SolidWorks Tutorial

Bearing Puller

Preparatory Vocational Training
and Advanced Vocational Training

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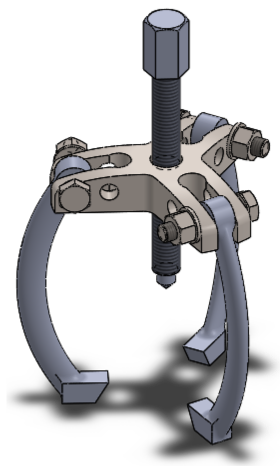
Document Number:

Tutorial: Bearing Puller

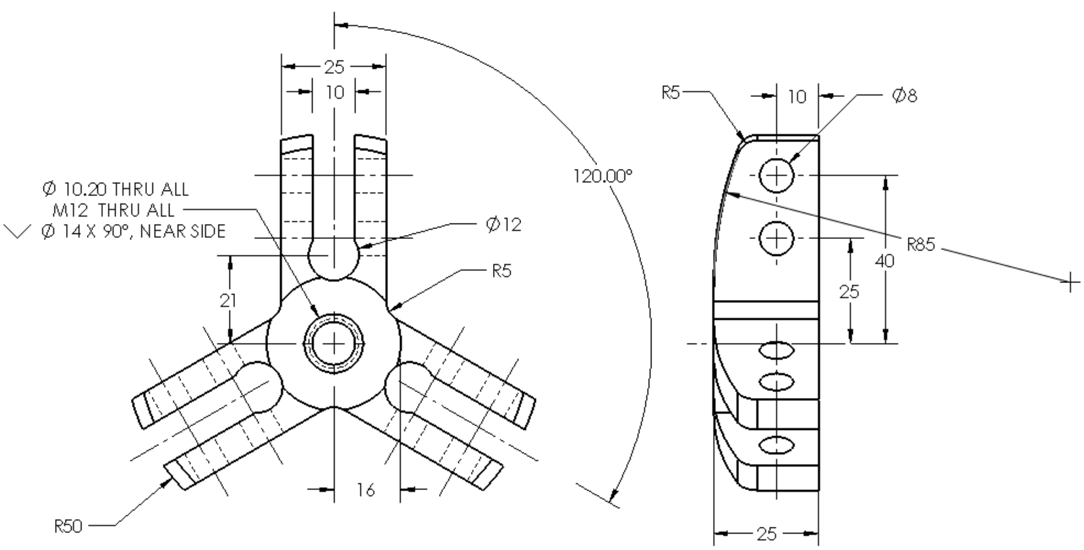
In this tutorial, we will build a bearing puller. This product consists of three parts. We will learn a few new functions in this tutorial. We will also perform a simple analysis on some of the parts.



Main Bridge

Work plan

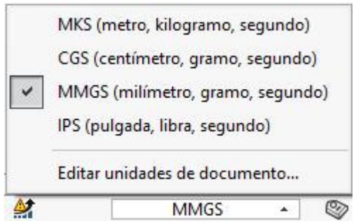
The first part we will make is the main bridge. We will make this according to the drawing below.



Make a plan! How would you handle this part? Make a plan for yourself and compare it with the plan we have developed for this tutorial.

1 Start SolidWorks and open a new part.

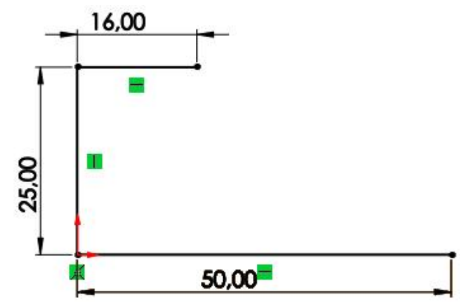
2 Set the units for the part as MMGS at the bottom right of the SolidWorks screen.



3 Select the **Front Plane** and make a **sketch** like in the illustration on the right.

The **sketch** consists of four lines and three dimensions.

Make sure the left bottom corner of the sketch is at the **origin**.



4 Create an arc:

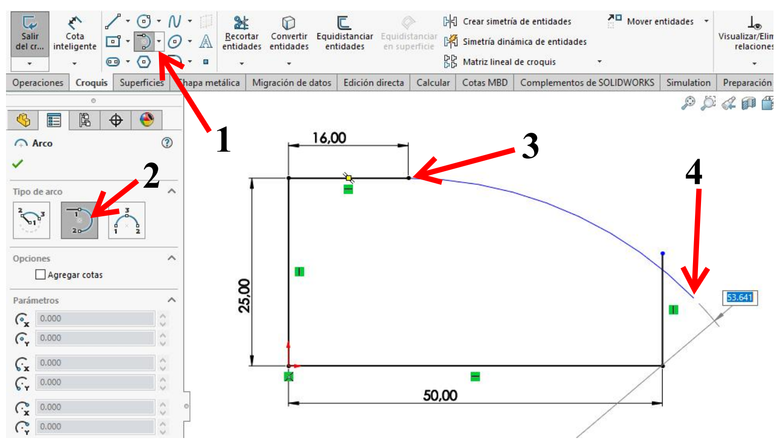
1 Click on **Arc** in the **CommandManager**.

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

3 Click on the right end of the upper horizontal line.

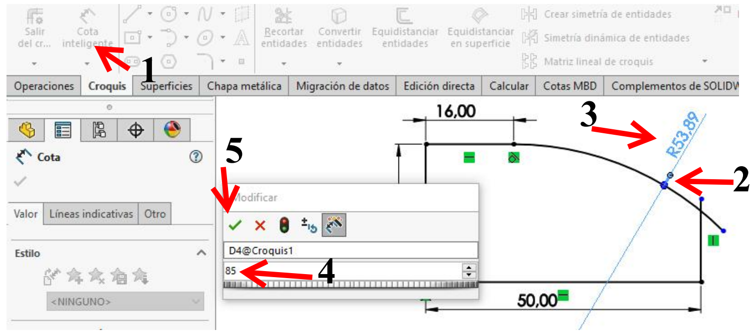
4 Put the end of the arc at about the same location as in the drawing. The exact spot is not relevant at this point.

5 Push the <Esc> key to end the line command.



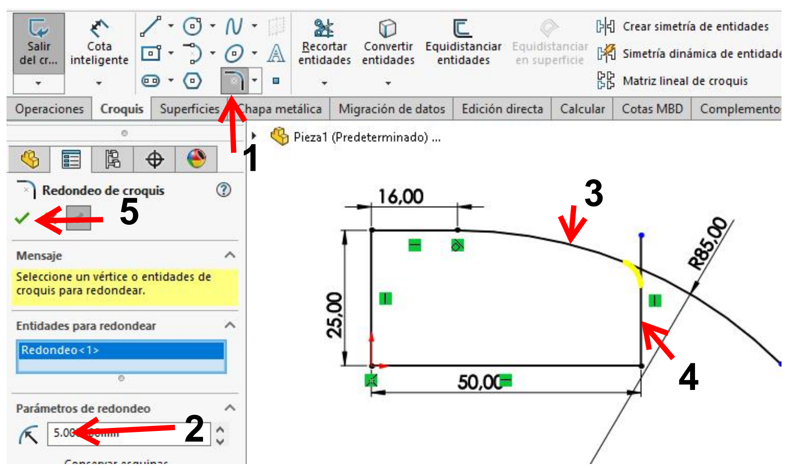
5 Set the dimension for the arc you have just drawn:

1. Click on **Smart Dimension** in the **CommandManager**.
2. Click on the **arc**.
3. Set the dimension.
4. Change the radius of the arc to **85**.
5. Click on **OK**.



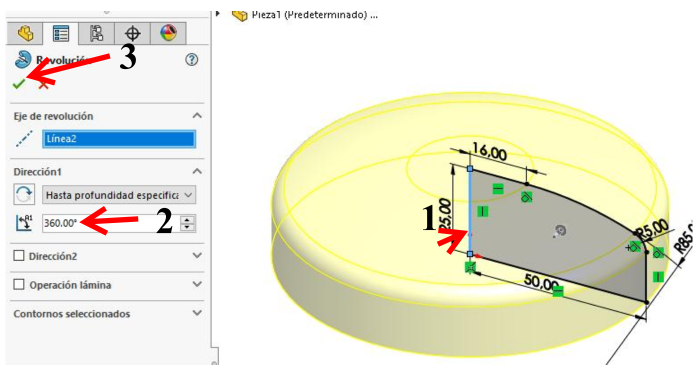
6 Make a curved edge between the arc and the vertical line.

1. Click on **Sketch Fillet** in the **CommandManager**.
2. Change the radius to 5 mm in the **PropertyManager**.
3. Click on the arc, to the left of the vertical line.
4. Click on the vertical line, just below the arc.
5. Click on **OK**.



