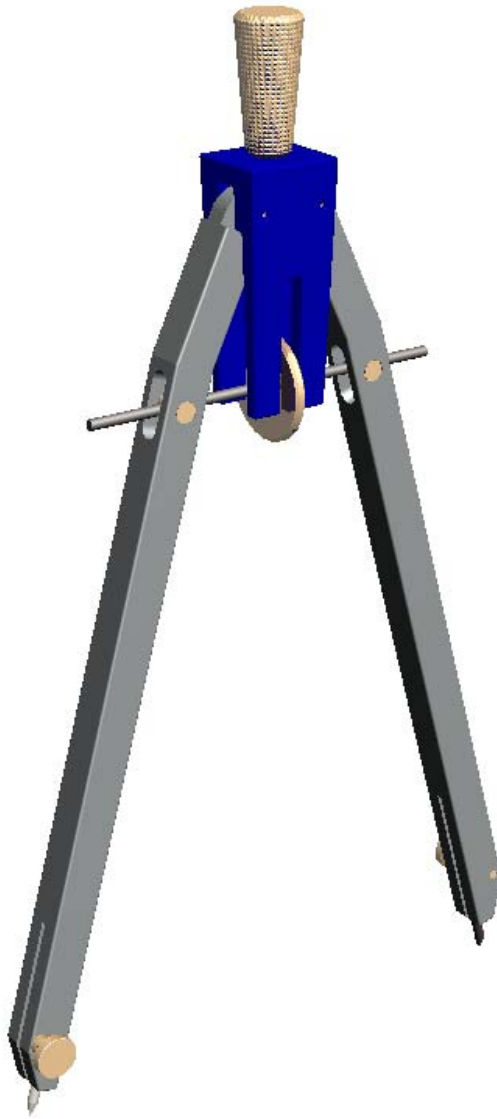


Pro/DESKTOP Tutorial

Drafting Bow Compass



Michael Flowers 2005

Objectives:

- To develop confidence with the Pro/DESKTOP software.
- To learn to utilize extrude, project, revolve, round, and chamfer features.
- To learn when to apply dimensions and constraints.

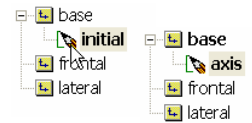
Getting Started:

1. Run the Pro/DESKTOP program by clicking the icon on your desktop.
2. Click on the **New Design** icon located in the upper left corner of the screen.
3. **Maximize** the design window so it fills the screen.
4. **Save** the file as *handle.des*.
5. Click on the **Tools** pull down menu and click on **Options...** at the bottom of the list. Click on the **Units** tab and make sure **Model & Paper** distances are set to inches and click **OK**.



Creating the Handle: (Revolve & Chamfer)

You will be creating an axis and a profile to use the Revolve feature.



1. On the base workplane rename the **initial** sketch to **axis**.
2. Draw a vertical line 2" up from the origin to serve as your axis. (fig 1)
3. Create a **New Sketch** on the base workplane named **profile**.
4. Draw the handle profile starting at the origin. (fig 2)
5. Add the dimension constraints to the handle profile. (fig 3)

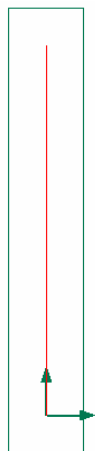


fig 1

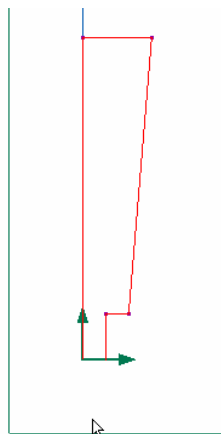


fig 2

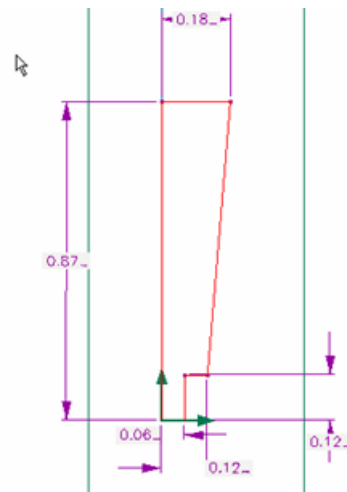
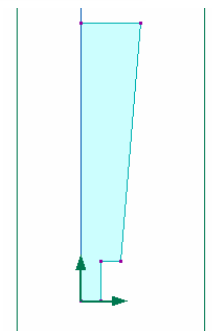
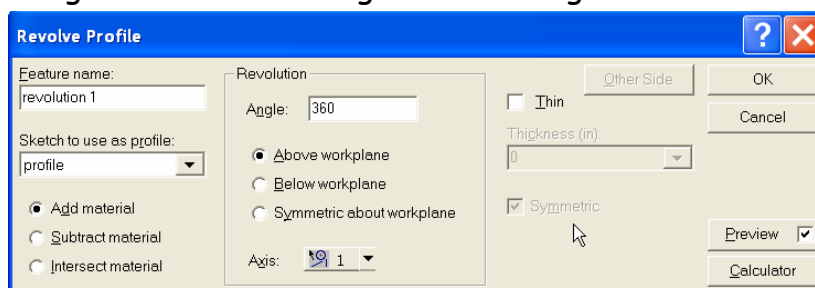
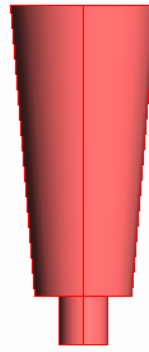


fig 3

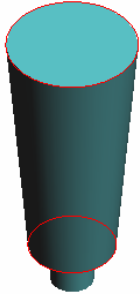
6. Choose the **Revolve Profile** icon  on the Features toolbar.
7. Set the following values in the dialogue box making sure the axis is selected.



