SolidWorks Tutorial

Clamp

Preparatory Vocational Training
and Advanced Vocational Training

Dassault Systèmes SolidWorks Corporation,
175 Wyman Street
Waltham, Massachusetts 02451 USA
Phone: +1-800-693-9000

Outside the U.S.: +1-781-810-5011
Fax: +1-781-810-3951
Email: info@solidworks.com
Web: http://www.solidworks.com/education

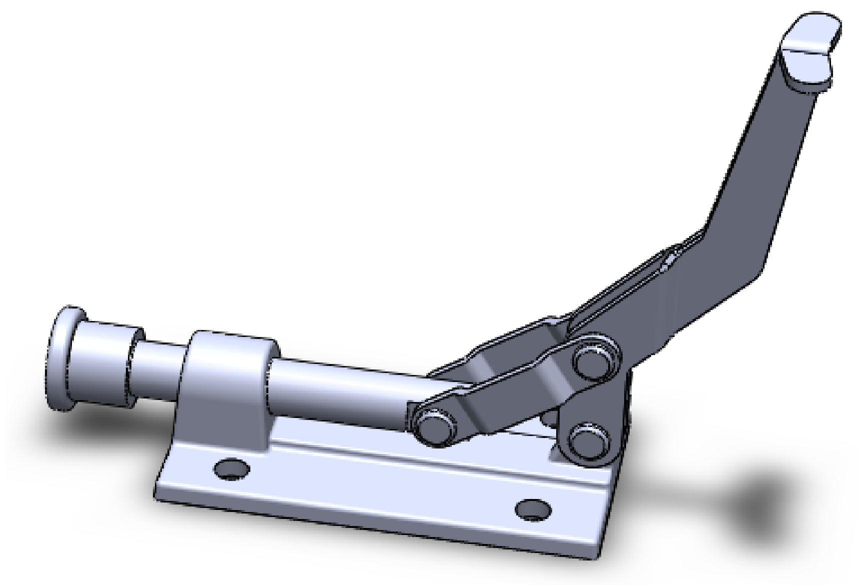
Document Number:

Tutorial: Clamp

In this tutorial we are going to make a clamp. Many of the topics we will use you have seen already, but we are also going to show you some new tools, including;

- Movements in an assembly.
- The creation of a rendering with PhotoView 360.

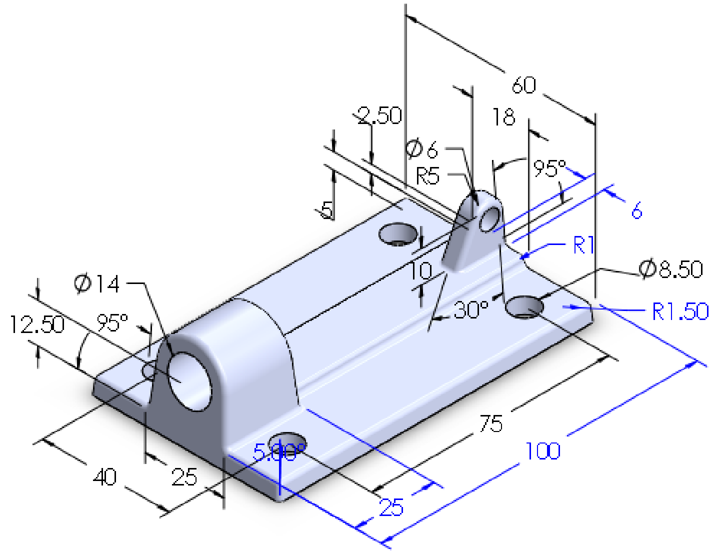
First, we are going to model the parts, and then we will make the assembly, in which you can see the exact movement of the product. Finally, we are going to make a rendering in PhotoView 360.



Base

Work plan

The first part we are going to make is the base. In the illustration below you can see the dimensions.



First, you will make a work plan. How would you build this part?

The main problem in this part is that almost all the vertical planes are at an angle of 5°, which is often the case with castings. To achieve that angle in the model, we use a new feature: **Draft**.

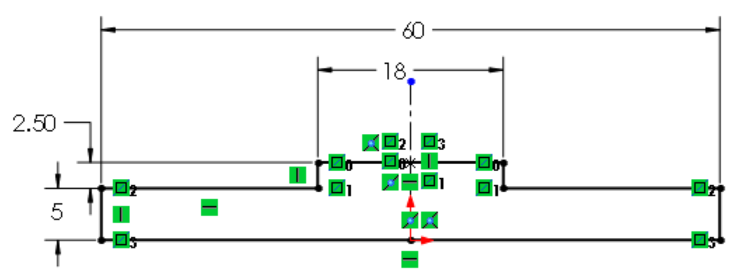
Make a plan by yourself for how to create this model.

1 Start SolidWorks and open a new part. Set the units to MMGS.

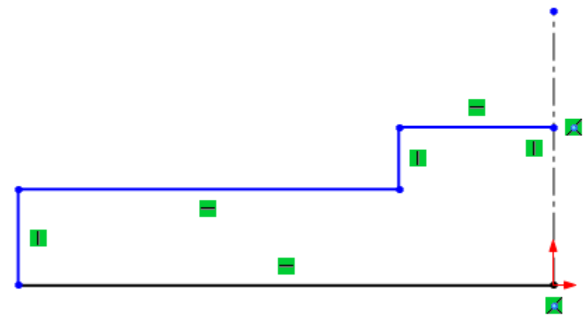
2 Select the **Front Plane** and make a sketch like you see in the illustration on the right.

Can you build this sketch by yourself? Good, continue to step 6.

If you cannot build this sketch, then follow the next steps.

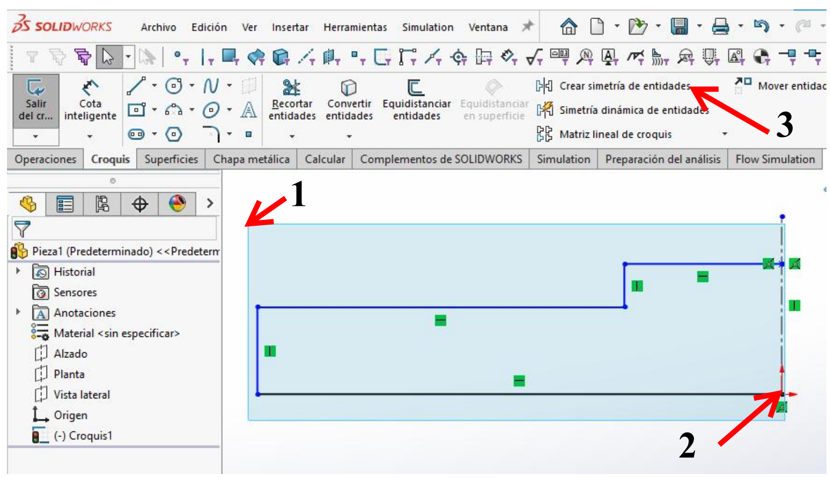


3 Draw the lines as shown on the right. Note the position of the origin.

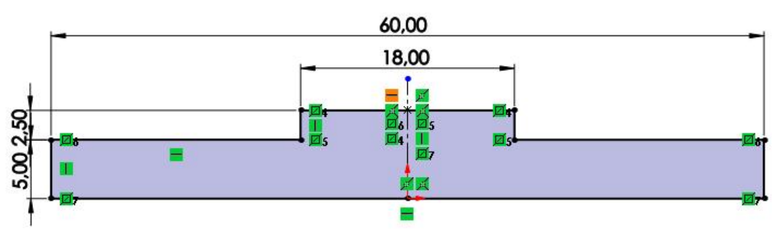


4 Now, select the whole sketch (all lines and the centerline). The easiest way to do this is by drawing a frame around the whole sketch.

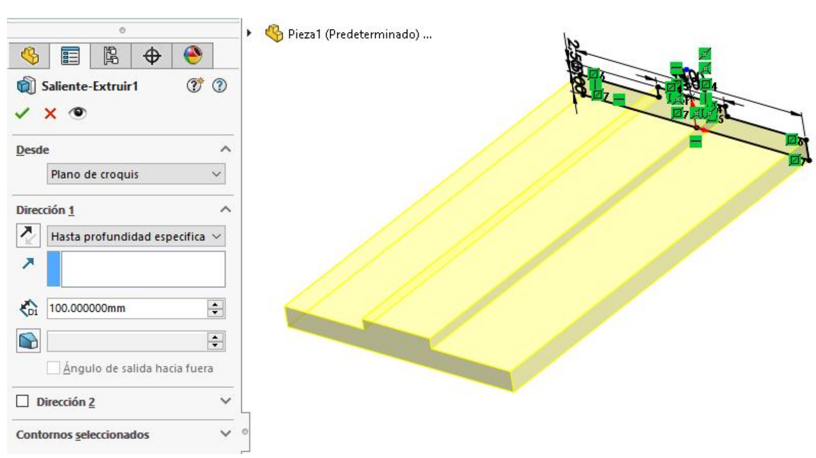
Next, click on **Mirror Entities** in the **CommandManager**.



5 Set the dimensions in the sketch as shown on the right.



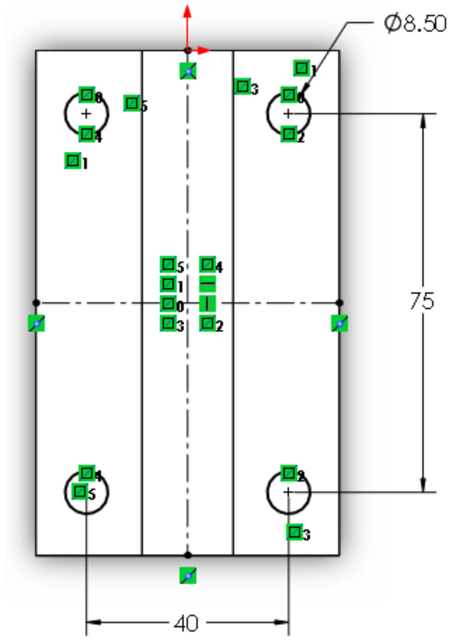
6 Extrude the sketch over a length of **100 mm**.



7 We are now going to make the mounting holes. Create a sketch on the upper surface of the model as shown in the illustration on the right.

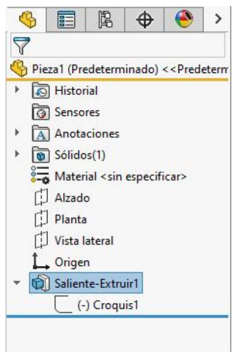
Can you build this sketch by yourself? Great! Continue to step 14.

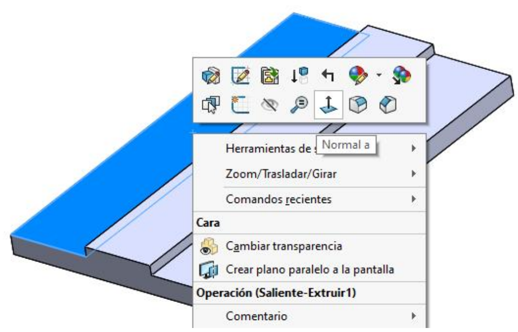
If you cannot build this sketch, then follow the next few steps.



8 First, select the plane where you want to make the sketch.

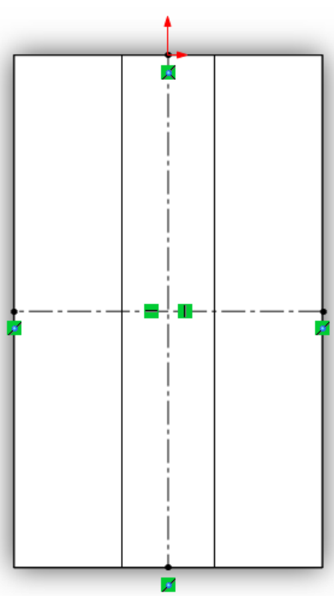
Click on **Normal To** in the menu that appears.



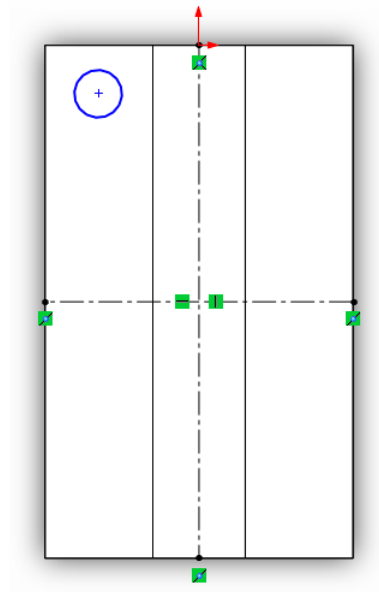


9 Next, draw the two **centerlines**, as illustrated on the right.

Be careful to draw the centerlines in the exact center of the model. To see if this worked out properly, you can verify it with the **Midpoint** symbols. These are at the end of the centerlines.

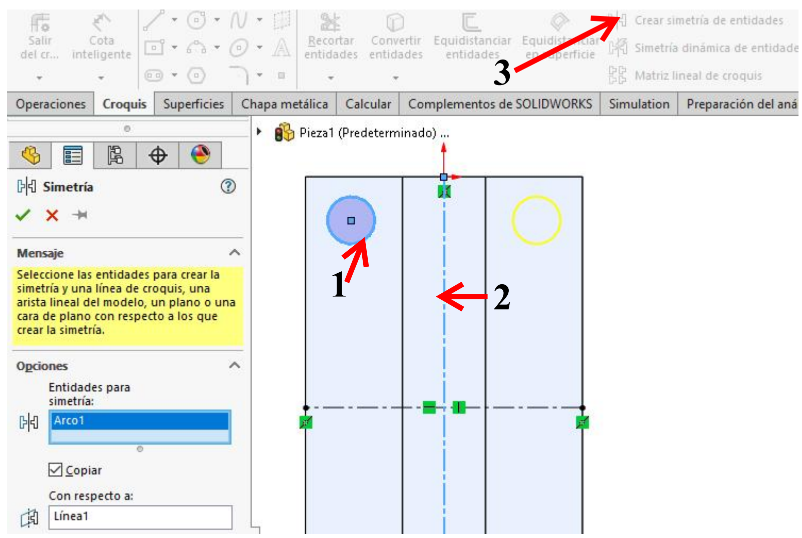


10 Draw a circle, similar to the illustration on the right.



11 Now mirror the circle:

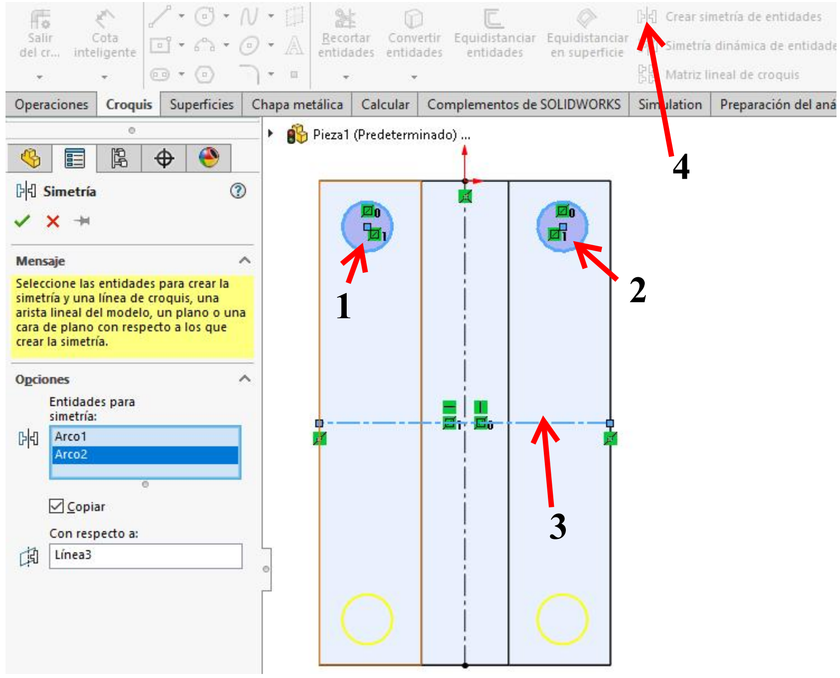
1. Select the circle.
2. Hold the <Ctrl> key and select the vertical centerline.
3. Select **Mirror Entities** in the **CommandManager**.



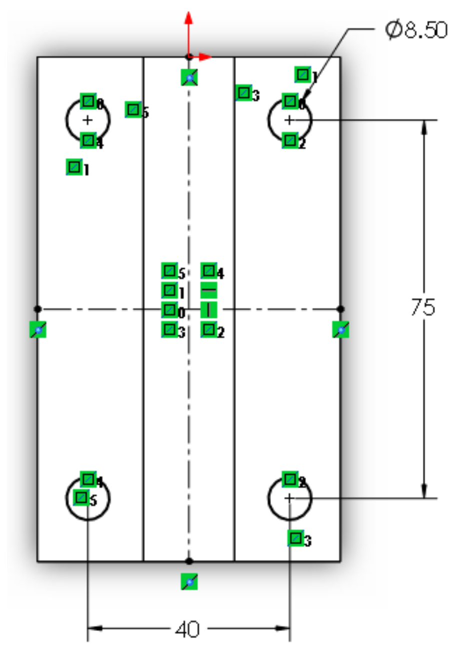
12 The two circles we have created will be mirrored a second time:

1-3 Select the two circles we have already drawn before and the horizontal centerline. Use the <Ctrl> key.

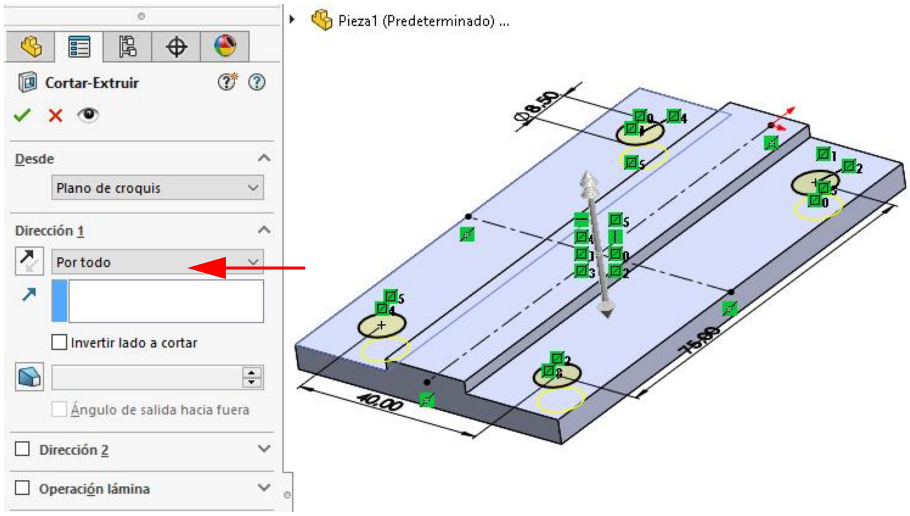
4 Select **Mirror Entities** in the **CommandManager**.