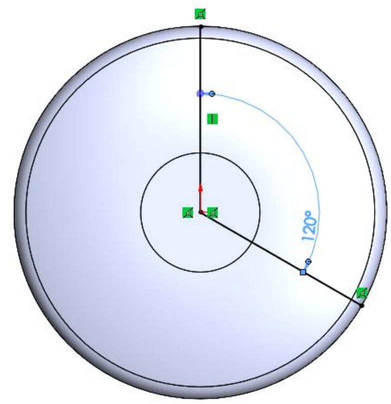
7 Click on the **Features** tab in the **CommandManager** and next on **Revolved Boss/Base**.

8 Next, you have to set the rotation axis:



1. Click on the left vertical line in the sketch.
2. Make sure the rotation angle in the **PropertyManager** is set to **360 degrees** (a complete circle).
3. Click on **OK**.

9 The basic form is ready. We will now remove three triangles from this body.

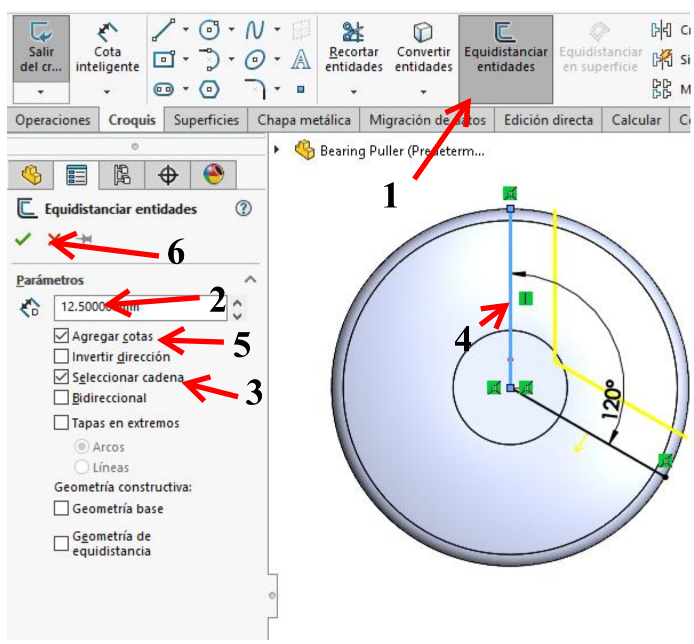


Select the Top Plane and create a sketch like in the illustration on the right.

The sketch consists of two lines emanating from the origin: one line goes straight up and the other runs downwards at an angle of 120 degrees to the first line. Both lines cross the outside edge of the part.

Set the dimension of 120 degrees between the two lines.

10 Make a parallel copy of both lines.



1. Click on **Offset Entities** in the **CommandManager**.
2. Change the distance in the **PropertyManager** to 12.5 mm.
3. Make sure the option **Select chain** is selected.
4. Click on one of the two lines in the sketch.

You can now see a preview. Both lines from the sketch are copied.

5. If the lines are copied in the wrong direction, click on **Reverse** in the **PropertyManager**.
6. Click on **OK**.

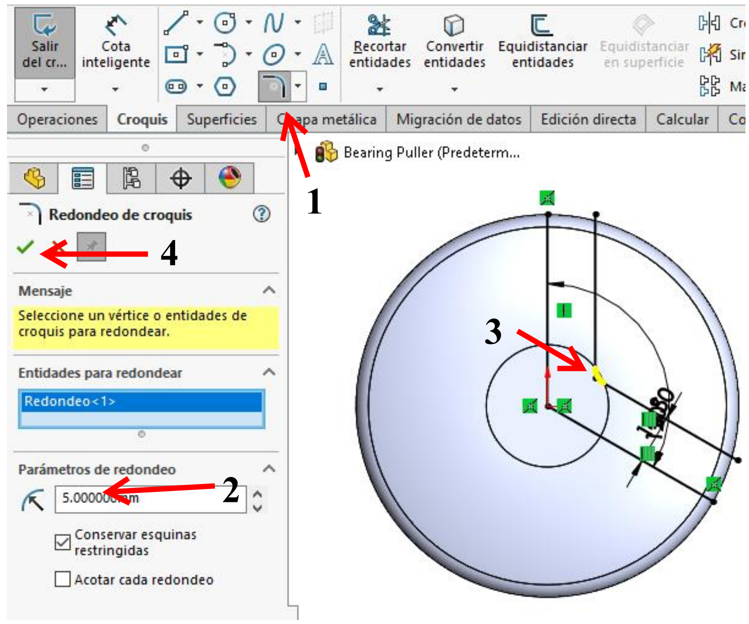
11 Round off the corners between the two lines.

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

2 Check to make sure that the radius is still 5 mm (you set this in step **7** already, and it should have remained in SolidWorks.

3 Click on the corner of both copied lines.

4 Click on **OK**.



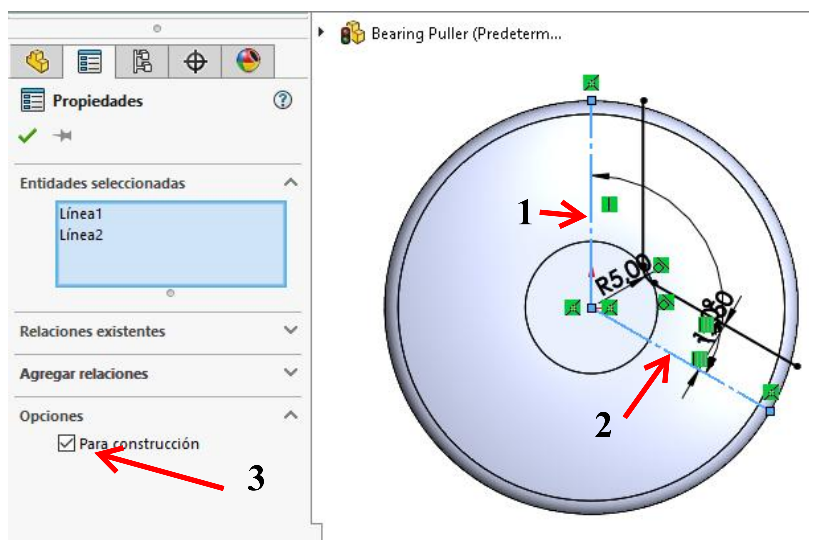
12 Next, we will make construction lines from the first two lines we have drawn.

1 Select the first line.

2 Hold the <Ctrl> key on your keyboard and select the second line.

3 Check the option **For construction** in the **PropertyManager**.

The two line will now be displayed at centerlines.



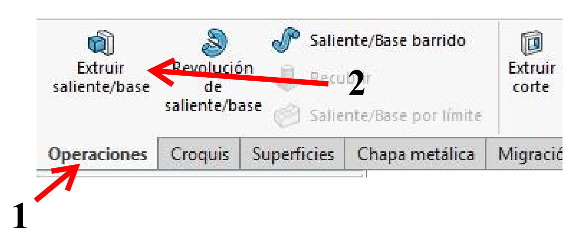
Tip: We have also used centerlines in other tutorials. These lines are actually auxiliary lines. When you use a sketch to make an extrusion, for example, SolidWorks only uses the ‘real’ lines and not the auxiliary lines.

In step **12** you have seen that you can easily change a ‘real line’ (or circle or arc) into an auxiliary line and vice versa. For this option, the **For construction** box in the **PropertyManager** must be checked.

13 Next, we will cut a corner from the model.

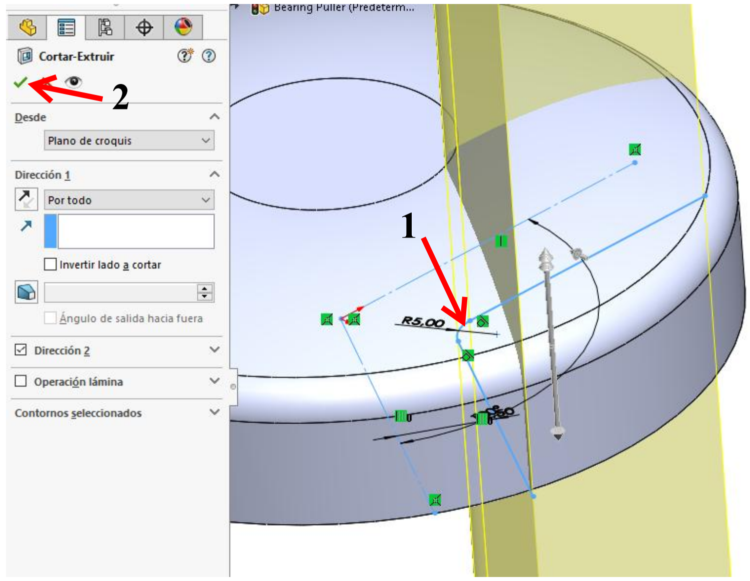
1 Click on the **Features** tab in the **CommandManager**.

