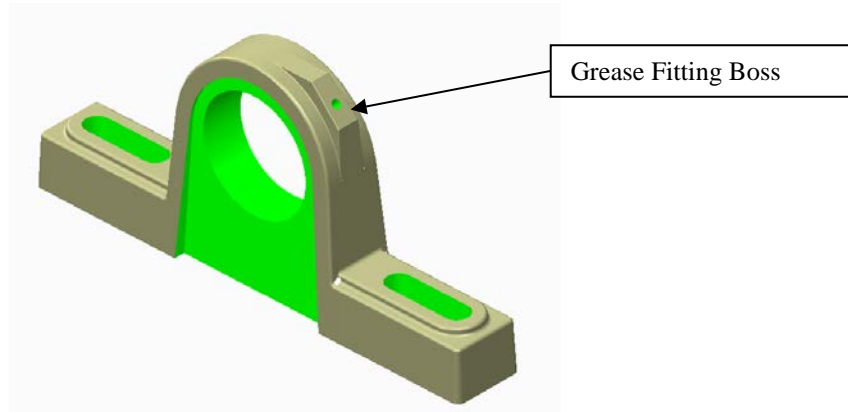


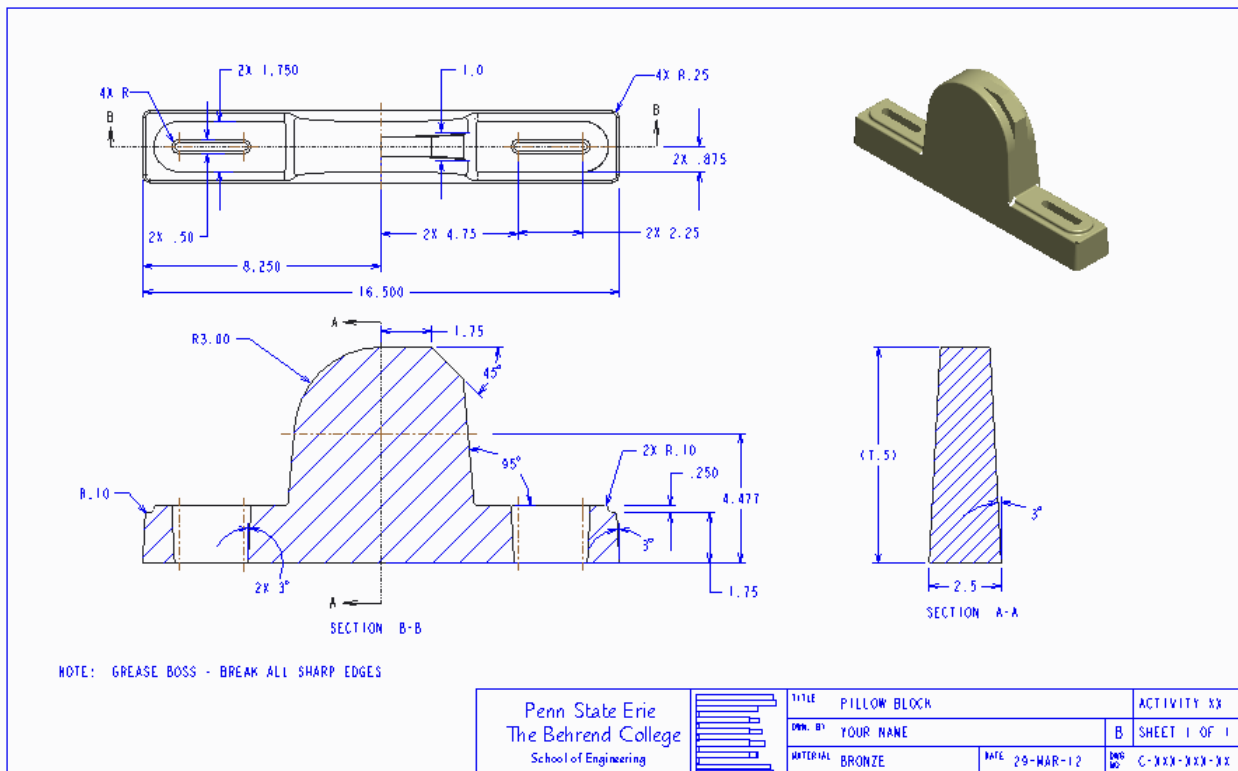
Custom Pillow Block Design  
 Protrusion, Cut, Round, Draft (Review)  
 Drawing (Review)  
 Inheritance Feature (New)  
 Creo 2.0

Rotatable pdf files:

[Casting](#)  
[Machining](#)



The general design of the pillow block is shown in the figure above. Dimensions of features of the casting are shown below. This is part 1 so the part number (drawing number) will be C-306-00X-01 where X is your section number. You are also prompted for activity number. This is activity 1.



Read up on Inheritance Features (Help Center, Search on inheritance feature, select About Inheritance Features). This will give you an overview of the capability and rules that need to be followed.