8. Your handle should appear like this:



9. Click on the **Select Edges** icon on the Design toolbar.

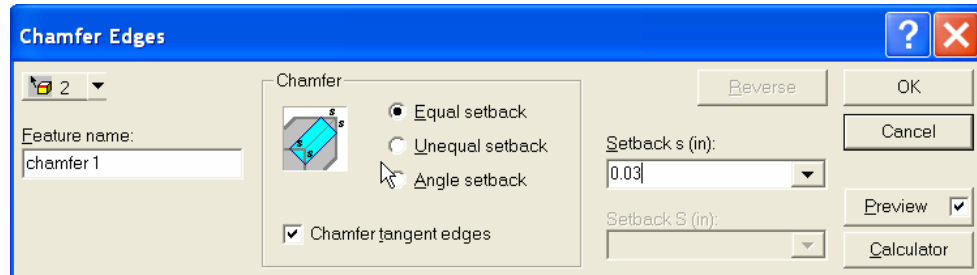


10. Click on the top edge of the tapered part of the handle and holding down the shift key click on the lower edge.

11. Click on the **Chamfer Edges** icon on the Features toolbar.



12. Set the following values in the dialogue box making sure the Setback value is correct.



13. Your handle is complete. Don't forget to save your work.



Finished Handle



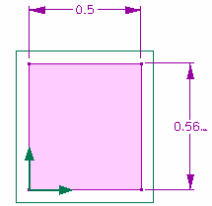
Brass, knurled material applied



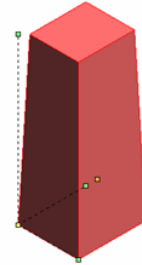
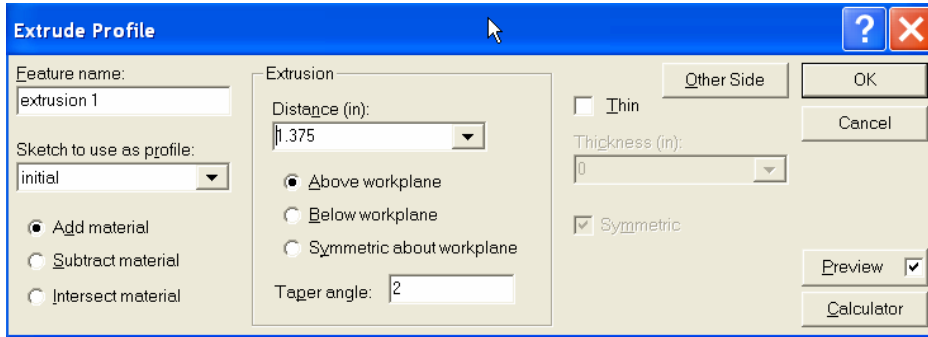
Creating the Leg connector: (*Extrude, Project, & Chamfer*)

1. Click on the **New Design** icon located in the upper left corner of the screen.
2. **Save** the file as *leg connector.des*.

3. Draw a .5 x .5625 rectangle at the origin on the initial sketch of the base workplane.



4. Choose the **Extrude Profile** icon  on the Features toolbar.
 5. Set the following values in the dialogue box making sure the distance and taper angle values are correct.



6. Create a **New Sketch** on the frontal workplane named **slot**.

7. Draw the slot profile as shown. (fig 4) Add dimensions. (fig 5)
 Using a fixed Vertical Construction line drawn up from the middle will help.

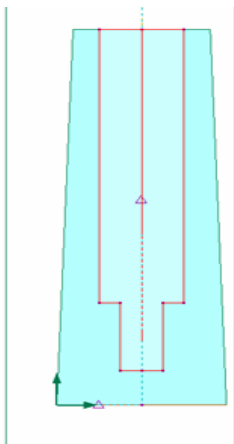


fig 4

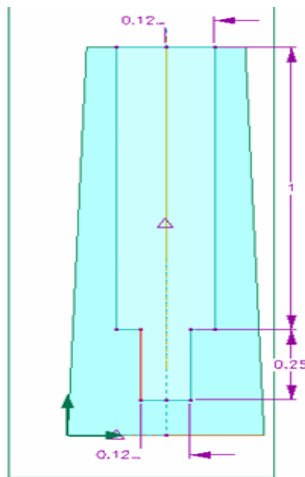
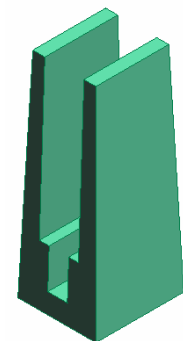
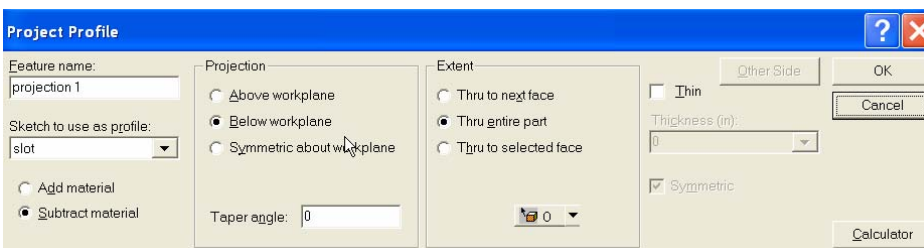
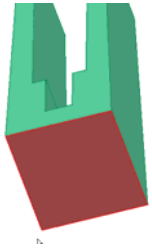


fig 5

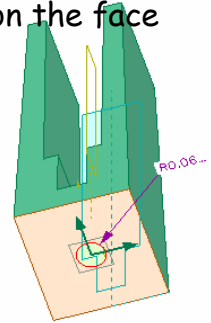
8. Choose the **Project Profile** icon  on the Features toolbar.

9. Set the following values in the dialogue box making sure you subtract material and below workplane





10. Select the bottom face of the leg connector. Right click on the face and pick **New Sketch**. Name the sketch **hole**. Workplane1 will also be created when you do this.



11. **Draw** a R.06 circle at the origin.

12. Choose the **Project Profile** icon  on the Features toolbar.

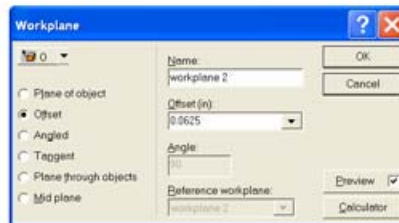
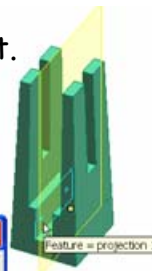
13. Subtract material, below the work plane, thru to next face.

14. Select the inside vertical face of the bottom slot.

15. Right click and pick **New workplane**.

Pick **Offset** and set the value at .0625"

This will create workplane 2



16. Create a **new sketch** on workplane2 called **slot1**.

Shift+W and draw the rectangle and two circles as shown in (fig 6). Using a fixed Vertical Construction line drawn up from the origin will help.

17. Choose the **Project Profile** icon  on the Features toolbar.

Set the following values in the dialogue box (fig 7) making sure you subtract material, symmetric about workplane, and thru entire part.

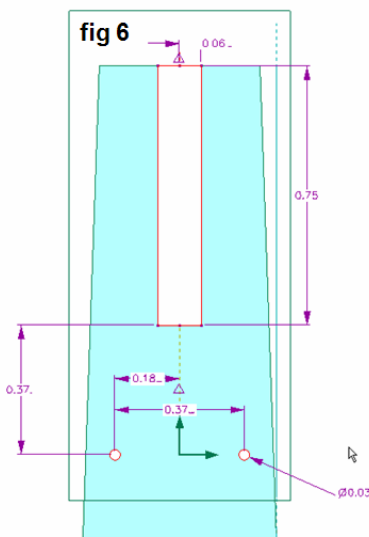
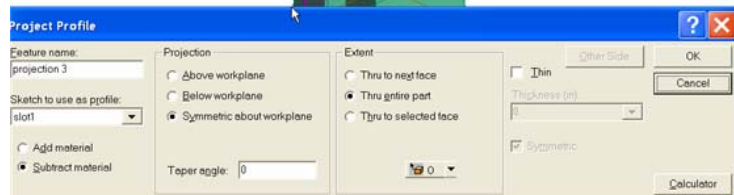


