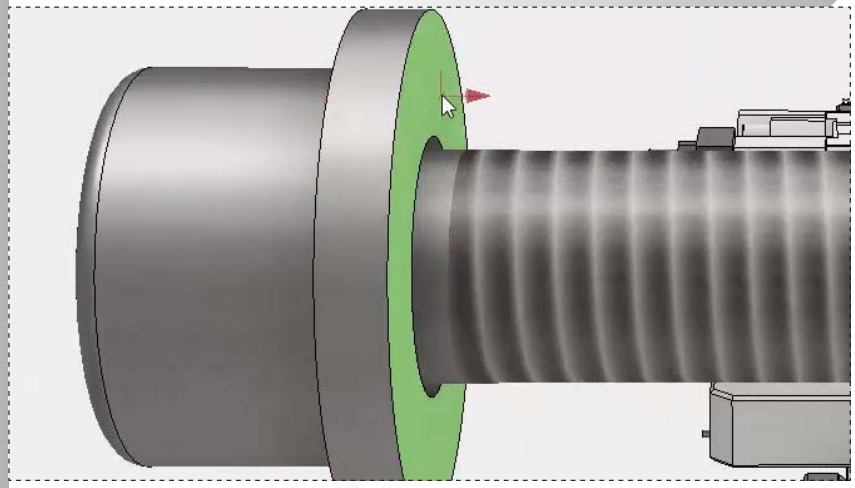
Right-click on the new bolt subassembly and select 'Copy' from the marking menu.

14.



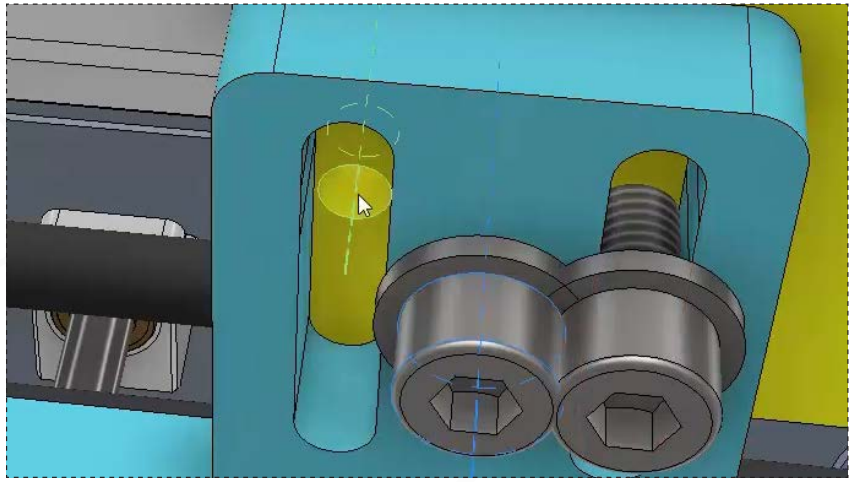
Right-click in empty space and select 'Paste' to insert another instance of the component. Note that it's listed as a subassembly within the feature tree, and not as an original bolted connection.

15.



Selecting 'Constraint' from the marking menu, create a 'Mate' between the front face of the bracket and this face on the washer.

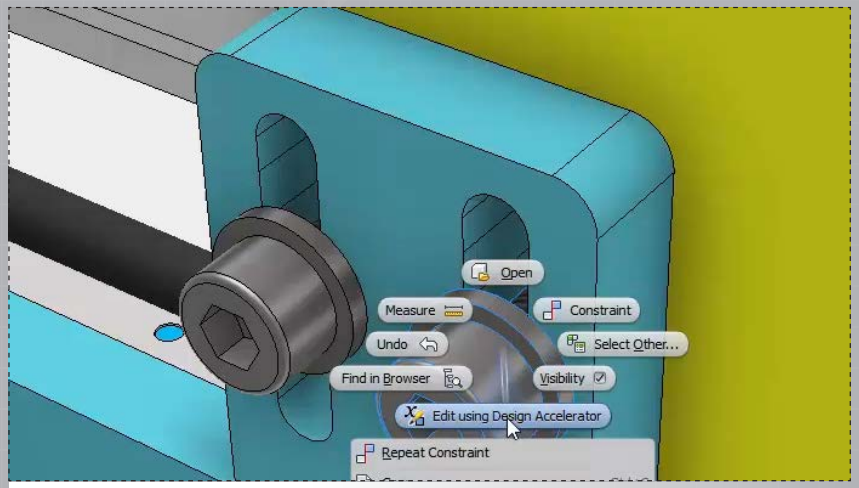
16.



Create another 'Mate' constrain between the centerline of the screw and the centerline of the remaining hole.

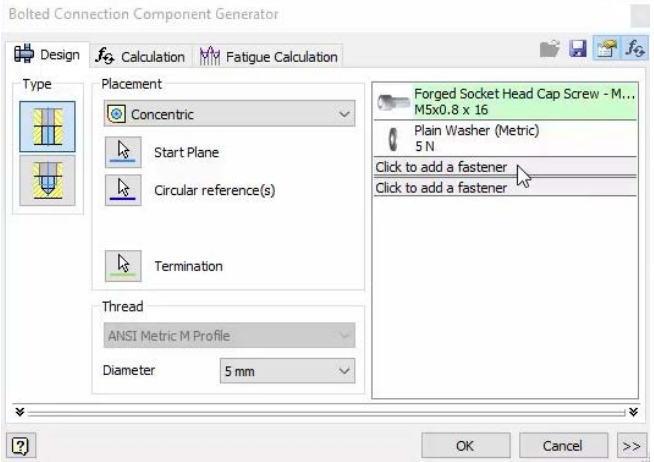
PART 1: DESIGN AUTOMATION

17.



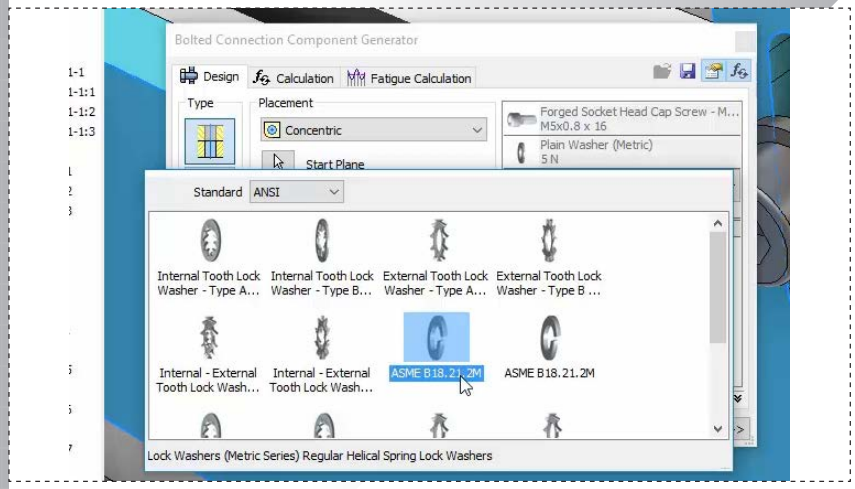
Right-click on the original bolt and select 'Edit' using 'Design Accelerator'.

18.



