ME&MTE100

Assignment 9.1
Clevis Mount, "Extended" version with Hints and Clarifications

Note: Check LEARN regularly… if anything is changed we will re-post this write-up.

Version: Nov. 14, 2016

Thanks to former TA's Dilpreet Bath and Andrew Carnovale for their contributions to this assignment.
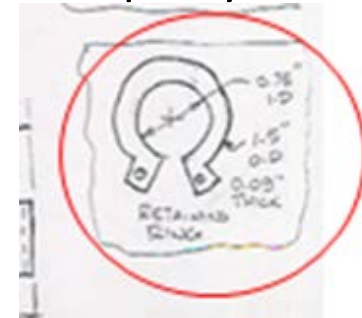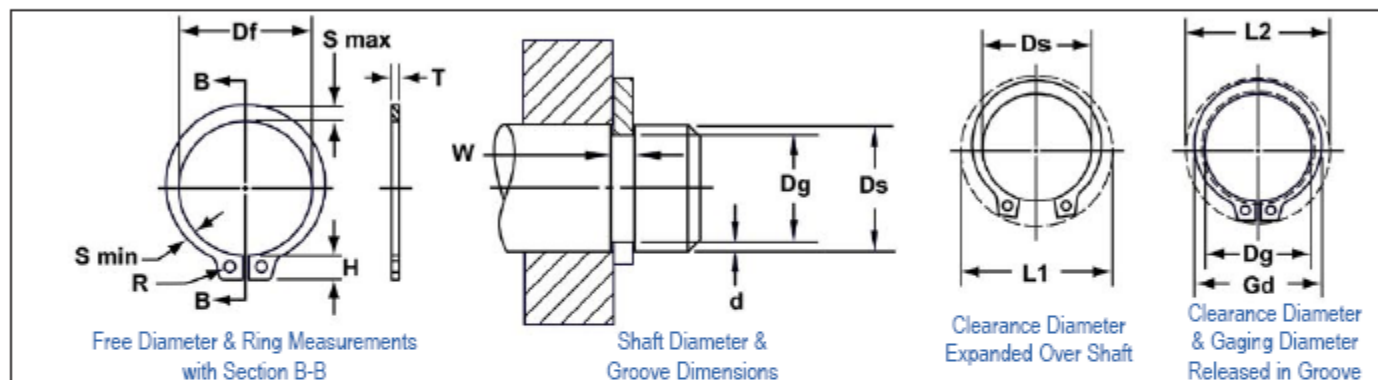
Introduction to Assignment

- You meet with the design team from your company. They are working on an industrial robot. During the meeting a sketch is made (by you, due to your superior freehand skills) of a clevis mount for a hydraulic actuation cylinder on the robot arm.

- After the meeting, you are asked to create a solid model assembly of the mount from the sketch. This can later be inserted as a sub-component into other drawings and put into place in the robot master assembly model.

The Sketch make during the Meeting (same as the redrawn version on pg 9-4 of notes)

CLEVIS MOUNT SKETCH

Required for assignment:

1. Create solid models in SolidWorks of the mount, washer, pin and retaining ring.

2. Position them to form an assembly.

3. Note: because this sketch was generated during a meeting, there will be some dimensions missing. If a dimension does not exist (look carefully) choose an appropriate dimension.

4. There are not many specific details for the retaining ring - complete the design of this retainer yourself. It does not have to be complex…just use something that makes sense.

5. There may be other small modifications that you feel will improve this design. (Remember fillets and chamfers?) This is also up to you to incorporate (or not, this is optional).

Once installed in the groove of a shaft, the shoulder holds an assembly in place.

Free Diameter & Ring Measurements with Section B-B

Shaft Diameter & Groove Dimensions

Clearance Diameter Expanded Over Shaft

Clearance Diameter & Gaging Diameter Released in Groove

RING NO.	SHAFT			GROOVE SIZE						RING SIZE & WEIGHT					CLEARANCE DIA.		î THRUST LD. (lbs.) Sqr. Corner Abutment	
	DIAMETER			DIAMETER		WIDTH		DEPTH	FREE DIAMETER		THICKNESS***		Weight Per 1000 Pcs.	Expand-ed over shaft	Released in groove	Ring Safety factor of 4	Groove Safety factor of 2
	Ds DEC	Ds FRAC	Ds mm	Dg	Tol.	W	Tol.	d	Df	Tol.	T	Tol.	lbs.	L1	L2	Pr	Pg
**SH-0012	0.13	1/8	3.2	.117		.012		.004	.112		.010		.018	.222	.214	112	35
**SH-0015	0.16	5/32	4.0	.146		.012		.005	.142		.010		.037	.27	.260	132	55
**SH-0018	.188	3/16	4.8	.175	±.0015 .0015*	.018	+.002	.006	.168	+.002 -.004	.015		.059	.298	.286	244	80
**SH-0019	.197	-	5.0	.185		.018	-.000	.006	.179		.015		.063	.319	.307	254	85
**SH-0021	.219	7/32	5.6	.205		.018		.007	.196		.015		.074	.338	.324	284	110
**SH-0023	.236	15/64	6.0	.222		.018		.007	.215		.015		.086	.355	.341	315	120
SH-0025	.250	1/4	6.4	.230		.029		.010	.225		.025		.21	.45	.43	599	175
SH-0027	.276	-	7.0	.255		.029		.010	.250		.025		.23	.48	.46	660	195
SH-0028	.281	9/32	7.1	.261		.029		.010	.256		.025	±.001	.24	.49	.47	670	200
SH-0031	.312	5/16	7.9	.290		.029		.011	.281		.025		.27	.54	.52	751	240
SH-0034	.344	11/32	8.7	.321		.029		.011	.309		.025		.31	.57	.55	812	265
SH-0035	.354	-	9.0	.330	±.002 .002*	.029		.012	.320		.025		.35	.59	.57	832	300
SH-0037	.375	3/8	9.5	.352		.029		.012	.338	+.002 -.005	.025		.39	.61	.59	883	320
SH-0039	.394	-	10.0	.369		.029		.012	.354		.025		.42	.62	.60	954	335
SH-0040	.406	13/32	10.3	.382		.029		.012	.366		.025		.43	.63	.61	964	350
SH-0043	.438	7/16	11.1	.412		.029		.013	.395		.025		.50	.66	.64	1035	400