fig 7




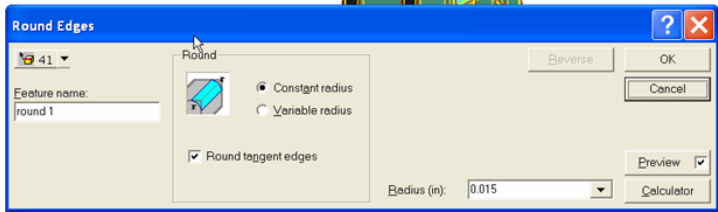
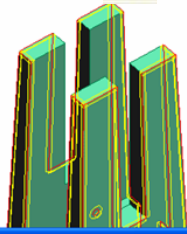
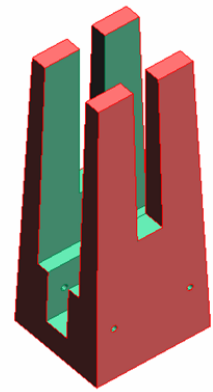
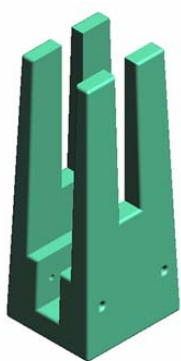
18. Select all the exterior faces of the leg connector (fig 8).
19. Click on the **Round Edges** icon  on the Features toolbar. Set the radius to .015"

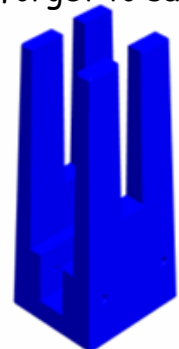
fig 8



20. Your leg connector is complete. Don't forget to save you work.



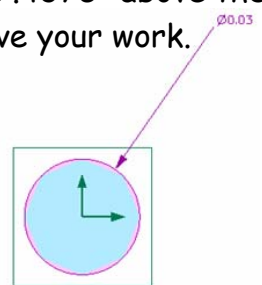
Finished Leg connector



Plastic, injected material applied
Blue color

Creating the Connector pin: (*Extrude*)

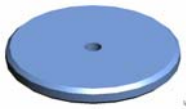
1. Click on the **New Design** icon located in the upper left corner of the screen.
2. **Save** the file as *connector pin.des*.
3. Draw a .03 diameter circle at the origin on the initial sketch of the base workplane.
4. **Extrude** the circle .4375" above the workplane. Your connector pin is complete. Don't forget to save your work.



Finished Connector Pin

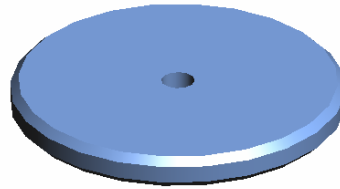
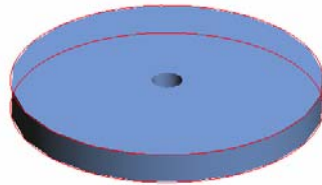
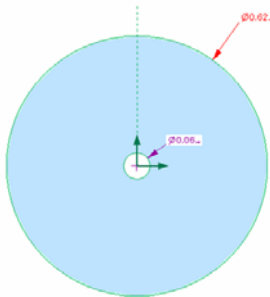


Brass, antique material applied

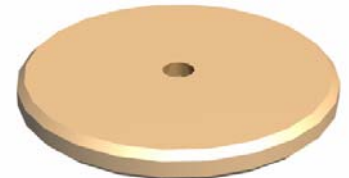


Creating the Center Wheel: (*Extrude & Chamfer*)

1. Click on the **New Design** icon located in the upper left corner of the screen.
2. **Save** the file as *center wheel.des*.
3. Draw a .0625 diameter circle and a .625 diameter circle at the origin on the initial sketch of the base workplane.
4. **Extrude** the circle .0625" above the workplane.
5. Click on the top and bottom edges of the center wheel.
6. Click on the **Chamfer Edges** on the Features toolbar. Set the value of the Setback *s* to .015" Your center wheel is complete. Don't forget to save your work.



Finished Center Wheel



Brass, polished material applied

Creating the Center Screw: (*Extrude*)

1. Click on the **New Design** icon located in the upper left corner of the screen.
2. **Save** the file as *center screw.des*.
3. Draw a .0625 diameter circle at the origin on the initial sketch of the base workplane.
4. **Extrude** the circle 2.25" above the workplane. Your center screw is complete. Don't forget to save your work.
5. **Bonus:** Try to add threads to your center screw by using the **Sweep Profile along Helix** feature.

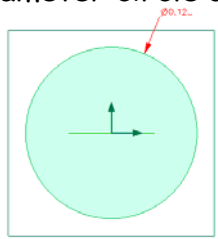
Finished Center Screw

Steel, machined material applied

Creating the Center Pin: (*Extrude & Project*)



1. Click on the **New Design** icon located in the upper left corner of the screen.
2. **Save** the file as *center pin.des*.
3. Draw a .125 diameter circle at the origin on the initial sketch of the base workplane.

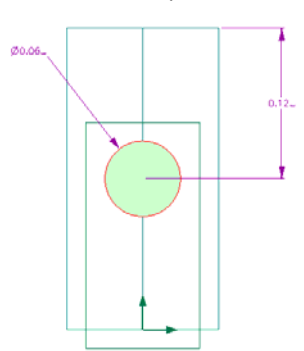


4. **Extrude** the circle .25" above the workplane.

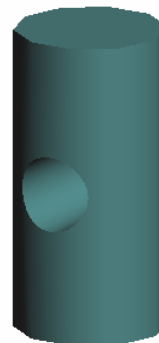
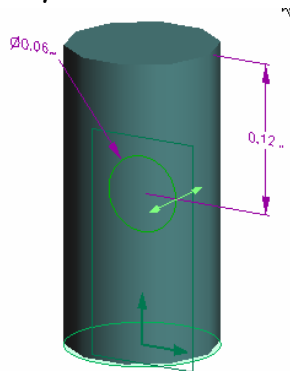


5. Create a **new sketch** on the frontal workplane called **hole**.

6. Draw a .0625 diameter circle .125" down from the top of the cylinder.



7. **Project** the circle. Make sure you; subtract material, symmetric about workplane, and thru entire part. Your center pin is complete. Don't forget to save your work.



Finished Center Pin

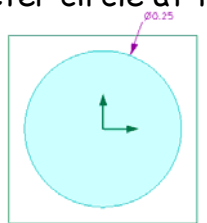


Brass, polished material applied

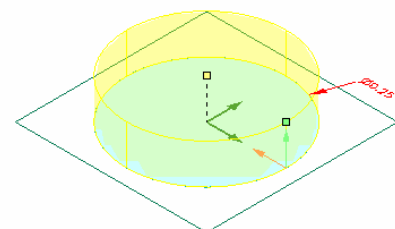


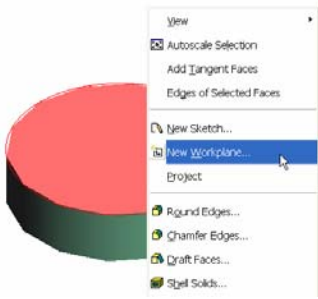
Creating the Thumb screw: (*Extrude & Round*)

1. Click on the **New Design** icon located in the upper left corner of the screen.
2. **Save** the file as *thumb screw.des*.
3. Draw a .25 diameter circle at the origin on the initial sketch of the base workplane.