2 Click on **Extruded Cut**.



14 You can see a small arrow in the model that indicates from which side of the sketch the material will be removed.

1. Make sure these arrows point outwards. Click on it when you need to change the direction.
2. Click on **OK**.



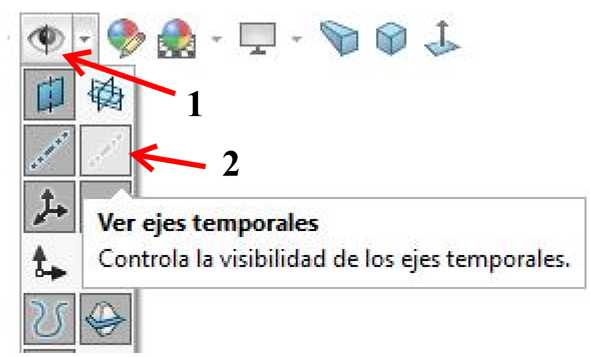
Tip: In most cases you will use a closed sketch for an **Extruded Cut**. In the case of a circle or a square, you will only make a hole in the shape of that sketch.

In the last step, we used an open sketch to make an **Extruded Cut**. It is handled in the same way except for two differences:

1. An **Extruded Cut** with an open sketch will always go through the entire depth of the model (**Through All**). You cannot set a depth.
2. SolidWorks needs to know from which side the material has to be cut away. You must pay attention to the little arrow, which indicates the cutting side. By the way, you can also change this direction in a closed sketch and cut away the material from the inside or outside of the sketch boundaries.

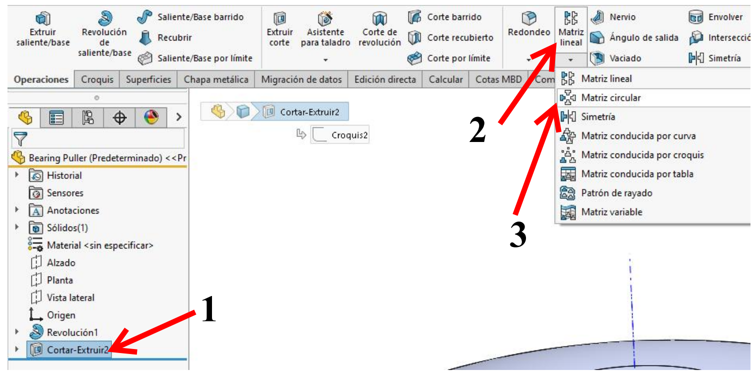
15 For the next feature we need an auxiliary line that runs through the middle of the model. This axis exists in the model already but is not visible with the standard (default) settings.

1. Click on the **Hide/Show Items** icon.
2. Make sure the button **View Temporary Axes** is set.



16 Next, we can copy the part with the cut three times around the axis.

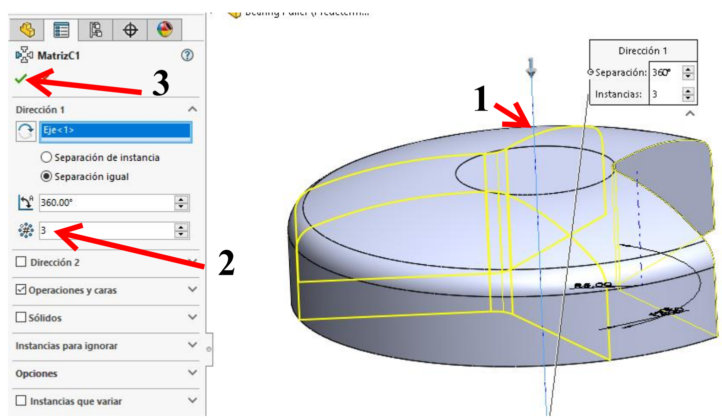
1. Select the last feature: **Cut-Extrude1** in the **FeatureManager**.
2. Click on the arrow below **Linear Pattern** in the **CommandManager**.
3. Click on **Circular Pattern**.



17 Select the centerline that runs through the middle of the model.

Change the number of copies in the **PropertyManager** to **3**.

Click on **OK**.



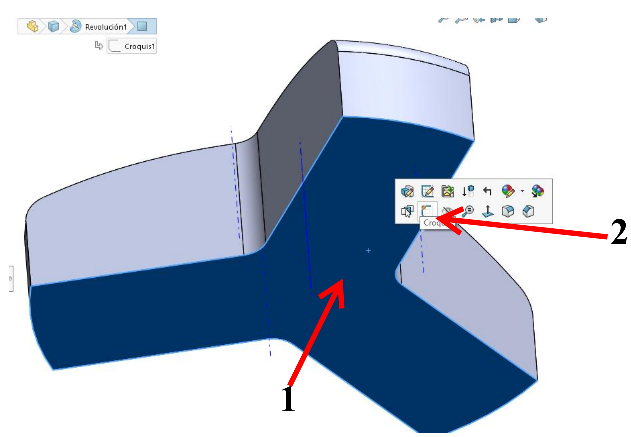
Tip: Notice that in the three last steps we first selected the feature in the **FeatureManager** and then selected the **Circular Pattern** command. At this point, SolidWorks 'understands' that you want to use this command for the selected item and automatically adjusts the settings in the **PropertyManager**.

You can also do this in the reverse order by giving the command first and then selecting the elements in the **PropertyManager**.

SolidWorks does not have a preference for how you do it. You will have to find out for yourself the approach that works best for you.

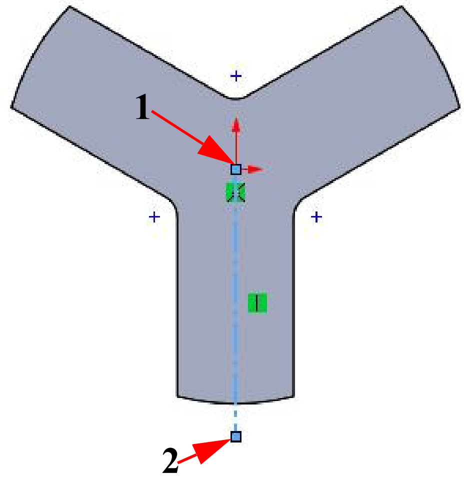
18 We will now make a sketch on the lower surface of the model. Rotate the model so you can see the bottom plane of the part.

1. Click on the surface to select it.
2. Click on **Normal To** in the menu that appears.



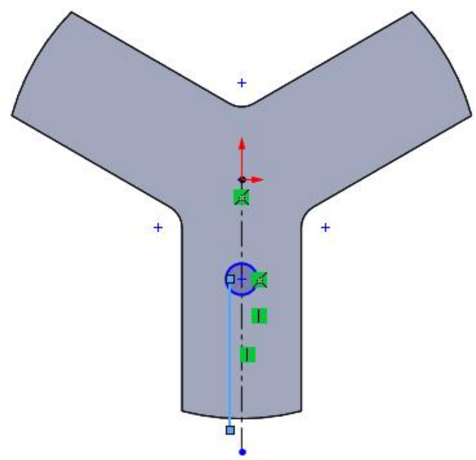
19 Draw a **Centerline**.

1. Put the first point right on the **origin**.
2. Set a second point at a random distance directly below the **origin**.



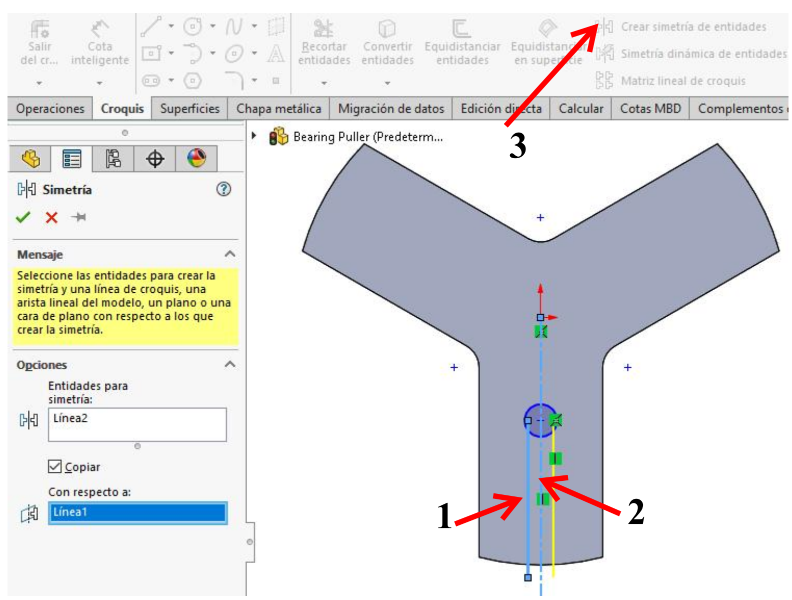
20 Draw a circle and a line at the locations indicated on the right.

The midpoint of the circle must be on top of the centerline.