Assignment Submission Requirements:

Note: Requirement 1. (below) can be completed following Lesson 9 but Requirement 2. will require the instructions for creating detailed drawings from Lesson 10. That is one of the reasons that you have two weeks for this assignment.

1. Show on a single sheet… two shaded pictorial views* of your solid model assembly. The views should show each end of the assembly clearly. *Create jpegs of your SolidWorks views. Include a border and informal title block on this sheet. Call this "CLEVIS MOUNT ASSEMBLY".

    **Use this icon to enter different parts on the same drawing sheet.

2. Create two dimensioned drawing sheets on A size paper:

    (You will need to use next weeks instructions for this.)

    1. Sheet One: a completely dimensioned drawing of the base plate.
    2. Sheet Two: dimensions for the pin, washer and ring**.
    3. You can create your own title block etc., or use a standard template (see hints).
       Choose an appropriate number of orthographic/pictorial views for each part (your decision).

3. If you did make modifications to the sketch design… include a sheet <u>briefly</u> describing the changes you made and why you made them.
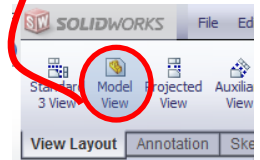
    **Click on browse and select the part you want to enter to the current drawing
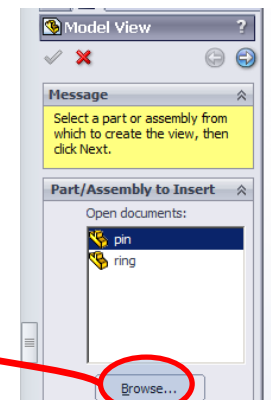
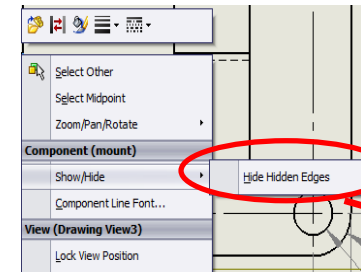# Assignment 9.1 Hints & Clarifications

An alternative way to create a detail drawing:

1. Have a model part open in SolidWorks and then click on File →
    Make Drawing from part. Select drawing under templates on the
    dialog box that appears. On the sheet format dialog box select the
    size of drawing that you would prefer. I would recommend **A (ANSI)
    Landscape** because the default title block doesn't take a lot of space
    and it fits nicely on A size paper.

2. Insert the views using the view palette (on the left). If you want to
    add more views from the current view go to view layout and then
    select projected views.