13 Add the dimension as shown in the sketch.



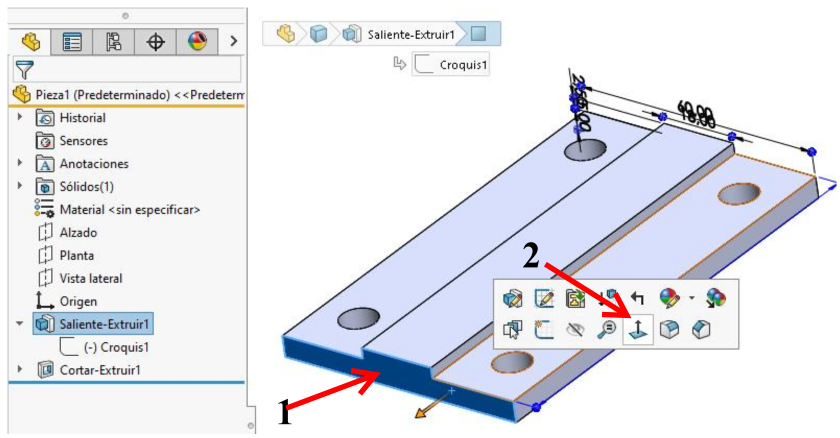
14 Make an **Extruded Cut** from the sketch with the depth **Through All**.



Tip: In these two sketches we have mirrored some entities. This not only saves time because you have to draw less, but the mirrored parts also remain constrained to each other and will always be symmetrical.

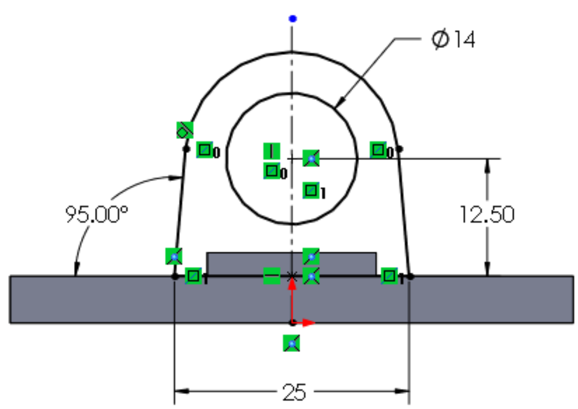
15 Now, select the front plane of the model and select **Normal To**.

Make a sketch on this plane.

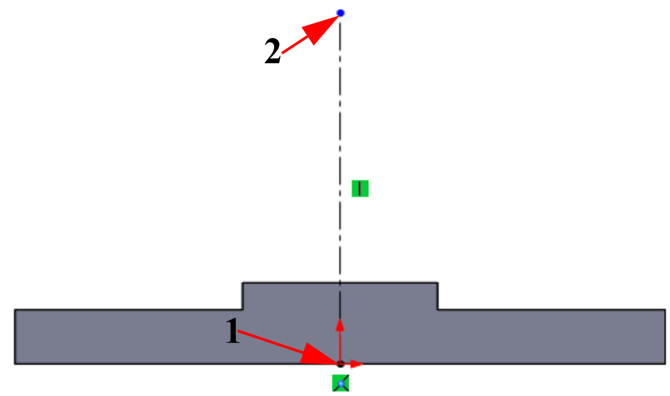


16 Can you build this sketch all by yourself? Great! Continue to step 25.

If you cannot build this sketch, then follow the next steps.



17 First, draw a **centerline** from the origin vertically upwards. The exact length does not matter.

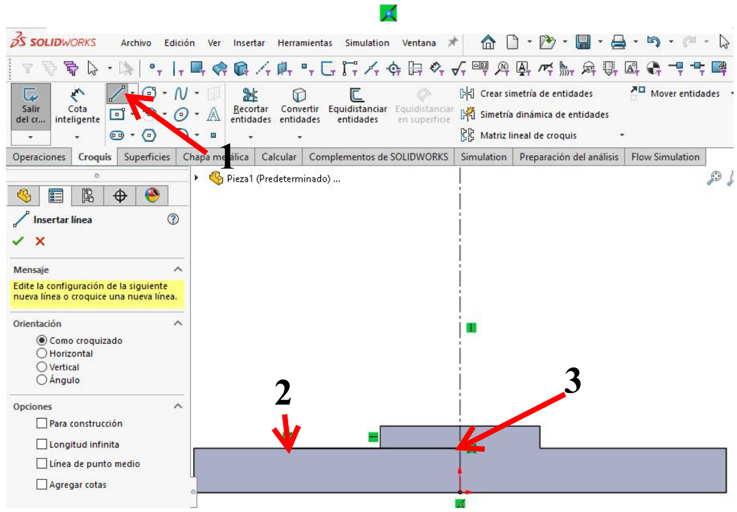


18 Draw a horizontal **line** as illustrated on the right.

The beginning of the line is at the upper surface of the model.

The endpoint is on the vertical centerline.

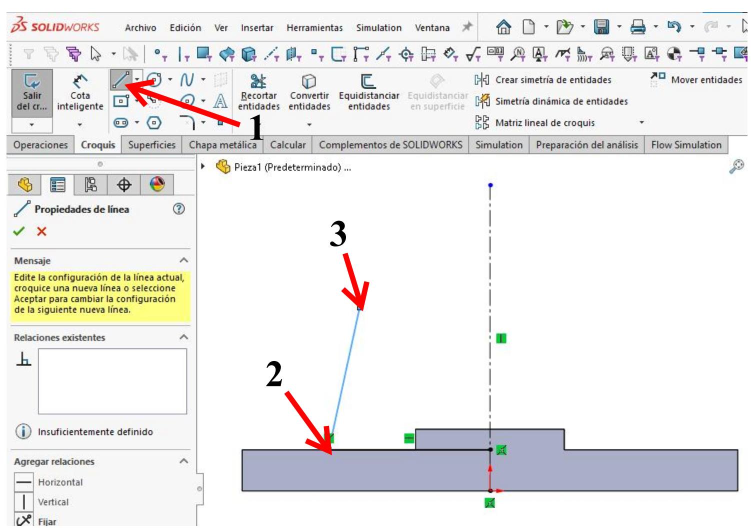
Push the <Esc> key to abort the line command.



19 Now, draw a second **line** as shown.

The beginning of the line is exactly on the beginning of the last line you drew.

The line is not positioned vertically but at a slight angle in relation to the vertical centerline.



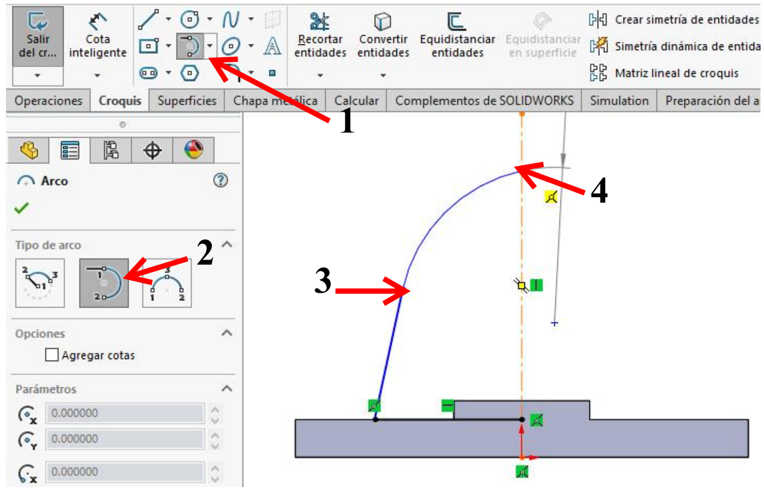
20 Create an arc:

1 Click on **Arc** in the **CommandManager**.

2 Click on **Tangent Arc** in the **PropertyManager**.

3 Click on the endpoint of the line you have just drawn to get the first point of the arc.

4 To get the endpoint of the arc, click on the centerline as shown.

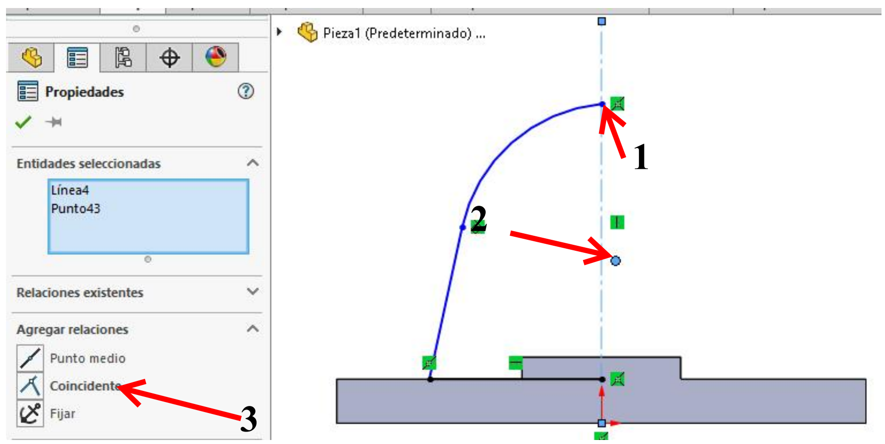


5 Click the <Esc> key to abort the command.

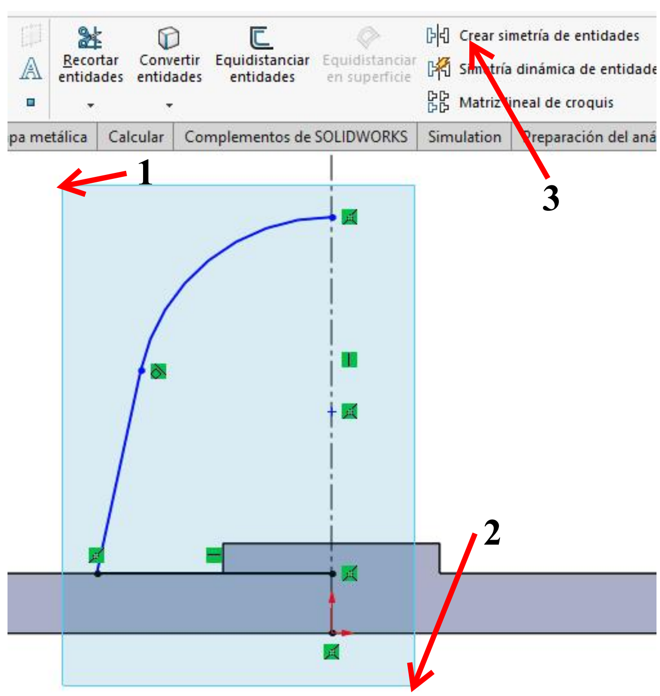
21 Select the centerline.

Hold the <Ctrl> key and select the center of the arc. This is marked as a little **+**.

Click on **Coincident** in the **PropertyManager**.

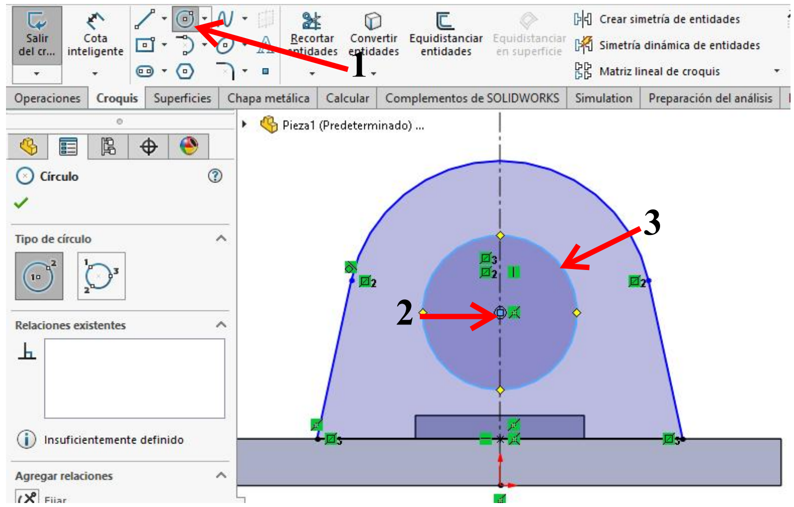


22 Select the whole sketch (including the centerline), and click on **Mirror Entities** in the **CommandManager**.

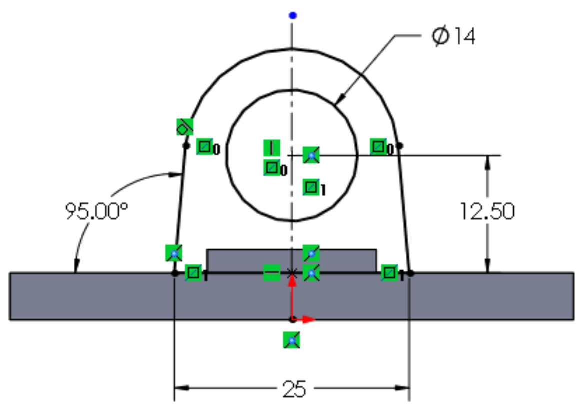


23 Next, you have to draw a circle.

Put the center of the circle on the center of the arc.

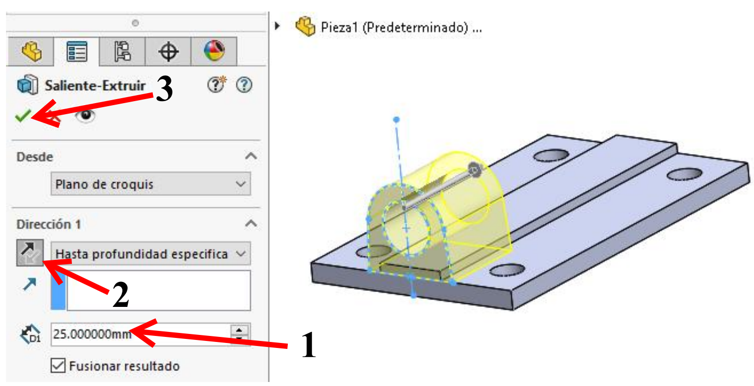


24 Set the dimensions in the sketch as shown.



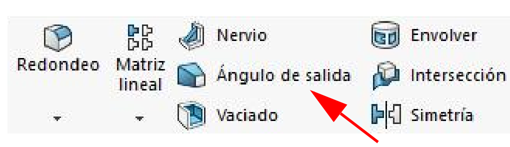
25 Extrude this sketch.

1. Set the depth to **25 mm**.
2. Make sure your extrusion extends in the right direction with **Reverse Direction**. Rotate the model to its isometric position. Otherwise, you will not be able to see this.
3. Click on **OK**.



26 We are going to set all vertical planes at an angle of 5°. For this we will use a new feature: **Draft**.

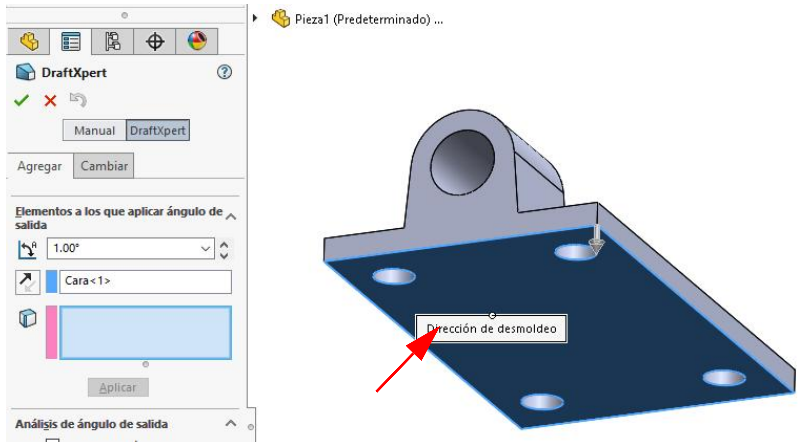
Click on **Draft** in the **CommandManager**.



27 First, we select the **Neutral Plane**. This is the partitioning plane from the mold or matrix.

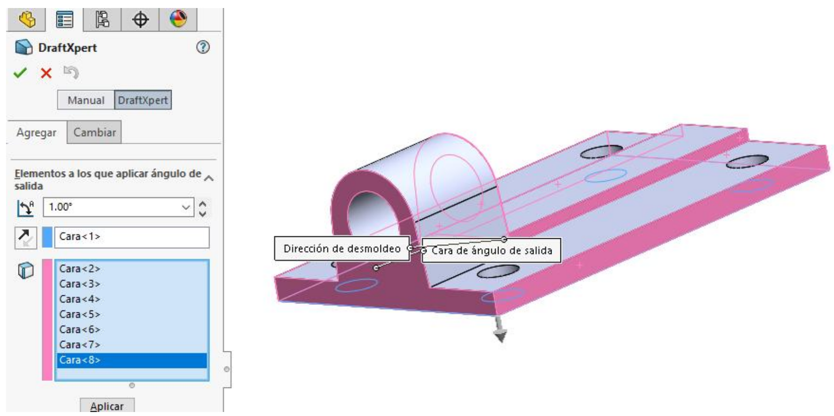
Rotate the model so you have a good view of the bottom.

Select the bottom plane.



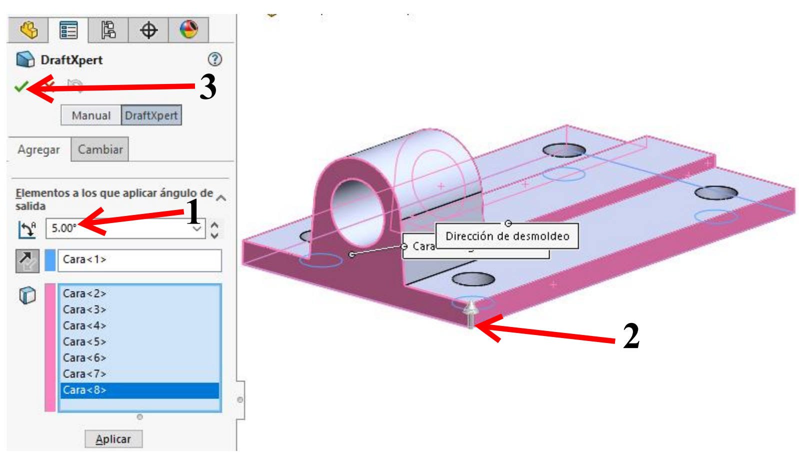
28 We can now select the planes that we want to tilt.

