

Inventor Assignment #6

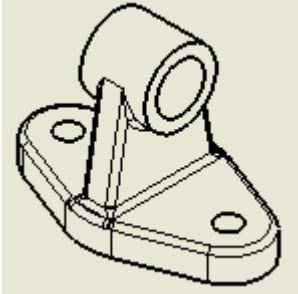
Reading Assignment:

Read Chapter 5 in Parametric Modeling with Autodesk Inventor 2009, by Randy Shih

Computer Assignment:

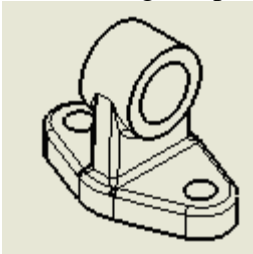
1. Create a solid model (a metric part) of the object shown in Exercise 2 of Chapter 6 in the text. Once the part has been created, change several of the dimensions as described below:
 - A. Let the following dimensions be control parameters (i.e., no change necessary, but these fixed values will be referred to by other dimensions):
 - 100 mm dimension (suppose it is called dx)
 - one of the 15 mm hole dimensions (called dy)
 - 30 mm hole dimension (called dz)
 - B. Change the following dimensions so that they are defined using the following parametric relations. Add more parametric relations or modify your expressions to preserve two planes of symmetry. The front view should be symmetric left and right and the right side view should remain symmetric front and back. Go back and correct the relationships if, later in the assignment, symmetry is not preserved. 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.
 - 46 mm hole – $46/30$ times dz
 - 40 mm dimension – 40% of dx
 - both 18 mm dimensions – 10% of dx + 8 mm
 - second 15 mm hole – dy
 - 67 mm dimension – 0.67 times dx
 - 76 mm dimension – 76% of dx
 - R40 round – 40% of dx
 - R22 round – 22% of dx
 - C. Use R3 as specified for all fillets and rounds (UON)
 - D. 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 four other sheets to be submitted (described below).
2. Create a drawing using an A-size (portrait) sheet. Create front, top, right side, 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.

3. Save your part file before proceeding. Make the following modifications to the part and save it under a different file name in each case. Modify the existing drawing file (A-size, portrait), but include a description of the modification (below) in the title block.



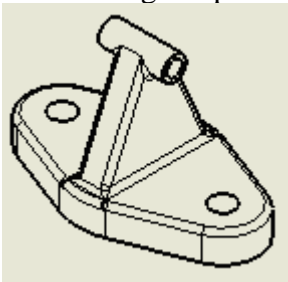
Original Part (unmodified)

- A. Original part except change the control parameter dx from 100 to 75



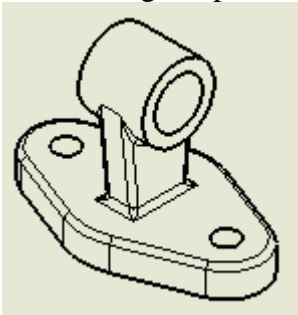
$dx = 75 \text{ mm}$

- B. Original part except change the control parameter dz from 30 to 10



$dz = 10 \text{ mm}$

- C. Original part except change the parametric relation 0.67 of dx to $1/3$ of dx



4. Staple together the 5 printouts required and submit them to the instructor.