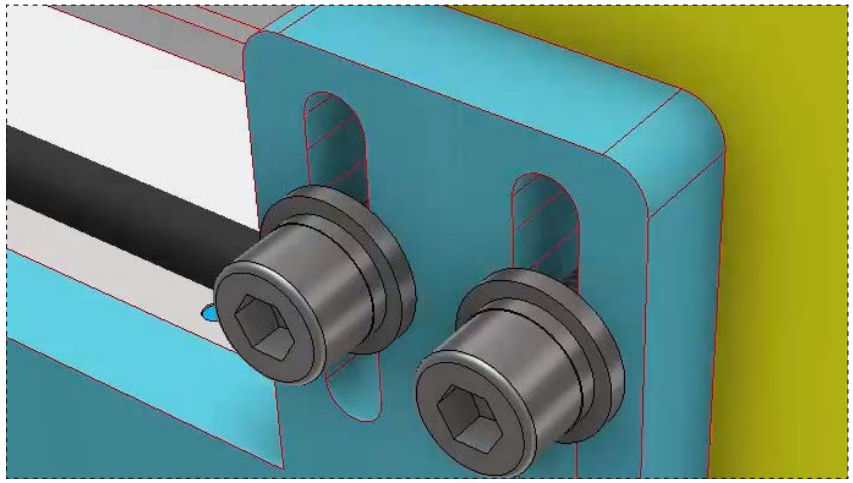
Click to add a fastener beneath the existing washer listing.

19.



Insert an instance of the 'ASME B 18.21.2M' lock washer to the stack, and then drag it in between the two existing components in the list.

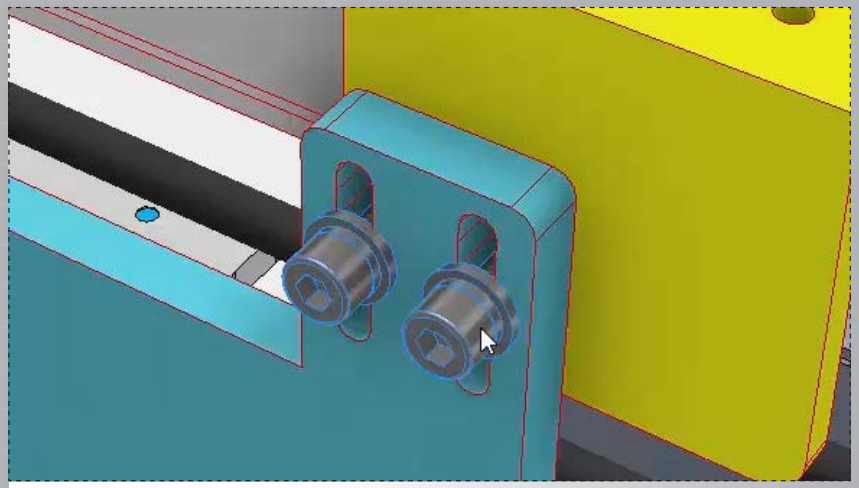
20.



Click 'OK' to apply the change, and notice how the copied subassembly updates to reflect the change as well.

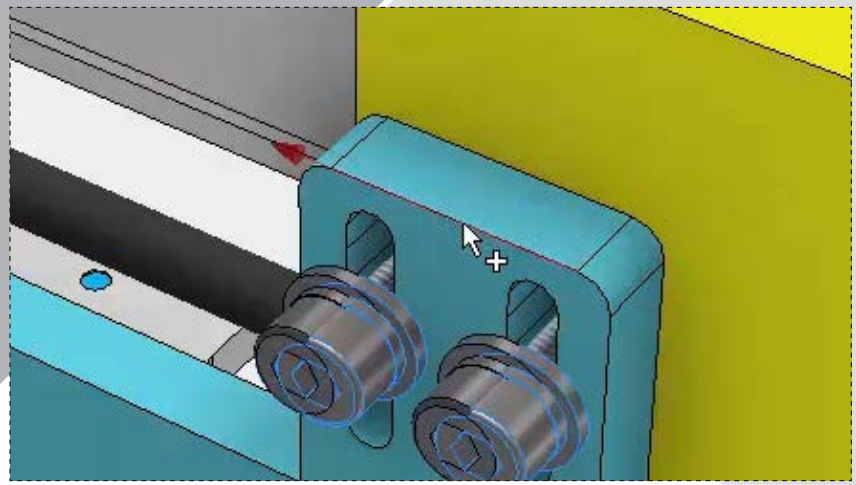
PART 1: DESIGN AUTOMATION

21.



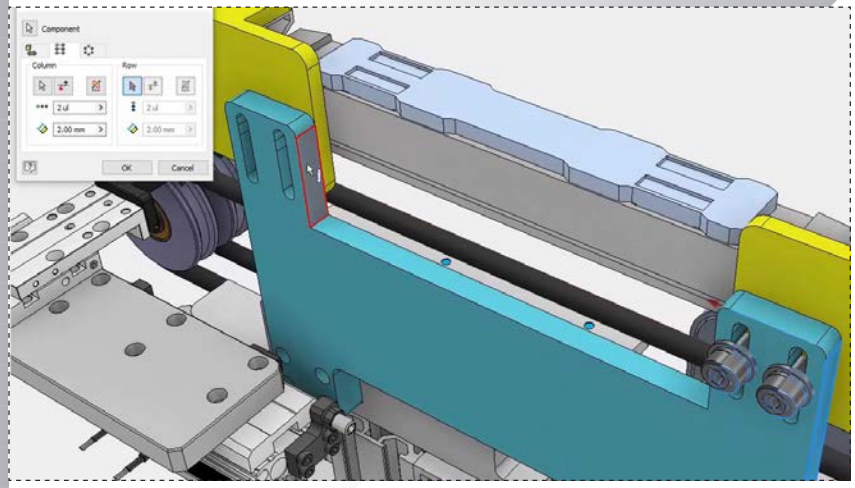
Activate the ‘Pattern’ command from the ‘Assemble’ tab in the ribbon. Select both of the bolt subassemblies for the components to pattern.

22.



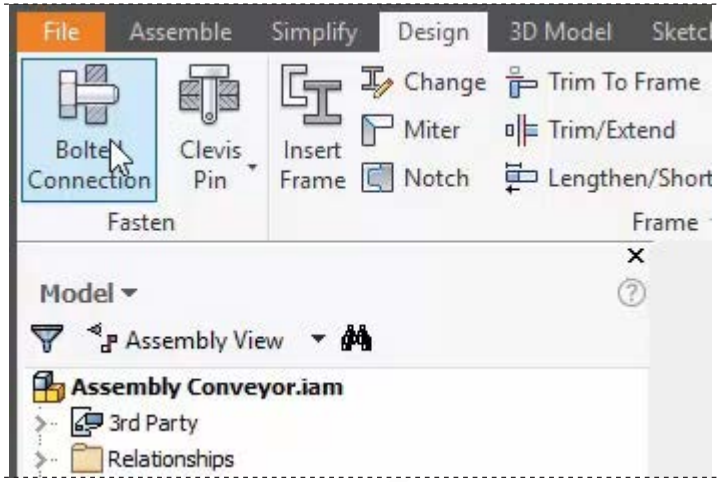
From the rectangular pattern tab in the dialog, select an edge along the X axis to define the first pattern direction.

23.



Choose ‘Measure’ for the distance option, and then select the faces shown. Click ‘OK’ to apply the pattern.

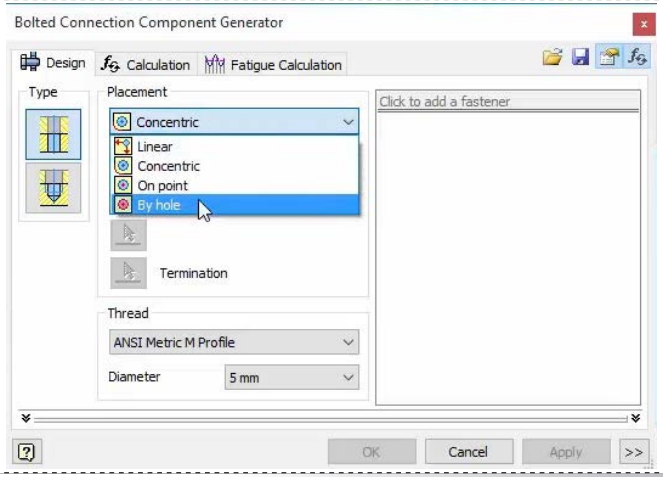
24.