Using Inheritance Features:

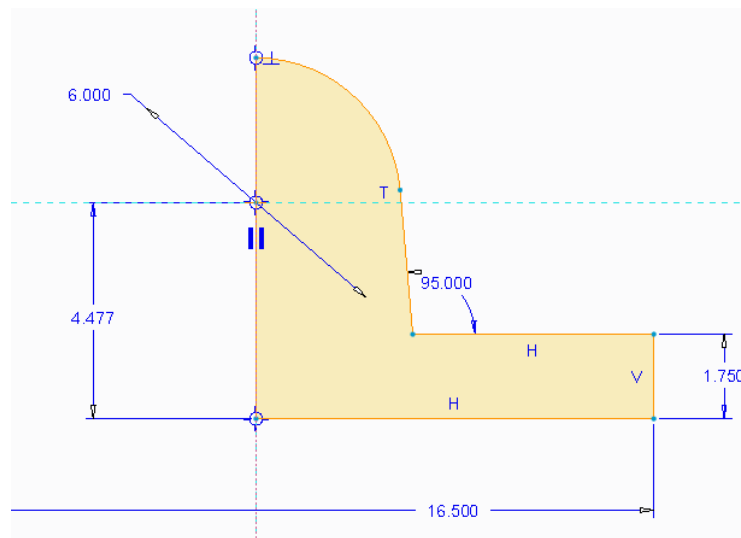
Create a casting model (pillow\_block\_cst.prt). Only features that will be cast are included. Use a width of .5 for the top of the vertical slots. This will allow ample material for machining purposes.

Latest Update -2/4/2014

**Important Note 1:** You must use the Start part when doing this assignment. If you do not, then you will not have any of the predefined Layers set up. Without pre-defined layers, you cannot hide PRT\_ALL\_CURVES.

**Important Note 2:** Changes of the Cast dimensions must be done at the Cast part level, not on the inheritance feature on the Machined part. If you do this, the update of features of the casting will not be reflected at the machined level. This also holds true for changing the number of decimal places in shown dimensions at the drawing level.

Create one half of the part and then *Edit - Mirror* (you need to have the model name highlighted (selected) in the model tree) for the other half. This will also duplicate your base datums. If you do this, make sure that you add centerlines into your basic sketch and dimension using the centerline to give you the overall size. An example is below. Remember, dimensioning in this fashion requires four picks. Pick the end of the entity, the centerline and the same end of the entity again before placing the dimension. The length of the “leg” is a function of the tangency, the 95 degree angle and the 4.477, 1.75, 6 and 16.5 dimensions. Note that the sketch is not mirrored, the part is.

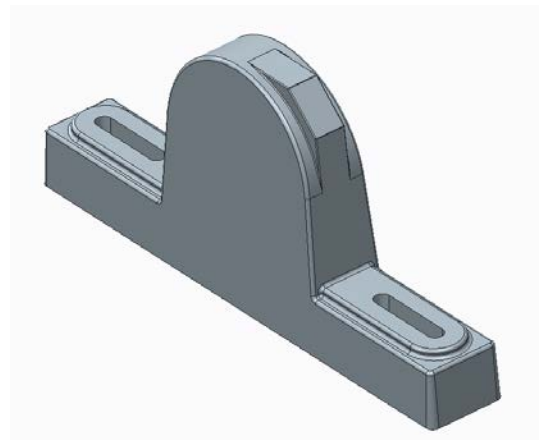


Add the R.25 vertical rounds *before* drafting (*Model Tab, Engineering Group, Draft*) the part. All other rounds will be added **after** the draft has been added. This will be discussed in depth in a future lecture.

All rounds are R.1 except for the rounds on the grease boss, which are .030 (indicated by the NOTE on the drawing). You may need to create the .030 rounds by selecting surfaces rather than edges. The casting drawing **will not** reflect these rounds as this clutters the drawing too much. They can be added and suppressed however.

The boss for the slotted mounting holes is going to require some thought and attention on the designers' part. The 4.75 locating and 2.25 length dimensions **convey design intent**. Notice that the rounded end of the boss has the same center location as the slot. How are you going to model this geometry so that when the slot length changes, that the rounded portion of the boss knows to move with it?

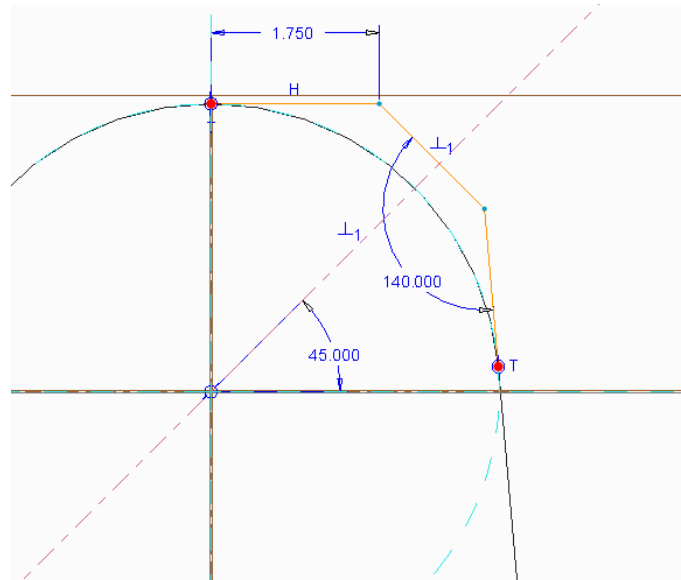
Hint: The Up to Surface option would be a good thing to use when creating the boss. The slot boss will be composed of multiple features.



Draft is added to all “cast” surfaces, **excluding** the small vertical surface on the sides of the boss for the slotted mounting holes. Use the under side surface of the part for the draft hinge.

Create the grease boss last. When you do this use the mid plane datum as the sketching plane, creating three object lines to describe the geometry as shown below. The location of the plane that you will use to define the draft hinge is a plane that is positioned at the center of the 6” diameter.