Click on all vertical planes as shown in the illustration on the right. There are 7 planes in total. To select them all, you will have to rotate the model.



29 Next, you have to set two more items.

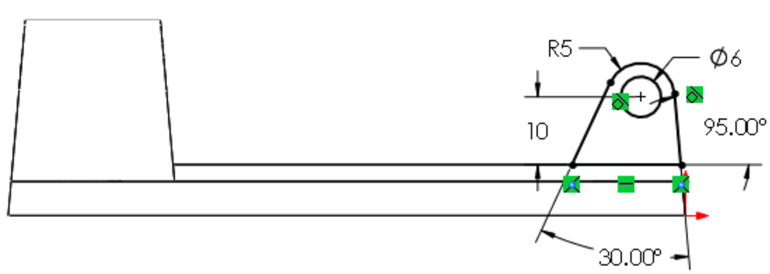
1. Set the **Draft Angle** to **5°** in the **PropertyManager**.
2. In the model, the angle direction is indicated by an arrow. Make sure this arrow points upward. You can change direction by clicking on the arrow.



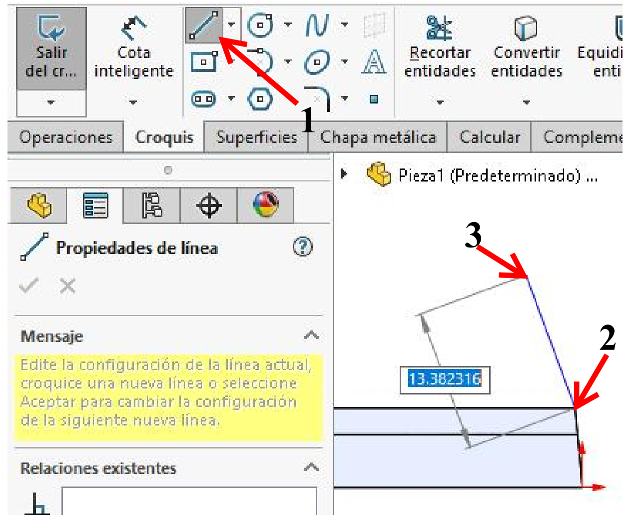
3. Click on **OK** in the **PropertyManager**.

30 Select the **Right Plane** in the model and make the sketch as shown.

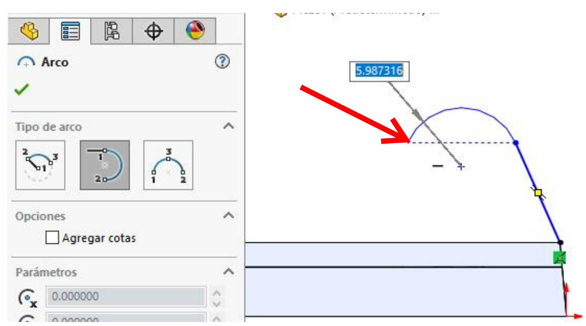
If you can do it yourself, then continue to step 37, if not, follow the next few steps.



31 Draw a line similar to the one in the illustration.



32 Use the **Autotransitioning** technique that we used before when we wanted to draw an arc using the line command.

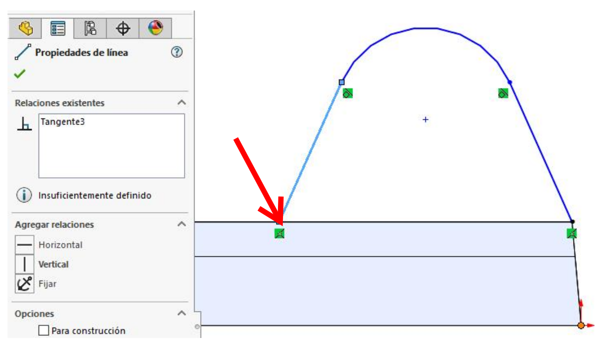


1. Move the cursor away from the last point that you drew.
2. Return the cursor exactly to the last point again (do NOT click on it!).
3. Move the cursor away and you will be drawing an arc.
4. Click as shown in the illustration to set an arc.

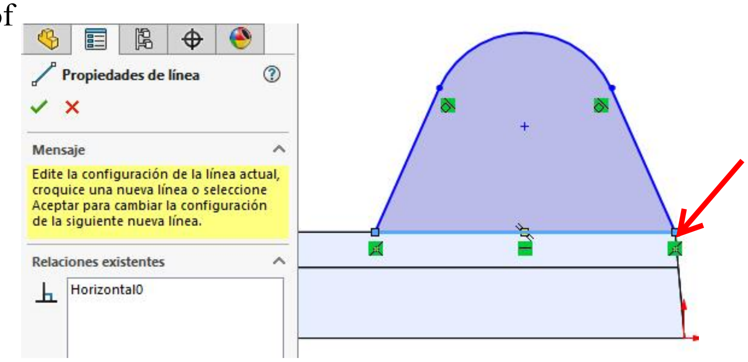
33 Click on the spot as shown on the right.

Use the dotted auxiliary line: it is aligned to the arc.

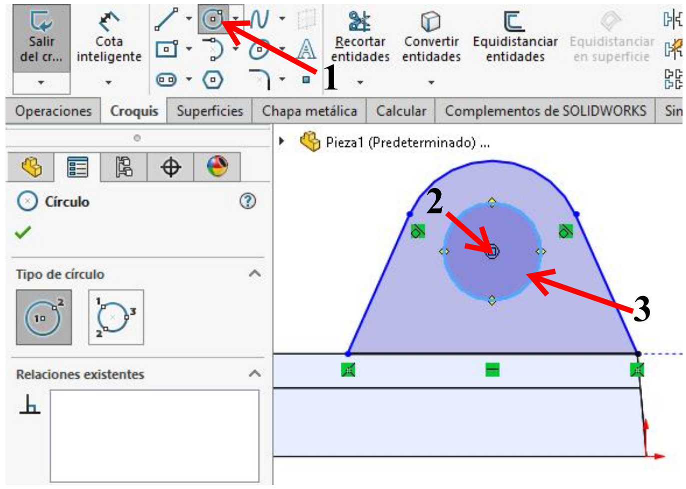
Note the two yellow icons near the cursor. These must be visible at the moment that you set the end point.



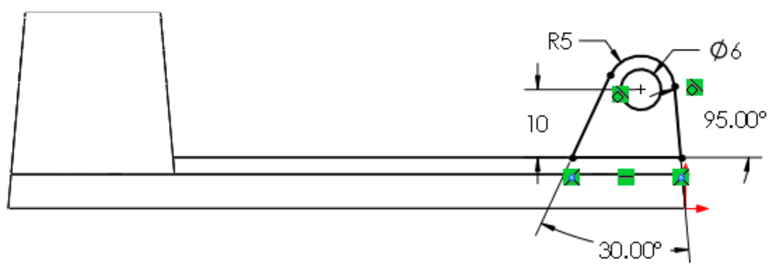
34 Click on the beginning of the first line now.



35 Draw a circle with its midpoint on the midpoint of the arc.

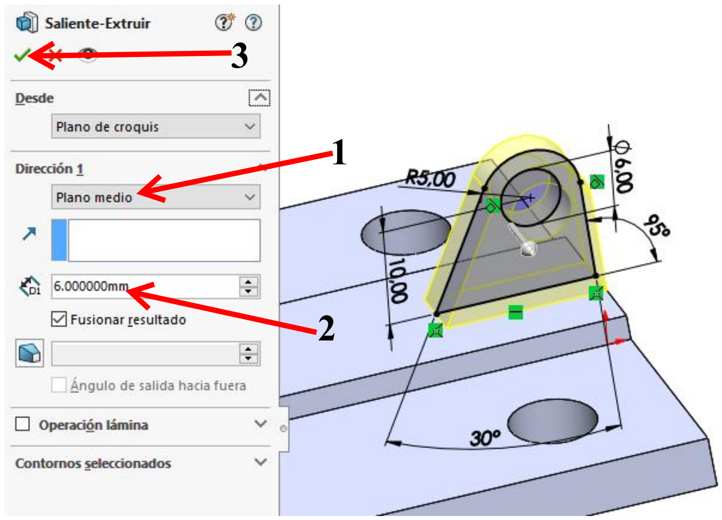


36 Set the dimensions as shown on the right.



37 Extrude this sketch.

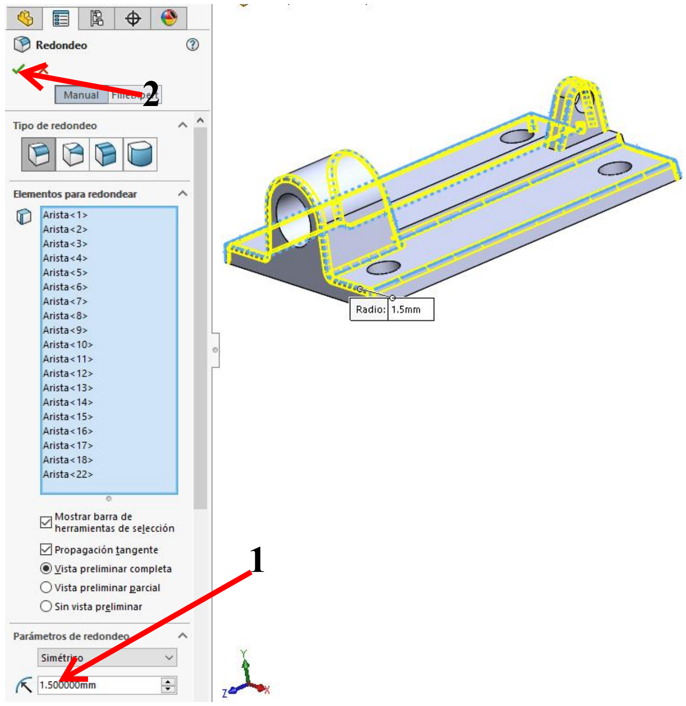
1. Select the option **Mid Plane** in the **PropertyManager**.
2. Set the distance to **6 mm**.
3. Click on **OK**.



38 Round the corners from the model with the **Fillet** feature.

Set the radius to **1.5 mm** and select the edges as shown on the right.

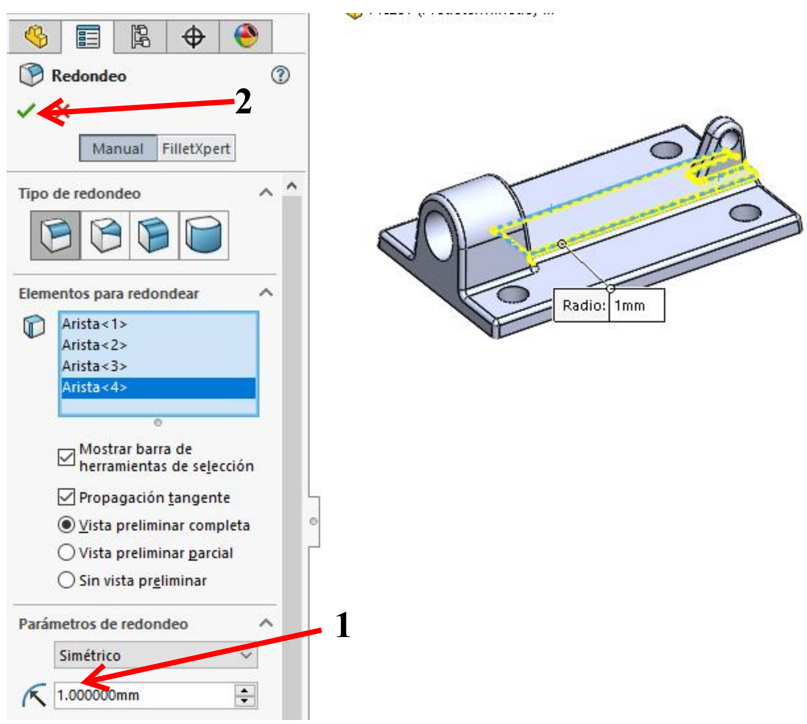
Click on **OK**.



39 Use the **Fillet** feature again to round off the rest of the edges. Do this using a radius of **1 mm**.

40 The first part of the clamp is now ready.

Save it as: base.SLDPRT.



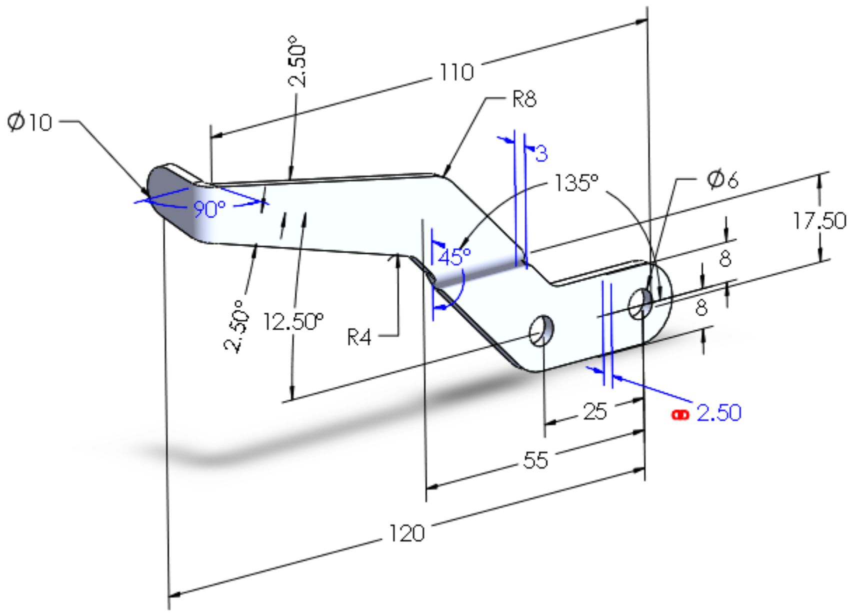
Arm

Work plan

The next part we will create is half of the arm. This part is made from sheet metal, so we will be using the SolidWorks **Sheet Metal** functions.

To make this part you need to use two new features:

1. **Jog**, which allows you to make a double bend in a part.
2. **Sketched bend**, which allows you to draw a line on a sheet of metal that will act as a bending line.



Making this part is actually very simple.

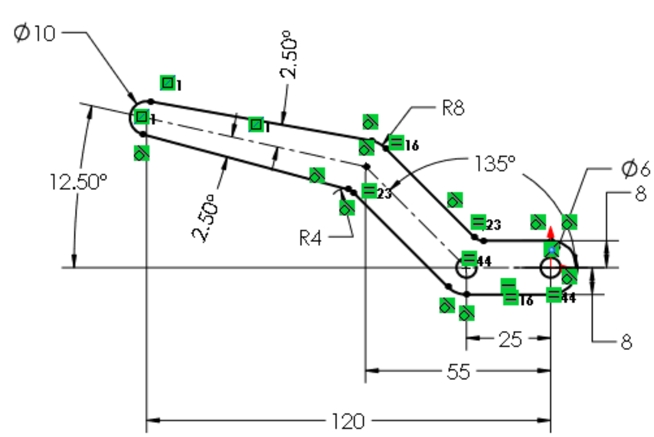
1. Use sheet metal. While making this part is easy, the sketch we have to make is fairly complicated.
2. Next we will **Jog** the part.
3. Finally, we will bend the sheet with the **Sketched Bend** command.

41 Open a new part and set the units to MMGS.

Select the **Right Plane** and make the sketch as shown on the right.

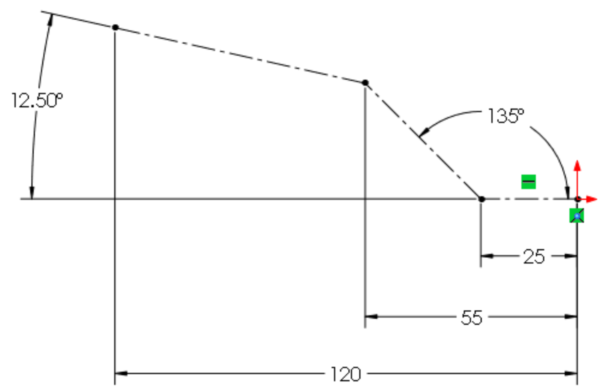
Did you succeed? Continue with step 56.

If you need help, follow the next few steps.



42 Draw three centerlines on the **Right Plane** first, as shown on the right. Draw the first centerline horizontally from the origin to the left.

Set the dimensions as shown in the illustration.



43 To offset sketch entities:

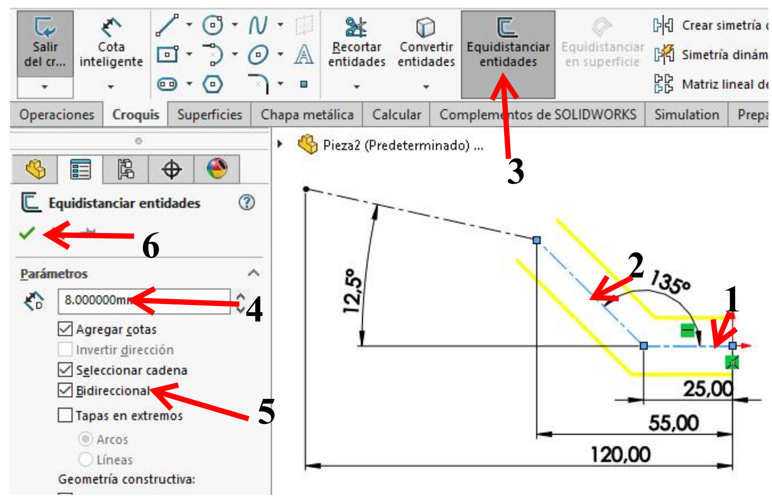
1,2 Select the two bottom centerlines (use the <Ctrl> key).

3 Click on **Offset Entities** in the **CommandManager**.

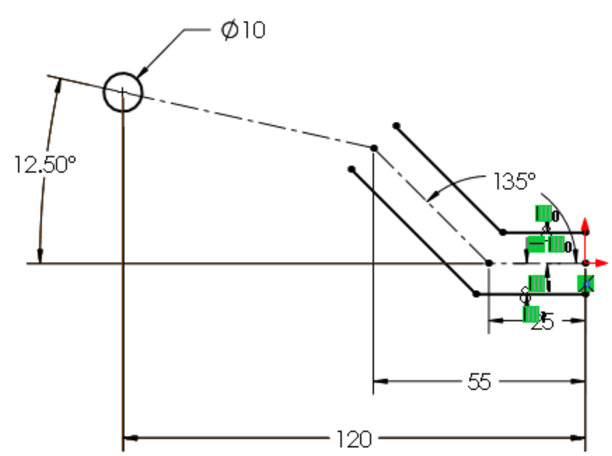
4 Set the distance to **8 mm** in the **PropertyManager**.