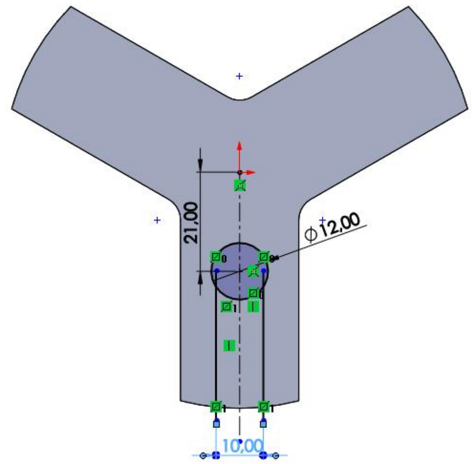


21 Make a mirrored image of this line at the other side of the centerline.

1. Select the line and the centerline (hold the <Ctrl> key).
2. Click on **Mirror Entities** in the **CommandManager**.

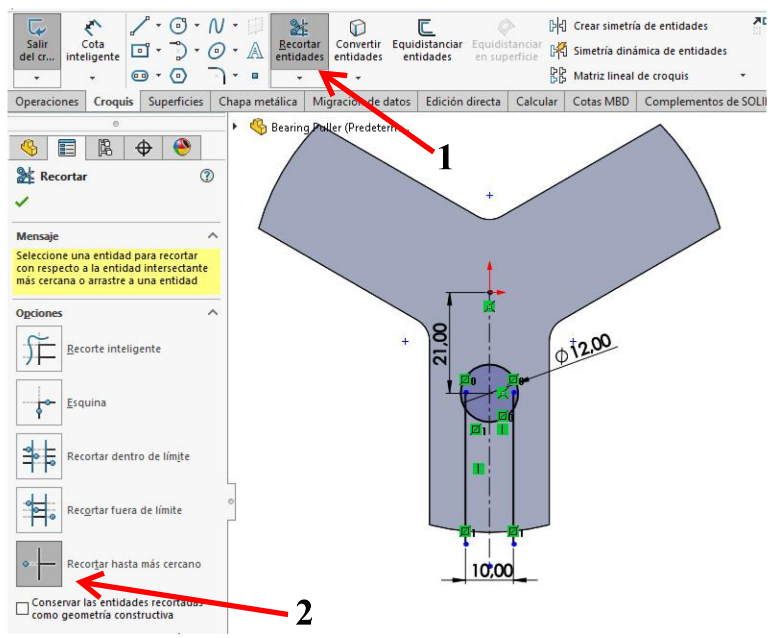


22 Now, set the three dimensions you see in the illustration on the right. Do this using **Smart Dimension** and change the values.



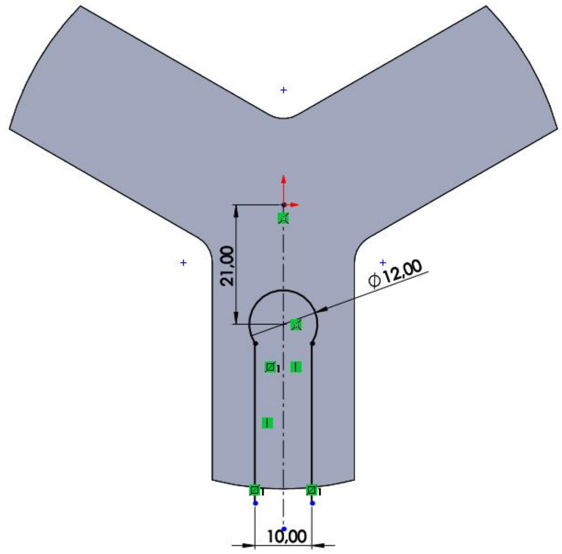
23 Click on **Trim Entities** in the **CommandManager**.

Select the option **Trim to closest** in the **PropertyManager**.

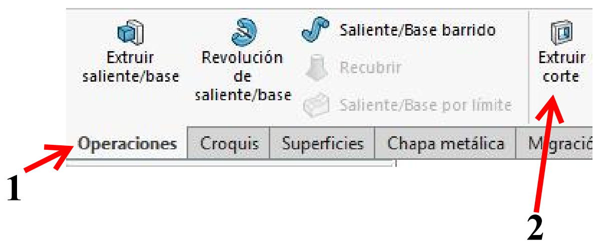


24 Next, click on the parts of the sketch that must be removed. Make sure you end up with a sketch similar to the one on the right.

Should the dimension of 10 mm disappear as a result of the trimming command, resize that item by using **Smart Dimension** again in the sketch.

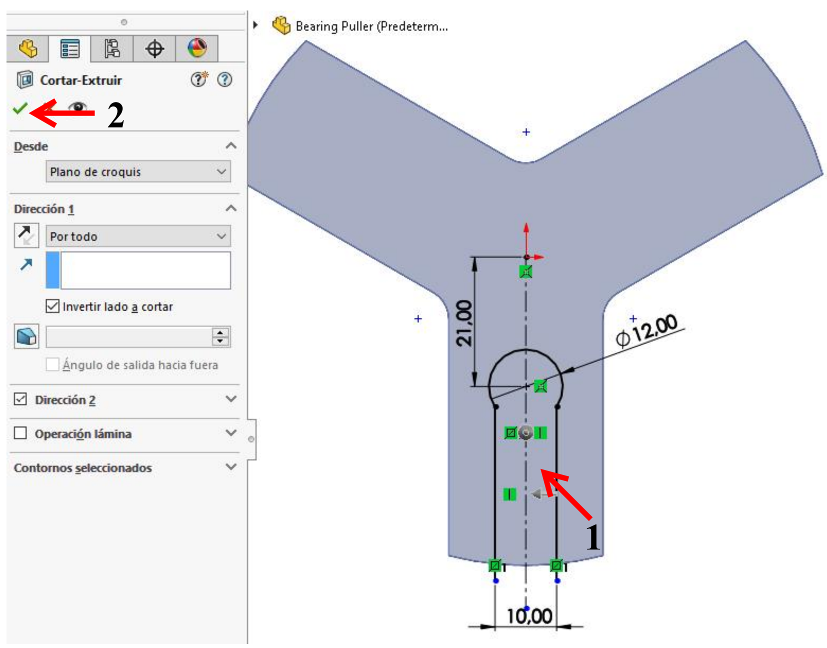


25 Click on the **Features** tab in the **CommandManager** and then on **Extruded Cut**.



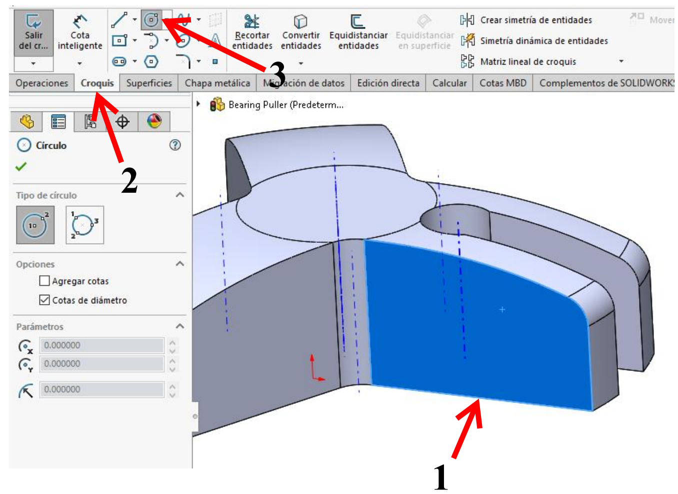
26 You must pay attention to which direction the material is removed from because the sketch is not entirely closed.

1. Make sure the little arrow that sets the direction is pointing inward.
2. Click on **OK**.

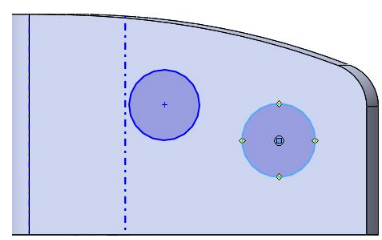


27 Next, we have to make some holes.

1. Select the plane as indicated in the illustration.
2. Click on the **Sketch** tab in the **CommandManager**.
3. Click on **Circle**.

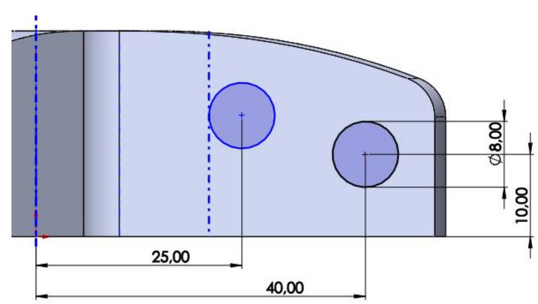


28 Rotate the model with **Normal To**, and draw two circle at random positions like in the drawing on the right.



29 Use **Smart Dimension** to set four dimensions in the sketch, and change their values as indicated on the right.

Push the <Esc> key to close the **Smart Dimension** command.