There is a centerline added at 45 degrees through the center of the hole. Use edge on the outer radius, add in a horizontal line on top, create a line perpendicular to the centerline and add a line tangent to the arc. The length of the line normal to the centerline is a function of the 1.75 dimension and the tangency. Extrude this both sides with a total depth of 1.



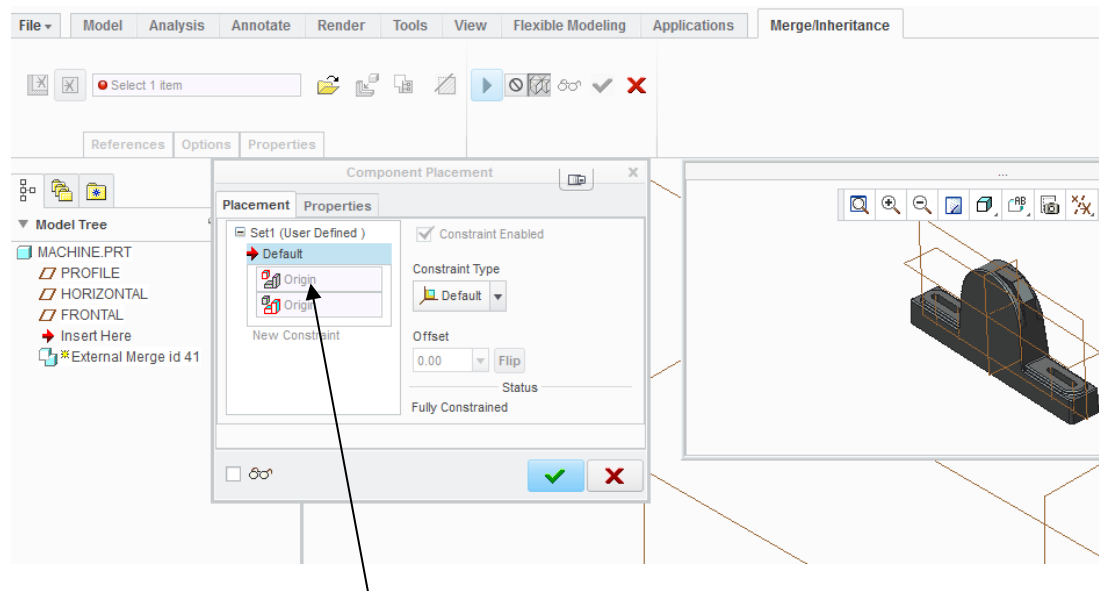
From the PTC Knowledge Base:

“The intent of the inheritance feature is to only bring across colors of part surfaces. **If the color has been applied to the entire part then it will not be seen in the inheritance part.**”

Apply a new color to all surfaces (WF 5 and Creo: Select the Appearance Gallery icon). Create or select a new color (bronze) and apply this to **every** surfaces.

WF 5 and Creo: You will have to select each surface individually.

Create a machining model (pillow\_block\_mch.prt). The first feature that you should produce will be the inheritance feature. In the Get Data region of the Model tab, select Merge/Inheritance.



Select your cast part. Assemble it using the Default Constraint type. This should work as long as you have started with the Start part in both cases. Otherwise you might have to assemble this part the long way using datums.

Create the horizontal hole with a diameter of 4.0 (Extrude).

Create the .375 deep cut (in the front view), and mirror it to the back side.

Change the color of the entire part to green. Go back to the cast part and change the 16.5 dimension to 20 and regenerate. Return to the machined part and regenerate. The inheritance feature should now be bronze and any additional features (the hole and cuts) should be green. In order to force the inheritance feature to update, a change had to have been made to the original geometry. Note that the bosses for the slots did not update at this time. This is intended functionality (design intent) for this model.

Change the 20 in the cast part back to 16.5, regenerate, return to the machined part and regenerate.

Create the .25 inch deep spot face on the back side.

Add features to effectively mill out the slots to the required diameter.

Add in the 3/8 diameter threaded hole for the grease fitting. To create the threaded hole, use the hole wizard.

Other hints:

Protrusions are generally (but not always) closed sections. Making them so will prevent the dreaded "unattached feature" error message.

Dimensioning a radius is done with one pick, a diameter with a double click.

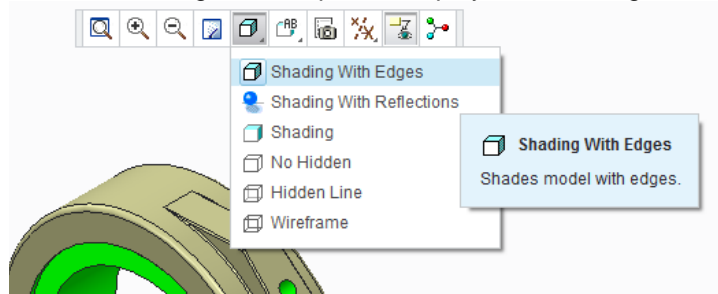
Rename your features to something intelligent.

If you have grouped features, expand the group.

Generate a Word document (Use the Snipping tool) with **both** the cast part and the machined part (on a single sheet) as shown on the following page. It is critical that you move the part model to the upper left corner of the Pro/E window and you crop the window so the model tree and part are clearly shown. Try to keep the parts the same size. Use the default isometric orientation. Enlarge the cropped images to fill the page. Make sure you don't change the aspect ratio when enlarging.