Access the ‘Design’ tab in the ribbon and click ‘Bolted Connection’ again.

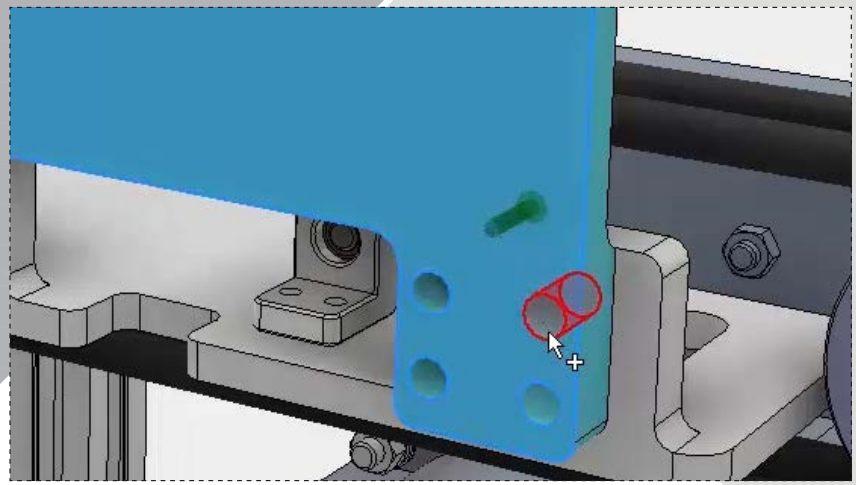
PART 1: DESIGN AUTOMATION

25.



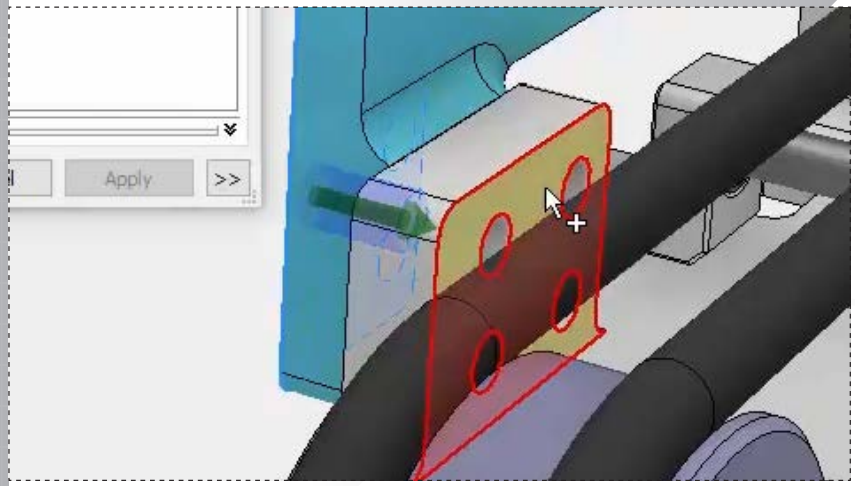
Define the ‘Placement’ as ‘By hole’.

26.



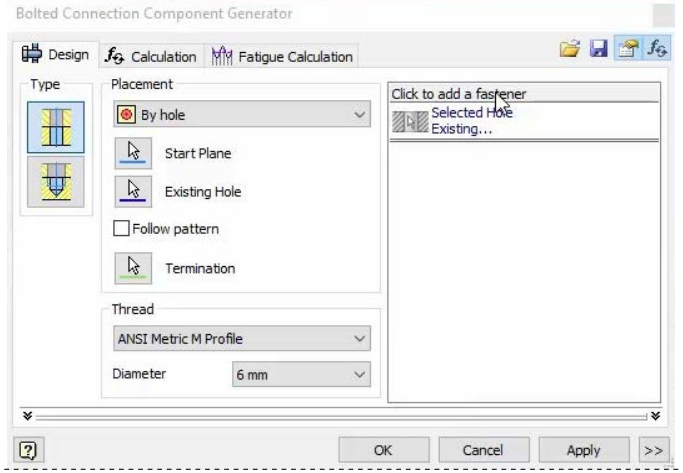
Select the front face of the blue bracket for the ‘Start Plane’, and then click on one of the holes at the bottom of the bracket for the location.

27.



Select the face on the opposite side of the mount for the ‘Termination’ definition.

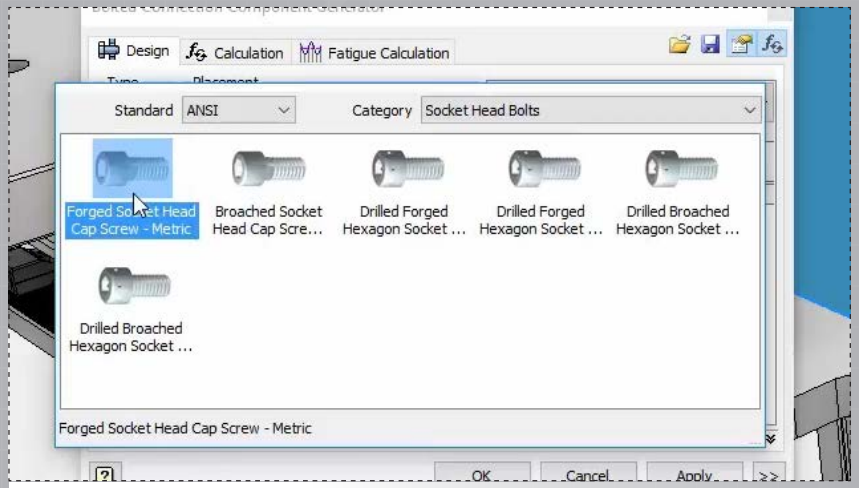
28.



Select ‘Click to add a fastener’ in the dialog.

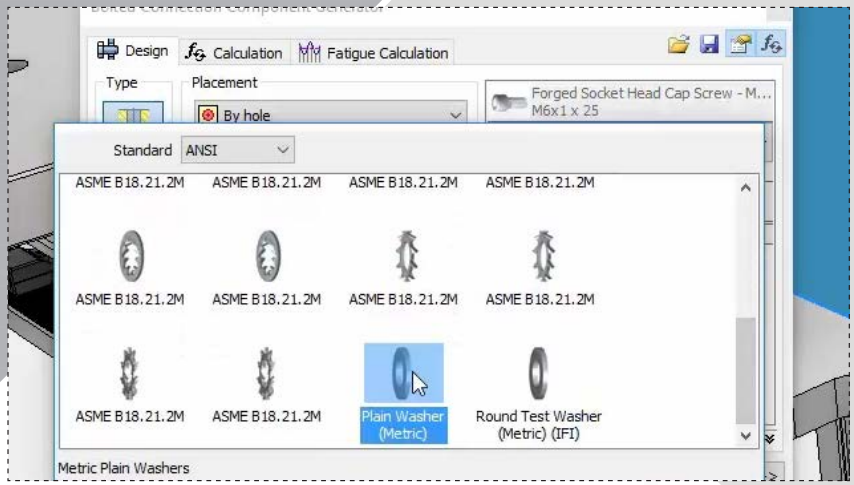
PART 1: DESIGN AUTOMATION

29.



