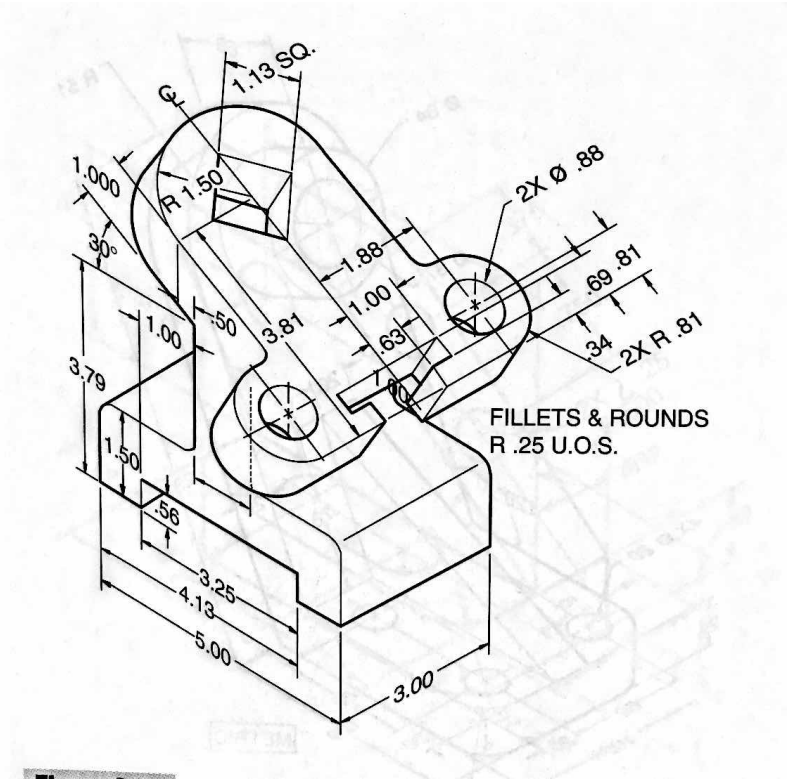
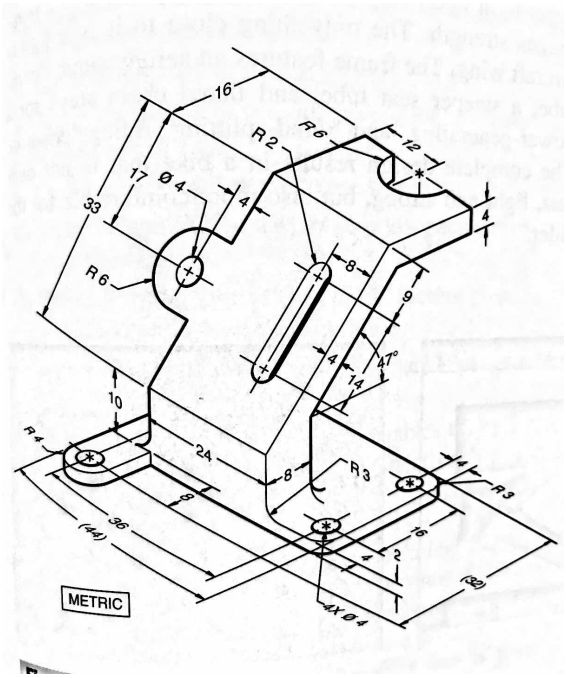
Lab 6