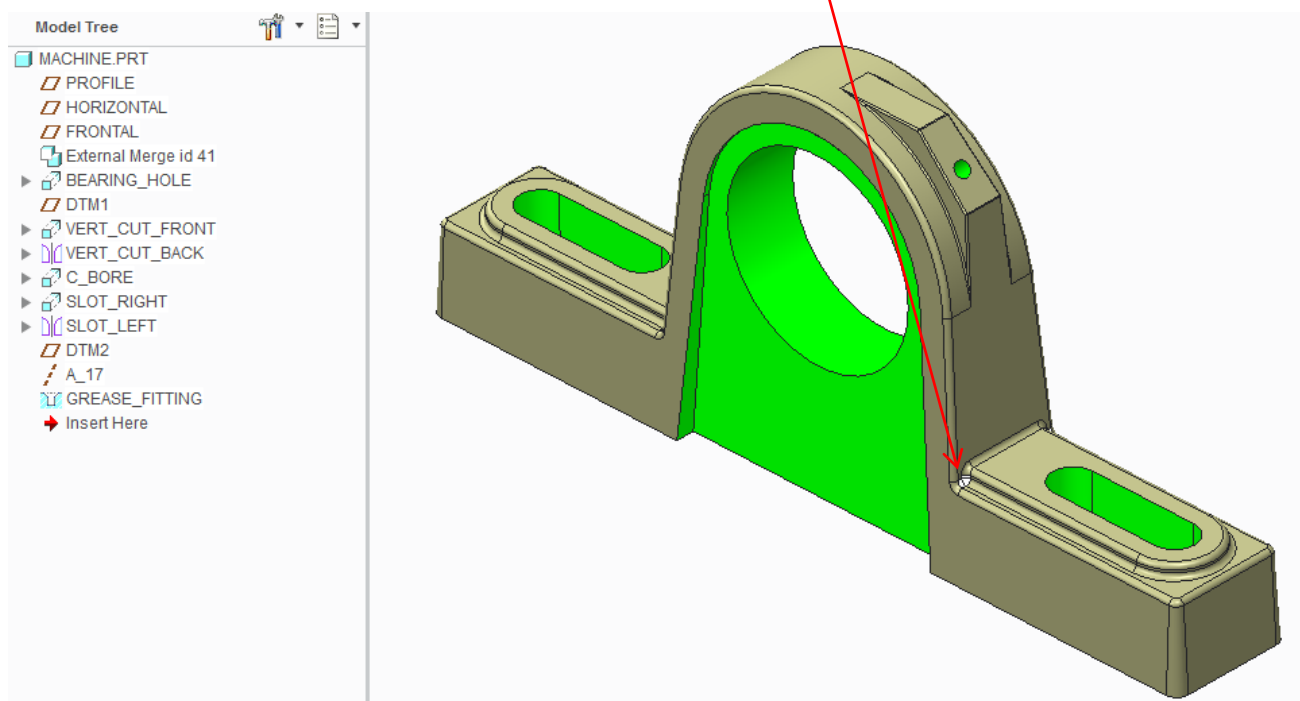
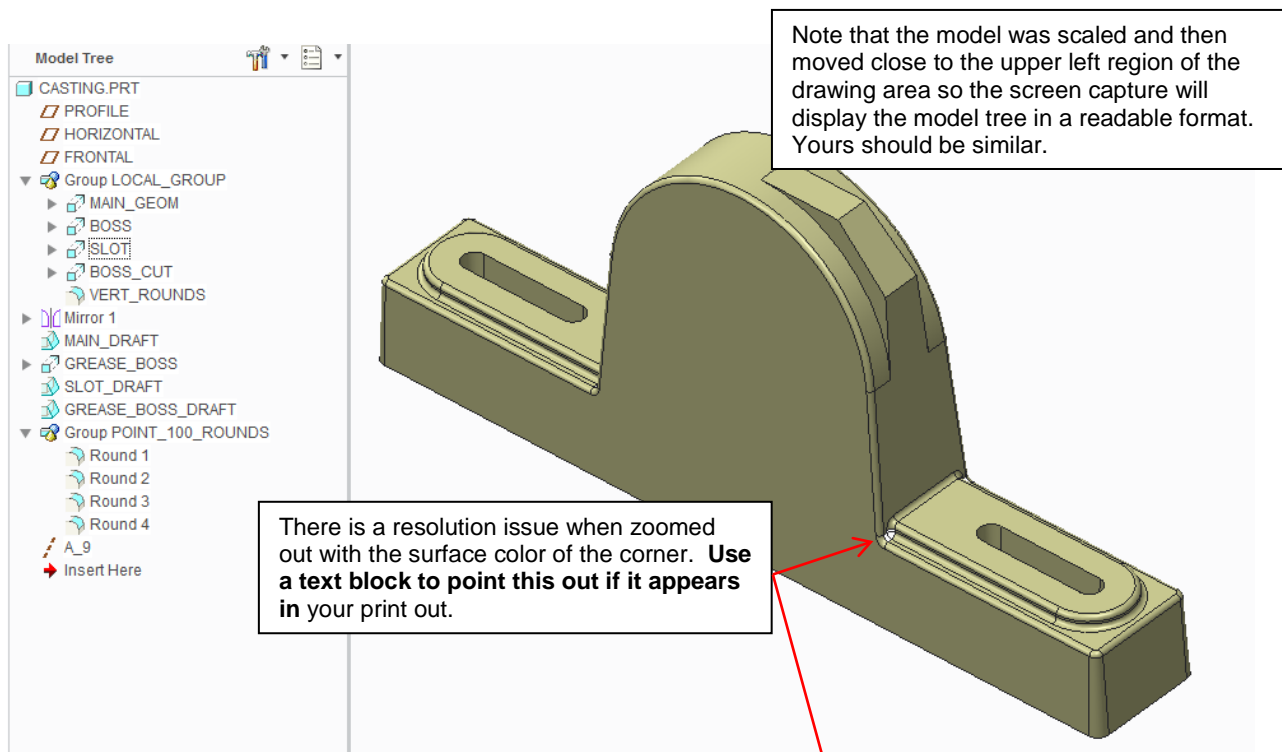
Note that the background color for Creo is white. You no longer need to use the F1 and F2 functions keys to set this.