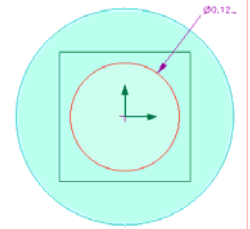
4. **Extrude** the circle .0625" above the workplane.



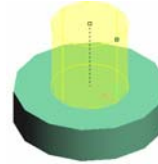


5. Select the top face of the extruded circle. Right click and pick **New Sketch**. Name the sketch **cylinder**. Workplane1 will also be created when you due this.

6. Draw a .125 diameter circle at the origin on workplane1.

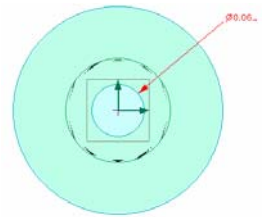


7. **Extrude** the circle .125" above the workplane.

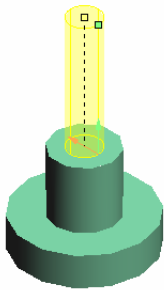


8. Select the top face of this extruded circle. Right click and pick **New Sketch**. Name the sketch **threads**. Workplane2 will also be created when you due this.

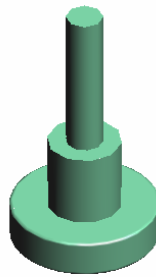
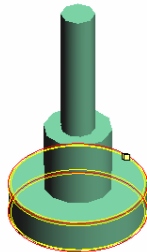
9. Draw a .0625 diameter circle at the origin on workplane2.



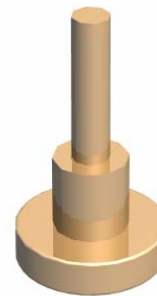
10. **Extrude** the circle .25" above the workplane.



11. **Round** the two edges of the bottom cylinder with an R .005" Your thumb screw is complete. Don't forget to save your work.



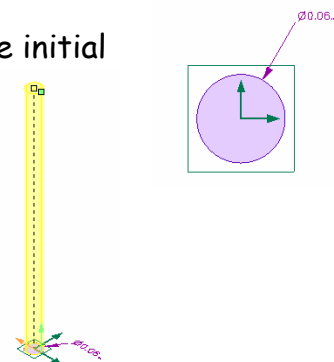
Finished
Thumb Screw



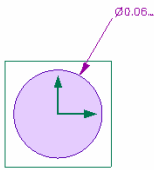
Brass, polished
material applied

Creating the Metal point: (*Extrude*)

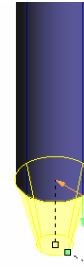
1. Click on the **New Design** icon located in the upper left corner of the screen.
2. **Save** the file as *metal point.des*.
3. Draw a .0625 diameter circle at the origin on the initial sketch of the base workplane.
4. **Extrude** the circle 1.00 " above the workplane.



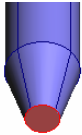
5. Create a **new sketch** on the base workplane called **taper**.



6. Draw another .0625 diameter circle at the origin on the taper sketch of the base workplane.

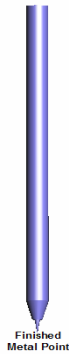
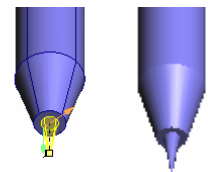


7. **Extrude** the circle .0625" below the workplane and at 15-degree taper angle.



8. Select the bottom face of the taper. Right click on the face and pick **New Sketch**. Name the sketch **needle**.

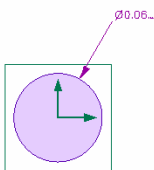
9. **Extrude** the circle .042" above the workplane and at 9-degree taper angle. Your metal point is complete. Don't forget to save your work.



Creating the Lead point: (*Extrude*)

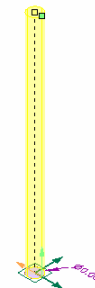
1. Click on the **New Design** icon located in the upper left corner of the screen.

2. **Save** the file as *lead point.des*.

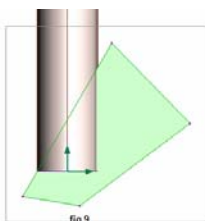


3. Draw a .0625 diameter circle at the origin on the initial sketch of the base workplane.

4. **Extrude** the circle 1.00 " above the workplane.

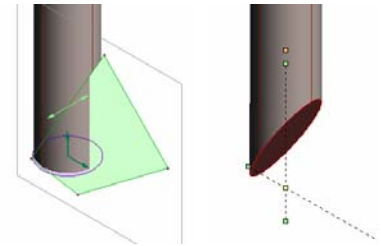


5. Create a **new sketch** on the frontal workplane called **slant**.



6. Draw any polygon similar to the one in (fig 9). Make sure to have it slant across the cylinder and pass through the lower left corner.

7. **Project** the polygon. Make sure you; subtract material, symmetric about workplane, and thru entire part. Your lead point is complete. Don't forget to save your work.



Finished Lead Point



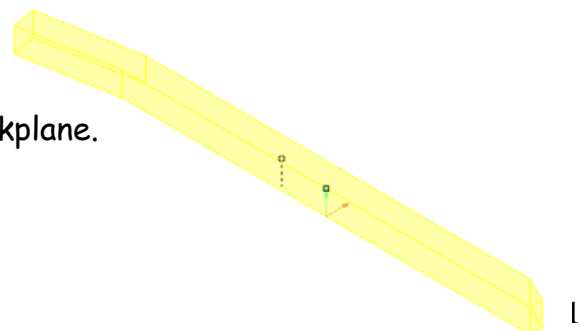
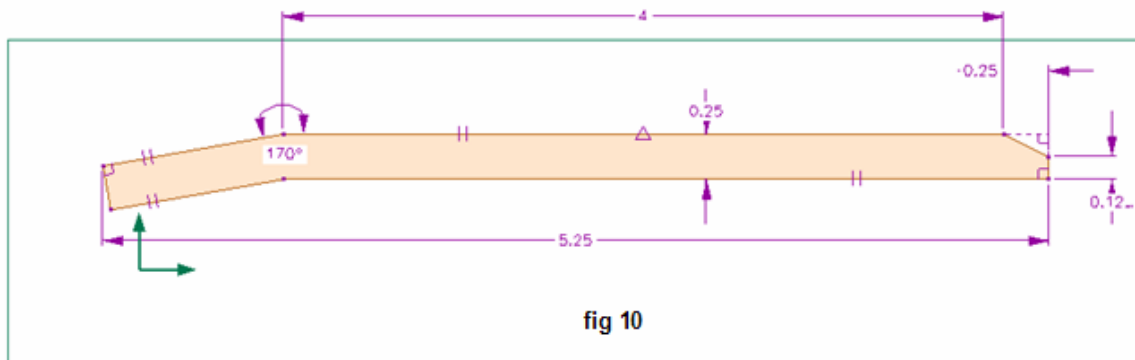
Graphite material applied

Creating the Compass leg: (*Extrude, Project, & Round*)



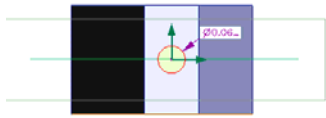
1. Click on the **New Design** icon located in the upper left corner of the screen.
2. **Save** the file as *compass leg.des*.