Due date: Monday November 21, 9:00AM

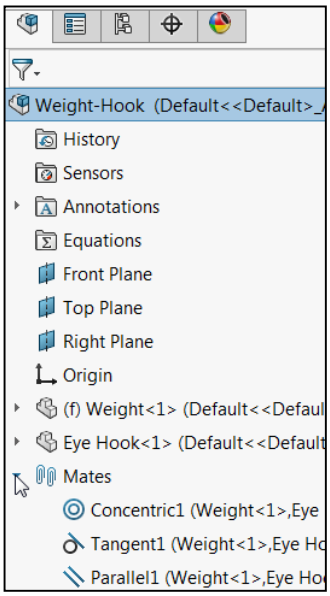
1. The lab extracts segments from the SOLIDWORKS Tutorial book where noted.
2. Files needed for this lab are uploaded to Courseworks.
3. For this Lab assignment, start a new Word Document and type your name/uni/date at the top. Use this document to collect the screen shots where noted and to answer all questions in the Lab exercises. Click "Save As" and use type "PDF". Use file name, “UNI_Lab5_WordDoc.pdf”.
4. Upload and Submit “UNI_Lab6_WordDoc.pdf”.
5. Upload and Submit all associated SOLIDWORKS PART, ASSEMBLY, and DRAWING files in a zip folder using file name, “UNI_Lab6_PartFiles.zip”.
6. Upload and Submit additional PDF files of the engineering drawings where indicated using the file name, “UNI_Lab6_DRAWINGNAME.PDF”
7. Ask in lab if you still have questions.

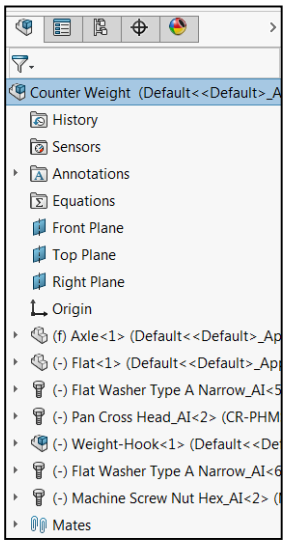
PROBLEM 1. Create the following Solidworks Parts and create the corresponding engineering drawings w/ the appropriate auxiliary views.





Problem 2:





Create the Counter Weight Assembly. Perform Exercise 3.15 in the SOLIDWORKS Tutorial book.

Copy all needed components from Courseworks.

Copy and open other required components (fasteners) from the SOLIDWORKS Toolbox. Apply SmartMates to all fasteners in the assembly. Use the Pack and Go tool to break and save all library references.

Create the **Counter Weight** assembly as illustrated. The Counter Weight consists of the following components and sub-assembly:

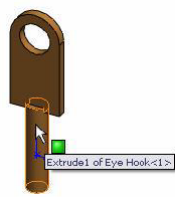
- Weight-Hook sub-assembly. (You create).
- Axle component. Fix to the origin.
- Flat component.
- Flat Washer Type A. From the SOLIDWORKS Toolbox. (SmartMate).
- Pan Cross Head Screw. From the SOLIDWORKS Toolbox. (SmartMate).
- Flat Washer Type A. From the SOLIDWORKS toolbox. (SmartMate).
- Machine Screw Nut Hex. From the SOLIDWORKS Toolbox. (SmartMate).

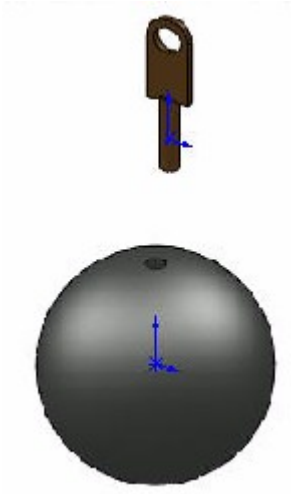
Read the section on SmartFasteners and SmartMates in SOLIDWORKS help.

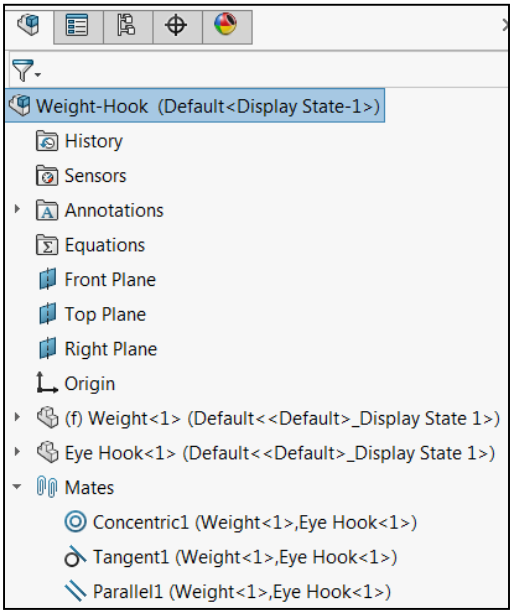
Item 2A:

Create a sub-assembly called Weight-Hook. The sub-assembly consists of two components: Weight and Eye Hook.