We want the edges of the part to display as black edges. Select the option below to set this.



**Print** this on a **color** printer so the difference between the cast and machined geometry is obvious.

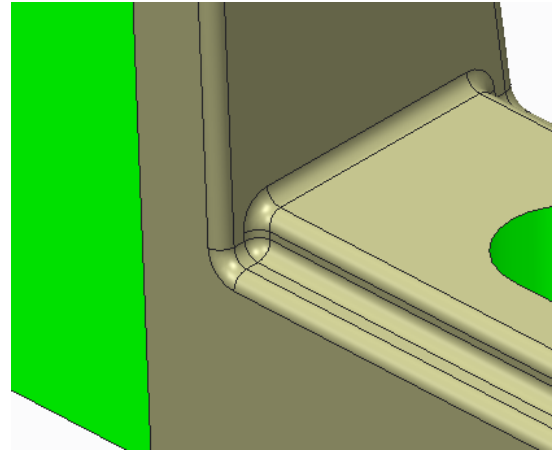
Your Name



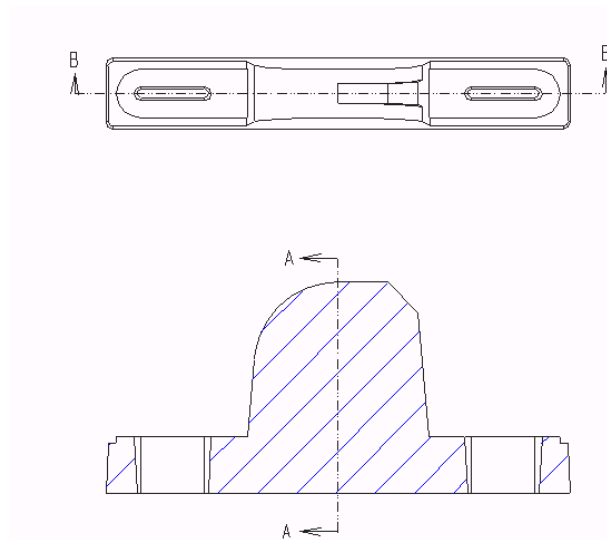
Note that the corner actually has the appropriate color in the corner.

Create a detail casting drawing similar to that shown on page 1.

Note that the **top** view of the fully dimensioned drawing has the most of the tangent lines hidden. This is a manual operation. It used to be done with *View –Drawing Display – Edge Display*. In WF 5 and Creo, on the Layout Tab in the Edit area select *Edge Display* and use the *Erase Line* function in the Menu Manager dialog box.



The cross hatch lines are for Bronze (Copper). Right click on the hatching and set its Properties. In the Mod Xhatch menu select Retrieve and select the appropriate material. The tolerance area in the title block should be filled in. Number of decimal places should be as shown.

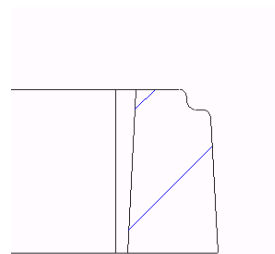


**Print** this drawing of the casting; no label required.

Hints on the Casting Drawing:

The problem with simply suppressing the rounds in the model is that we would be missing rounds that need to be detailed.

We need these rounds:

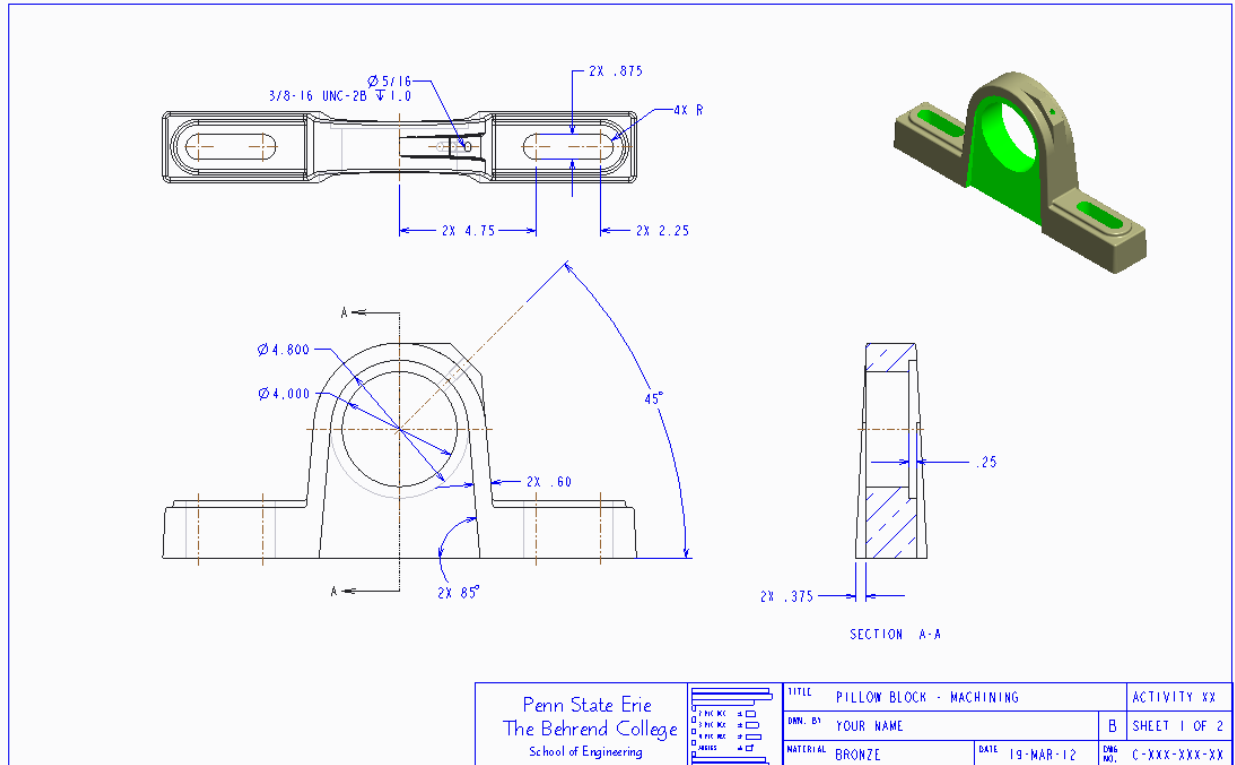


Since we need these rounds to show up in the front view, but not in the top view (too busy) we will have to manually hide edges.

The .030 rounds can be suppressed as we will take care of this detail with a note on the drawing (Grease Boss: Break all sharp edges).

Create a machining drawing as shown below. Pay attention to the number of decimal places shown.

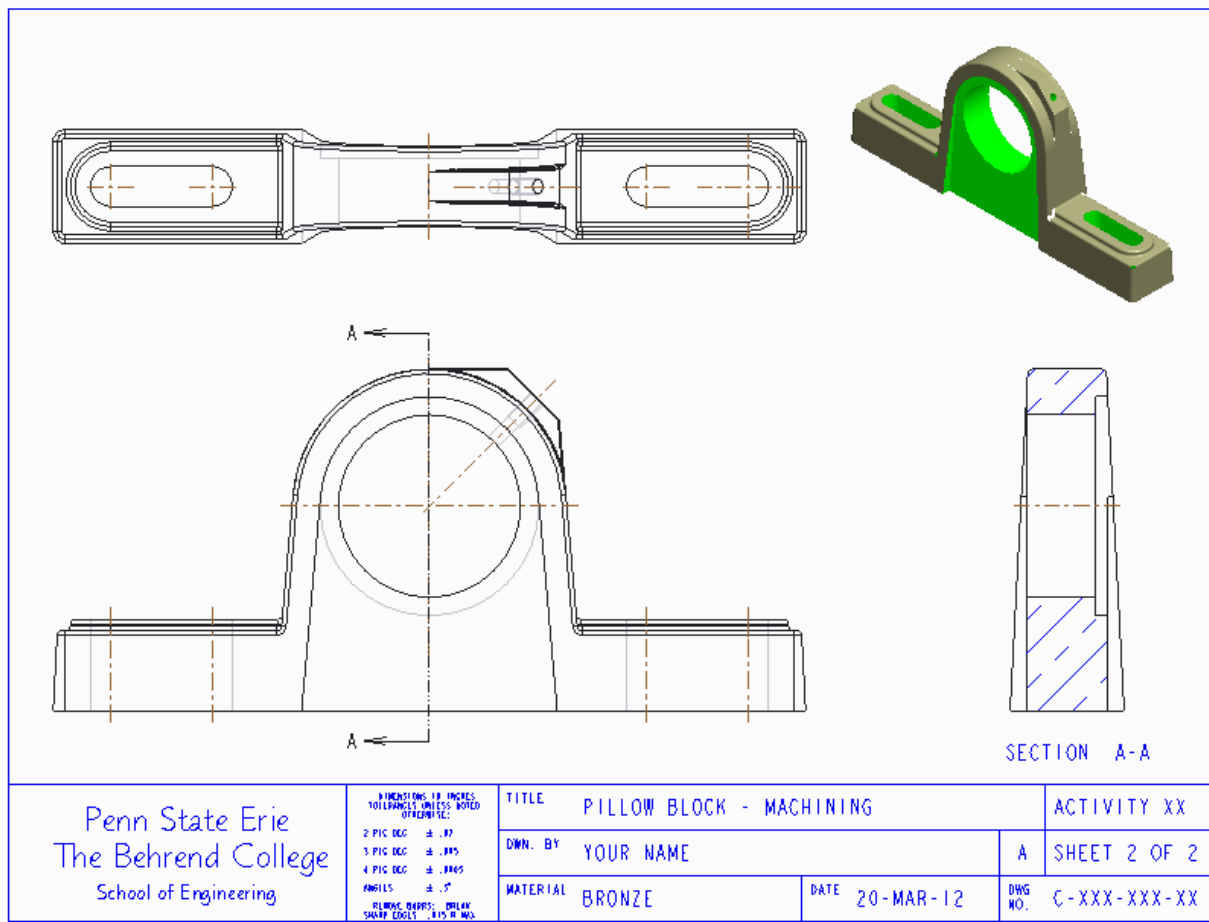
The **front** view has tangent lines hidden through the use of the *Properties – View Display – Tangent edges display style* set to None. Make sure that the orthographic views have hidden lines where required. You should be well aware of the fact that screen captures shown in on-line assignments have hidden lines shown as gray lines which do not translate well to these documents.