3. To edit title block, right click on the sheet and select "Edit Sheet
    Format".

4. To change the scale right click on Sheet1 on the bottom left corner
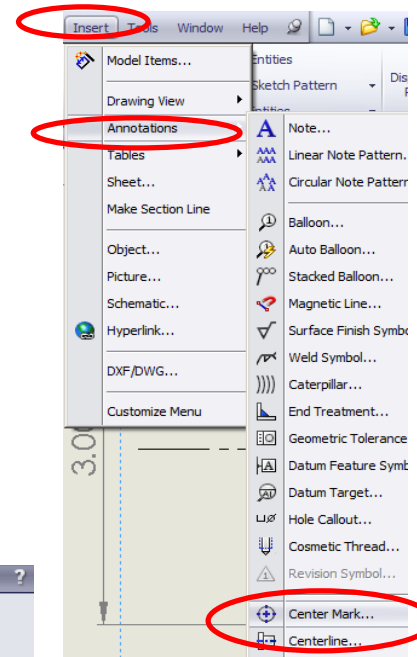   and select properties and then change the scale.
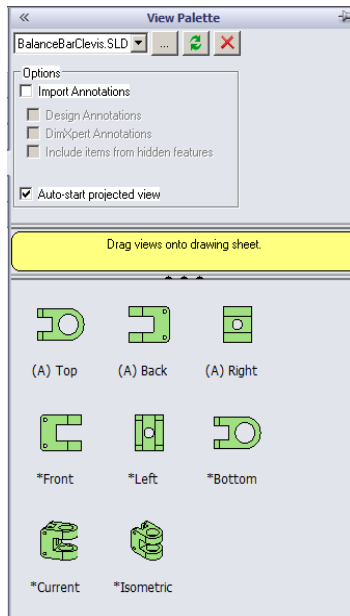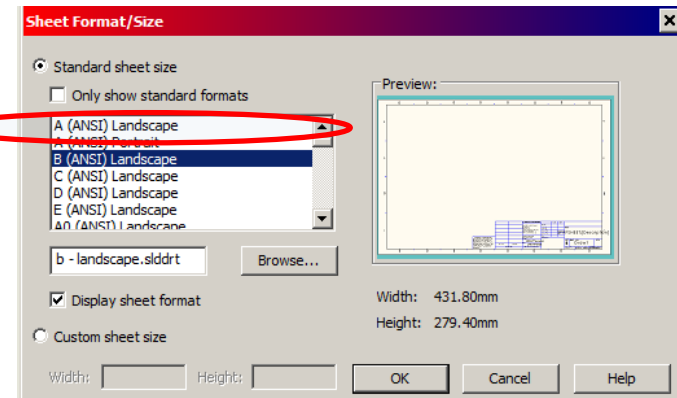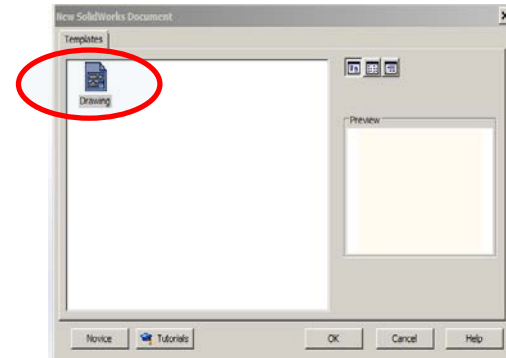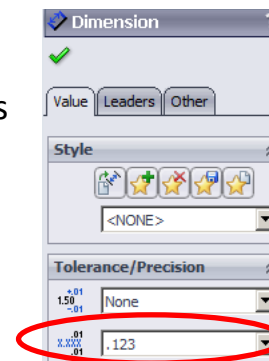
Assignment 9.1 Hints & Clarifications, continued

4. To show hidden lines… Select any of edge of a particular view you want to show hidden lines for and then Right click and in Shoe/Hide select show hidden edges.

Since my hidden edges are already shown that is why it says hide hidden edges.

5. Add centerlines by going to sketch and then clicking the dropdown menu beside the line symbol and selecting centerline. Also, you can add center marks by going to Insert and then annotations and selecting center mark.

6. Don't worry too much about using baseline dimensions on this part… just try and make sure that there is nothing missing if a machinist were to fabricate it.

7. Precision: Holes & hole locations to 3 decimal places, all else to 2 decimal places. You can change the number of decimal places by clicking the dimension and changing the Tolerance/Precision value in the dialog box that appears on the left. If you want to add text beside your dimension go to Annotation and select "NOTE". **STYLE**

8. All this time you should see "Sheet#" on the top of the SolidWorks window to indicate that you are in 2D drafting mode.

Assignment 9.1 Hints & Clarifications, again continued

9. For the Pin, Washer and Retaining Ring drawing, pictorial views are not mandatory for these simple components. Also, it does not take many orthographic views to describe these parts. The view scale can be 1:1 for these components.

10. Retaining Ring: To keep matters simple – just dimension the ID, OD, and thickness. (Don't worry about dimensioning the other small details on this part.)

11. In order to dimension chamfers go to annotation and then click note and then click on the chamfer you want to dimension and enter the dimension. HINT: you can put the degree symbol in Solidworks by holding down Alt and then pressing 0176. (Alt + 0176)

12. In general: Creating the solid models should be fairly straightforward. It may be more challenging to generate the dimensioned drawings. Just make sure you have included all the dimensions required to fabricate each component and don't break any of the more important dimensioning rules that you have been taught. The TA's will be asked to be as reasonable as possible with their marking.
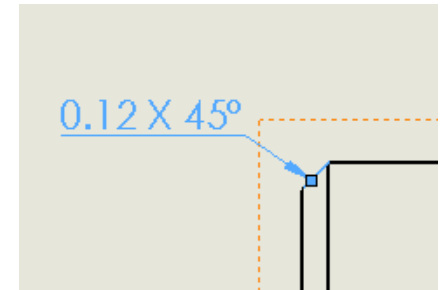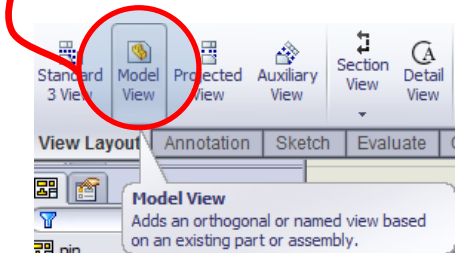
Once you finish this assignment… give yourself a pat on the back… you have come a long way!