Insert all needed mates.

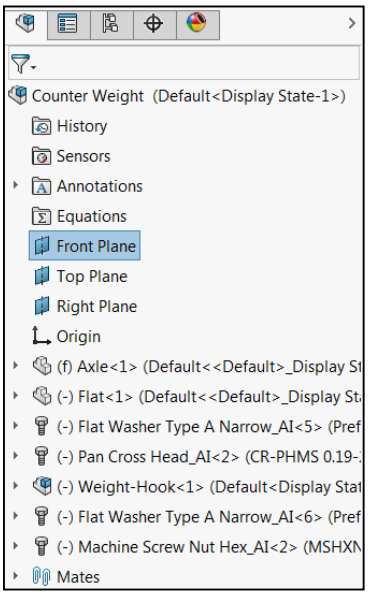






Item 2B:

Create the Final Assembly (Counter Weight) as illustrated.



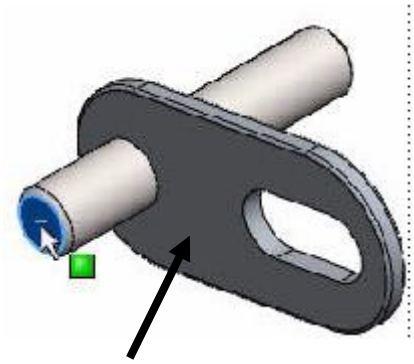
Think about the first component.
Insert the Axle component.

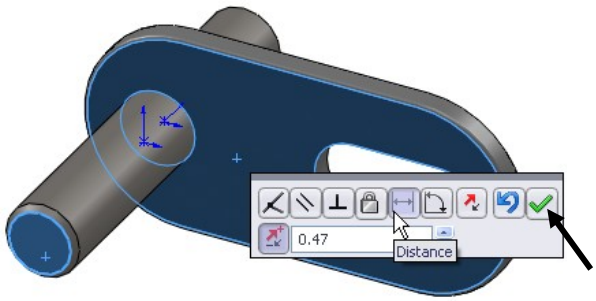
Insert the FLAT part as illustrated.

Insert a Concentric mate between the cylindrical face of the Axle and the cylindrical face of the small Flat hole component.

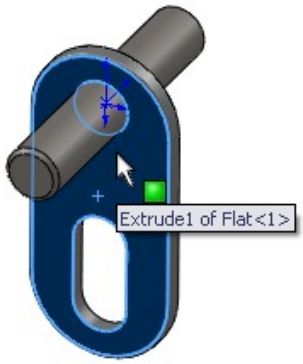
Insert a Distance mate (.47in) between the flat face of the Axle and the front flat face of the Flat component.

You can all so insert a Width mate to replace the distance mate.

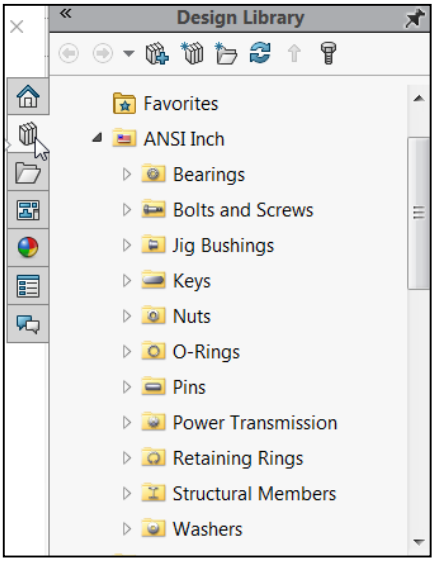




In the next section, insert components (fasteners) from the SOLIDWORKS Toolbox.



Insert a Washer from the SOLIDWORKS Toolbox. Use the SOLIDWORKS SmartMate feature.