5 Check the option **Bi-directional**.

6 Click on **OK**.

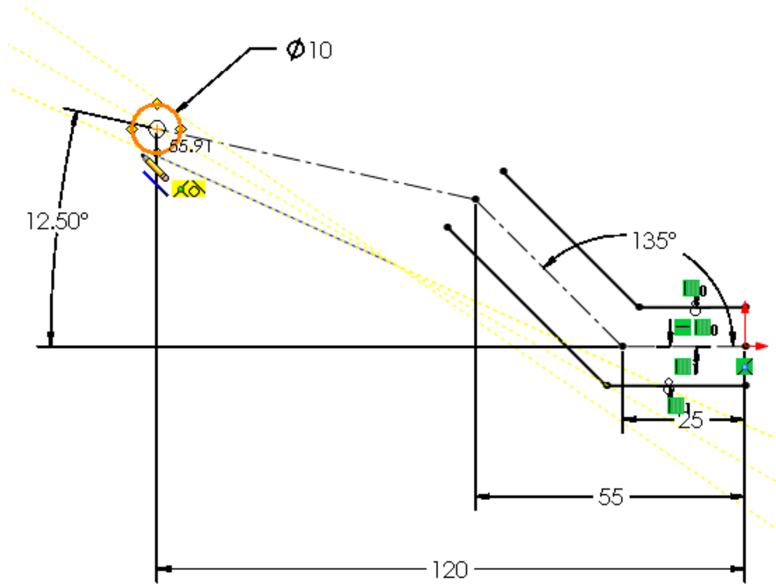


44 Draw a **circle** with the midpoint on the left end of the centerline. Set the dimension to **Ø10 mm**.



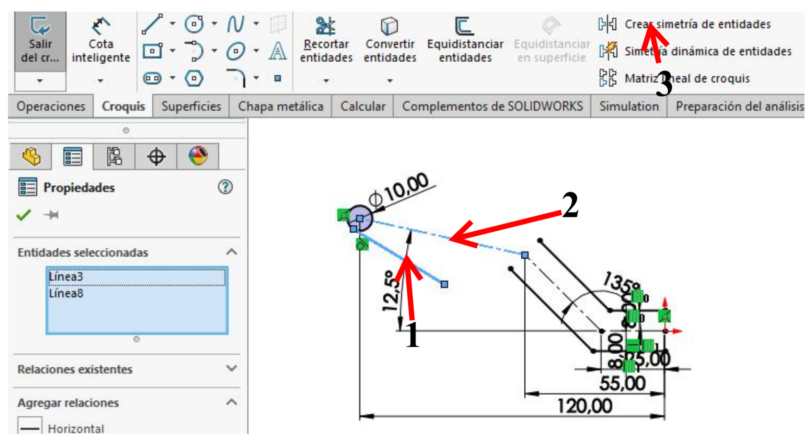
45 Next, draw a line.

1. Set the beginning at a random spot, as shown on the right.
2. Set the second point on the circle. Make sure it touches the circle at the correct spot. You can tell by the coincident and tangent icons that pop up at the cursor.
3. Push the <Esc> key on the keyboard to abort the **Line** command.

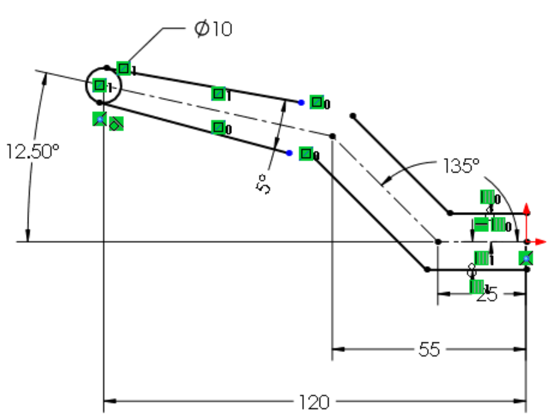


46 Select the line and the centerline as shown on the right.

Click on **Mirror Entities** in the **CommandManager**.

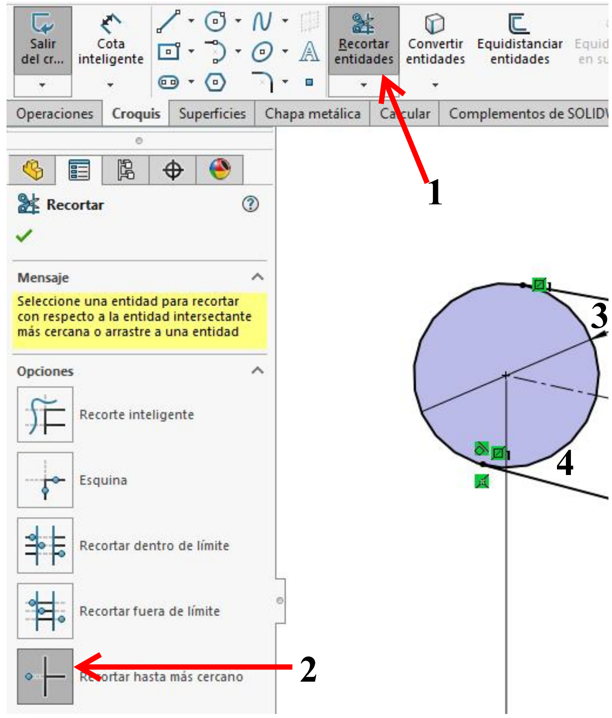


47 Set the angle between the lines to **5°**.



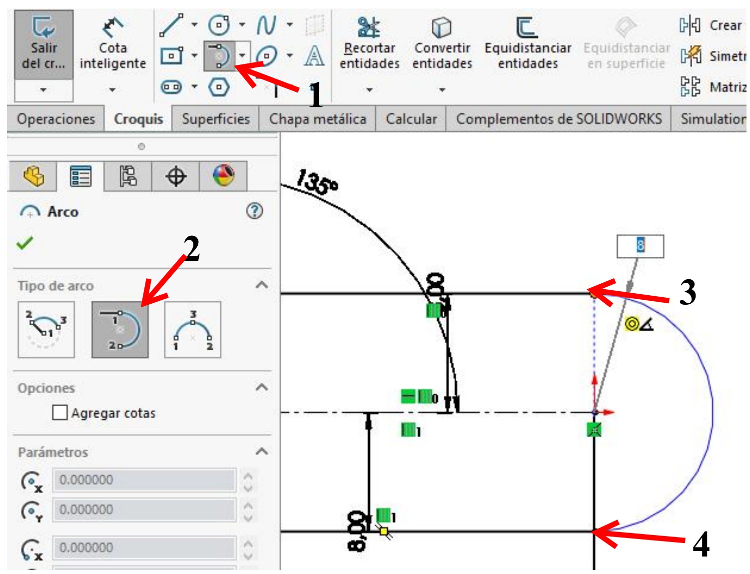
48 Next, we will trim the part of the circle that lies between the lines.

1. Click on **Trim Entities** in the **CommandManager**.
2. Click on **Trim to closest** in the **PropertyManager**.
3. Click on the parts of the circle that need to be removed.



49 We need another half circle at the other end of the sketch.

1. Click on **Arc** in the **CommandManager**.
2. Click on **Tangent Arc** in the **PropertyManager**.
3. Click on the end of the upper line.
4. Click on the end of the bottom line.



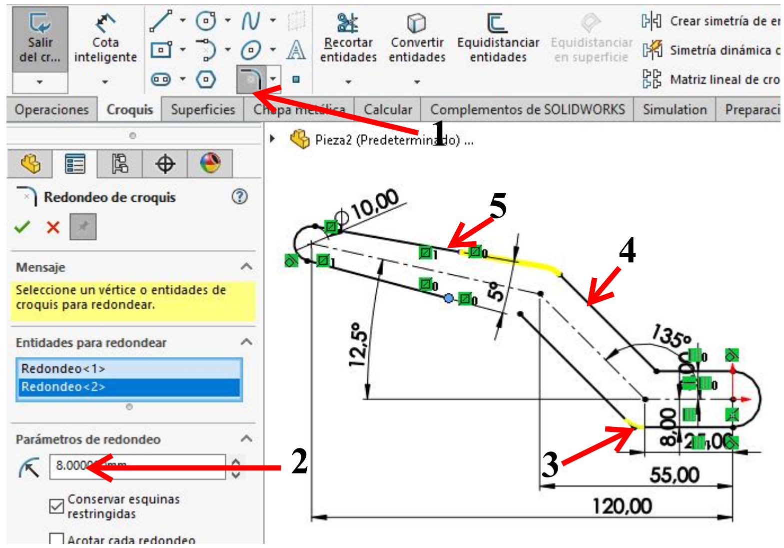
50 We want to round the four corners now.

1 Click on **Sketch Fillet** in the **CommandManager**.

2 Set the radius to **8 mm** in the **PropertyManager**.

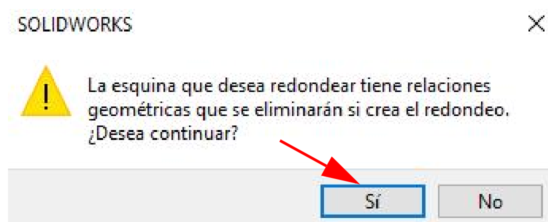
3 Click on the bottom corner as shown.

4,5 Click on both line which we want to connect with an arc.



51 A message appears. Click **Yes**.

Click on **OK**.



Explination!

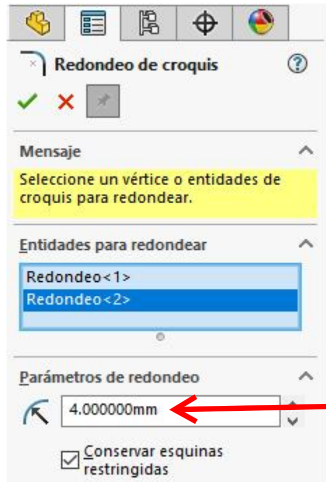
What does the message in step 51 mean?

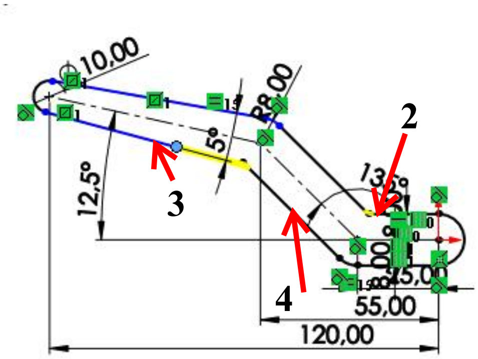
The upper sloped lines in the sketch are mirrored lines (from step 46). For this reason, the lines are connected together by a relation: they are symmetrical around the centerline and equally long.

When you want to round one of these lines, their lengths will not be equal anymore. The symmetry will be disconnected or destroyed and that is what the software warns you about.

The lines were black (fully defined) but after you click on **Yes** and the symmetry is disconnected, they will turn blue (not fully defined). We will show you how to resolve this later.

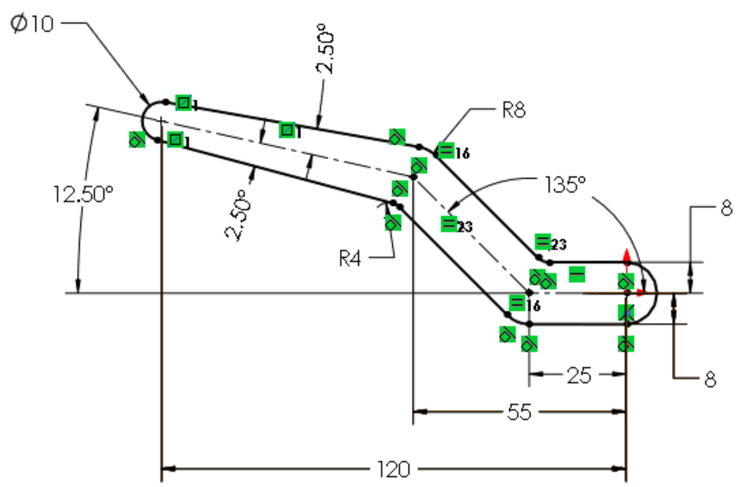
52 Set the radius to **4 mm** and round the two other corners the same way.





53 To return to a fully defined sketch, you have to follow the next few steps:

1. Remove the dimension of **5°**.
2. Add two angles of **2.5°** instead.

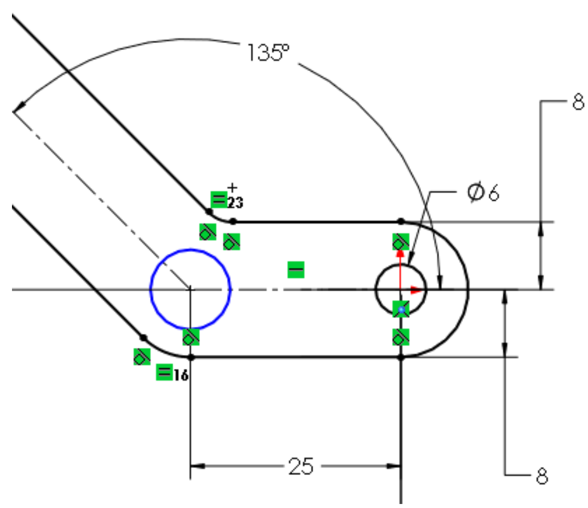


54 Finally, we have to draw two holes.

Draw two circles as shown on the right.

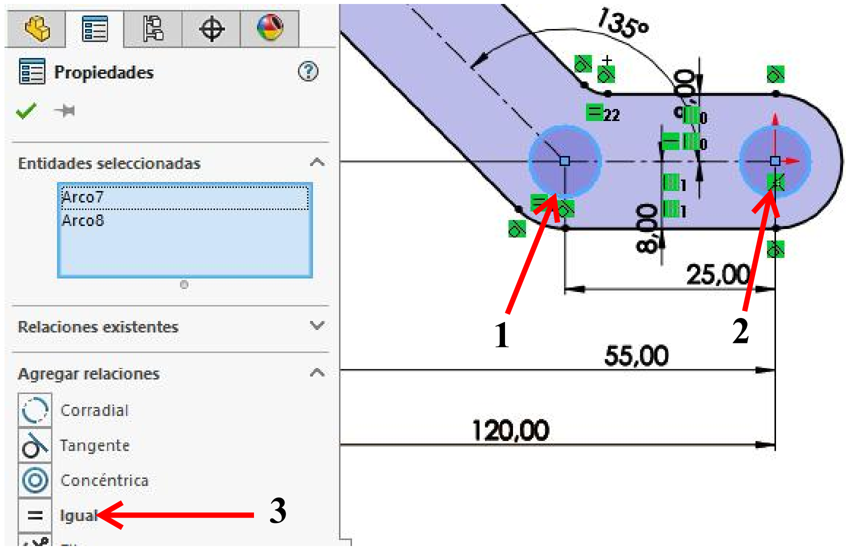
The midpoints are on the ends of the bottom centerline.

Set the size for one of the holes to Ø6 mm.



55 Select both (use the <Ctrl> key).

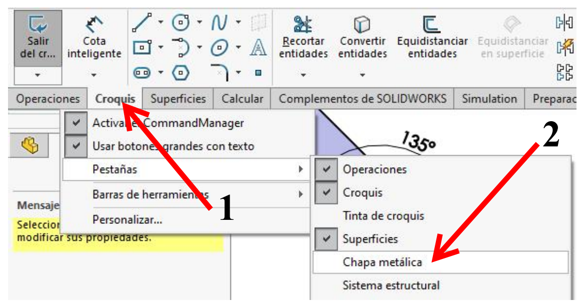
Click on **Equal** in the **PropertyManager**.



56 We will make a part with sheet metal from this sketch.

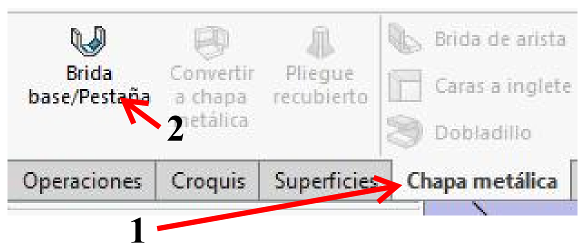
Make sure the tab **Sheet Metal** is displayed in the **CommandManager**.

If not, right-click on one of the other tabs and select the **Sheet Metal** function in the pop-up menu.



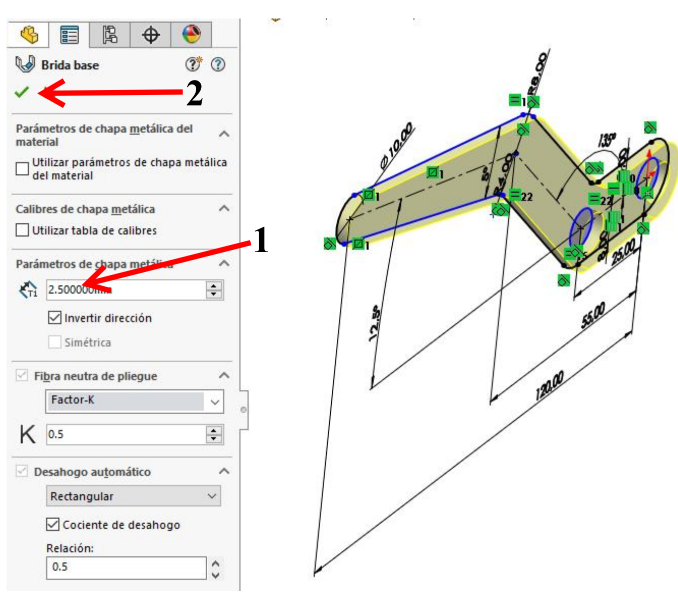
57 Click on the **Sheet Metal** tab in the **CommandManager**.

Click on **Base Flange/Tab**.



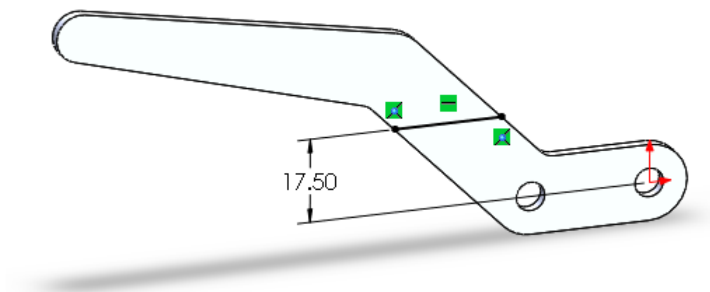
58 Set the thickness for the material to **2.5 mm** in the **PropertyManager**.

