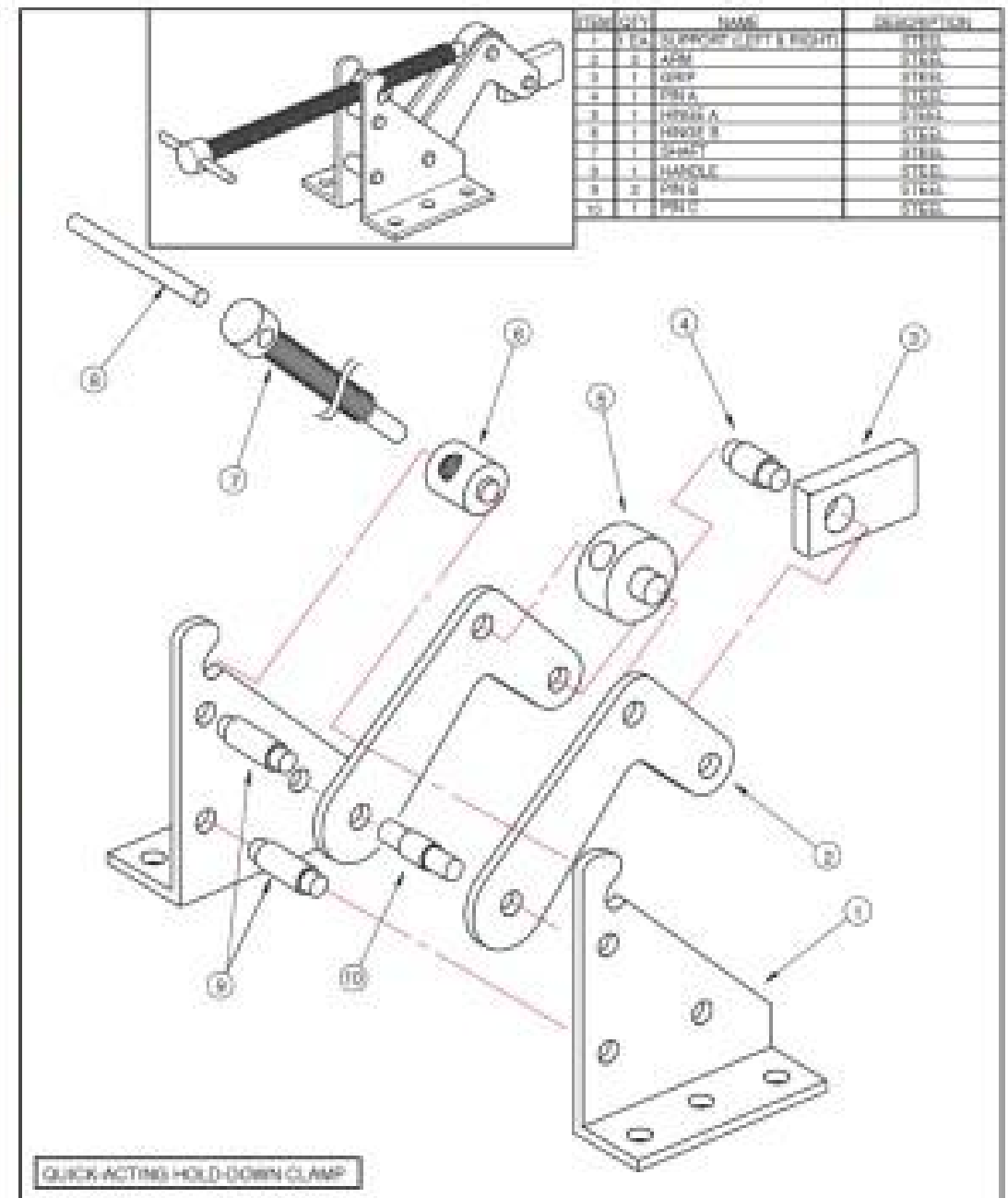
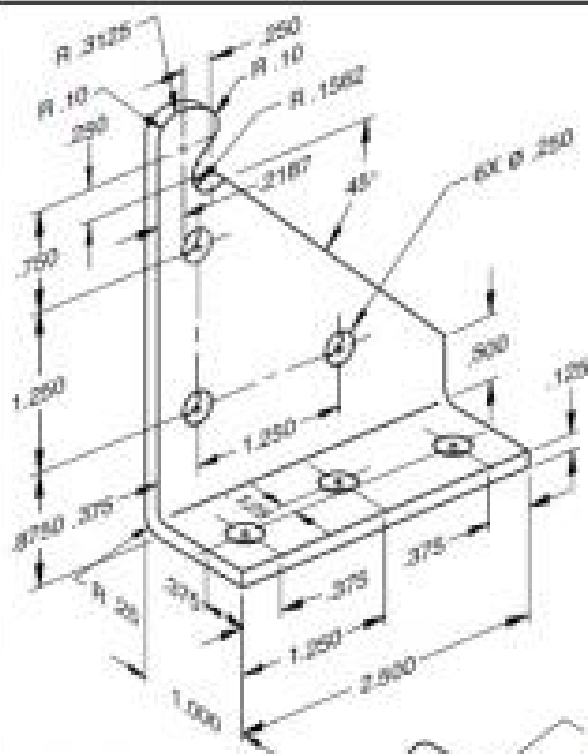