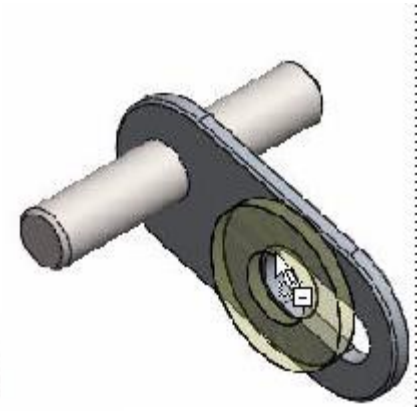
Display the SOLIDWORKS Design Library.

Expand the Toolbox folder.
Expand the ANSI Inch folder.
Expand the Washers folder.

Click the Plain Washers (Type A) folder. Various types of Machine washers are displayed.

Drag and drop the Preferred-Narrow Flat Washer Type A on the upper arc edge of the Flat component as illustrated (Zoom-in to select the correct edge).

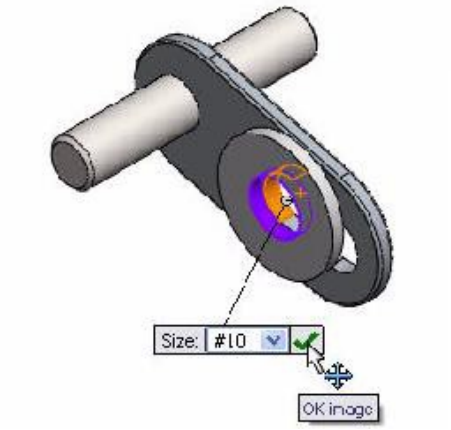
Note the icon feedback symbol.



The Preferred - Narrow Flat Washer Type A PropertyManager is displayed. The Washer in your picture will be displayed larger than needed at this time.

Select 10 for size. View the PropertyManager.

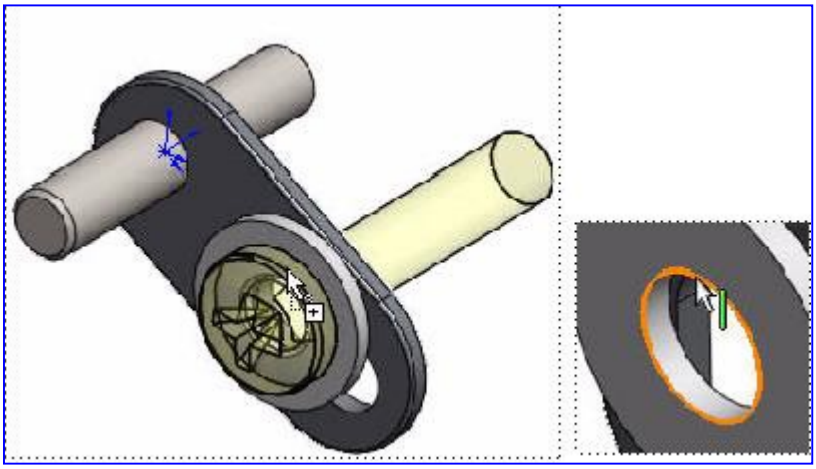
Accept the default settings.



Do not insert a second Washer at this time.

Display an Isometric view. View the Flat Washer Type A Narrow part, FW 0.19 in the FeatureManager. You just created a Concentric and a Coincident mate using the SW SmartMate feature.

Insert a Pan Cross Head screw from the SOLIDWORKS Toolbox. (SmartMate)



Expand the Toolbox folder.

Expand the ANSI Inch folder.

Expand the Bolts and Screws folder. View your options.

Click the Machine Screws folder. Various types of Machine Screws are displayed.

Drag and drop the Pan Cross Head Screw on the edge of the Washer as illustrated. (Zoom in to select the proper edge). Note the icon feedback symbol. The Pan Cross Head PropertyManager is displayed.