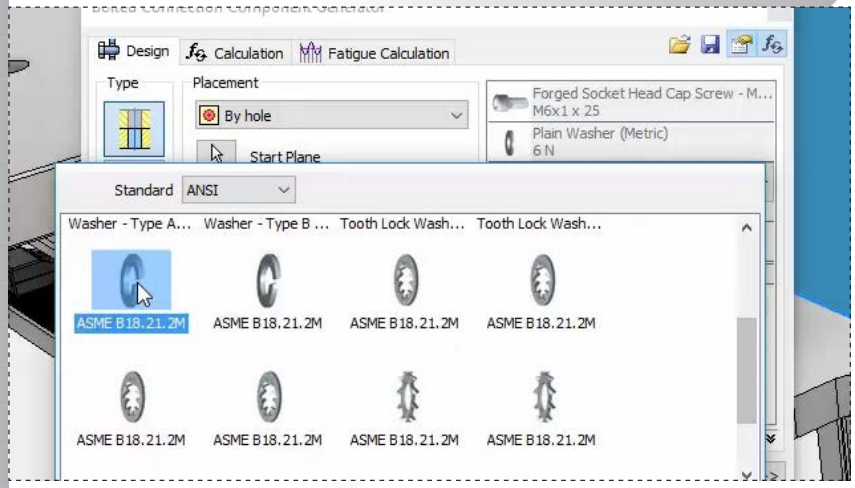
Filter to 'ANSI, Socket Head Bolts', and select the 'Forged Socket Head Cap Screw - Metric' option again.

30.



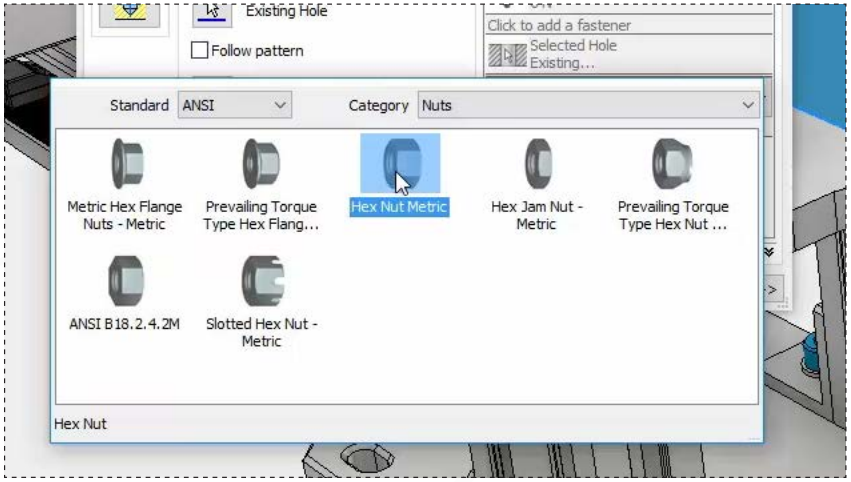
Click to add a fastener below the screw, inserting a 'Plain Washer (metric)'.

31.



Add a third fastener to the list, choosing the 'ASME B18.21.2M' lock washer, and then drag it in between the screw and plain washer in the list.

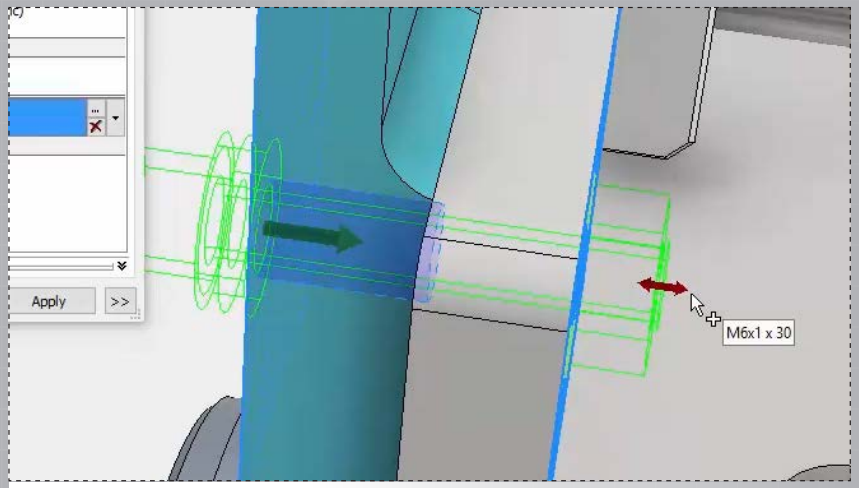
32.



Select 'Click to add a fastener' below the 'Selected Hole' listing, and then choose the 'Hex Nut Metric' component within the 'ANSI, Nuts' category.

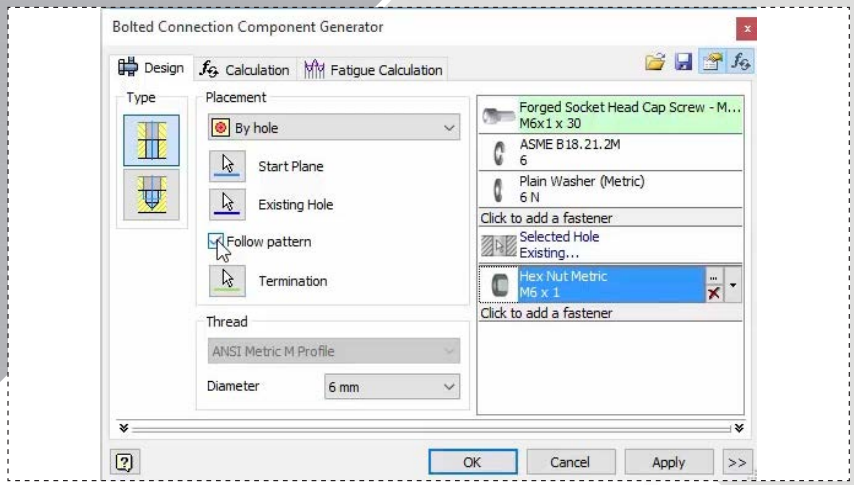
PART 1-2: DESIGN AUTOMATION

33.



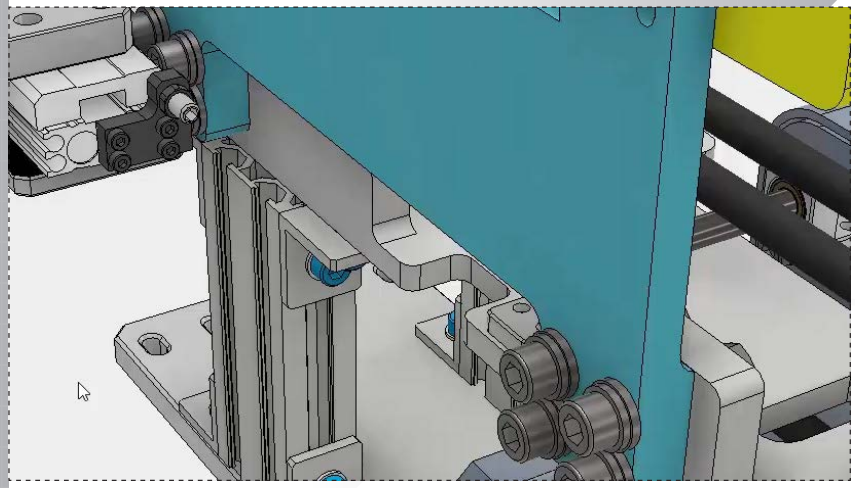
Adjust the cap screw length to 30mm.