Exercises are shown below:

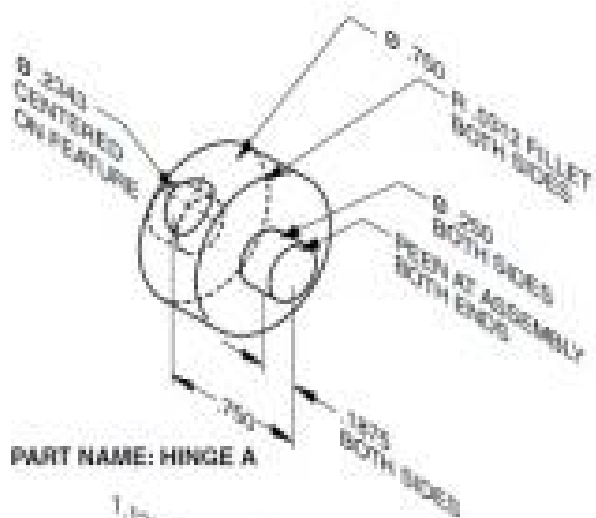
1- (20 pts.) Upload the entire assembly and also each part in the assembly you modeled in chapter 14 on blackboard. Also upload the assembly in the drawing sheet with the bill of materials (name it: tut.assembly.lab8and9; tut.collar.lab8and9; tut.bearing.lab8and9; tut.base-plate.lab8and9; tut.cap-screw.lab8and9)

2- (30 pts.) Create the 3D model of the quick-acting hold-down clamp assembly (name the assembly as: Ex2.lab8and9) upload the parts for the assembly. It is up to you what name you choose for the parts, however the names of the parts are also given. Make sure that the assembly is moving in an appropriate way. (you don't have to make the threads on the non-standard parts like part 7. However, making the threads provides extra credits)