Click on **OK**.

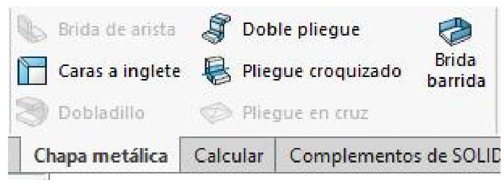


59 We will now make a double bend in the sheet. This is called a **Jog**.

Select the flat surface from the model and make the sketch as shown: it consists of one horizontal line and a dimension.

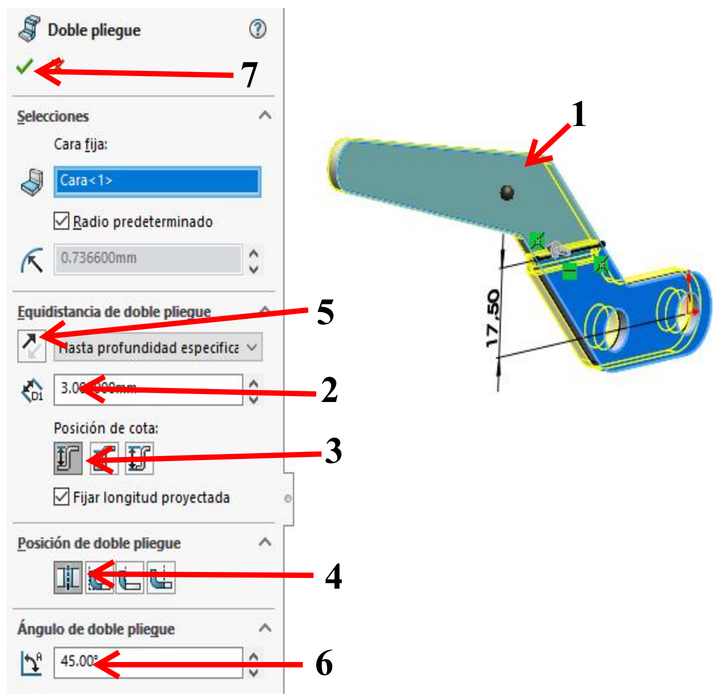


60 Click on **Jog** in the **CommandManager**.



61 To add the **Jog**:

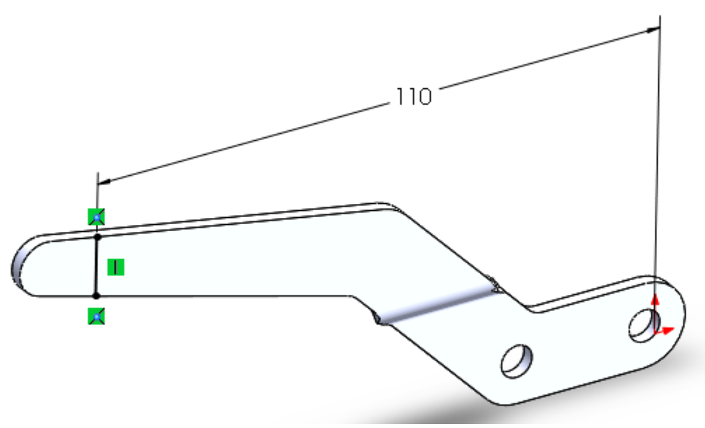
1. First, click on the part of the model that must be fixed. Click on the spot as indicated.
2. Set the distance to **3 mm**.
3. This distance is called the **Outside Offset**.
4. Select the option **Bend centerline** to set the position of the jog.
5. Make sure that the jog goes backwards with the **Reverse Direction** command as shown in the illustration.
6. Set the **Jog Angle** to 45°.
7. Click on **OK**.



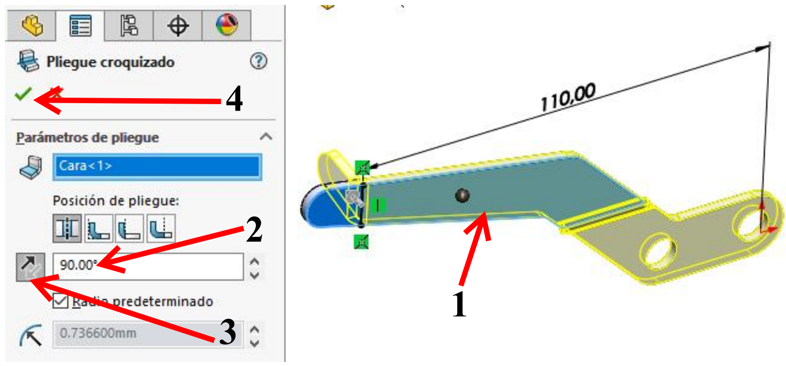
62 Next, we have to bend the upper end of the arm.

Select the plane as shown and make a sketch. Draw a vertical line and set the distance to **110 mm** from the origin.

63 Click on **Sketch Bend** in the **CommandManager**.



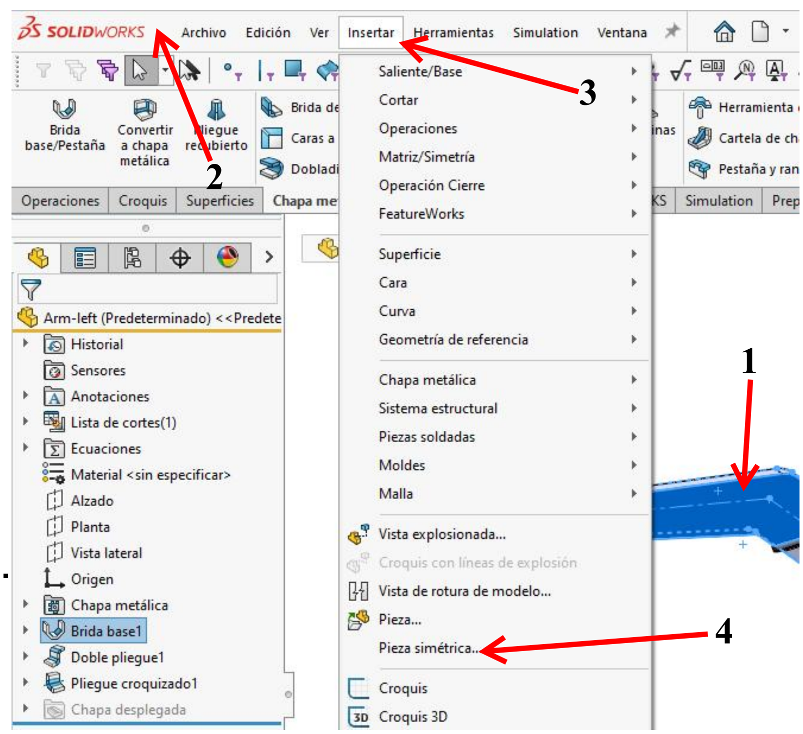
64 Set the following options:



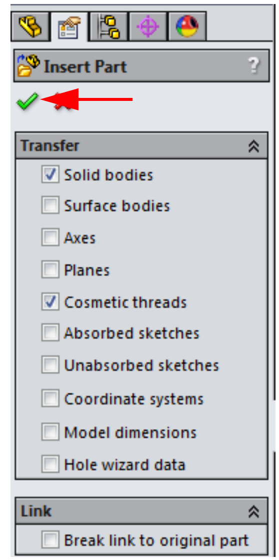
1. Again, you will have to indicate first which plane stays fixed. Click on the spot as indicated in the illustration.
2. Set the angle to **90°**.
3. Make sure that this part of the sheet metal is bending in the right direction with **Reverse Direction**. The arrow in model indicating the direction must point backwards.
4. Click on **OK**.

65 This model is now finished. Save it as: Arm-right.SLDPRT.

66 We need a mirrored copy from this part. This is very easy to create.



1. Select the plan as shown in the model. This is the ‘mirror’ for the mirror command (the mirror ‘axis’).
2. Open the pull-down menus.
3. Click on **Insert** in the pull-down menus.
4. Click on **Mirror Part...**



67 Click on **OK** in the **PropertyManager**.

68 A new file has opened containing the mirrored part.

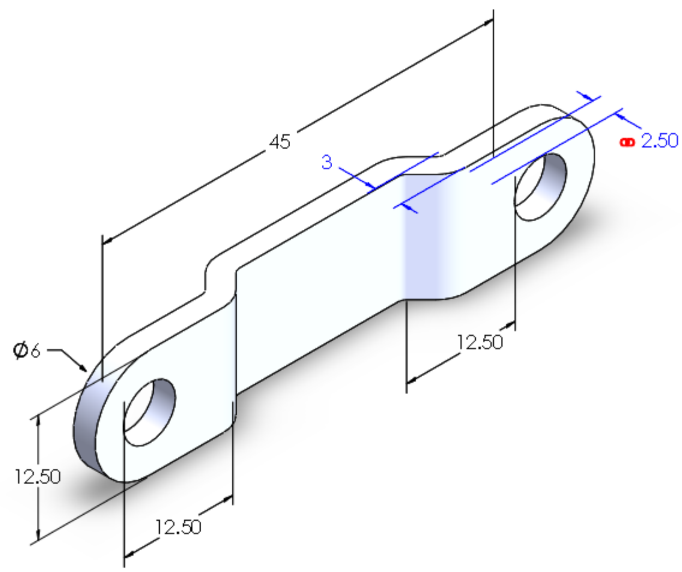
This part is constrained to the original part. If you change the original, the mirrored copy will also change.

Save this part as: Arm-left.SLDPRT.

Bracket

Work plan

The next part is a bracket. This is much simpler than the last part. How would you handle this? Make a plan!

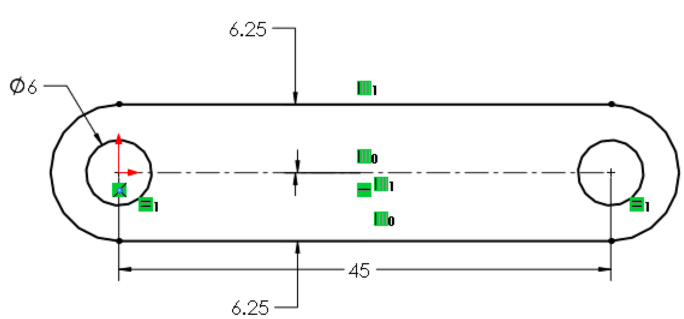


We will build this part in sheet metal too.

69 Open a part and set the units to MMGS. Make the sketch on the **Right Plane** as shown.

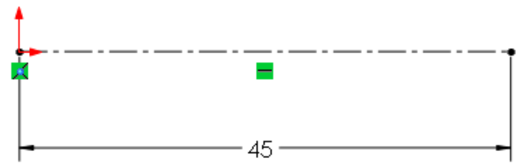
When done, continue to step 73.

If you have trouble, follow the next few steps.



70 Draw a **centerline** horizontally to the right from the origin.

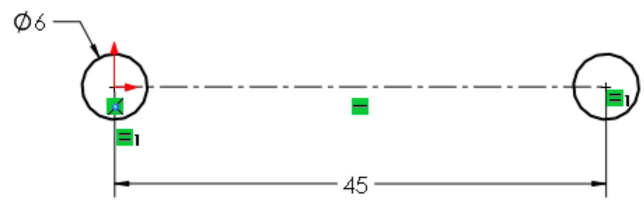
Set a dimension for the length: **45 mm**.



71 Draw two circles with the centers at both endpoints of the centerlines.

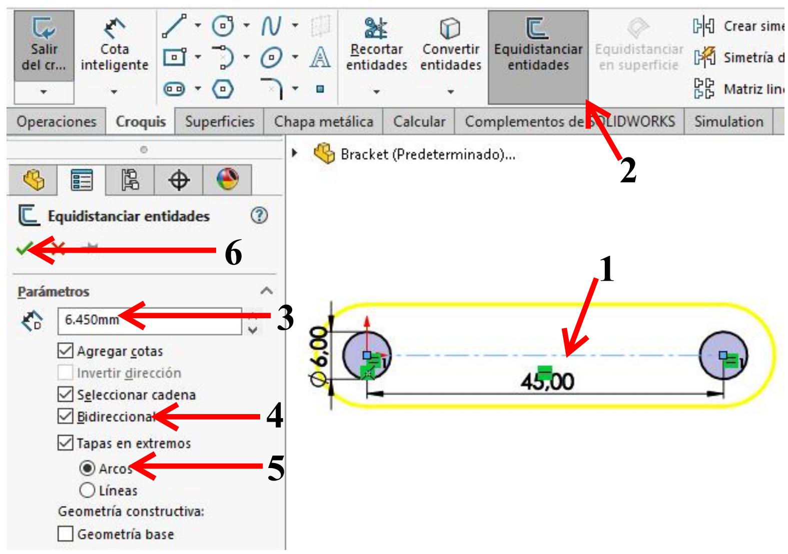
Set the dimension from one of the circles to **Ø6 mm**.

Select both circles and set an **Equal** relation.

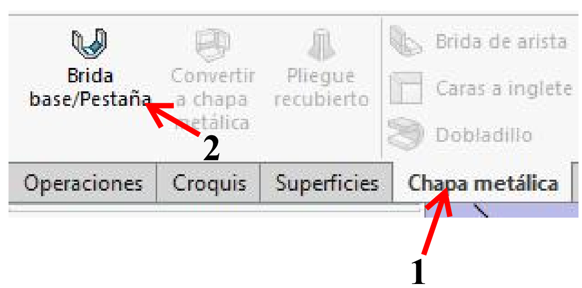


72 To offset sketch entities:

1. Select the centerline.
2. Click on **Offset Entities** in the **CommandManager**.
3. Set a distance of **6.25 mm** in the **PropertyManager**.
4. Check the option **Bi-directional**.
5. Check the option **Cap ends** and next check **Arcs**.
6. Click on **OK**.

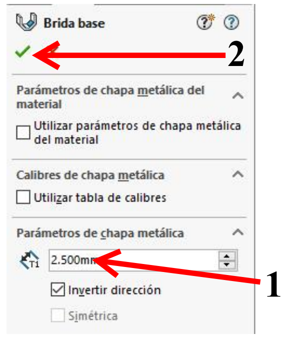


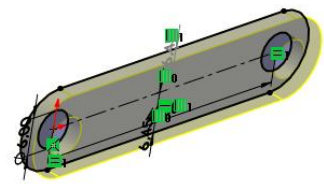
73 First, click on the **Sheet Metal** tab in the **CommandManager**, then on **Base Flange/Tab**.



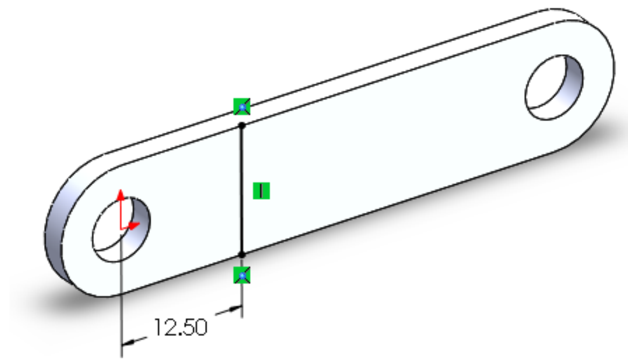
74 Set the thickness of the material to **2.5 mm** in the **PropertyManager**.

Click on **OK**.



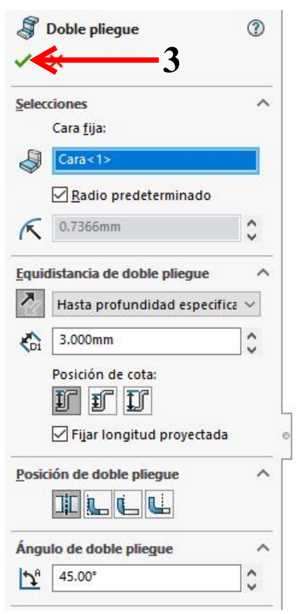


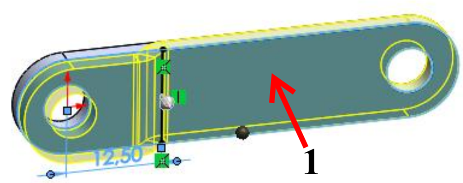
75 Make the sketch as shown. Draw a vertical **line** and set the dimension from that line to the center of the left hole to **12.5 mm**.



76 Click on **Jog** in the **CommandManager** and set the following features in the **PropertyManager**:

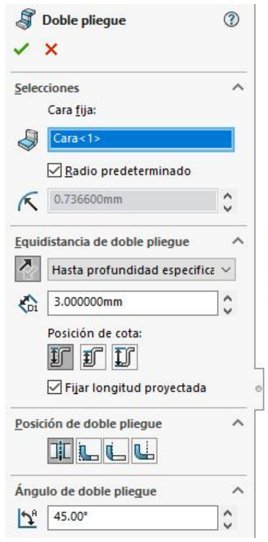
1. Click on the middle of the model to determine the fixed plane.
2. All the other settings will be the same as the last time you did this. So you do not have to change them. Check the settings with the illustration.
3. Click on **OK**.

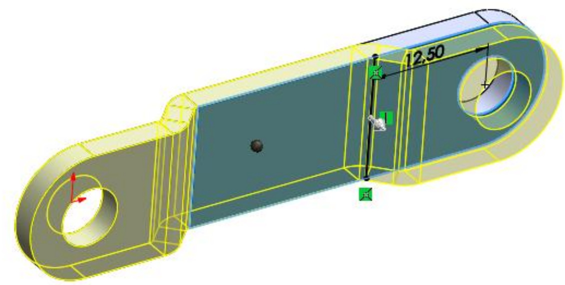




77 Make a second **Jog** at the other end of the bracket. Do exactly the same as you did in the last two steps, only now set the vertical line **12.5 mm** from the right hole.

78 Save the file as: Link.SLDPRT.

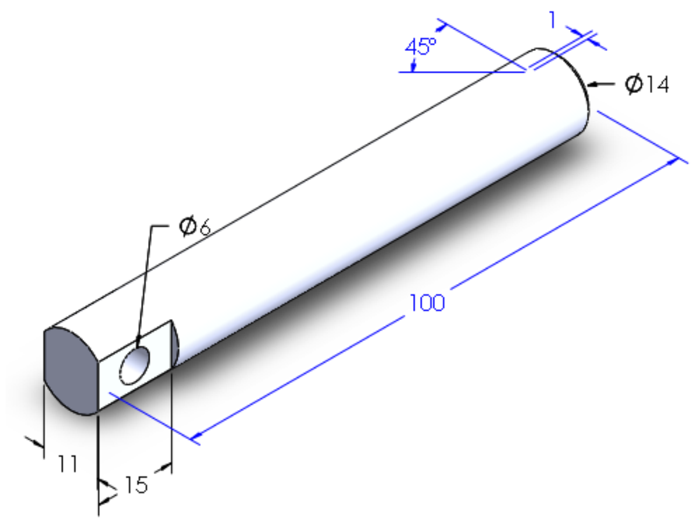




Rod

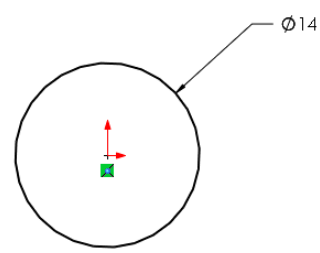
Work plan

We will make the pin now. This is a simple part that you can probably make by yourself without any problem. We only provide the main steps.