3. Draw, constrain, and dimension a polygon by using (fig 10). On the initial sketch of the base workplane.



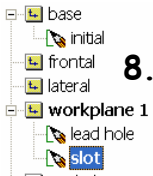
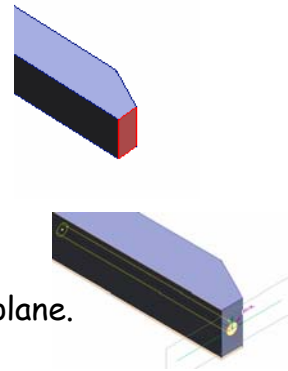
4. **Extrude** the polygon .25 " above the workplane.

5. Select the bottom face of the compass leg. Right click on the face and pick **New Sketch**. Name the sketch **lead hole**.



6. Draw a .0625 diameter circle at the origin.

7. **Extrude** the circle 1.00 " Subtract material, below the workplane.

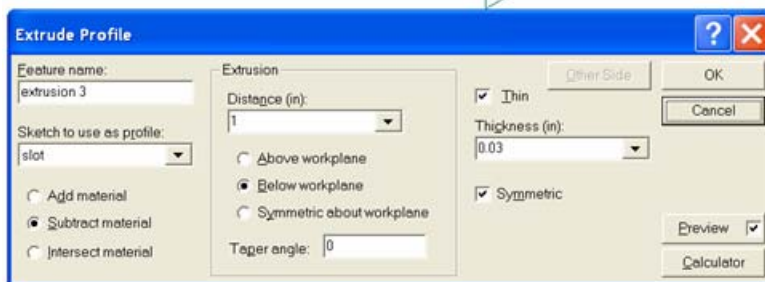
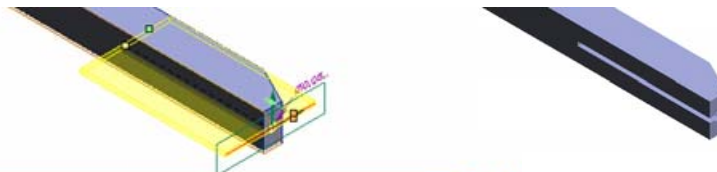


8. Create a **new sketch** on the workplane1 called **slot**.

9. Draw a horizontal line any length through the origin. Make sure it extends past the compass leg on both sides.



10. **Extrude** the line 1.00 " Subtract material, below the workplane, check Thin & Symmetric, and use .03" Thickness.



11. Select the side face of the compass leg. Right click on the face and pick **New Sketch**. Name the sketch **adjust hole**. Workplane2 will also be created when you due this.

12. Draw and locate the circle as shown in (fig 11).

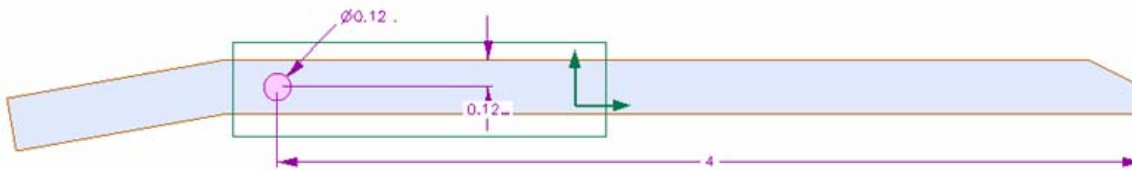
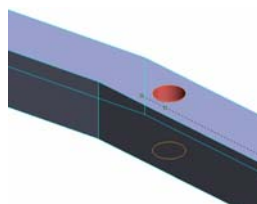


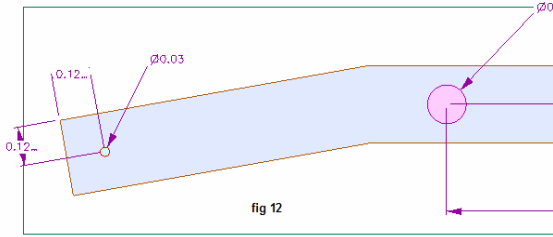
fig 11

13. **Project** the circle. Subtract material, below the workplane, thru entire part.

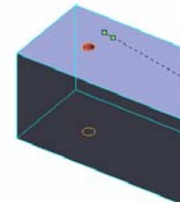


14. Create a **new sketch** on the workplane2 called **handle hole**.

15. Draw and locate the .03" diameter circle as shown in (fig 12).

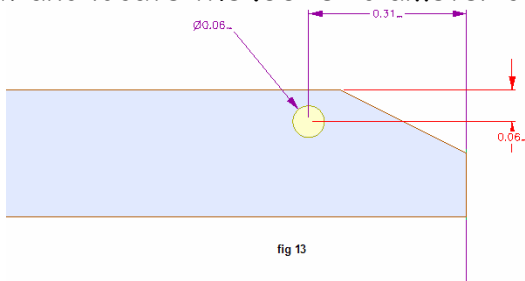


16. **Project** the circle. Subtract material, below the workplane, thru entire part.

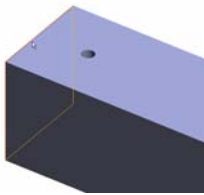
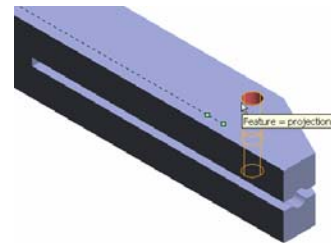


17. Create a **new sketch** on the workplane2 called **thumb screw hole**.

18. Draw and locate the .0625" diameter circle as shown in (fig 13).

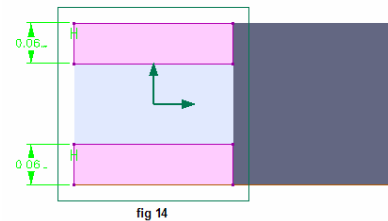


19. **Project** the circle. Subtract material, below the workplane, thru entire part.

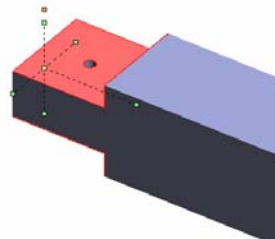
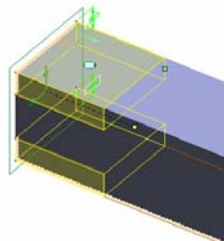


20. Select the top face of the compass leg. Right click on the face and pick **New Sketch**. Name the sketch **handle cut**. Workplane3 will also be created when you due this.

