

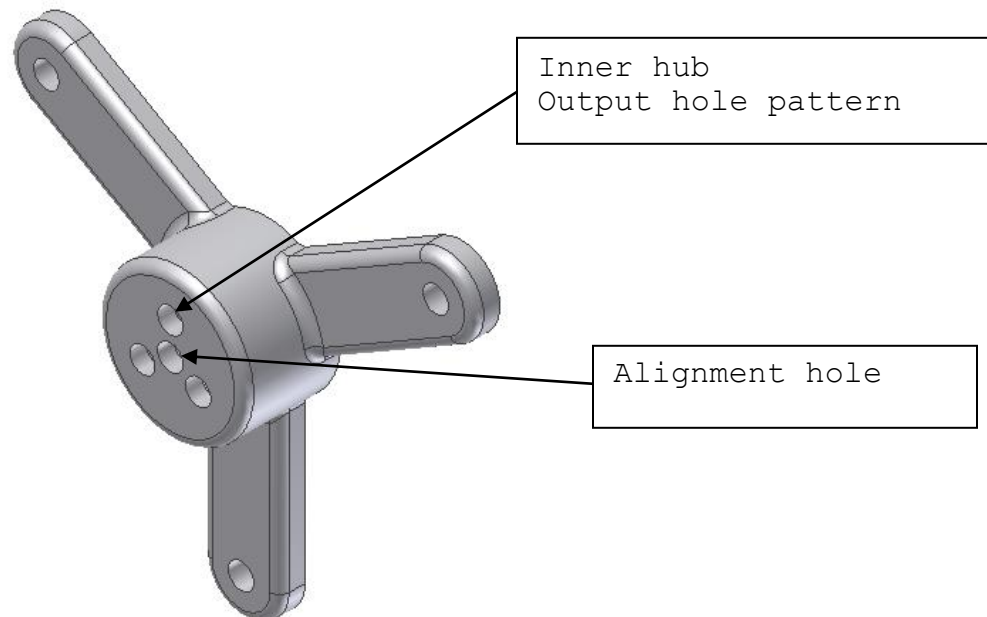
Inventor Assignment #6

Reading Assignment:

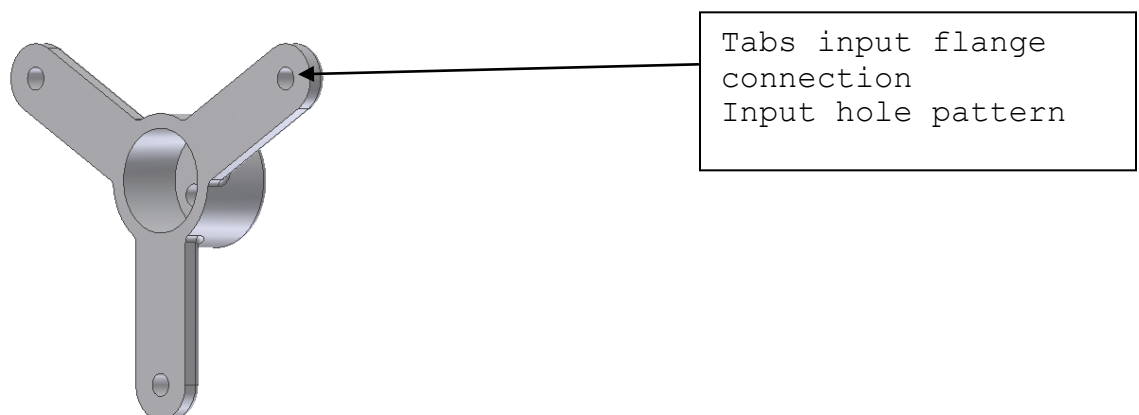
Read Chapter 6 in Parametric Modeling with Autodesk Inventor 2013, by Randy Shih. Learn about parameters in Inventor, renaming dimensions and using equations in parametric dimensions.

Computer Assignment:

1. Create a solid model (a metric part) of the object shown below with variable dimensions. The part is an adapter and some of the dimensions depend on the items connected to it. An output pulley is connected to the small bolt pattern on the inner hub.