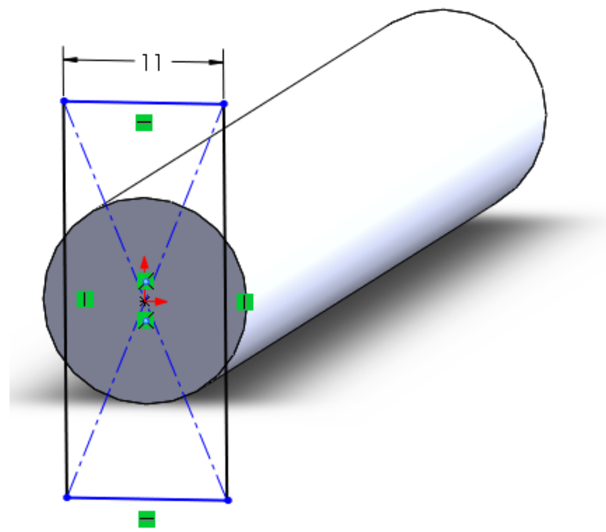


79 Open a new part and set the units to MMGS. Make the sketch on the **Front Plane** as shown. It consists only of one **circle**.

Extrude this circle to a length of **100 mm**.

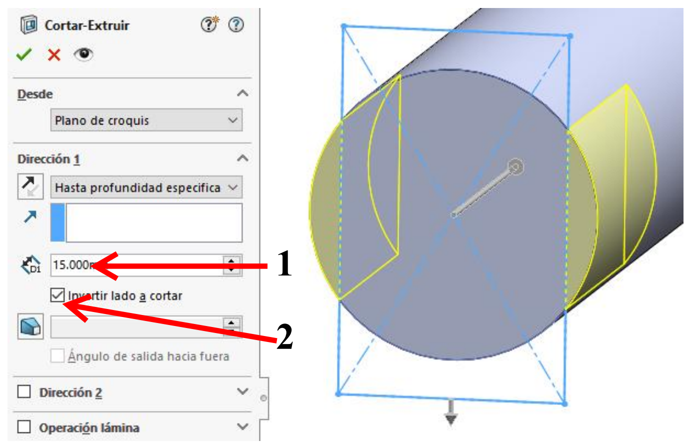


80 Make a sketch as shown. Use the center rectangle tool to make sure that the rectangle is exactly in the middle of the circle. The height of the rectangle does not matter.



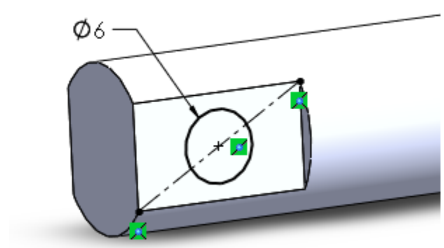
81 Make an **Extruded Cut** from this sketch.

1. The depth is **15 mm**.
2. Check the option **Flip side to cut** to make sure that the material on the **outside** of the rectangle will be removed and not on the inside, like we would do with a normal **Extruded Cut**.



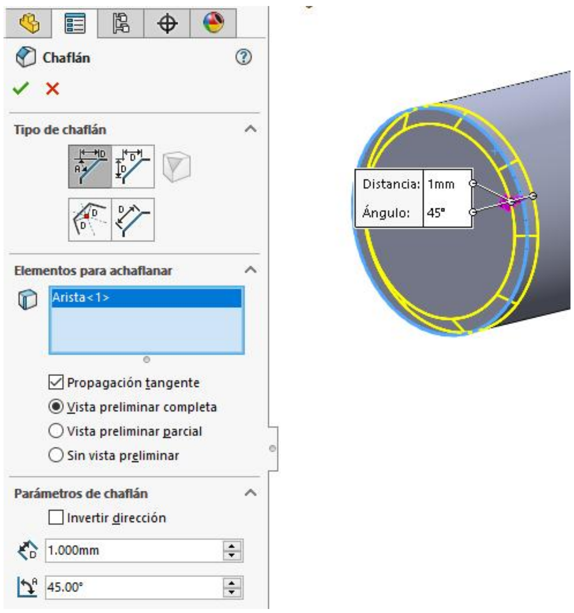
82 Make the sketch as shown. Draw the diagonal **centerline**. Next draw a **circle** on the midpoint of the centerline.

Make an **Extruded Cut** with a depth set to **Through All** for this sketch.



83 Finally, chamfer the end of the pin by **1 mm x 45°** using the **Chamfer** feature.

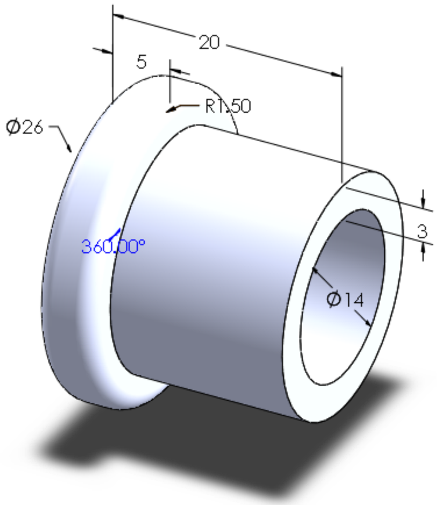
84 Save the file as Rod.SLDPRT.



Socket

Work plan

The next part is the cap. It only consists of one feature: a Revolved Boss.

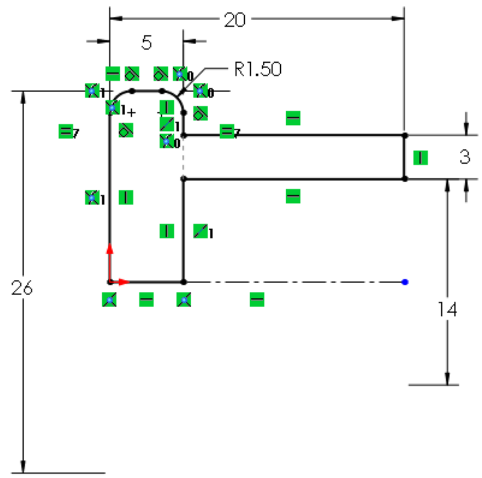


85 Open a new part and set the units to MMGS. Make the sketch on the **Front Plane** as shown.

Make the sketch complete without any fillets. Only when the sketch is done use the **Sketch Fillet** command.

Make a **Revolved Boss** over **360°** from this sketch.

86 Save the file as Socket.SLDPRT.

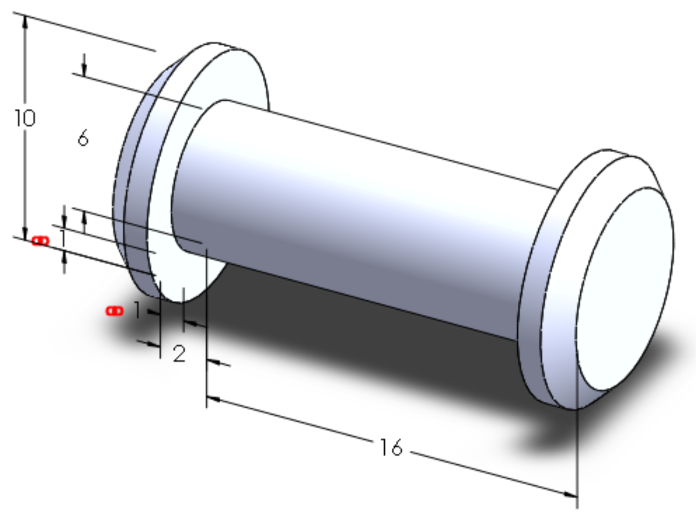


Rivet

Work plan

Finally, we have to build a rivet. This is also a part made from only one **Revolved Boss** feature.

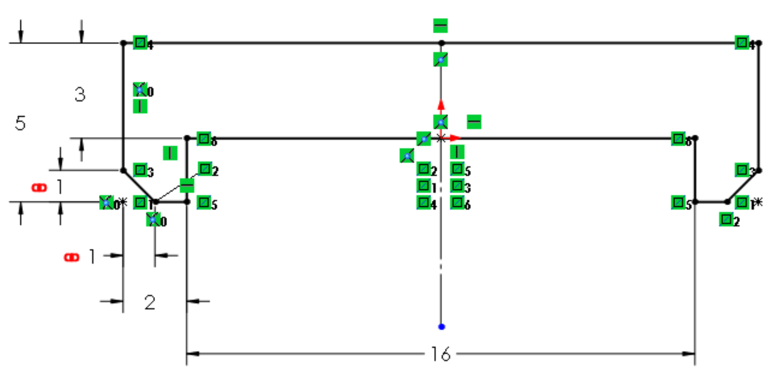
We need two lengths of rivets though: **16 mm** and **11 mm**. That is why we will make two configurations of this part.



87 Open a new part and set the units to MMGS. Make the sketch as shown on the **Front Plane**.

You can of course draw half of the sketch first and mirror it around the centerline.

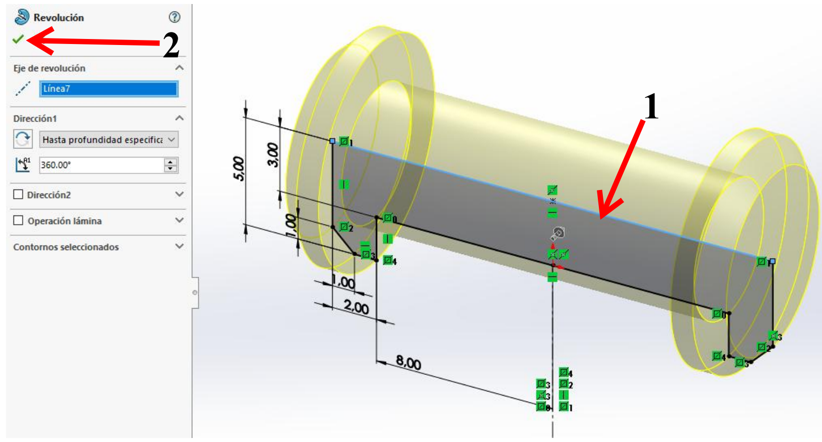
The sloped edges should be done with the **Sketch Chamfer** command.



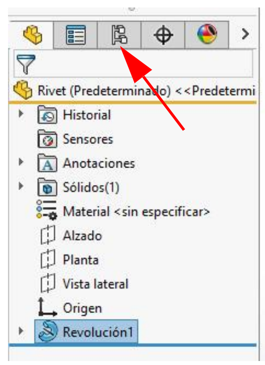
88 Select the upper horizontal line in the sketch. This will be our rotation axis.

Click on **Revolved Boss/Base**.

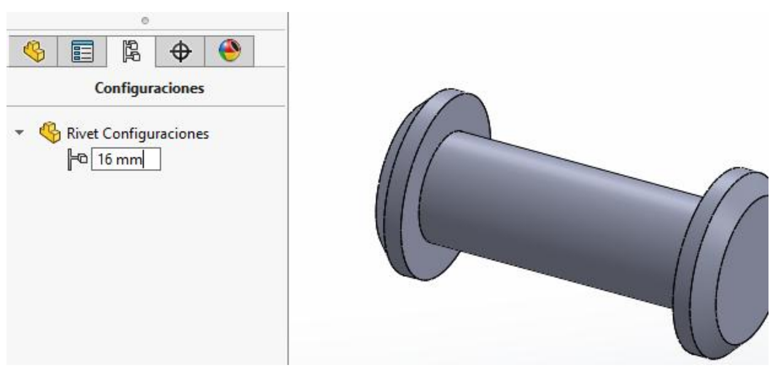
Click on **OK** in the **PropertyManager** to make the rotation.



89 Go to the **ConfigurationManager**.

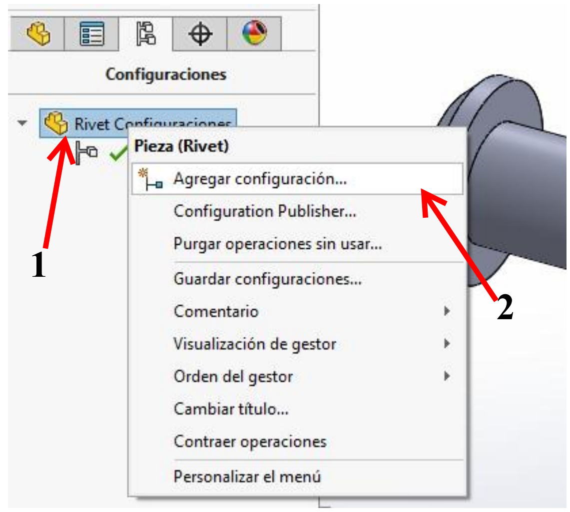


90 Change the name of the current configuration from **Default** to **16mm**.



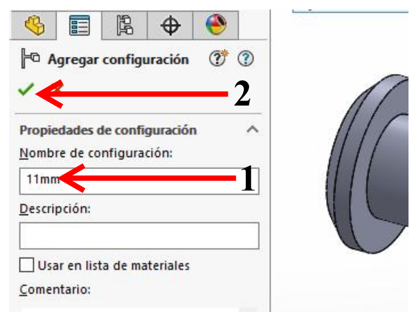
91 Add a new configuration.

1. Right-click on the upper line.
2. Click on **Add Configuration...**



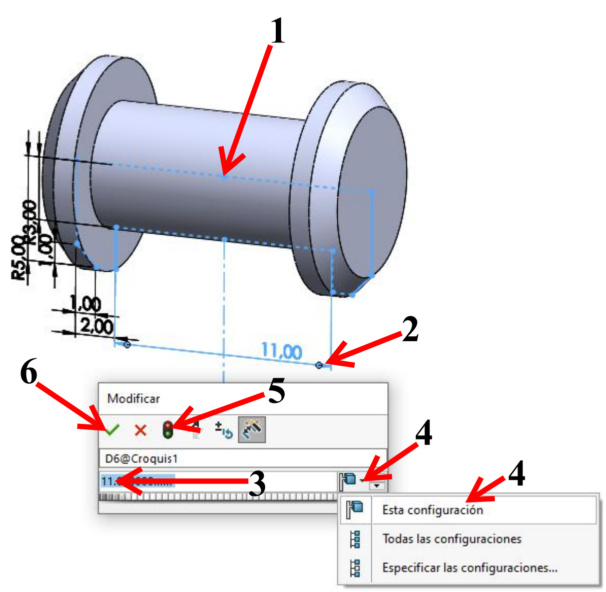
92 Name the new configuration **11mm**.

Click on **OK**.



93 Change the length of the rivet.

1. Double-click on the model. The dimensions appear.
2. Double-click on the dimension **16 mm**. The **Modify** menu appears.
3. Change the size to **11 mm**.
4. Select **This configuration**. The changed value will only be altered in the active configuration now and not in the other one.
5. Click on **Rebuild** to activate the changes.
6. Click on **OK**.



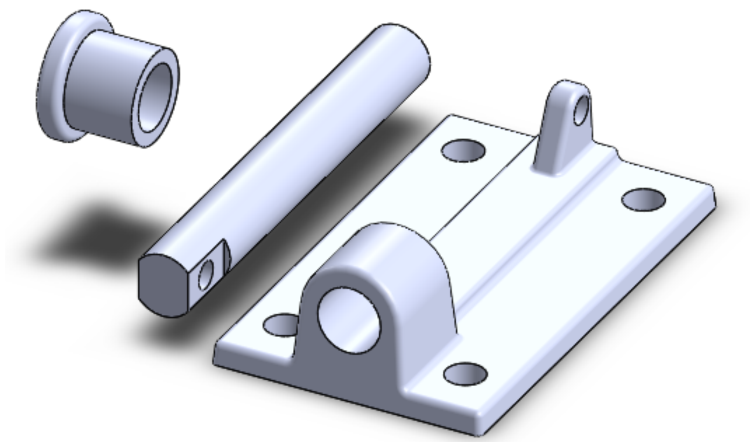
94 This part is ready too. Save it as Rivet.SLDPRT.

Clamp Assembly

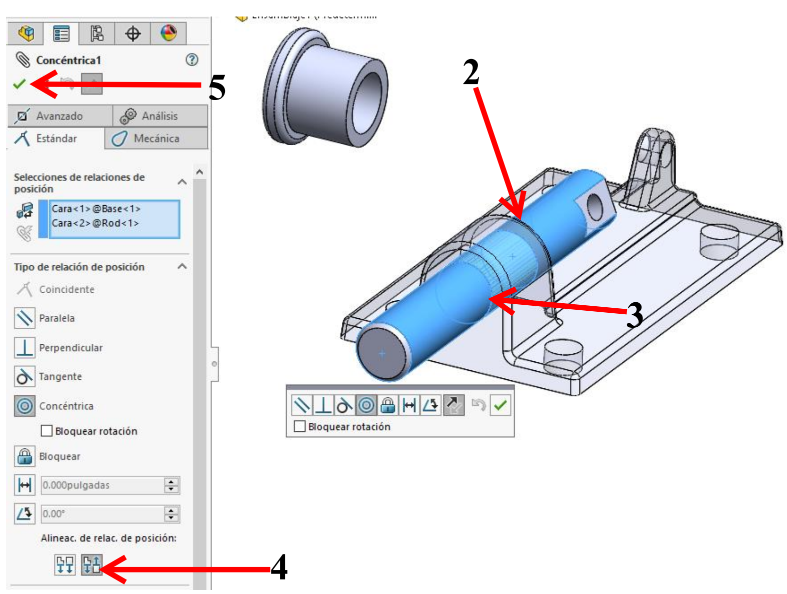
95 All parts of the clamp are now ready, so we can start building the assembly. Try it yourself first. If you need help, follow the steps below.

Open a new assembly and set the units to MMGS.

96 Place the base in the assembly, next the pin and the cap. You can place the components in random positions on the screen.