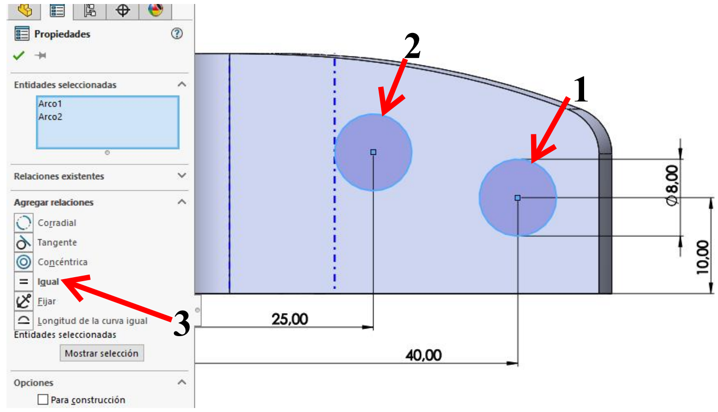


30 Next, set the circles to same size:

1 Select one of the circles.

2 Hold the <Ctrl> key and select the other circle.

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

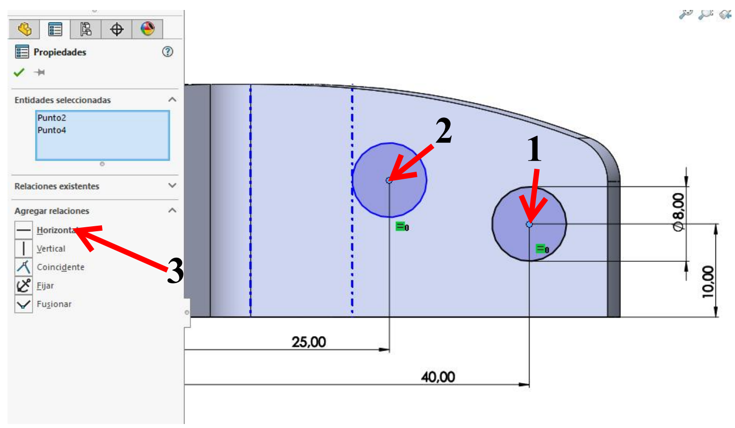


31 Next, set the circles to same height:

1 Select the midpoint of one of the circles.

2 Hold the <Ctrl> key and select the midpoint of the other circle.

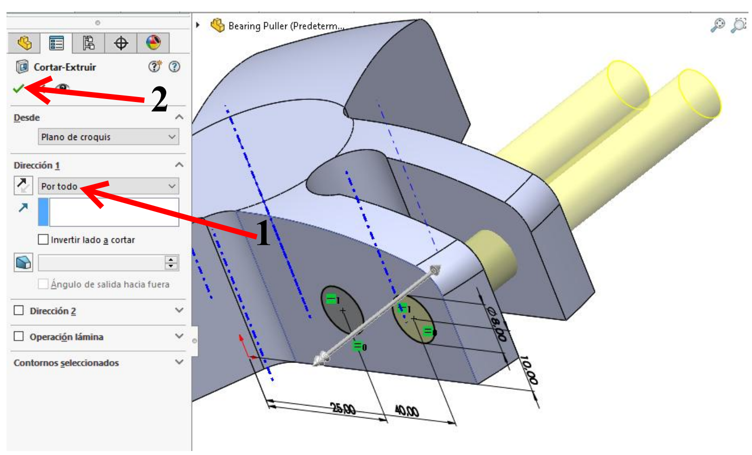
3 Click on **Horizontal** in the **PropertyManager**.



32 Click on the **Features** tab in the **CommandManager**, and after that on the **Extruded Cut**.

1 Set the depth to **Through All** in the **PropertyManager**.

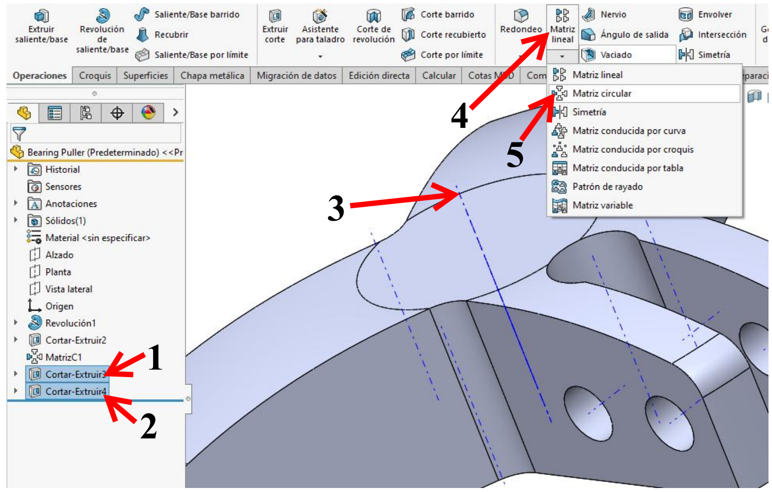
2 Click on **OK**.



33 We must now copy the holes we just made to the other 'legs'.

1,2 Select the last two features in the **FeatureManager**.

3 Select (holding the <Ctrl> key) the axis through the middle of the model.

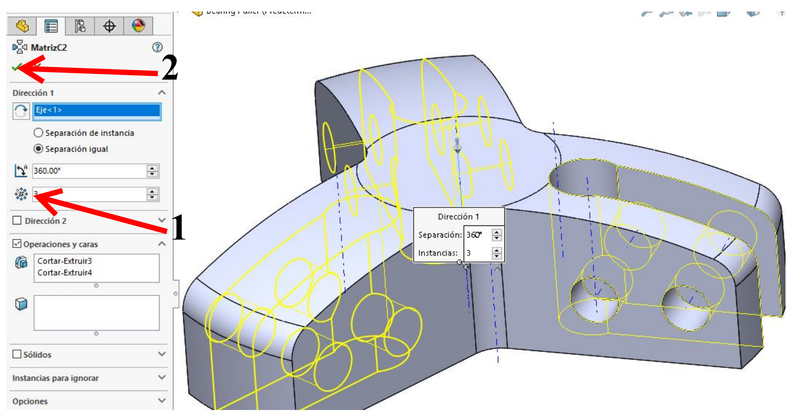


4 Click on the arrow below **Linear Pattern** in **CommandManager**.

5 Click on **Circular Pattern**.

34 Set the number of copies in the **PropertyManager** to **3**.

Click on **OK**.



35 Finally, we have to make the metric thread in the hole:

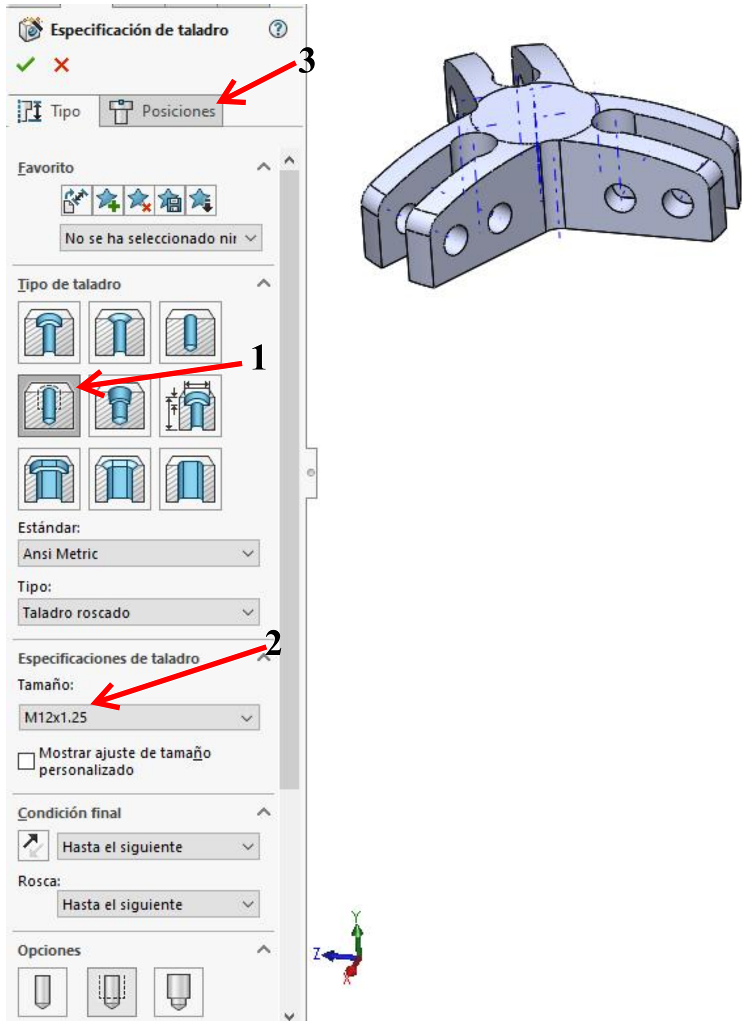
Click on **Hole Wizard** in the **CommandManager**.