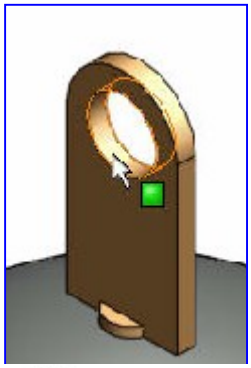
Select 10-24 for size. View your options.

Select **1** for Length.

Accept the default settings. View the Pan Cross Head part, CR-PHMS from the FeatureManager. Note: Threads are suppressed on all SW Toolbox components.

Insert the Weight-Hook sub-assembly.

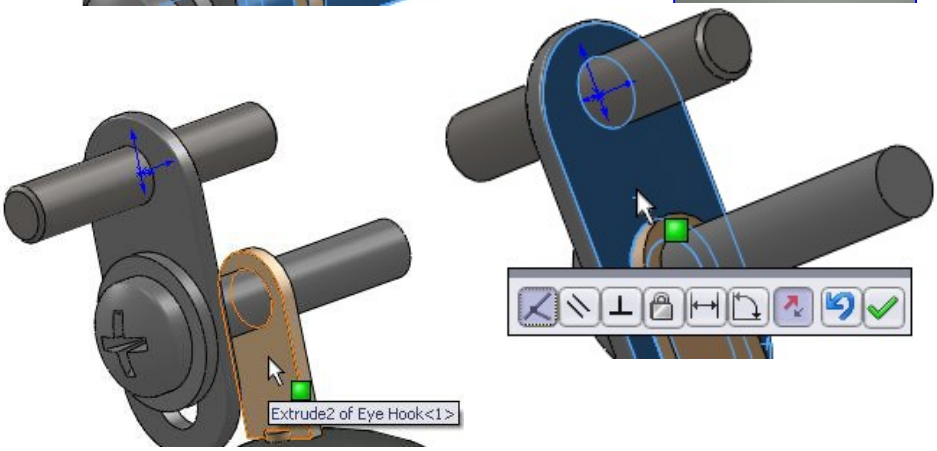
Click the Insert Components tool from the Assembly toolbar. Insert the Weight-Hook assembly. If Weight-Hook is an active document, double-click Weight-Hook in the Open documents box to open the assembly.



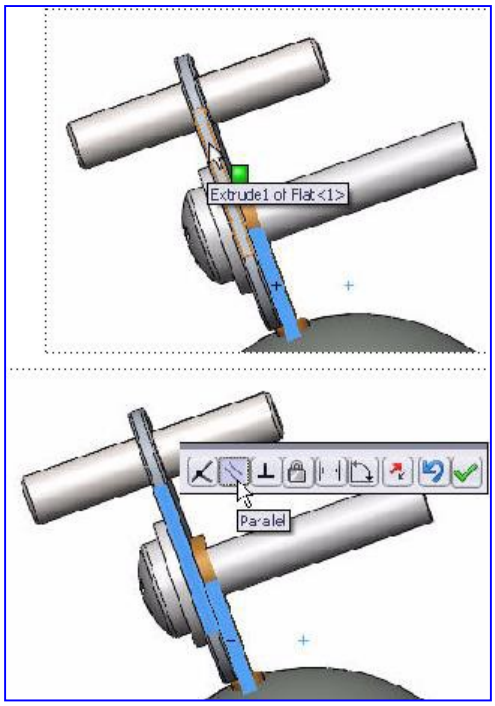
Create a Concentric mate between the cylindrical face of the Pan Cross Head shaft and the inside Eye Hook cylindrical face.

Create a Coincident mate between the front face of the Flat part and the back face of the Eye Hook.

Currently the Weight-Hook rotates about the Axle.



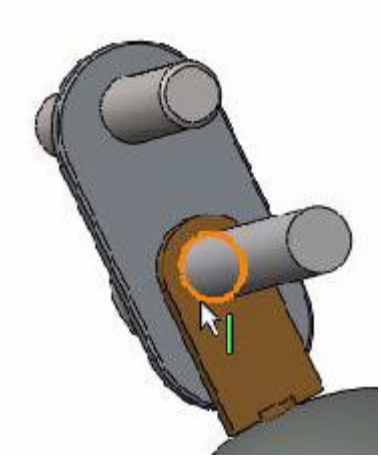
Insert a Parallel mate to control the Weight movement with the Flat component. Insert a Parallel mate between the vertical face of the Flat and the vertical face of the Weight-Hook.