These instructions refer to the drawing at the top of the following page:

Create an **A** size second sheet (2 of 2) that shows the part with no dimensions as shown on the following page.

The orthographic views **MUST** have a .375 scale factor, and the Isometric view is at .2. Move the Scale for the isometric off the drawing so it will not print. Note that all tangent lines are shown. The purpose of this sheet is so that the instructor can check to make sure all the appropriate rounds have been created, and to check feature sizes by overlaying the drawing with the solution of a known part size.



In all cases, the title block will come from the pro\_e subdirectory on the V: drive (courseware data disk). Use the a\_horiz\_English and b\_horiz\_English title blocks.

The front and isometric views are general views. The front and top view shows hidden lines, as well as all the tangent lines.

The right view is a sectional view with no hidden lines.

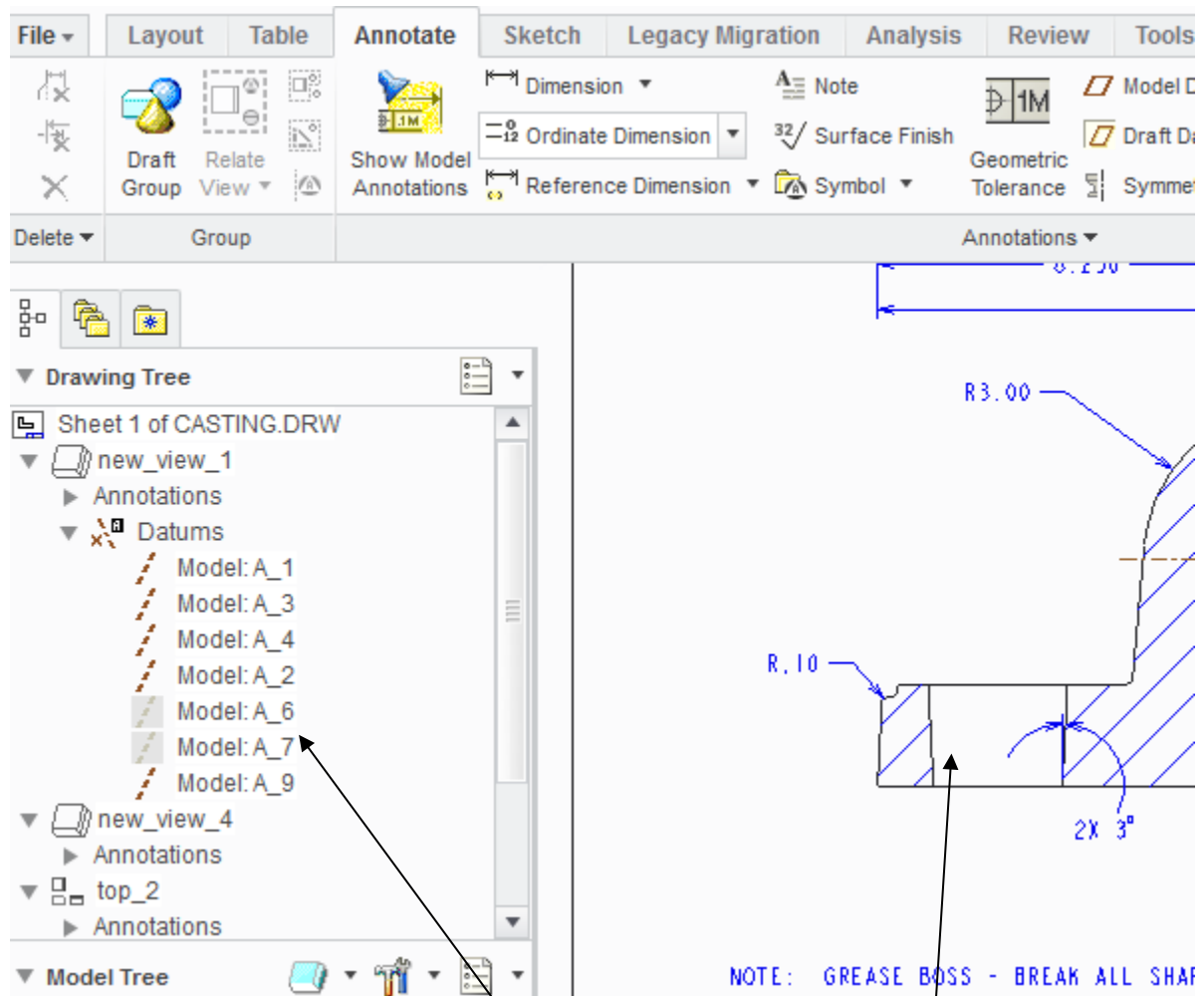
The isometric view is shaded. Change the view display using the View Display option in the Drawing View dialog box (see next page).

Show centerlines in the orthographic views. Hide the center-line for the grease fitting and the slots in the right side view as it is a section view without those features.

**Print** these two detailed drawings of the machined part (B size title block on A size paper); no label required. (This is the Level 7 documentation)



Additional Creo hint (similar to functionality in WF5):



Note the missing center line (axis) in the front view.