36 Set the following features in the **PropertyManager**:

1. The **Hole Type** is **Straight Tap**.
2. The **Size** is **M12**.

Check the other settings to make sure they match with the illustration on the right.

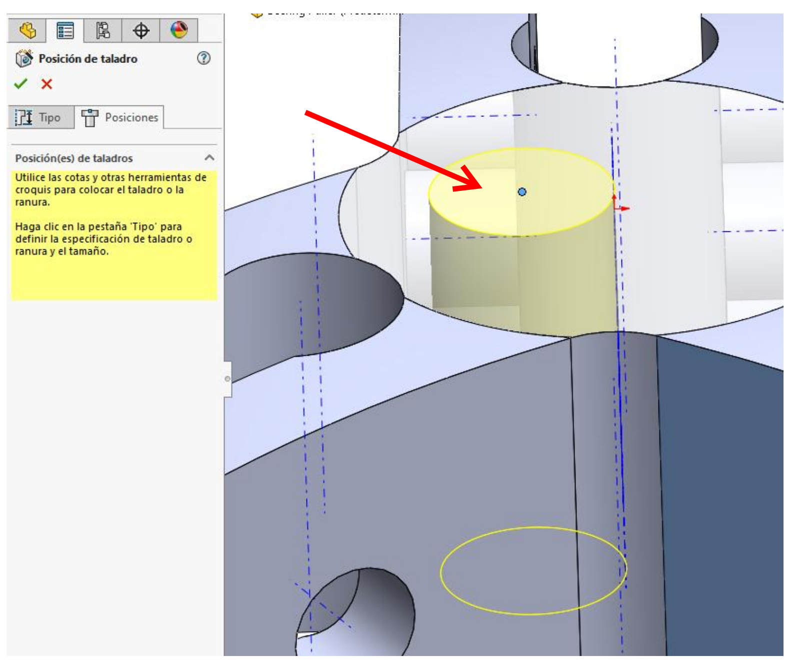
3. When everything is set properly, click on **Positions** to place the hole.



37 Set the hole on the top plane of the bridge at a random position.

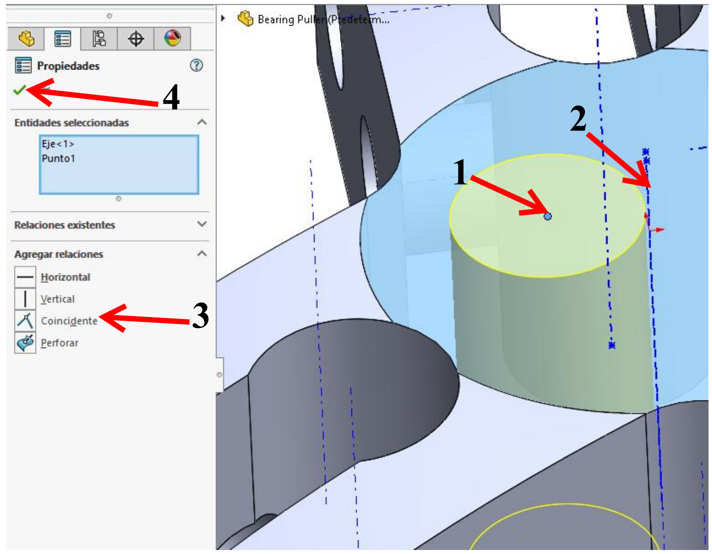
Actually, you are setting a **point** now, which will determine the position of the hole.

The point is on the plane, but it is not at the midpoint of the plane. To do this, we conduct an additional step.



38 Push the <Esc> key first.

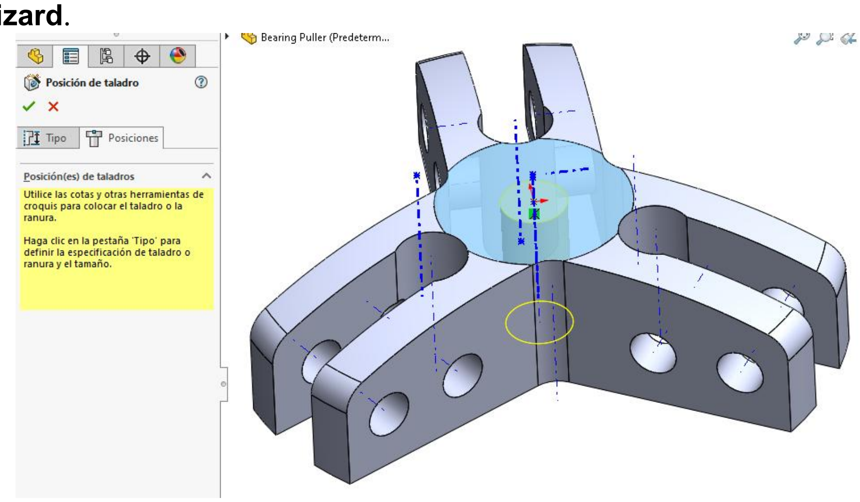
1. Select the point that you positioned in the last step.
2. Push the <Ctrl> key and select the axis we used before for the circular patterns.
3. Click on **Coincident** in the **PropertyManager**.
4. Click on **OK**.



The hole will now shift to the middle of the plane.

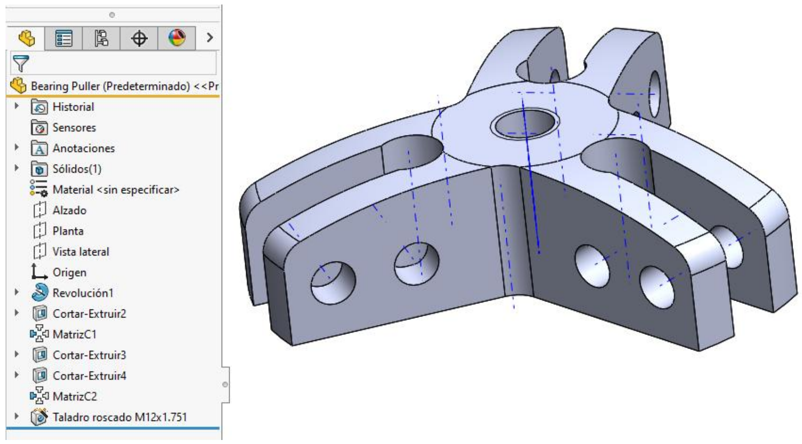
39 You can now return to the **Hole Wizard**.

Click on **OK**.



Tip: When you have to place a hole using the Hole Wizard (steps 37-38), you are actually making a sketch. By putting a point in that sketch, you are positioning the hole.

40 The model is now ready. Save it as: bridge.SLDPRT. First, create a new folder, so you can keep all files together.

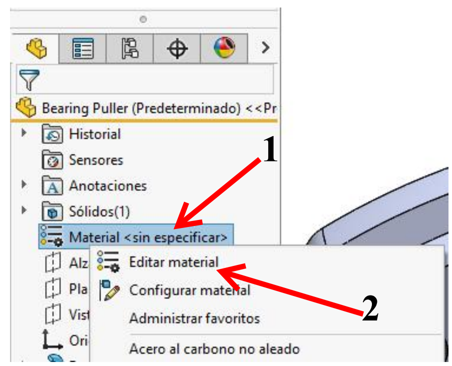


41 We would like to have more information about this model. What does it weigh? Where is the center of gravity? Is it strong enough?

To be able to answer these kinds of questions, we must first determine the kind of material to use to make the part.

1 Right-click on **Material** in the **FeatureManager**.

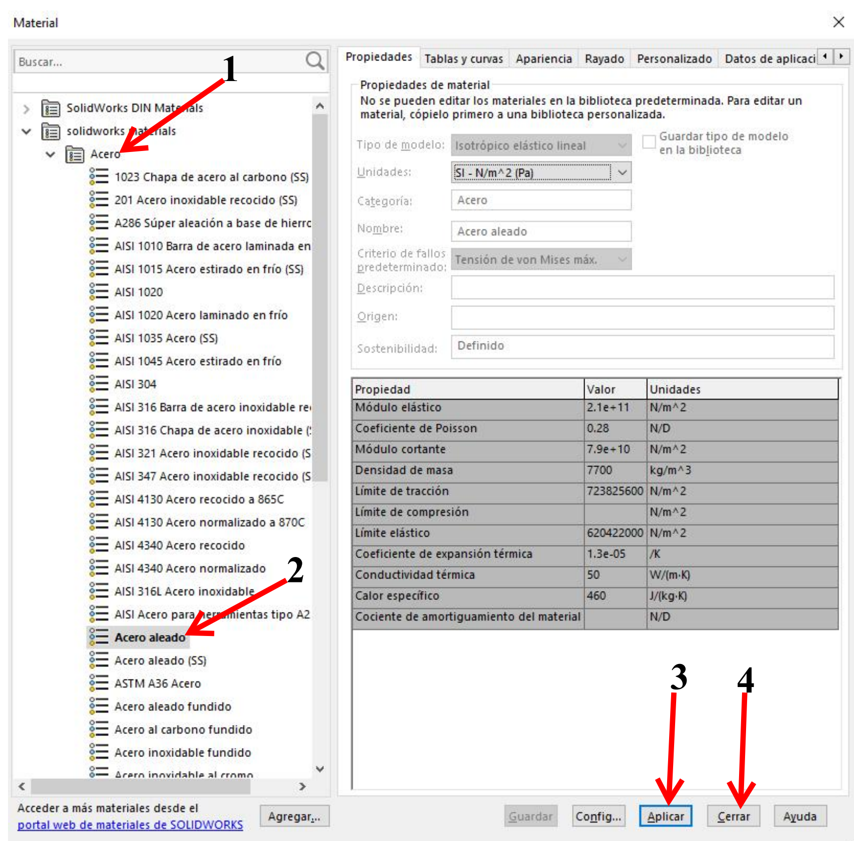
2 Select **Edit Material** in the menu.