Insert the second Washer from the SW toolbar. View the mating edge.

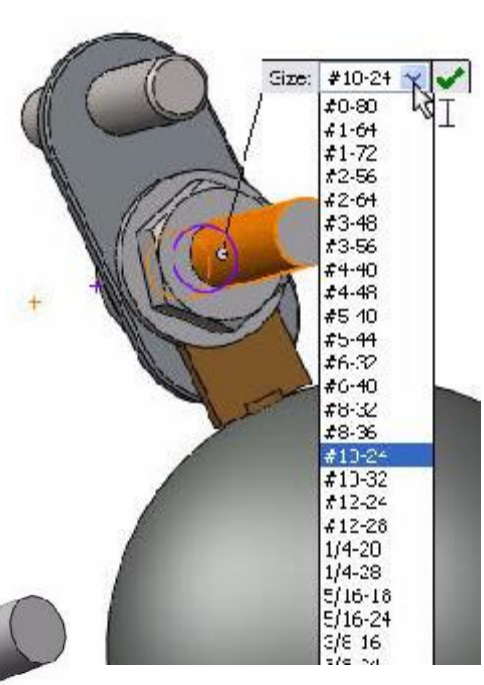
Follow the above procedures to insert your second Preferred-Narrow Flat Washer Type A component from the SW toolbox. Apply a SmartMate to the illustrated edge.

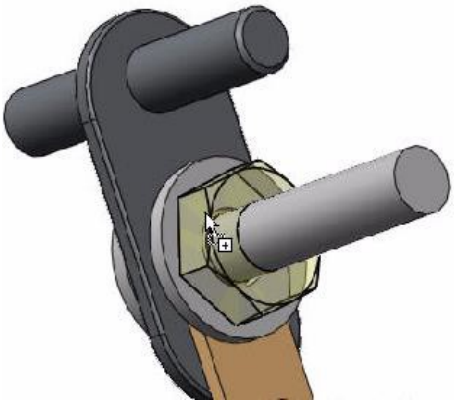
Zoom in if needed to select geometry.



Insert a Machine Screw Hex Nut from the SOLIDWORKS toolbox.

Select 10-24 for size. Accept the default settings. Apply a SmartMate to the circular edge of the second washer. Note: Threads are suppressed on all SW Toolbox components. Un-suppress all thread on the final assembly.



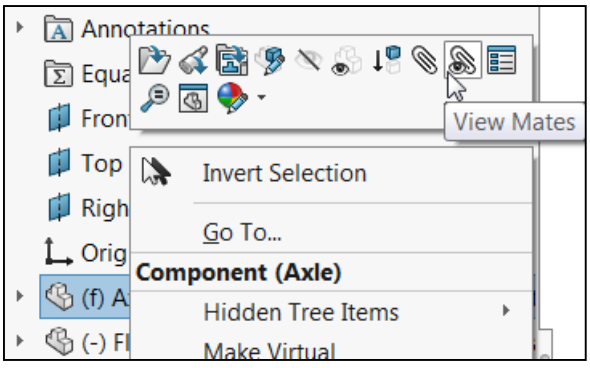


Un-suppress the Threads on your SOLIDWORKS Toolbox components.

Insert a Lock Mate on the Nut and Screw.

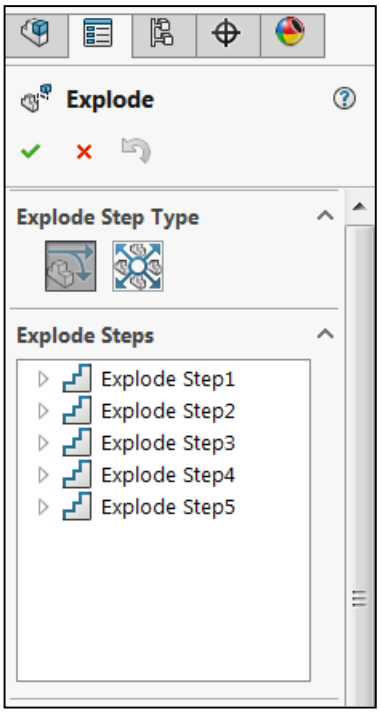
View the mates in your Assembly. Right-click Counter Weight in the FeatureManager as illustrated.