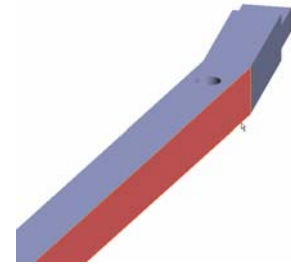
21. Draw and locate the two rectangles as shown in (fig 14). Make sure the rectangles line up with the edges of the compass leg.



22. **Extrude** the rectangles .25 " Subtract material, below the workplane.



23. Select the side face of the compass leg. Right click on the face and pick **New Sketch**. Name the sketch **adjust slot**. Workplane4 will also be created when you due this.



24. Draw and locate the slot as shown in (fig 15). Use construction lines to center the slot on the compass leg and the adjust hole.

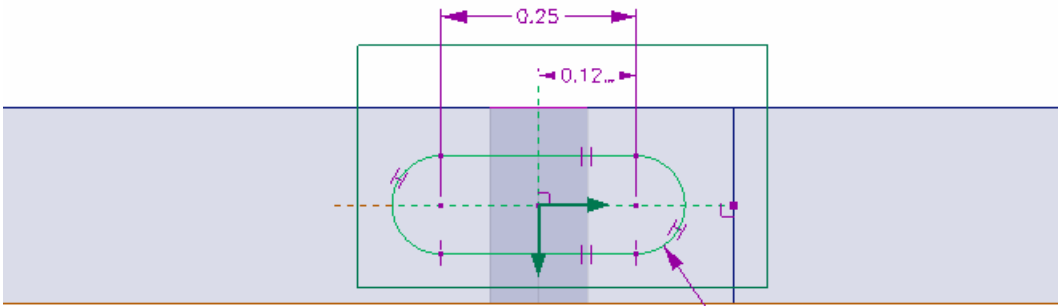
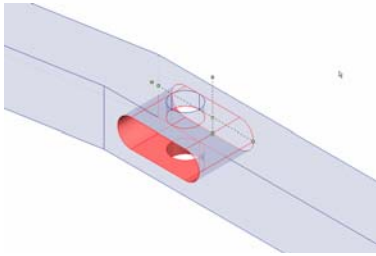


fig 15

25. **Project** the slot. Subtract material, below the workplane, thru entire part.



26. Select the edges of the compass leg as shown in (fig 16).

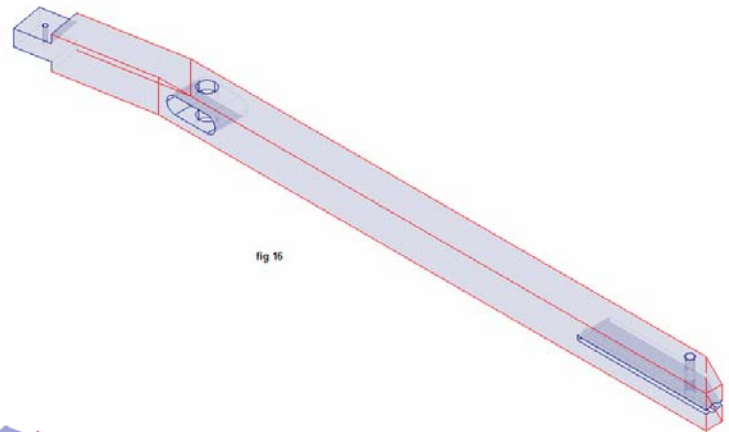
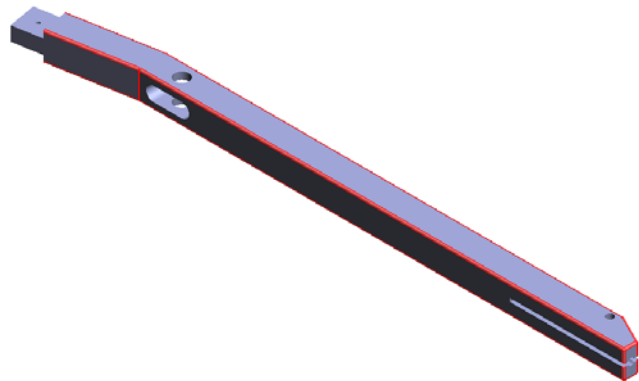
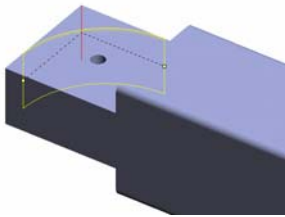
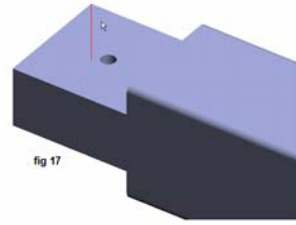


fig 16

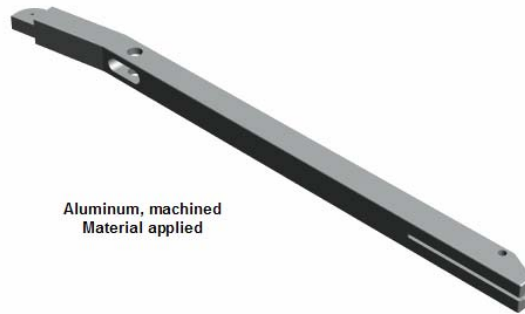
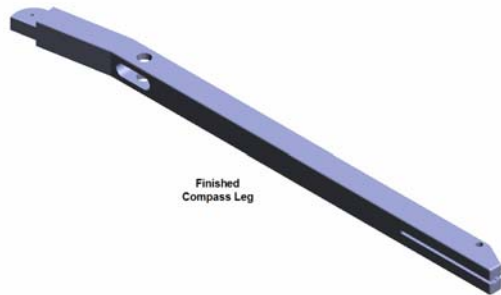
27. **Round** the edges of the compass leg with an R .015"