34.



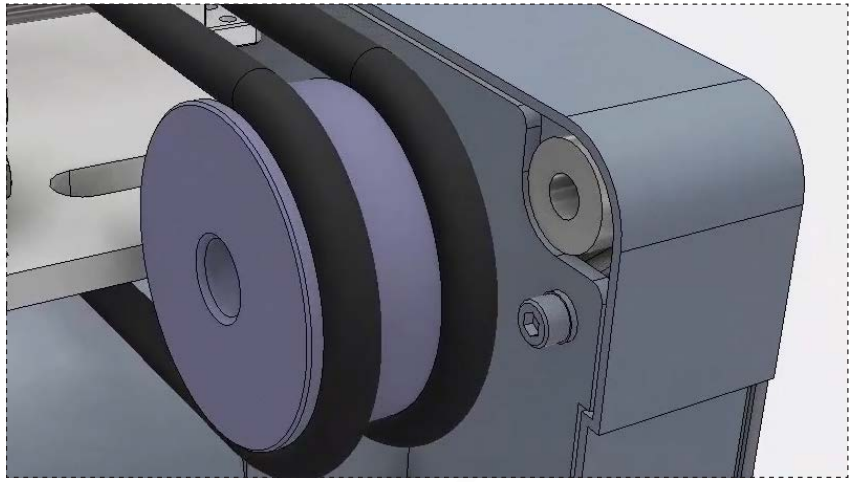
Check ‘Follow pattern’ in the dialog.

35.



Click ‘OK’ twice to place an instance at each hole in the pattern, and to accept the default ‘File Naming’. Save your progress to continue.

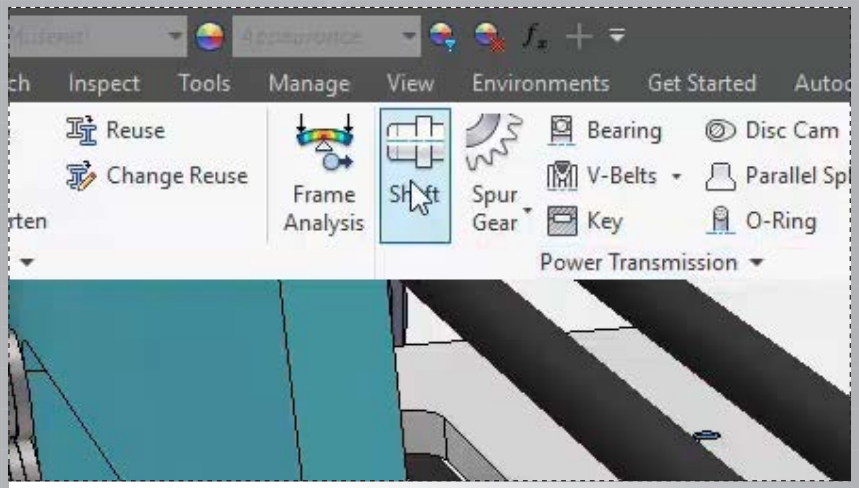
36.



Continue using ‘Design Automation’ to create a shaft connecting the motor and the drive pully.

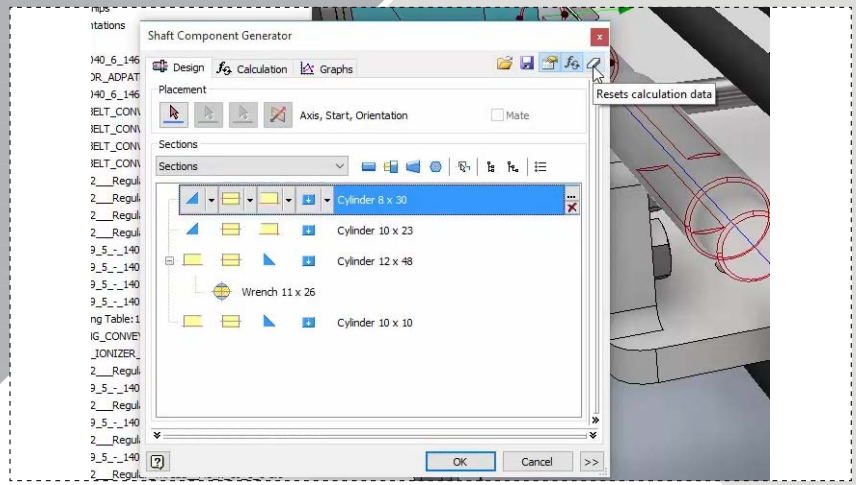
PART 2: DESIGN AUTOMATION

37.



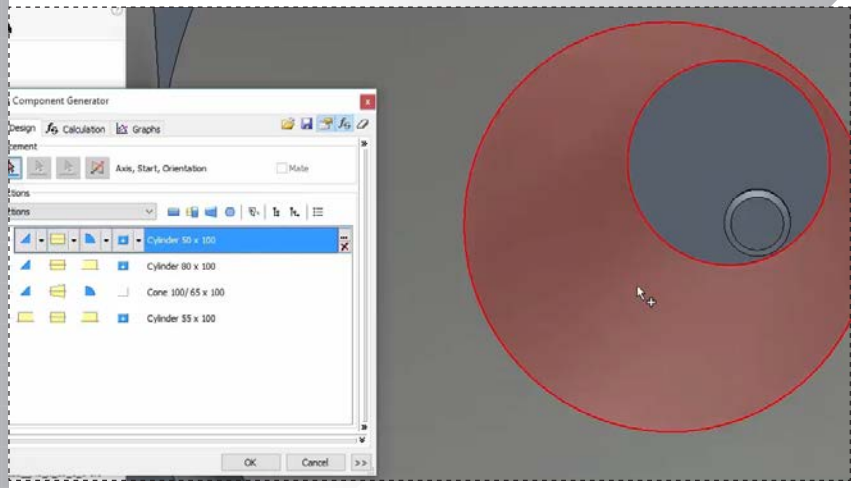
Activate the ‘Shaft’ command, located in the ‘Power Transmission’ panel on the ‘Design’ tab.

38.



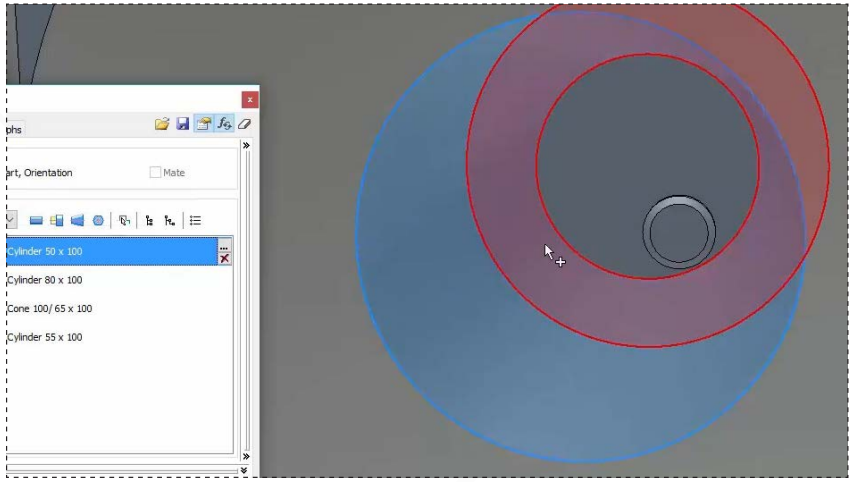
Click the eraser icon at the upper-right of the ‘Shaft Component Generator’ dialog to reset the calculation data, and then click ‘OK’.

39.