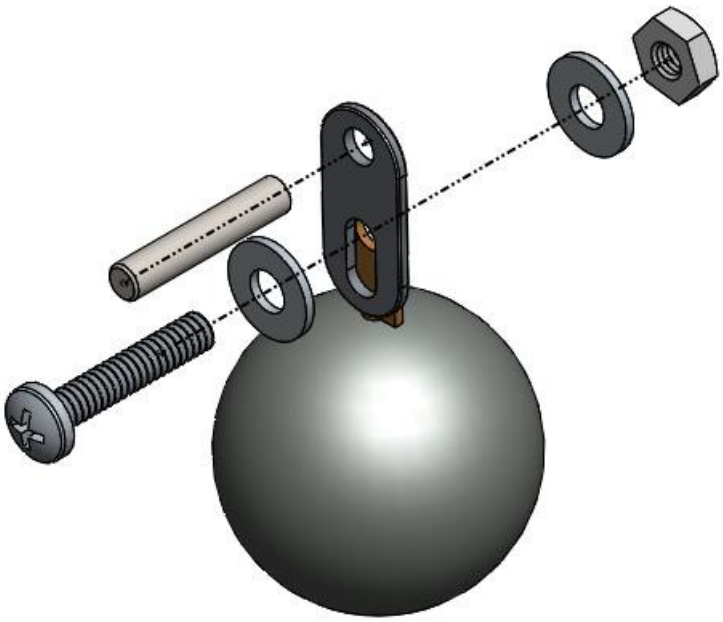
Click View Mates from the Context toolbar. The View Mates PropertyManager is displayed with a list of the component's mates.



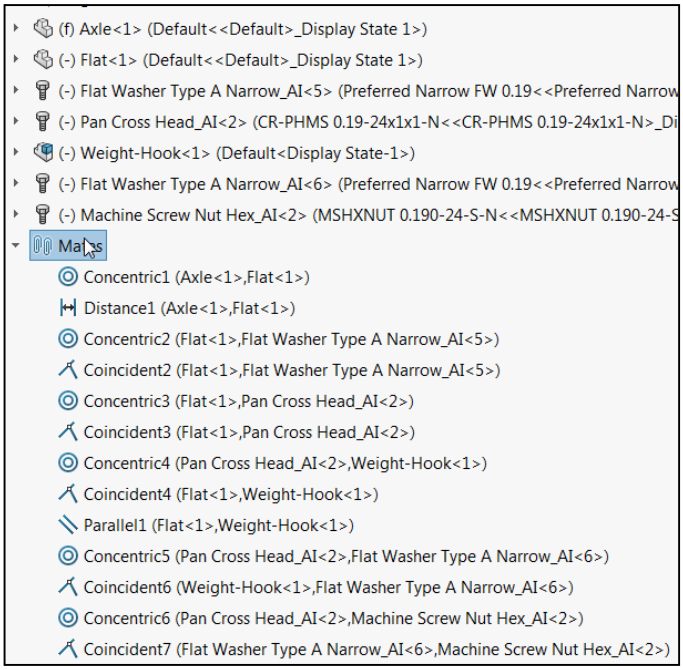
Item 2C:

Create an Exploded Assembly document with Explode Lines as illustrated.