97 To mate the components:



1 Click on **Mate** in the **CommandManager**.

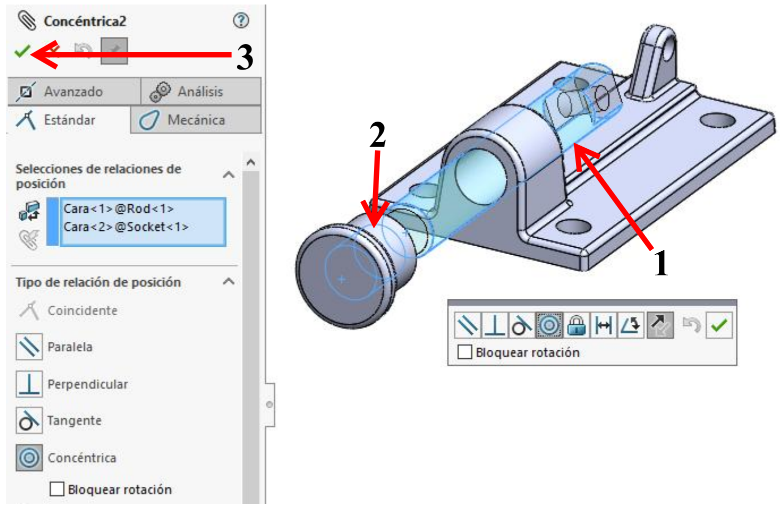
2,3 Select the two surfaces from the pin and the base as illustrated on the right.

4 Because the pin is in the wrong direction, you must click either **Aligned** or **Anti-Aligned** in the **CommandManager**. The pin is reversed now.

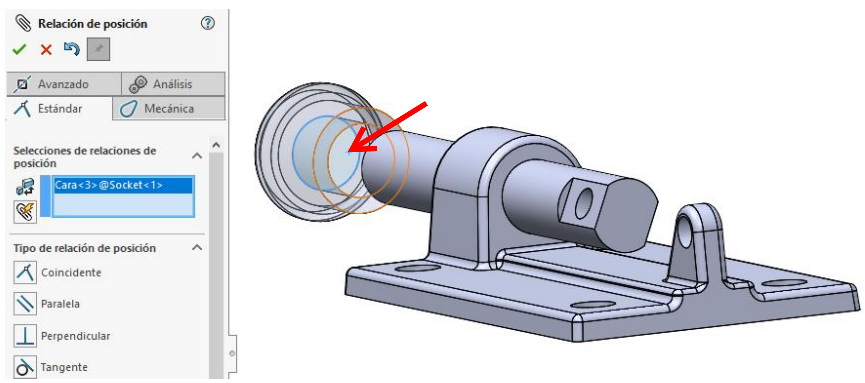
5 Click on **OK**.

98 Select the two surfaces as shown.

Click on **OK**.

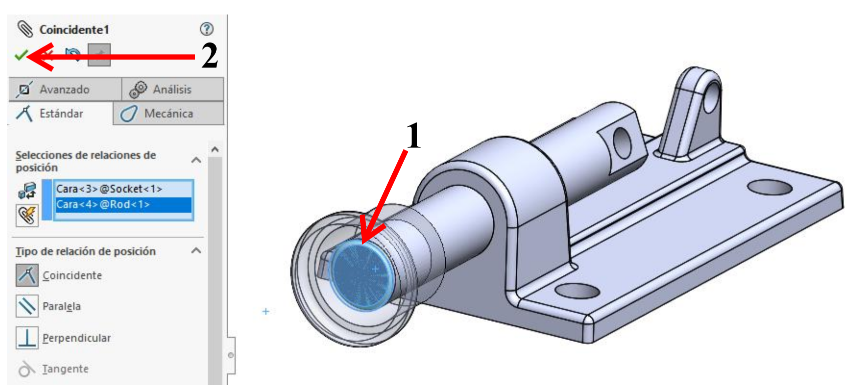


99 Select the face on the inside of the cap as shown.

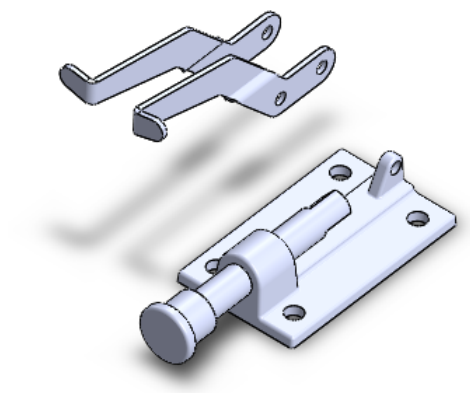


100 Rotate the model and select the face from the rod as shown.

Click twice on **OK** to end the **Mate** command.



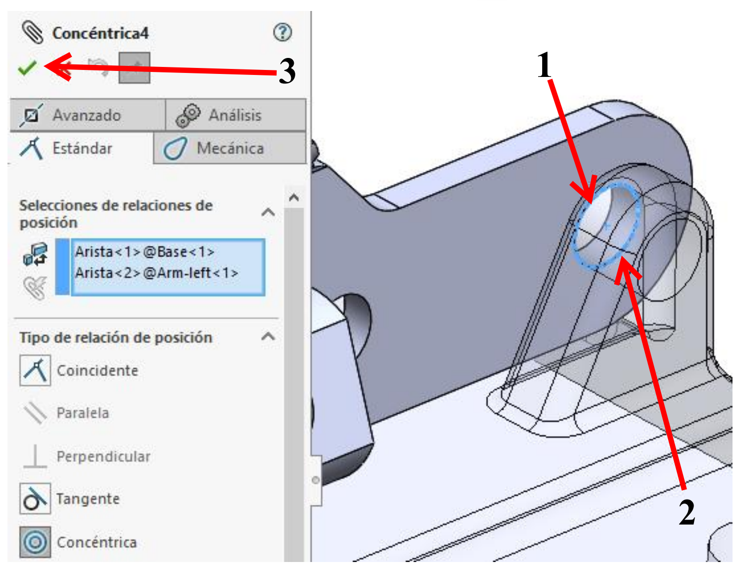
101 Use **Insert Component** to put the two arms in the assembly.



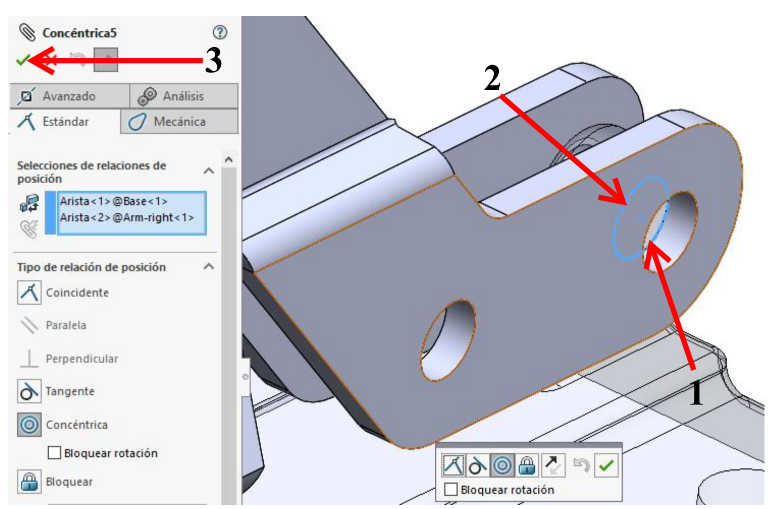
102 Click on **Mate** in the **CommandManager** again.

Select the two edges as shown.

Click on **OK**.

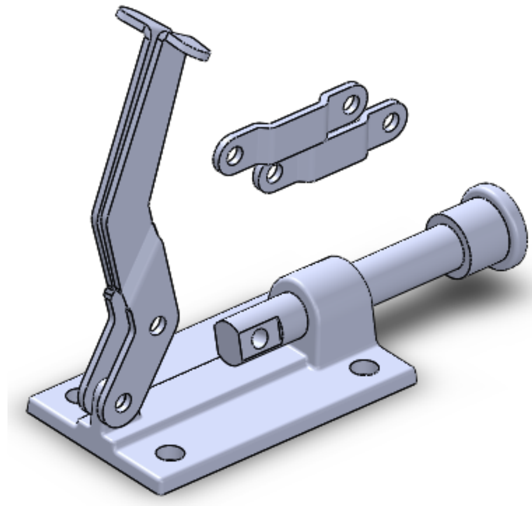


103 Rotate the model and do the same again for the other arm.



104 Try to drag the parts around the screen now. You will notice that you can only move the pin and cap left and right and rotate the arms. These movements are determined by the mates you have added.

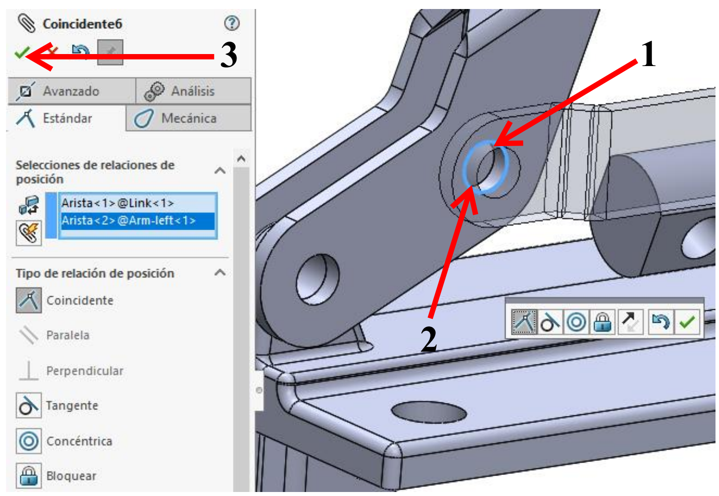
Add two brackets to the assembly.



105 Start the **Mate** command again and make a **Coincident** mate (not a Concentric mate!).

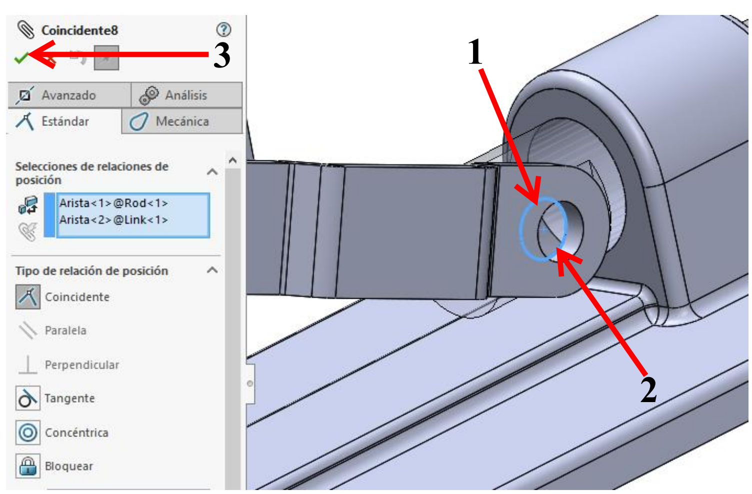
Select the two edges as shown on the right.

Click on **OK**.



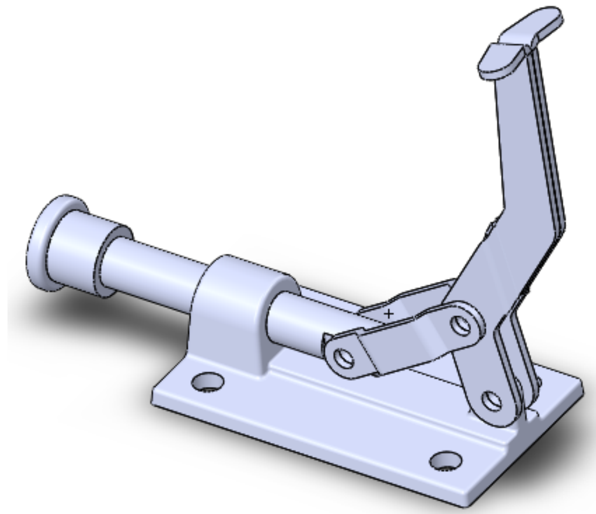
106 Select the two edges as shown.

Click on **OK**.



107 Set the other bracket as well.

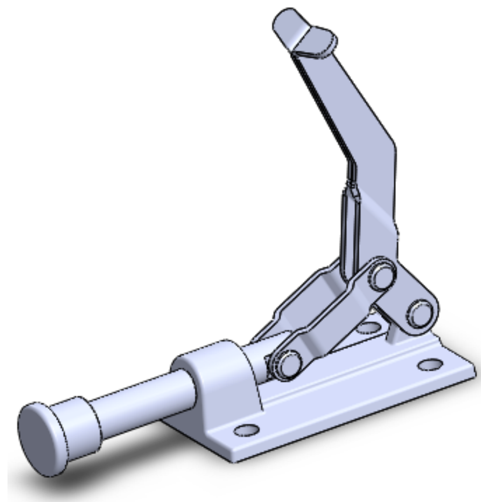
Use the option **Anti-Aligned** to reverse the bracket.



108 You can move the arm now and you will see the clamp functioning.

To finish the model, you need to add the rivets. You will need one rivet of **11mm** and two rivets of **16mm**.

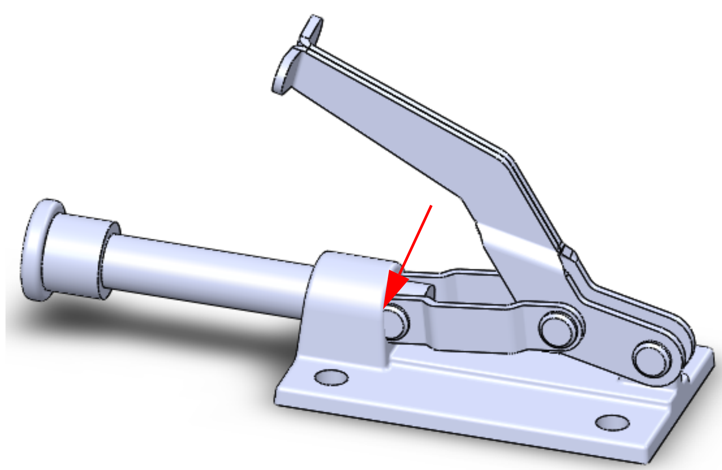
109 The assembly is ready now. Save the file as Clamp.SLDASM.



Checking the model.

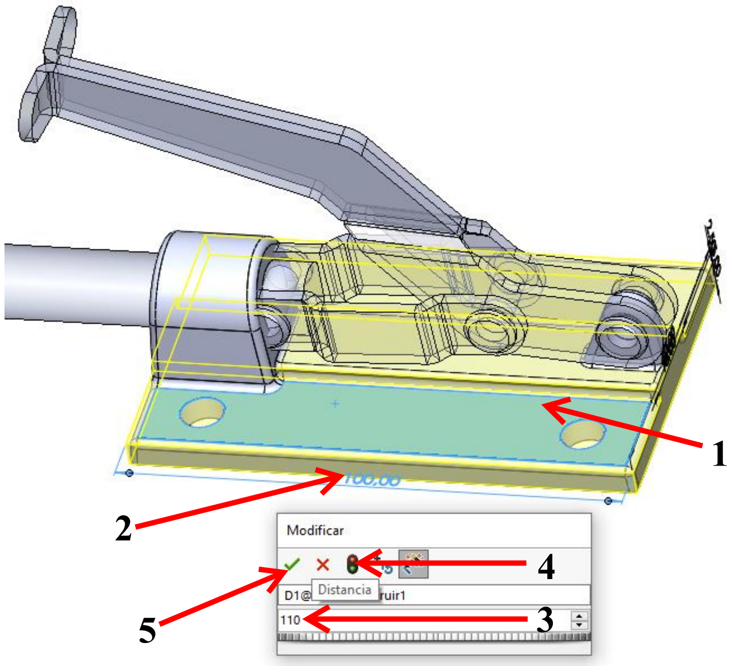
When you move the arm of the clamp you will notice that the brackets collide with the base.

To solve this problem, we need to extend the base a bit.



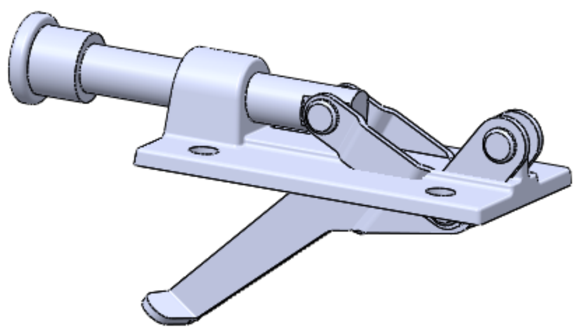
110 The easiest way to extend the size of the base is to do the following:

1. Double-click on the base. The dimensions appear.
2. Find the length (100) and double-click on this. The **Modify** menu appears.
3. Change the size to **110 mm**.
4. Click on **Rebuild** and check to see if the change is correct.
5. Click on **OK**.



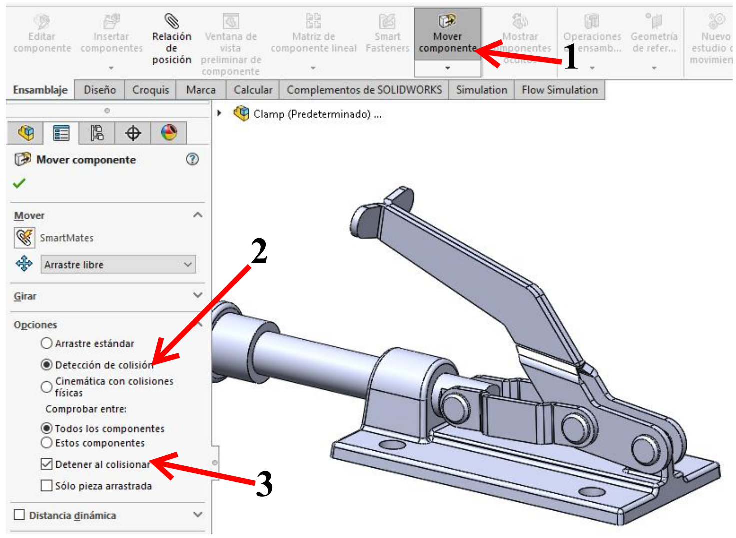
Checking the model

The arm for the pin can rotate 360 degrees and in the software, the arm goes right through the material of the base. This is not possible in the real world, so we want to limit the rotation of the arm.