Zoom into the drive portion of the motor and select the inner cylindrical face for the first ‘Placement’ definition.

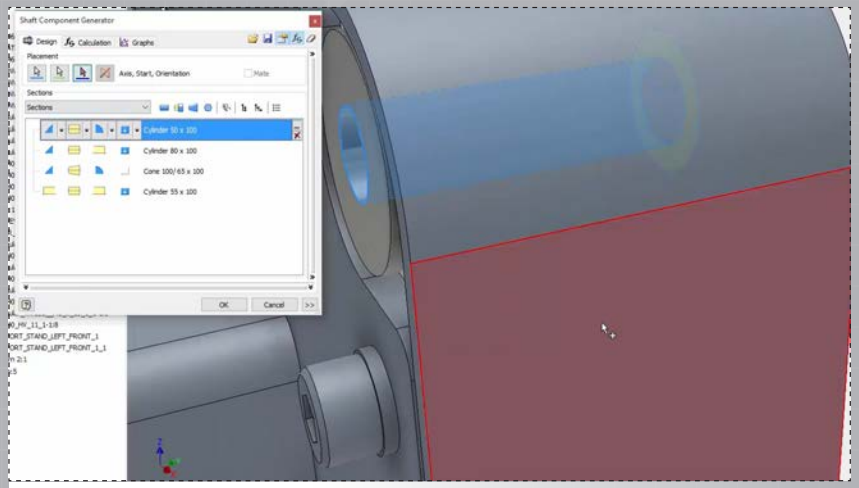
40.



Select the back face of the drive motor pully for the shaft’s ‘Start plane’ definition.

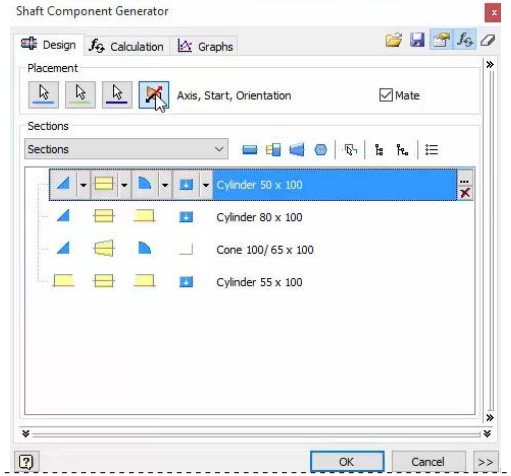
PART 2: DESIGN AUTOMATION

41.



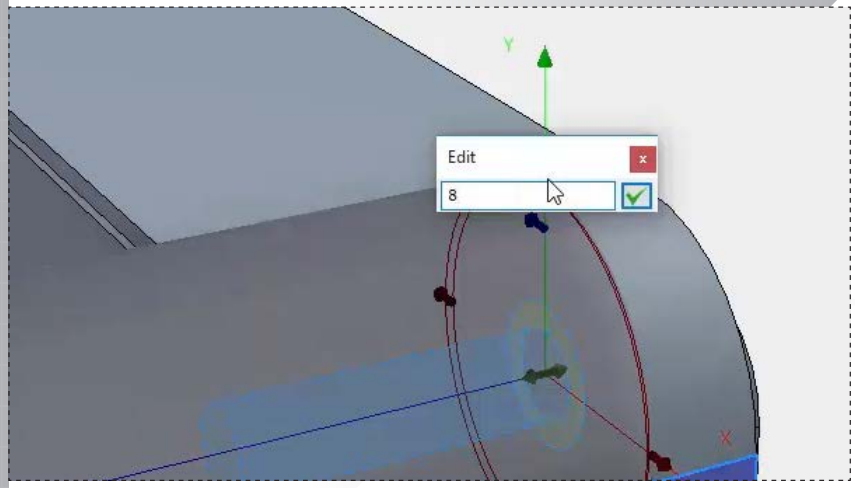
Select the outer face of the motor cover for the shaft's orientation definition.

42.



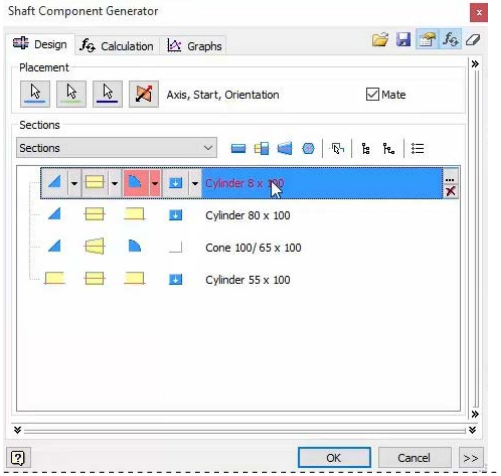
If required, use the flip direction button to ensure the shaft is oriented correctly.

43.



Double-click an arrow glyph for the first shaft section to enter a specific value of 8mm.

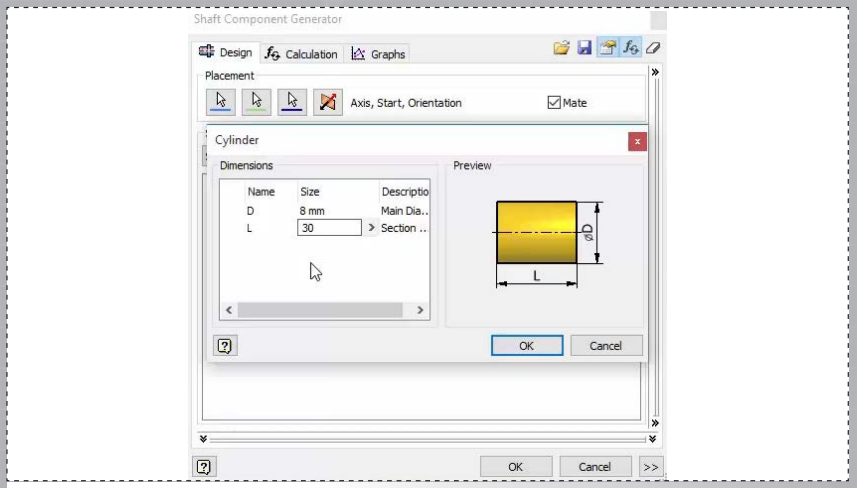
44.



Double-click the listing in the dialog to edit the section length.

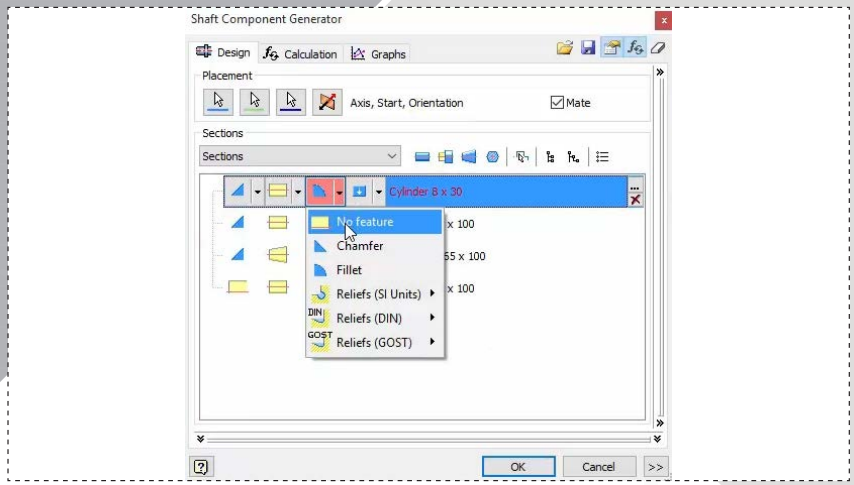
PART 2: DESIGN AUTOMATION

45.