42 Open the main group **Steel** by clicking on the **+** symbol.

Select **Alloy Steel** as the desired material.

Click on **Apply** and then **Close**.

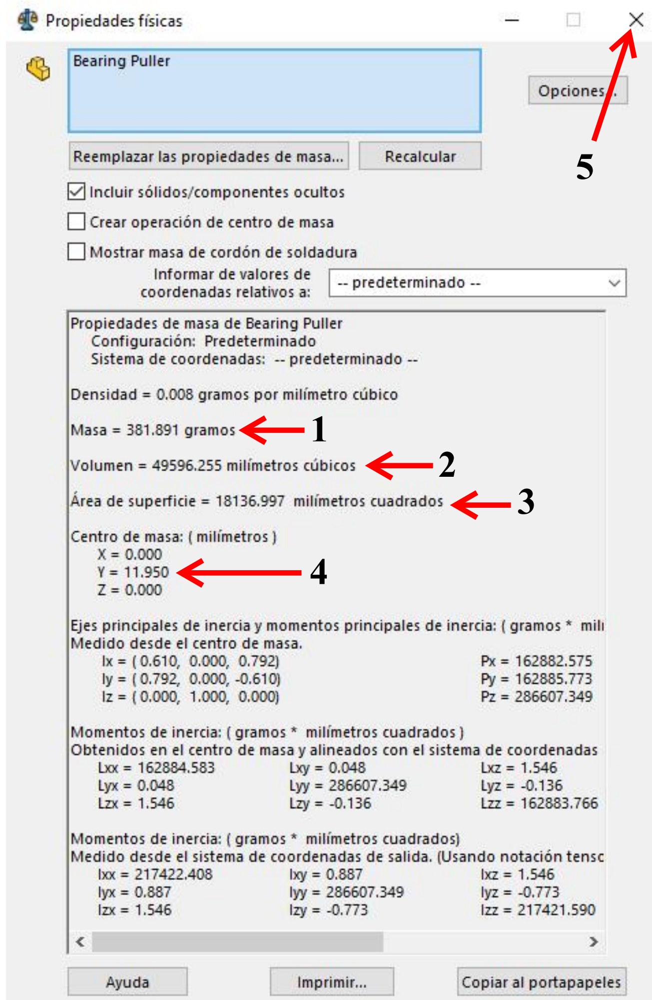


43 We can evaluate the data now.

1. Click on the **Evaluate** tab in the **CommandManager**.
2. Click on **Mass Properties**.

44 A menu appears, in which you can read the data, including:

1. The weight of the part.
2. The volume.
3. The total surface of the part. This could be important when a part has to be painted.
4. The coordinates of the point of gravity.
5. When you have finished reading the data, click on Close to close the window.

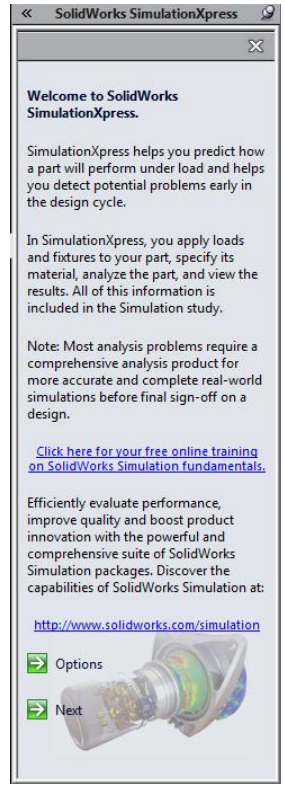


45 Next we want to know if the part is strong enough for our purpose. We want to be able to pull 600 kg (6000 N). To find out if our part is strong enough for this, we will use **SimulationXpress**.

Click on **SimulationXpress Analysis Wizard** in the **Evaluate** tab of the **CommandManager**.

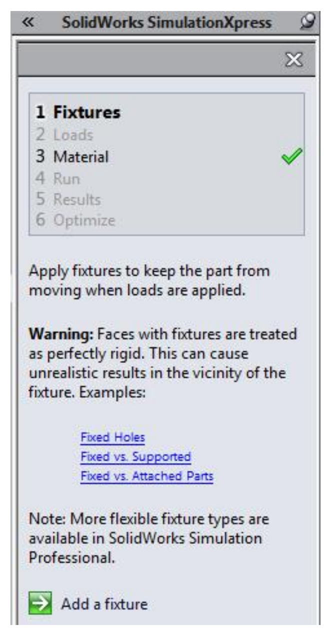
46 **SimulationXpress** starts as a wizard. You will be led through a number of steps and will get a result at the end.

Click on **Next** in the startup screen.



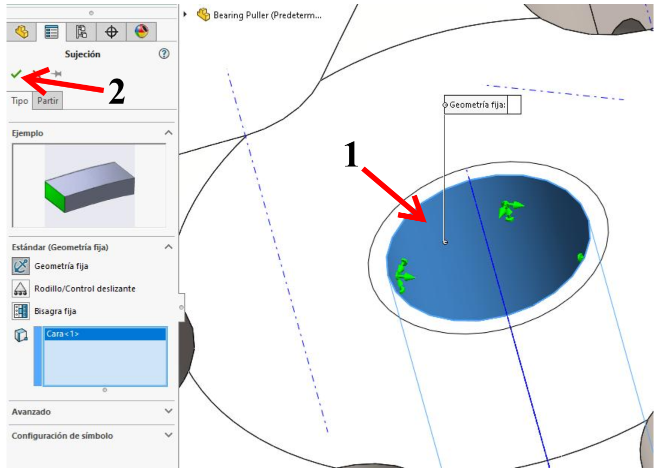
47 First, we need to add a fixture which will fix part of the bridge.

Click **Add a fixture**.

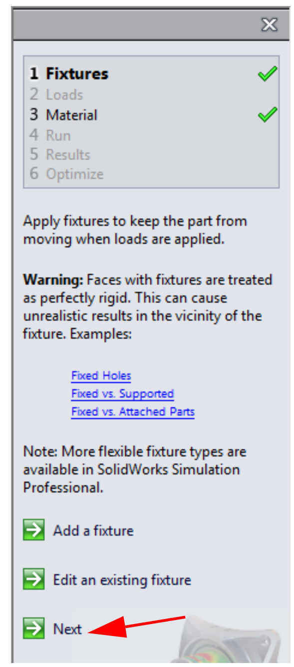


48 Select the inside face of the threaded hole in the model.

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

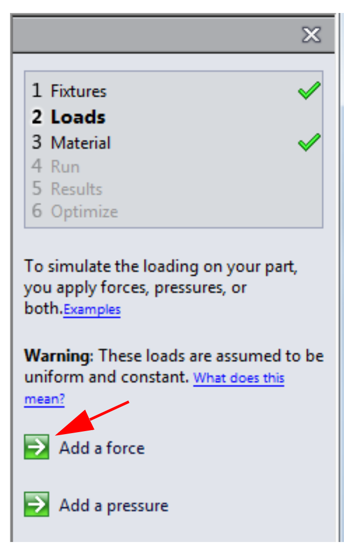


49 Click **Next** in the **SimulationXpress Wizard**.

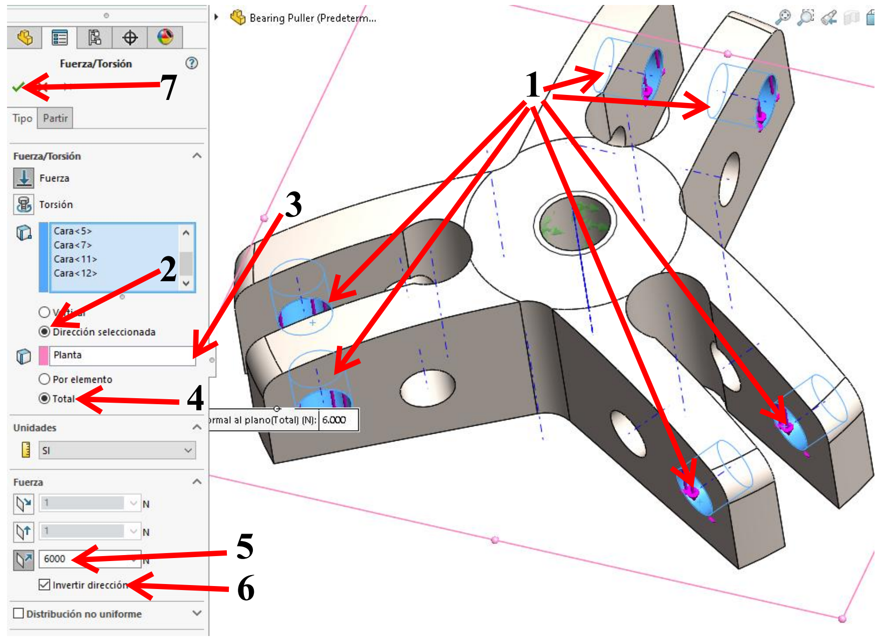


50 This next step is where we can set the **Load**.

Click on **Add a force**.