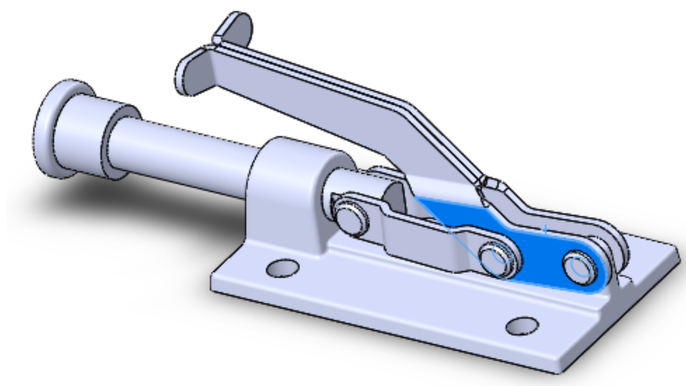


111 To find the most extreme positions, we will follow the next few steps:

1. Make sure the arm is pointing upward.
2. Click on **Move Components** in the **CommandManager**.
3. Select the option **Collision Detection** in the **PropertyManager**.
4. Check the function **Stop at collision**.



112 Move the arm again. Notice that the movement is limited to the position where two parts collide. At that point, the colliding parts turn blue.



Model Rendering

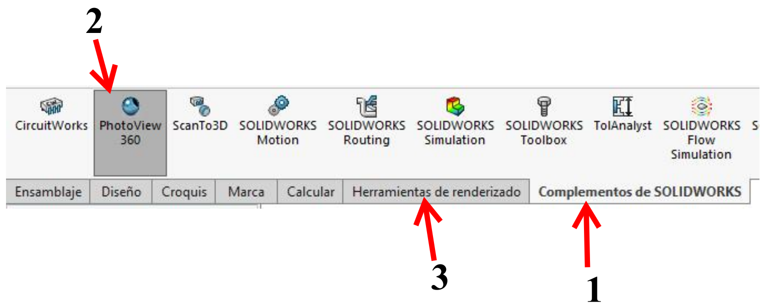
Work plan

Finally, we will make a rendering from this model. A rendering is a picture of the model with all features displayed as realistically as possible. You can use a rendering for many communication purposes, such as in a presentation.

To make a rendering in SolidWorks, we use an add-in called PhotoView 360. This is a very robust program with a wide range of capabilities. We will show you how to make a standard rendering using the default settings.

113 Check to see if **PhotoView 360** is activated.

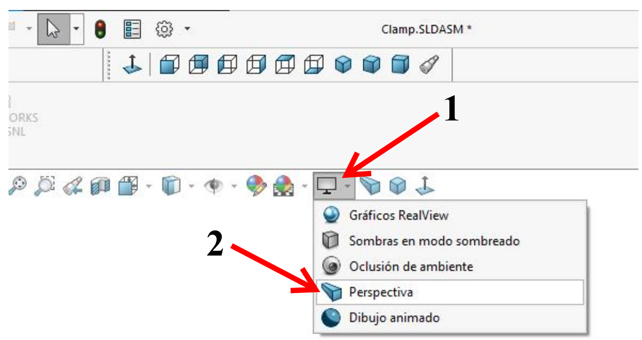
1. Click on the tab **Office Products** in the **CommandManager**.
2. Click on the **Photoview 360** button if not already selected.
3. Select the tab **Render Tools** in the **CommandManager**.



The buttons and functions of **Photoview 360** are now displayed in the **CommandManager**.

114 Put the model in a perspective view. This will give a more natural look than an isometric or diametric view.

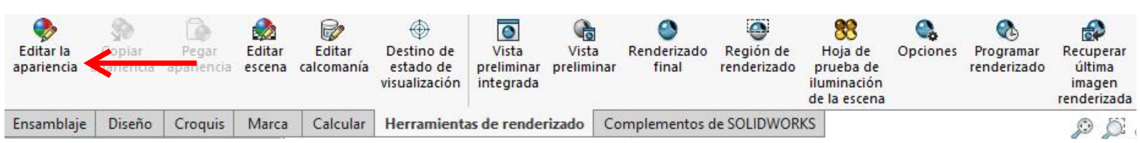
1. Click on **View Settings**.
2. Click on **Perspective**.



Rotate the model to establish the view that you want to show in the rendering.

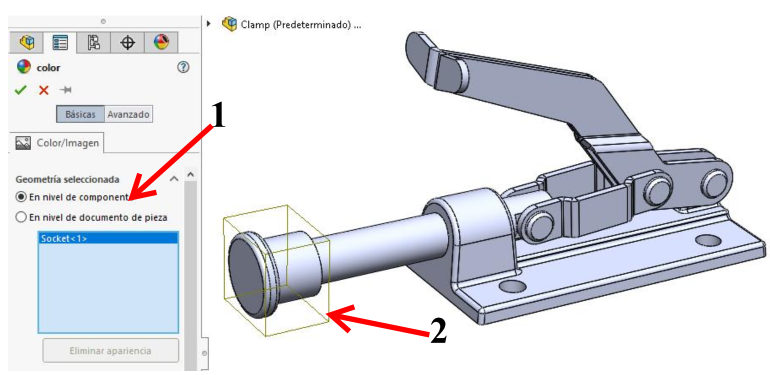
115 We will determine the kind of material for the different parts.

Click on **Edit Appearance** in the **CommandManager**.



116 Check the option **Apply at component level** in the **PropertyManager**.

Click on the cap in the model.



117 Apply an appearance to the cap.

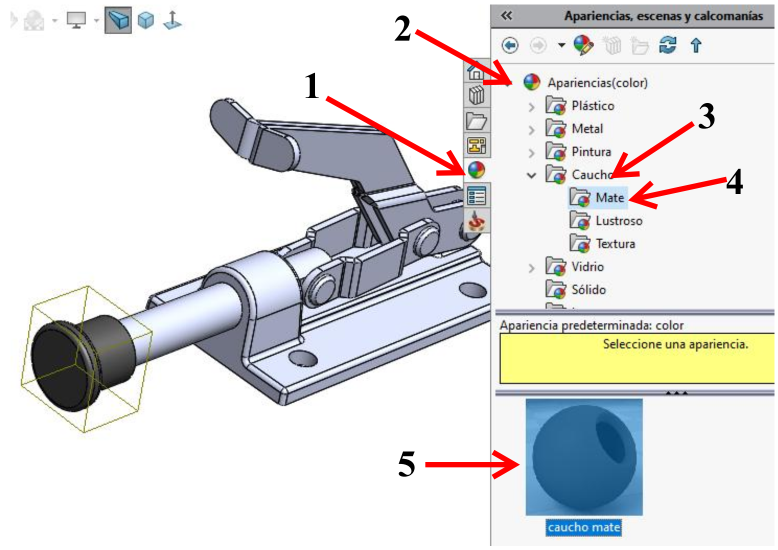
1 Click on the **Appearances, Scenes, and Details** in the **Task Pane** (on the right side of your screen).

2 Click on **Appearances**.

3 Click on **Rubber**.

4 Click on **Matte**.

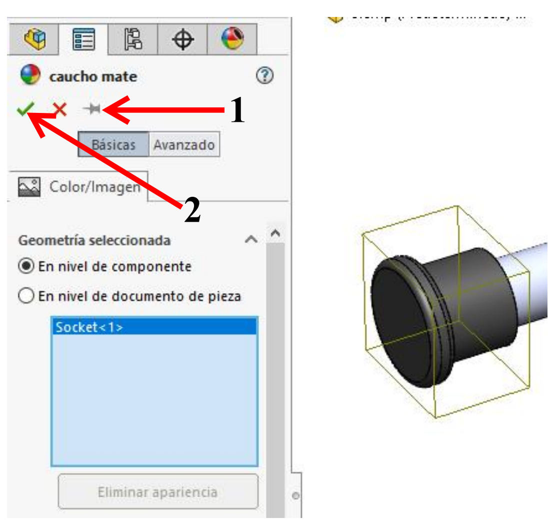
5 You will only find one kind of material in this category. Select it.



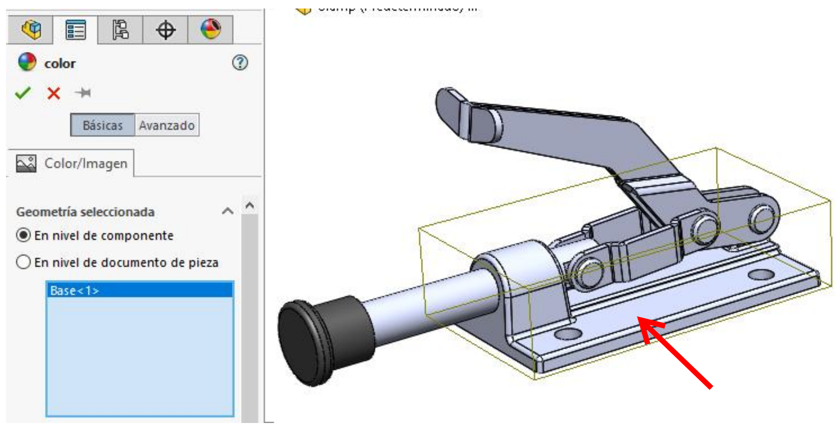
The cap now has the appearance of **matte rubber**.

118 Click on the pushpin in the **PropertyManager**. The **PropertyManager** will remain visible even after you have clicked **OK**. This will come in handy when you are applying appearances to several parts.

Click on **OK**.



119 Select the base in the model.

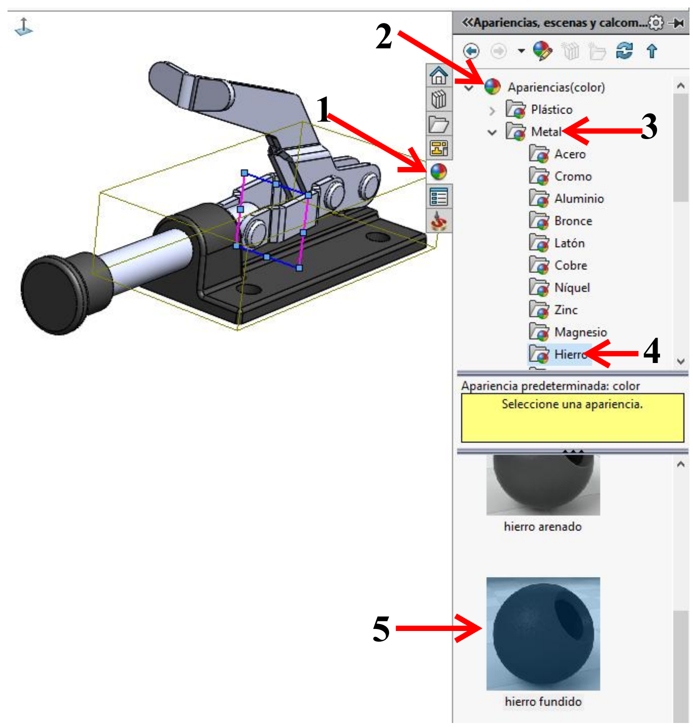


120 Select **Cast Iron**.

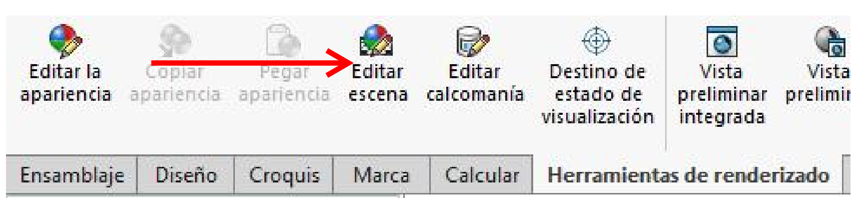
Click on **OK** in the **PropertyManager**.

121 You can do the same with all of the other parts yourself. You can also determine colors for the different parts.

Try this or keep the default settings.



122 Now that we have determined the appearances, we can set the scene around the product. The scene is the environment, the background, and/or the lighting. SolidWorks has a number of standard scenes.



Click on **Edit Scene** in the **CommandManager**.

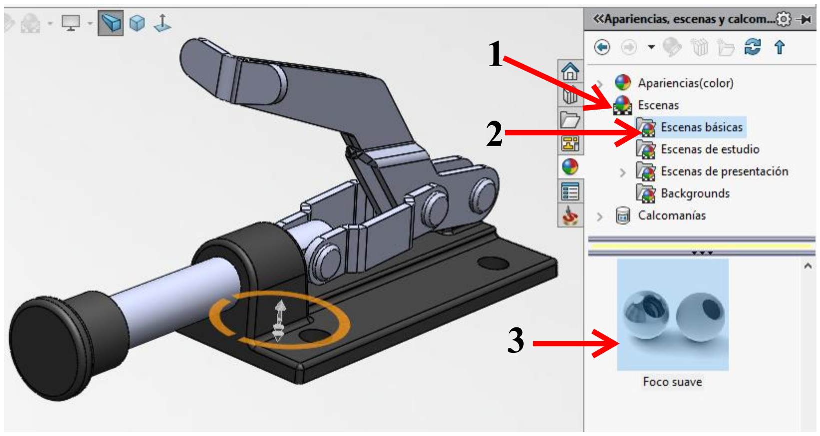
123 In the **Appearances, Scenes, and Details** task pane:

1 Click on **Scenes**.

2 Click on **Basic Scenes**.

3 Select the scene **Soft Spotlight**.

4 Click **OK** in the **PropertyManager**.

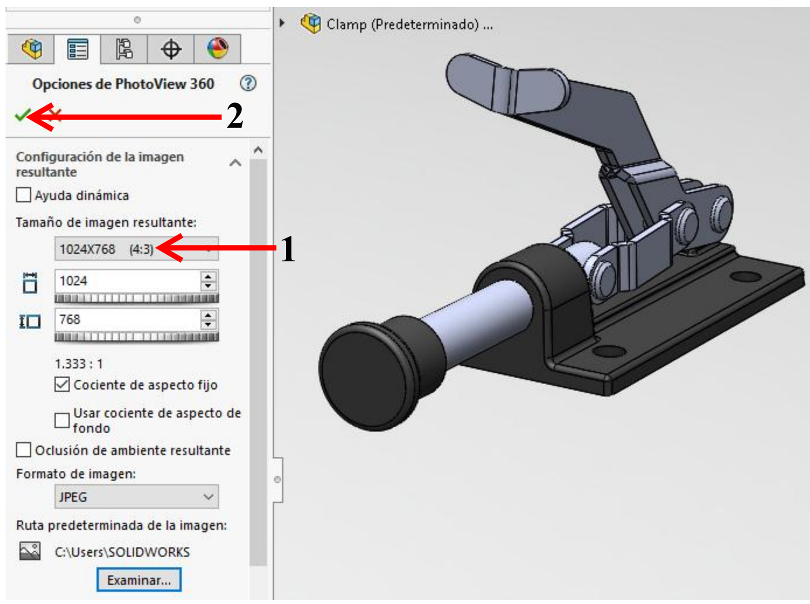


124 Select **Options** in the **CommandManager**.

Change the **Output image size** to **1024x768** (or a different size of your choosing).

Other settings such as the final render quality can be set here as well.

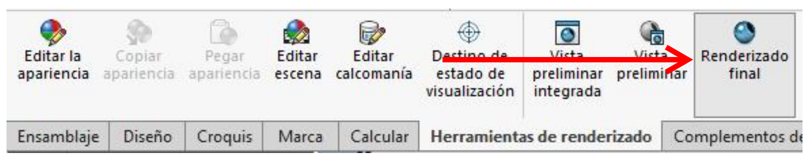
Click on **OK**.



125 Click on **Preview Window** in the **CommandManager**.

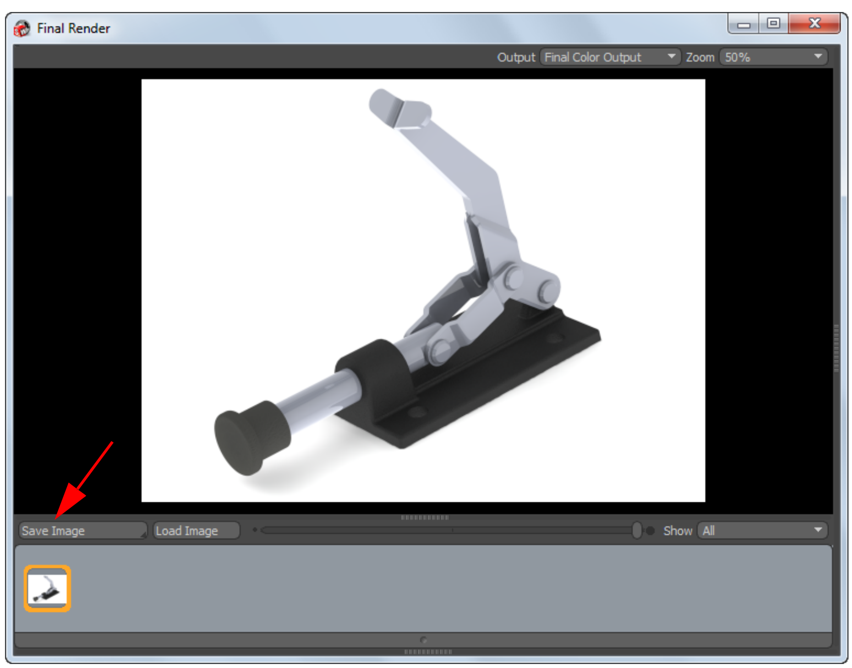
This provides a preview of what the final image will look like. Move and zoom in on the model in order to position it within the window.

126 When the image looks OK, click on **Final Render** in the **CommandManager**.



127 When the rendering is complete, the image can be saved by clicking **Save Image**. Save the image to a location of your choosing.

128 Try changing the scene, appearances or position of the model and create other rendered images.



Tip: Make sure to use the **Preview Window** before making the final rendering. This will save you a lot of time when making changes.

Tip: What you have just seen in PhotoView 360 is only the beginning of what you can do with this application. You can change whatever you like: the background, the appearances, the lighting, and so on. These steps are not included in this tutorial, but if you are interested, try them yourself.

What are the main features you have learned in this tutorial?

In this tutorial you have learned a lot of new tools.

- ❑ You have used **Jog** in the sheet metal features.
- ❑ You have used the **Draft** feature to slope faces in the model.
- ❑ You have seen how to limit the movement in an assembly.
- ❑ You have used PhotoView 360.
- ❑ The most important thing you have gained, however, is the practice the tutorial has provided in modeling and, even more importantly, making sketches.

This is the last tutorial in this series. When you have completed all twelve exercises and have done some additional practice, you should be able to work with SolidWorks quite well now.

To get even better, all you need to do is practice, practice, and practice some more!

Not all of the features in SolidWorks were presented in these tutorials. That would be virtually impossible, given the vast possibilities and features in the software.

You are now a SolidWorks ‘user’ and that means you can try and build something on your own. You will learn a lot from doing this. If you do not succeed with one or more functions, use the help function. It will help you to get on with your work.

Do not be afraid to try things yourself and keep on practicing. You will soon be able to call yourself a SolidWorks expert!