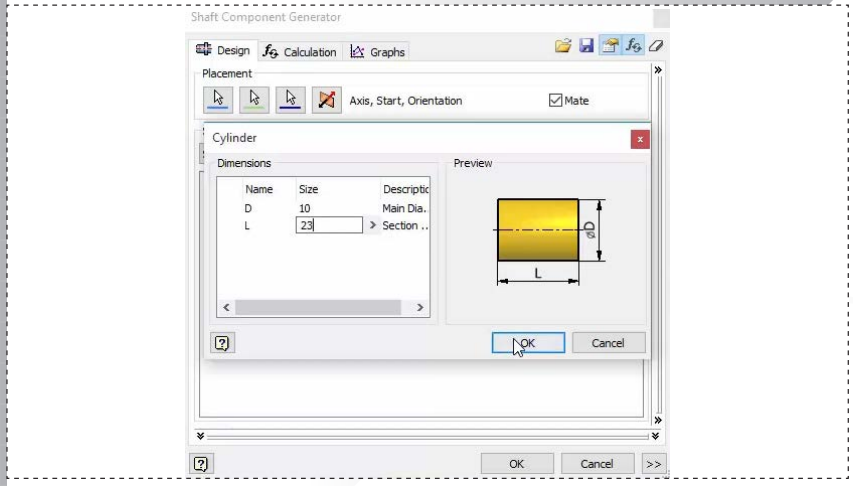
Enter a length of 30mm and click ‘OK’.

46.



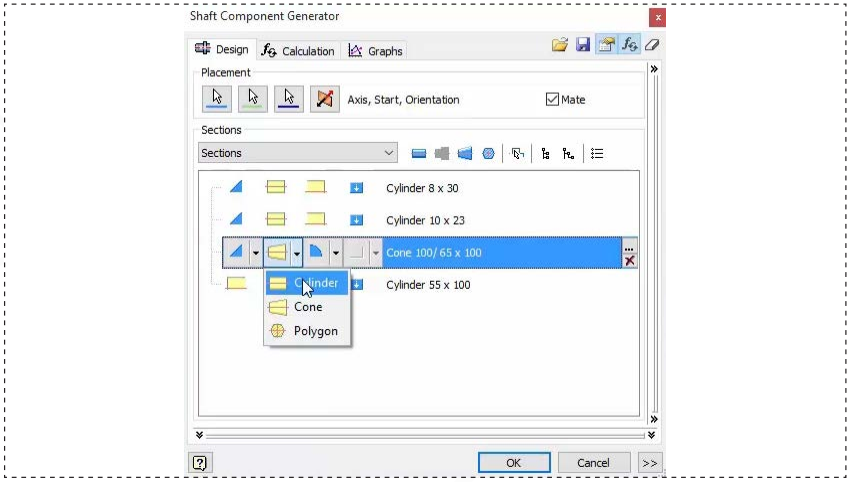
Define the second edge of the shaft section as having ‘No feature’.

47.



Double-click the next section in the list to edit it, setting a diameter of 10mm and a length of 23mm. Keep the default edge definitions as well.

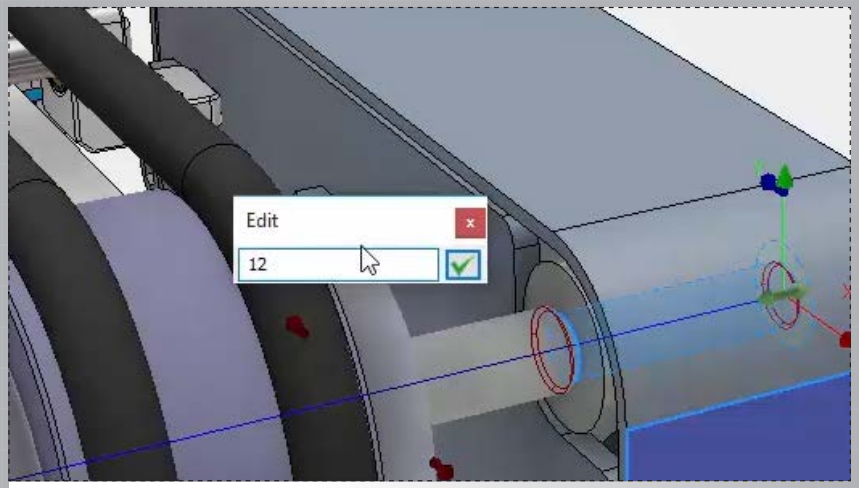
48.



Change the section type for the third listing to ‘Cylinder’ and select ‘Yes’ to confirm.

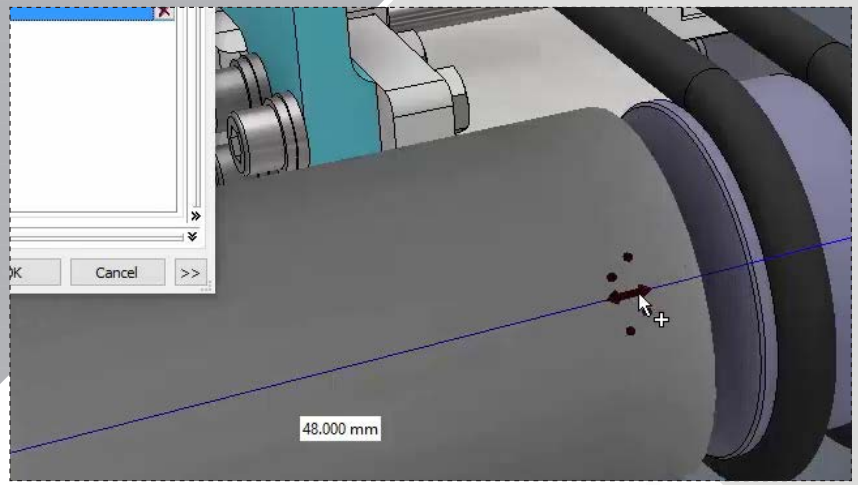
PART 2: DESIGN AUTOMATION

49.



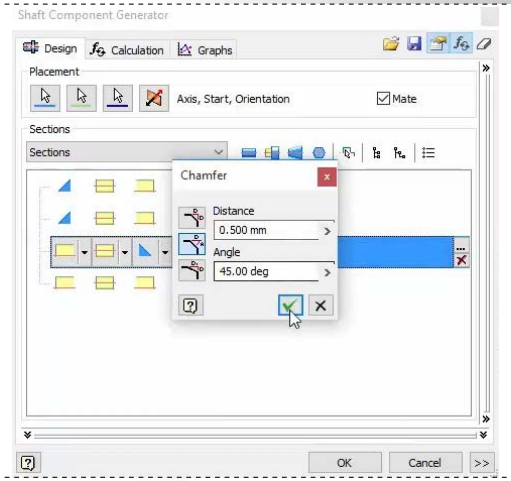
Adjust the third section's diameter, making it 12mm.

50.