28. Select the top edge of the compass leg handle section as shown in (fig 17).

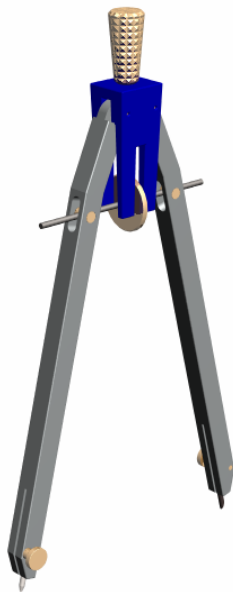


29. Round this edge with an R .1875" Your compass leg is complete. Don't forget to save your work.



ASSEMBLY

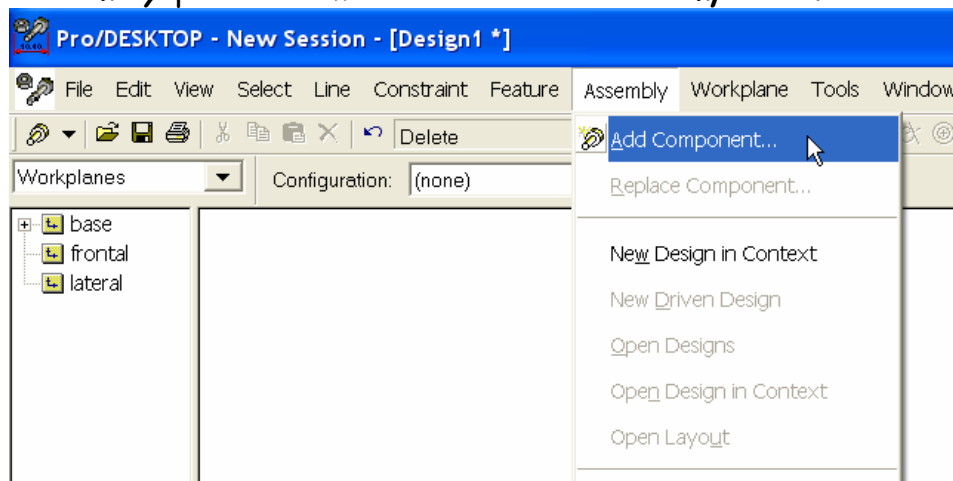
You now have all the parts necessary to assemble your Drafting Bow Compass. Some of the parts must be added more than once. Once you have the compass assembled you should be able to move one of the legs and watch the other leg move in and out. The center wheel should also move up and down as you move the leg.



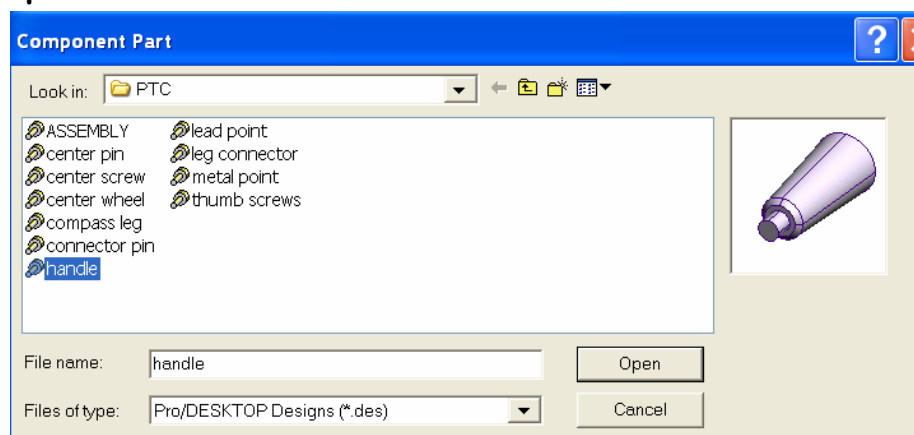
Assemble the parts of the Drafting Bow Compass:

(Center Axis, Mate, Align Offset, & Orient)

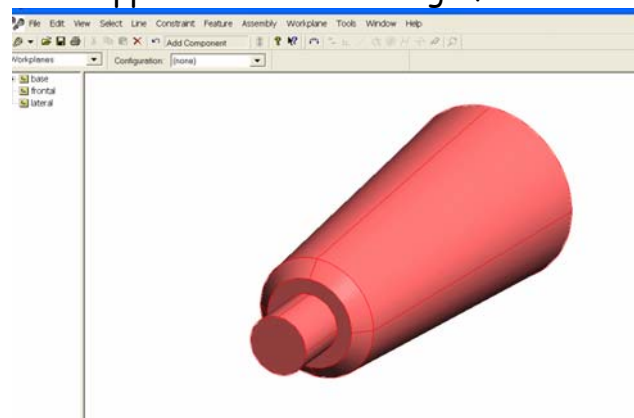
- Click on the **New Design** icon located in the upper left corner of the screen.
- **Save** the file as *compass assembly.des*.
- Use the **Assembly** pull-down menu and select **Add Component**.




- Find the folder that you saved all the compass parts in. Select the **handle** .des and pick **Open**.

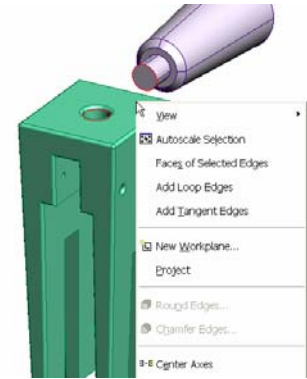


- The handle should now appear in the new design file window.

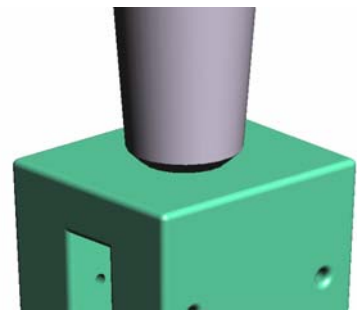
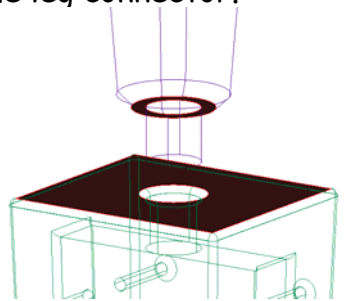


- Add the **leg connector** .des part the same way you added the handle. You can use the **Select Parts Tool**  to move the parts away from each other.

- Use the **Select Edges Tool** and select the bottom edge of the handle and the top edge of the hole on the leg connector. Remember to hold down the **Shift** key after you make your first selection to select addition
- Right click and select **Center Axis**.

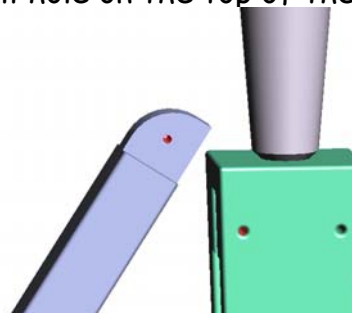


- Use the **Select Faces Tool** to select the bottom face of the shoulder on the handle and the top face of the leg connector.
- Right click and select **Mate**.



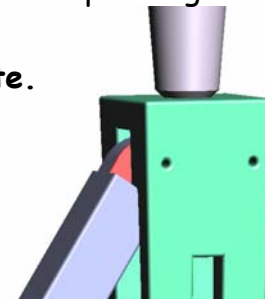
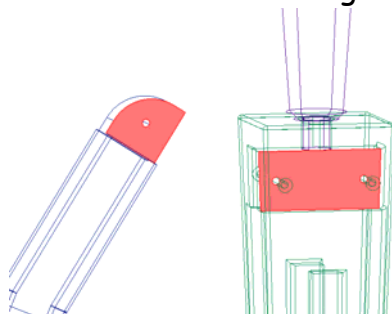
- Use the **Select Parts Tool** and select the handle. Right click and select **Fix Components**. This will keep the handle from moving as you add the additional parts.

- Use the **Assembly** pull-down menu and select **Add Component** to bring in your next part. Select the compass leg and add it twice so you have two legs.
- Use the **Select Faces Tool** to select the small hole on the top of the compass leg and the small hole on the leg connector.
- Right click and select **Center Axis**.