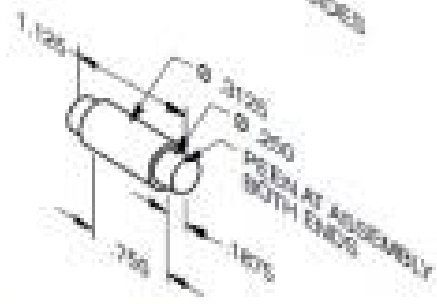




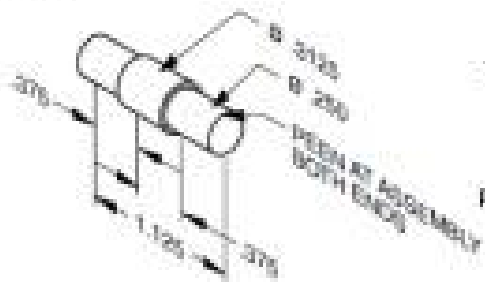
PART NAME: SUPPORT
RIGHT & LEFT NEEDED



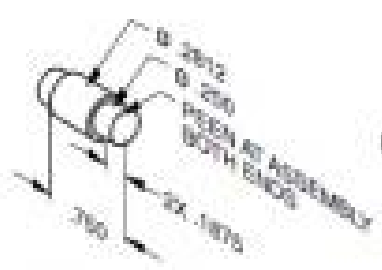
PART NAME: HINGE A



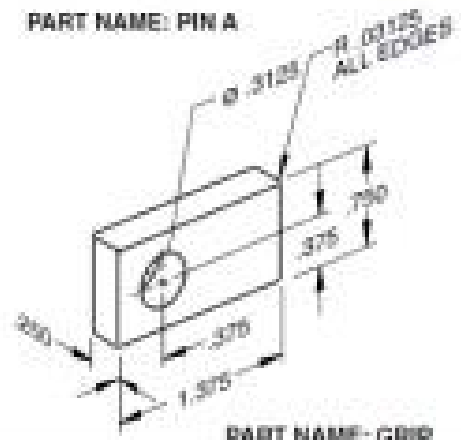
PART NAME: PIN B



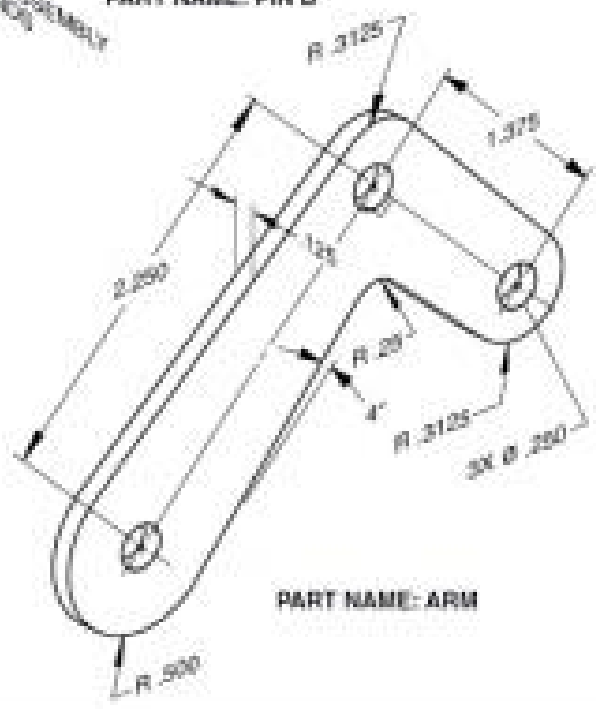
PART NAME: PIN C



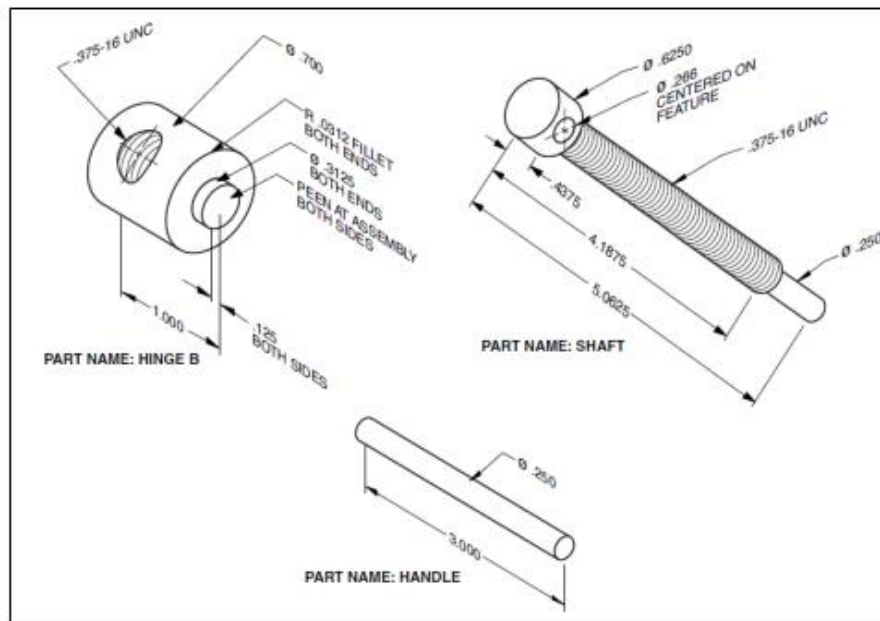
PART NAME: PIN A



PART NAME: GRIP



PART NAME: ARM



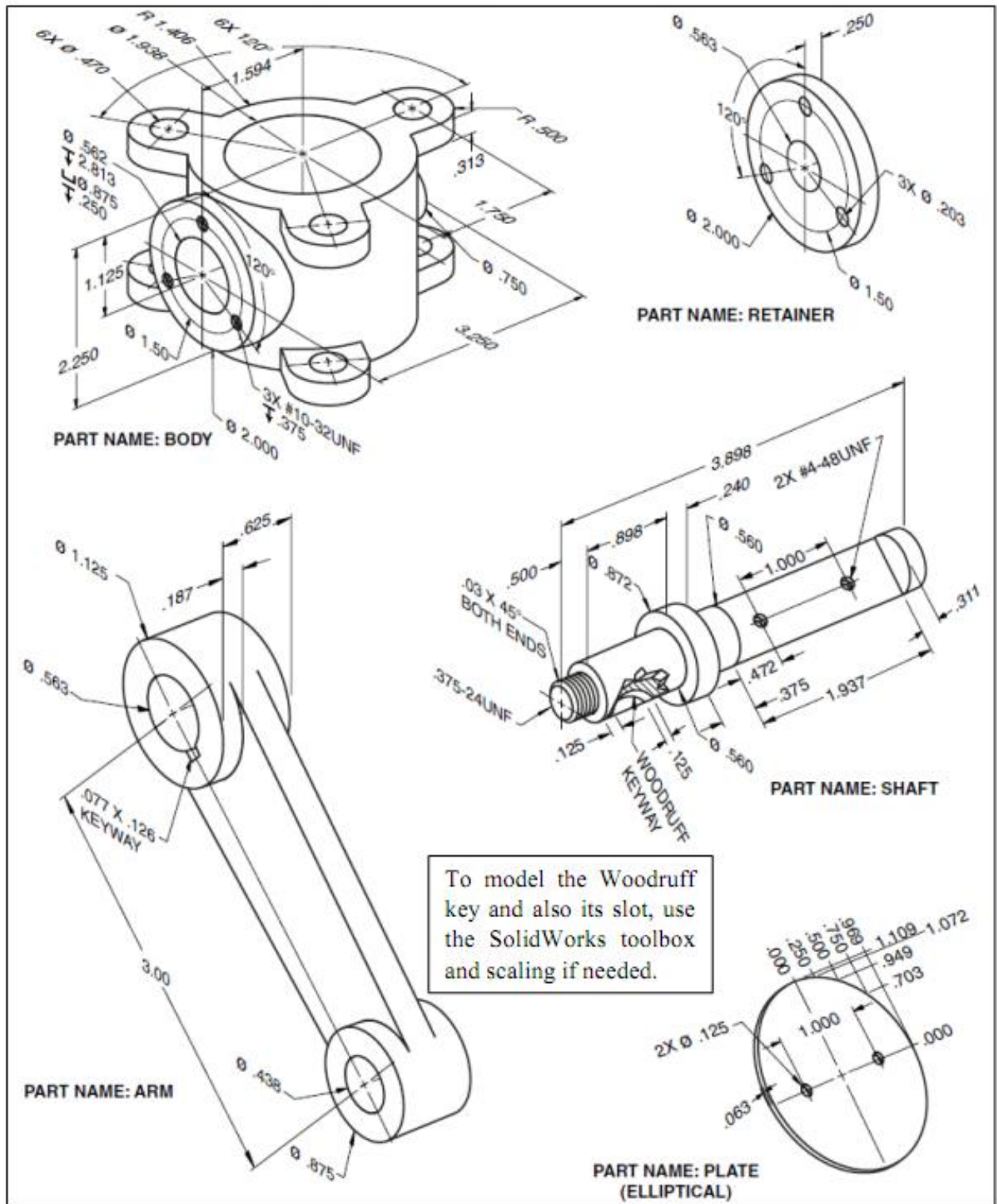
- 3- (50 pts.) (a) Create the 3D model of the assembly shown below (name the assembly as: Ex3.lab8and9) upload the parts for the assembly. It is up to you what name you choose for the parts, however the names of the parts are also given. (b) Create the exploded view of the assembly in the drawing sheet with the bill of materials (name the drawing as: Ex3.drawing.lab8and9). Make sure that the assembly is moving in an appropriate way. (you don't have to make the threads on the non-standard parts like part 4. However, making the threads provides extra credits)

ITEM	QTY	NAME	DESCRIPTION
1	1	BODY	CAST IRON
2	2	ROUND HEAD MACHINE SCREW	#4-48UNF X .250
3	1	PLATE	ALUMINIUM
4	1	SHAFT	STEEL
5	1	WOODRUFF KEY	
6	1	RETAINER	STEEL
7	3	ROUND HEAD MACHINE SCREW	#10-32UNF X .500
8	1	ARM	STEEL
9	1	HEX HEAD NUT	.375-24UNF

To model the bolt and nuts, use the SolidWorks toolbox and the description in the bill of materials.

ASSEMBLED SECTION

BUTTERFLY VALVE



You can use ellipse tool in SolidWorks. One method is to use the complete ellipse and enter the minor and major radii. Another method is to pass a curve from the given points. The first method is easier to apply.