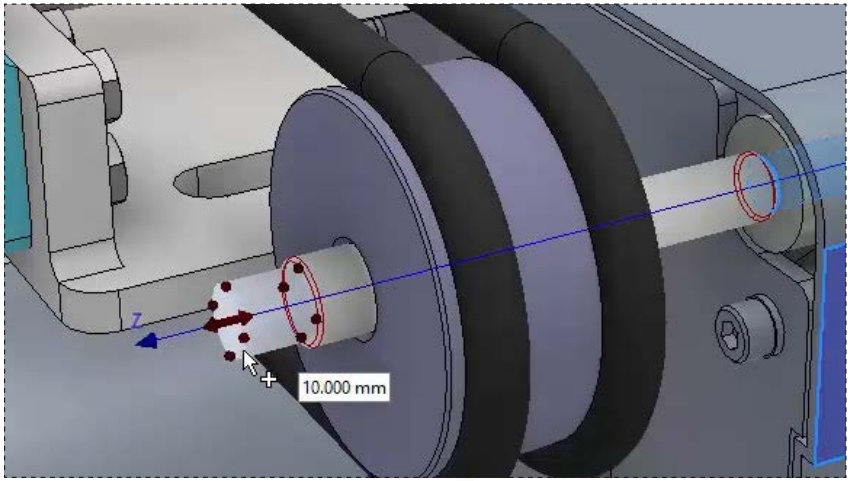
Drag the arrow glyph to set the length of this section at 48mm.

51.



Define a 'Chamfer' for the second edge of the section, using the values shown.

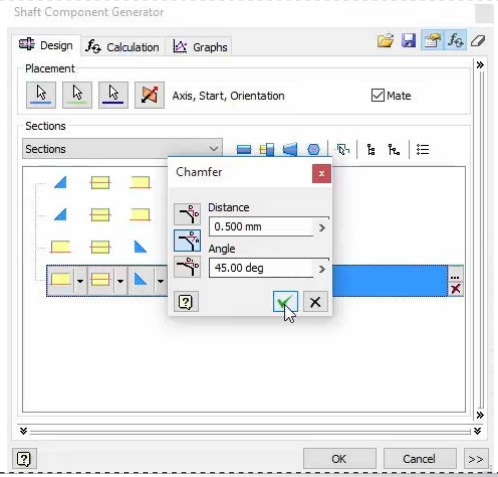
52.



For the last shaft section, drag both the diameter and length to 10mm.

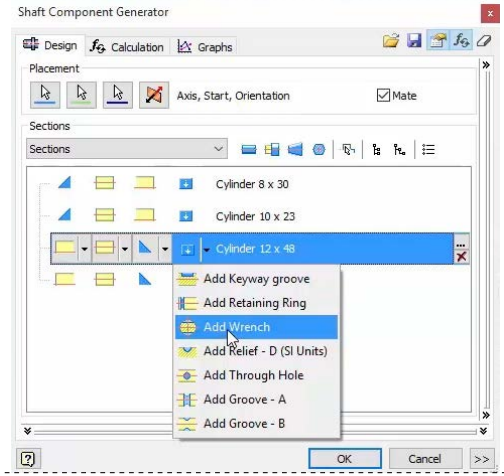
PART 2: DESIGN AUTOMATION

53.



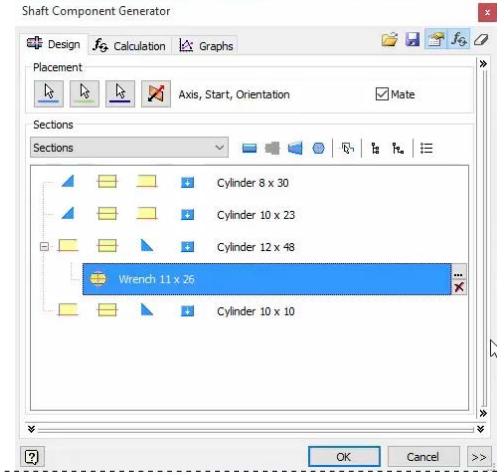
Define a ‘Chamfer’ for the second edge of the section, using a ‘Distance of 0.5mm’ and an ‘Angle of 45deg’.

54.



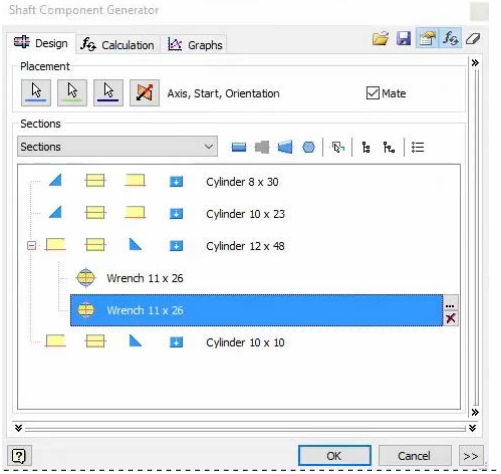
Access the section features drop-down for the third shaft section, and then select ‘Add Wrench’ to add some flat spots for the pully key onto.

55.



Define the feature at 26mm long and 11mm deep.

56.



Click ‘Add Wrench’ again to define a second feature with the same dimensions.

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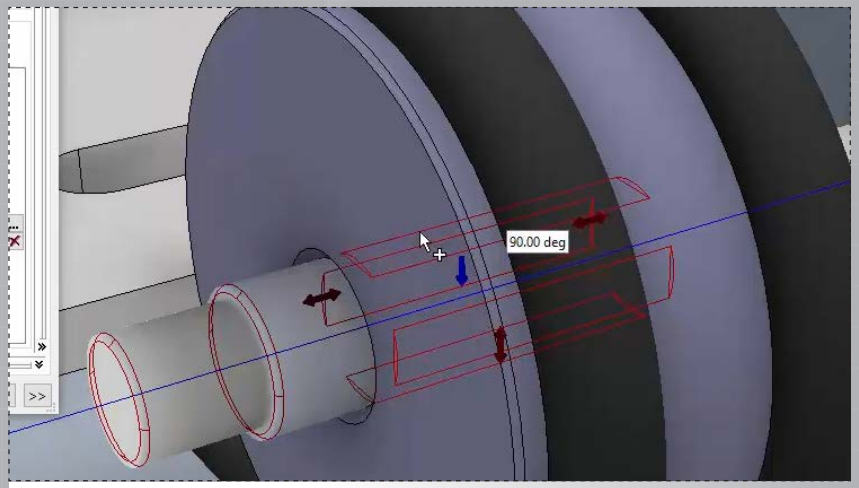
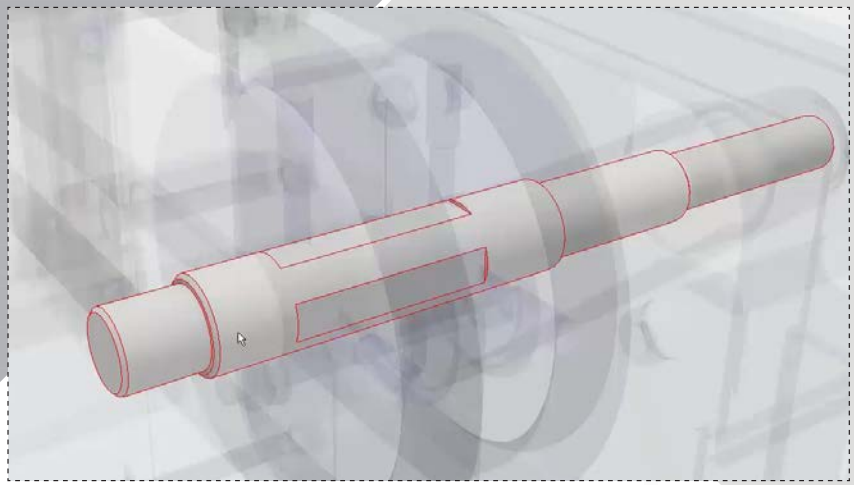
57.



Drag the blue arrow in the graphics area to rotate the second feature 90deg about the shaft. Click ‘OK’ twice to create the shaft and accept the default ‘File Naming’.

58.



Double-click on the shaft to view it clearly, and to confirm it was created as intended. Save all of your files to finish.

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