This was "erased" using the RMB

Trying to re-Show Model Annotation Axis has no effect. Expand the Drawing Tree to show the datums of the new\_view\_1 and note that Datum Model A\_6 and A\_7 is hidden (grayed out). RMB to unerase.

End Level 7

## Level 8.5

Make the following modifications to the model and **re-print** the casting and machining drawings. Note that it is **not** necessary, nor should you create a new drawing, as this is a fully parametric system.

The overall depth from 2.5 to 3.5.

The width of the boss with the slot from 1.75 to 2.25.

The length of the slot for the fasteners from 2.25 to 1.75.

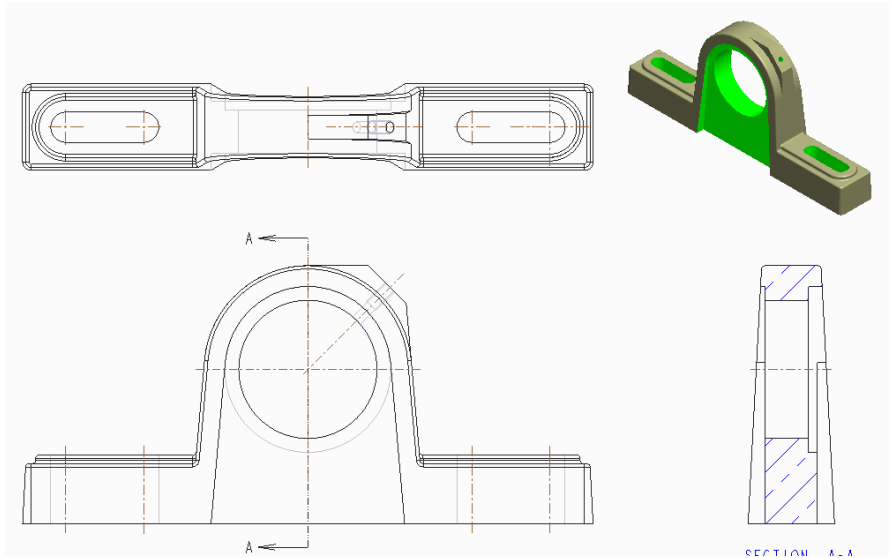
The diameter of the inner hole for the bearing from 4. to 3.

The depth of the facing on the front and back of the part from .375 to .5 (as shown in the right side view).

The geometry should appear as shown:

Make sure you have the hidden lines shown in the front and top views. They don't display too well in the screen captures shown.

Erase tangent lines that appear in front of the center lines!



End Level 8.5

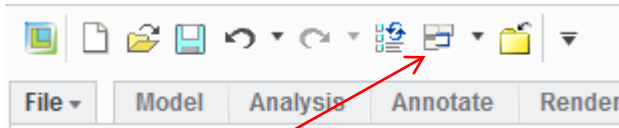
## Level 10

Answer the following questions (use a word processor) and attach to your documentation:

- 1) What are the differences between a Extruded (remove material) and Hole feature in Pro/E?
- 2) Explain the difference between weak and strong dimensions in sketcher (be specific). Hint: the difference isn't that they are different colors; it's how Pro/E reacts with the different dimensions as modifications are made to the sketch.
- 3) Is it better to use Edit Definition functionality or Edit when making dimensional changes for the level 8.5 portion of this assignment? Why?
- 4) What are the differences between Mirror by selecting the part name from the top of the model tree and then picking on Mirror from the Editing group or first selecting features then picking on Mirror from the Editing group?
- 5) In Drawing mode, how do you move a dimension from one view to another?

End Level 10

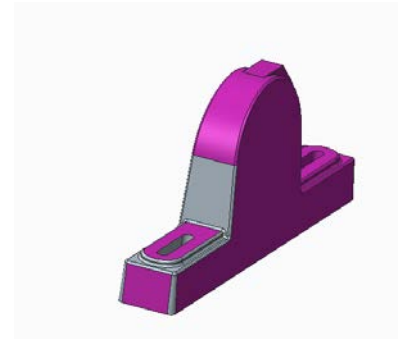
Hints:



To Switch and activate windows, select here.

Colors are set in the Appearance Gallery on the Render tab.

When setting the color of the inheritance part, select a “weird” color at first so it is obvious if you did not select all of the surfaces. You can then edit the “weird” color back to something a little more normal.



Required Documentation (with sheet order)

Level 7:

Standard Grade Cover Sheet

Color Print

Detail Casting Drawing (original size, fully dimensioned)

Detail Manufacturing Drawing (original size, fully dimensioned)

Detail Manufacturing Drawing (original size, to scale, no dimensions)

Level 8.5

All the level 7 documentation plus:

Detail Casting Drawing (modified size, fully dimensioned)

Detail Drawing (modified size, fully dimensioned)

Detail Drawing (modified size, to scale, no dimensions)

Level 10

All the level 7 & 8.5 documentation plus:

Answers to the eight questions.

Deduction (-1) for no color print (Level 7)

Deductions (.5 each) for minor errors (Level 7 and 8.5) such as:

Axis labels not turned off

Hidden line problems

Missing centerlines

Missing/Incorrect information in title block

Missing geometry (rounds, draft etc.)

Missing dimensions

Incorrect scale

Remember, we did an in-class activity regarding errors on drawings. You should recognize these errors by now.

Deduction (-1.5) if you create new detail drawings, rather than just clean up and re-print original drawings for Level 8.5.