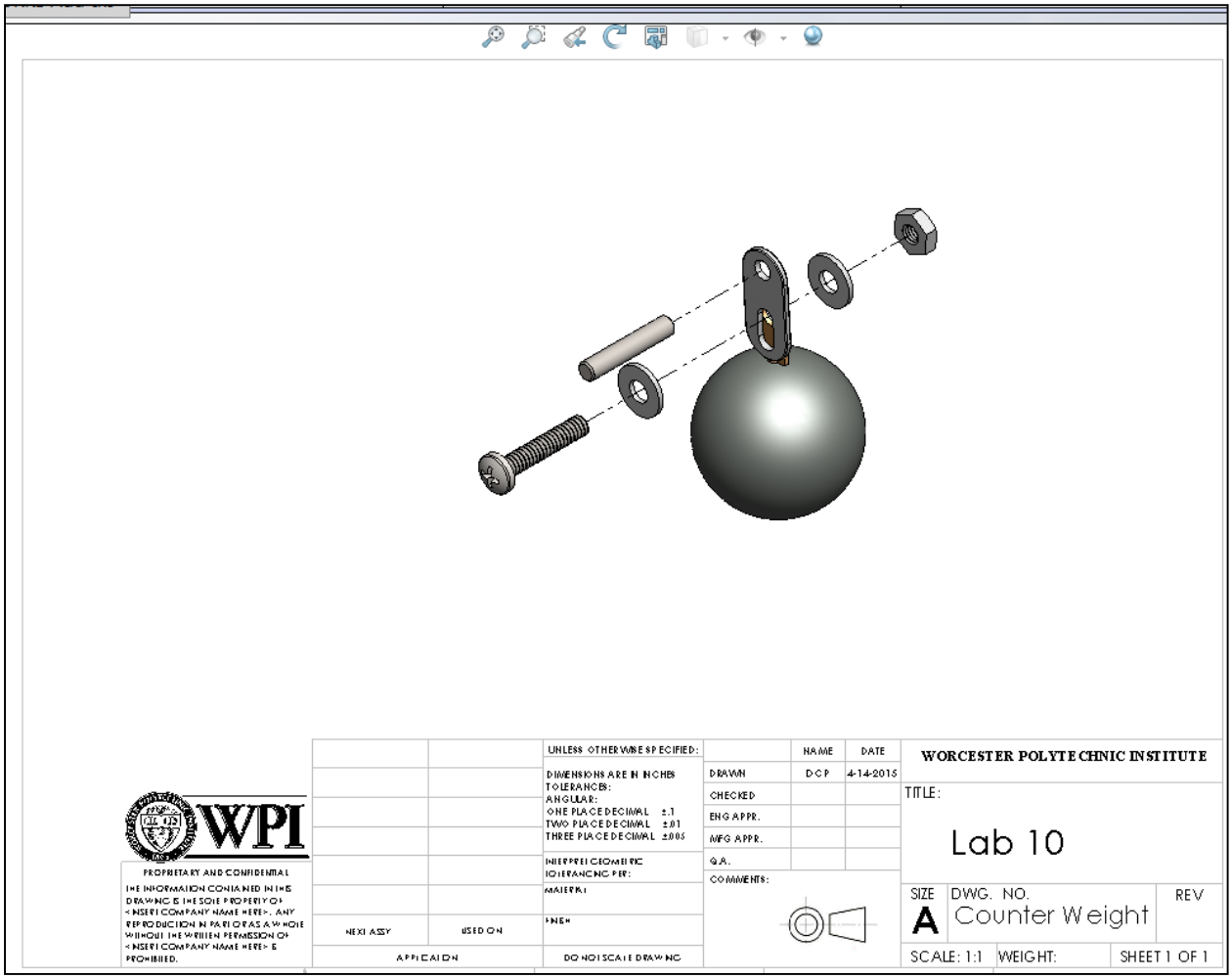
Submit the Exploded Assembly model with explode lines in an Isometric view with Shaded Edges and with an expanded Mates folder.



ME 3408 – Computer Graphics and Design
Prof. Kristin Myers

Item 2D:

Create an Isometric Shaded with Edges exploded A ANSI-Landscape - IPS Assembly drawing with Explode lines as illustrated. Sheet scale 1:1. Address the Title box and Custom Properties. Save the Assembly drawing document and all of the reference components including the SOLIDWORKS Toolbox components. Use the Pack and Go tool to save all Assembly drawings.



Submit the Assembly drawing in an Isometric view with Shaded Edges. Address all Custom Properties with a full Title block.