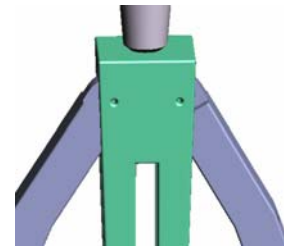


- Use the **Select Faces Tool** and select the top face of the compass leg and the inner face of the leg connector.

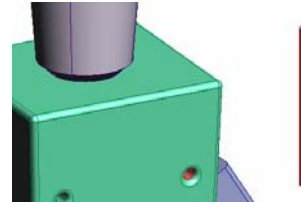
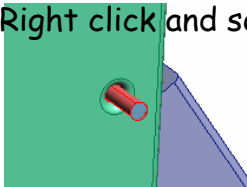
- Right click and select **Mate**.



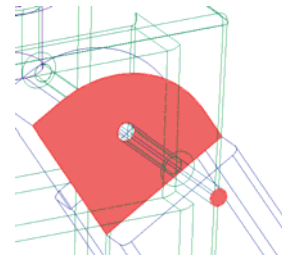
- Repeat the process to assemble the other compass leg.



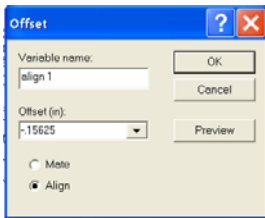
- Use the **Assembly** pull-down menu and select **Add Component** to bring in your next part. Select the connector pin and add it twice so you have two pins.
- Use the **Select Faces Tool** to select one of the small holes in the top of the leg connector and the outside face on the connector pin.
- Right click and select **Center Axis**.



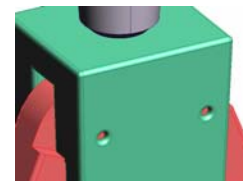
- Use the **Select Faces Tool** to select the outside face of the compass leg and the end face on the connector pin.



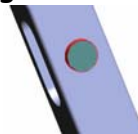
- Right click and select **Offset**. Check **Align** and set the Offset (in) at $-.15625$



- Repeat the process to assemble the other connector pin.



- Use the **Assembly** pull-down menu and select **Add Component** to bring in your next part. Select the center pin and add it twice so you have two pins.
- Use the **Select Faces Tool** to select one of the round holes in the slot at the top of the compass leg and the outside face on the center pin.



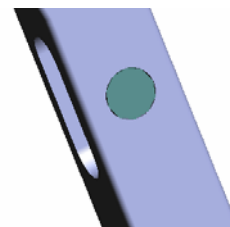
- Right click and select **Center Axis**.




- Use the **Select Faces Tool** to select the outside face of the compass leg and the end face of the center pin.

- Right click and select **Align**.

- Repeat the process to assemble the other center pin.



- You now need to assemble the center screw and center wheel before you can assemble them to the center pins in the compass legs.
- **Add Component** to bring in your both of these two parts. Select the center wheel and center screw.

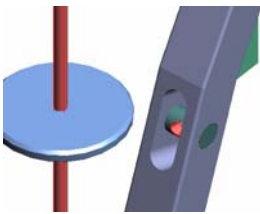
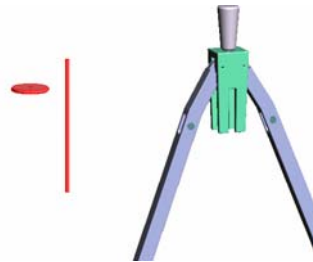
- Use the **Select Parts Tool**  to move the parts away from main compass.

- Use the **Select Faces Tool** to select the outside face of the center screw and the hole of the center wheel.

- Right click and select **Center Axis**.

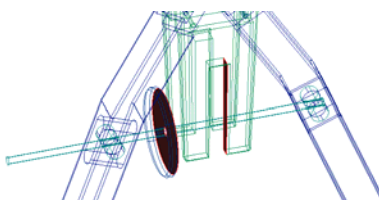
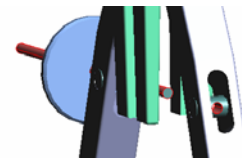
- Use the **Select Faces Tool** to select the outside face of the center wheel and the end face of the center screw.

- Right click and select **Offset**. Check Align and set the Offset (in) at -1.0625



- Use the **Select Faces Tool** to select the outside face of the center screw and the hole of the center pin.
- Right click and select **Center Axis**.

- Repeat the same face selection for the other center pin and again assemble using **Center Axis**.



- Use the **Select Faces Tool** to select an outside face of the center wheel and inside face of the slot on the leg connector.

- Right click and select **Offset**. Check **Mate** and set the Offset (in) at .03125

- You should now be able to use the **Select Parts tool** to pick one of the compass legs.

As you move the leg and the legs should open and close as the center wheel goes up and down.



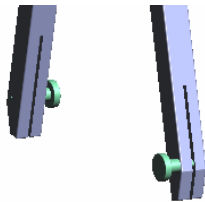
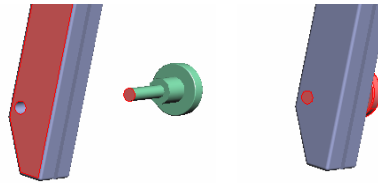
- Use **Add Component** to bring in two thumb screws.

- Use the **Select Faces Tool** to select the outside face of the thumb screw and the bottom hole of the compass leg.

- Right click and select **Center Axis**.



- Use the **Select Faces Tool** to select the end face of the thumb screw and the back face of the compass leg.
- Right click and select **Align**.

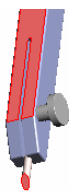
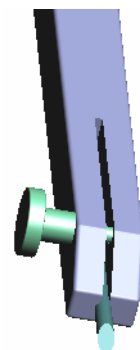


- Repeat the process for the other thumb screw but make sure to have it face the opposite direction on the compass leg.

- Use **Add Component** to bring in the metal point.
- Use the **Select Faces Tool** to select the circular face of the slot hole on the bottom of the compass leg and the face of the metal point.
- Right click and select **Center Axis**.
- Use the **Select Faces Tool** to select the bottom face of the compass leg and the flat end face of the metal point.
- Right click and select **Offset**. Check **Mate** and set the Offset (in) at $-.875$.



- Use **Add Component** to bring in the lead point.
- Use the **Select Faces Tool** to select the circular face of the slot hole on the bottom of the other compass leg and the face of the lead point.
- Right click and select **Center Axis**.
- Use the **Select Faces Tool** to select the bottom face of the compass leg and the flat end face of the lead point.
- Right click and select **Offset**. Check **Mate** and set the Offset (in) at $-.75$.



- Use the **Select Faces Tool** to select the outside face of the compass leg and the slanted end face of the lead point.
- Right click and select **Orient** and set the **Angle** at 30 .

Congratulations! You have completed your assembly of the Drafting Bow Compass. You can now take the assembly into the **Photo Album** and apply some different materials and render it.

Extra bonus: Try to animate the compass leg action if you want to challenge yourself.