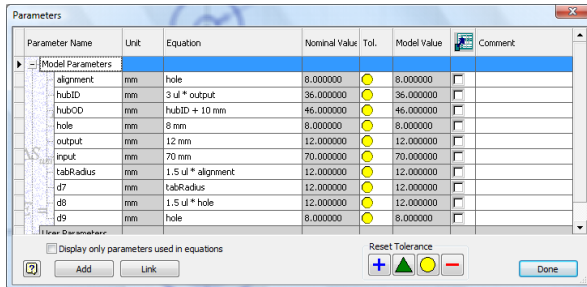
An input flange connects to the three tabs shown below:



Make the following dimensions so that they are defined using the following parametric relations. Ensure that your sketches are properly constrained. Hint, use the origin to project geometry onto sketch planes to keep things from moving around in undesired ways. As you create the part, make a note of the dimensions names Inventor uses (d1, d2, ...d8, etc.) as you apply the dimension to the following dimension parameters:

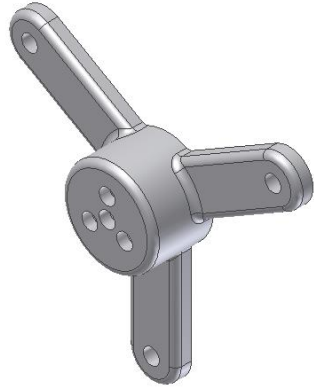
- hole = 8 mm (input and output)
- input (input flange hole radius) = 70 mm
- output = 12 mm radius
- tabRadius= 150% of hole diameter
- tabThickness = same as hole
- alignment = 8 mm diameter hole, make same as bolt pattern hole
- hubID = output * 3
- hubThickness = 5 mm
- hubOD = hubID + 2 × hubThickness
- hubLength = 30 mm
- fillets and rounds = 3 mm

- Open the parameters dialog box and revise the dimension names and create relational equations for required variable dimensions.
- Show the parameters table on the screen. Use PrintScreen to copy the table to the clipboard and paste it into a Word document. Crop the image so that only the table is seen and resize the image to fill the screen. Print the Word document showing the table and staple it to the three other sheets to be submitted (described below).

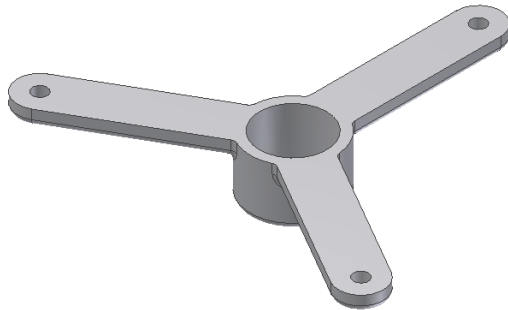


- Create a drawing using an A-size (portrait) sheet. Create front, top, and isometric views of the part with the following additional specifications:
 - Label each view.
 - Decide on a scale such that the views are as large as possible while not making the drawing too crowded. Use one of the following scales: 1=1, 1=1.25, 1=1.5, 1=1.75, 1=2, 1=2.25, etc.
 - Show dimensions on each view, except not on the isometric.
 - Pay particular attention to good style in dimensioning (i.e., follow rules covered in class)
 - Include centerlines for all holes, but not for any fillets or rounds.
 - Include the note for Rounds & Fillets (add UON).
 - Include appropriate title block information.
 - Print the drawing.
- Save your part file before proceeding. Make the following modifications to the part and print out each drawing version of the new part in each case. Modify the existing drawing file (A-size, portrait), but include a description of the modification (below) in the title block.

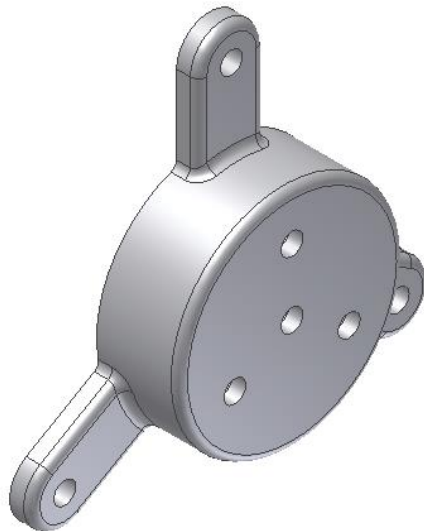
Fig. 1 Original Part (unmodified)



A. Original part except change the input flange bolt radius parameter from 70 to 100



B. Original part except change the output bolt pattern radius parameter from 20 to 30



4. Staple together the 4 printouts required and submit them to the instructor.