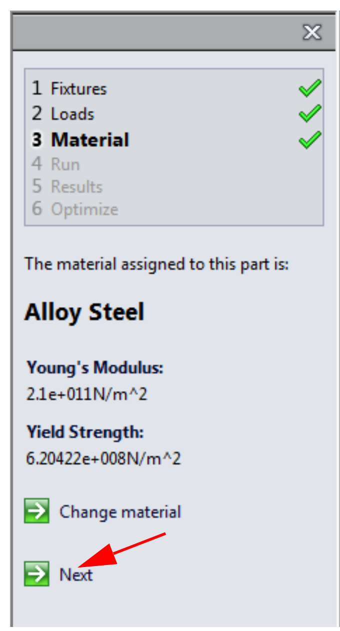
51 Set the following settings in the **PropertyManager**:



1. Select the six holes in which the arms will be mounted.
2. Check the option **Selected direction**.
3. Click on the **Top Plane** in the **FeatureManager**.
4. Select the option **Total**.
5. Set the force to **6000 N** (Newtons).
6. Check **Reverse Direction** in order to make the arrows point downward.
7. Click on **OK**.

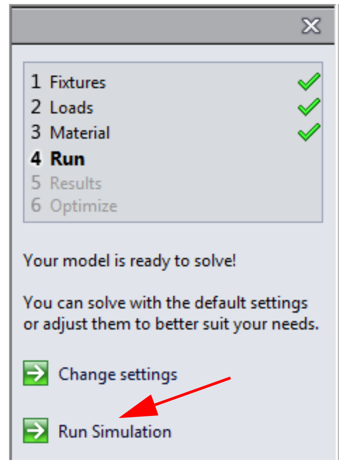
52 Click **Next** in the **SimulationXpress Wizard**.

53 The **Material** must now be set. We already did this so click **Next**.

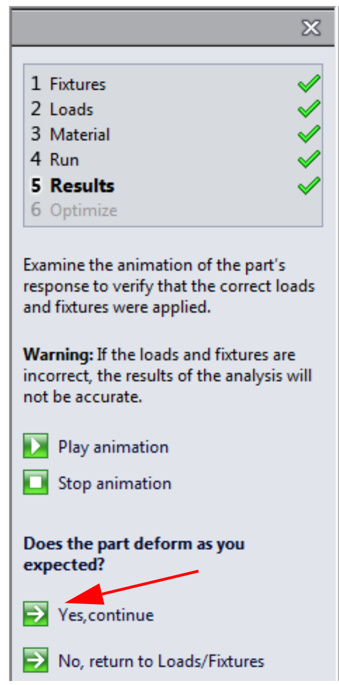


54 We are now ready to run the simulation.

Click **Run Simulation**.



55 Once the simulation is done, click **Yes, continue** if the part deforms as it should.

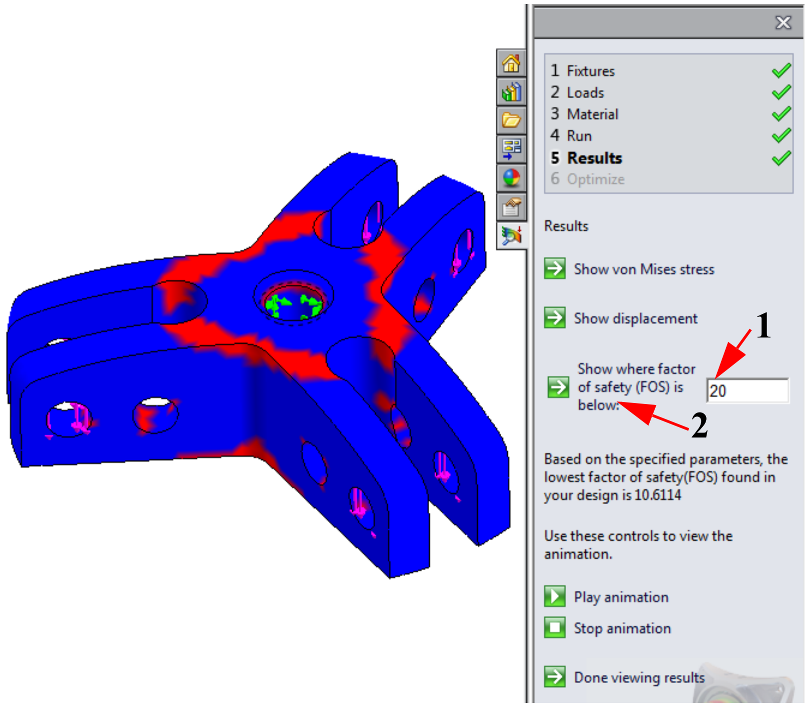


56 The result of the analysis is that the lowest factor of safety is 10.61. The part is strong enough (read the tip below).

Do you want to see the weak spots?

1. Set the FOS value to **20** (as an example).
2. Click on **Show where factor of safety (FOS) is below:**

You will see the weak spots in red now.



57 Click **Done viewing results** and then on **Next**.

Tip: The factor of safety (FOS) is a number calculated by the simulation. When the FOS value is less than 1, the part will collapse when the given forces are applied. When the FOS value is more than 1, the model is strong enough, maybe even too strong.

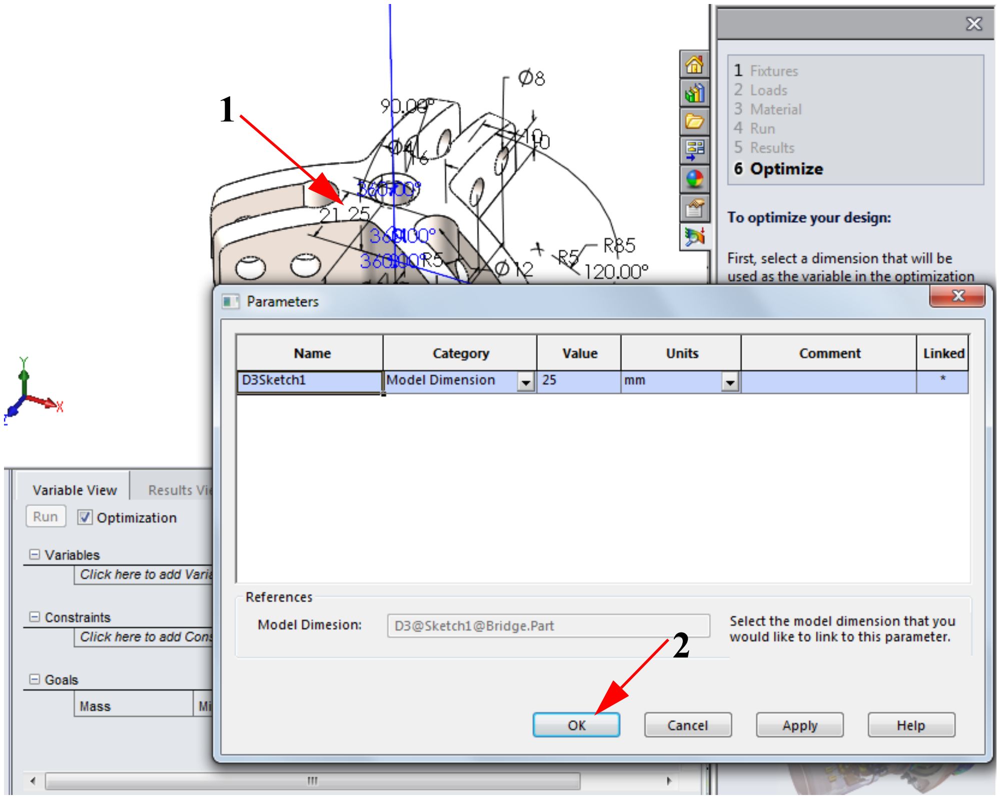
58 Because the calculated FOS value is 10.61, the construction of the model is obviously too heavy.

You can now decide to optimize the design by setting the FOS value to 1.

1. Click on **Yes**.
2. Click on **Next**.

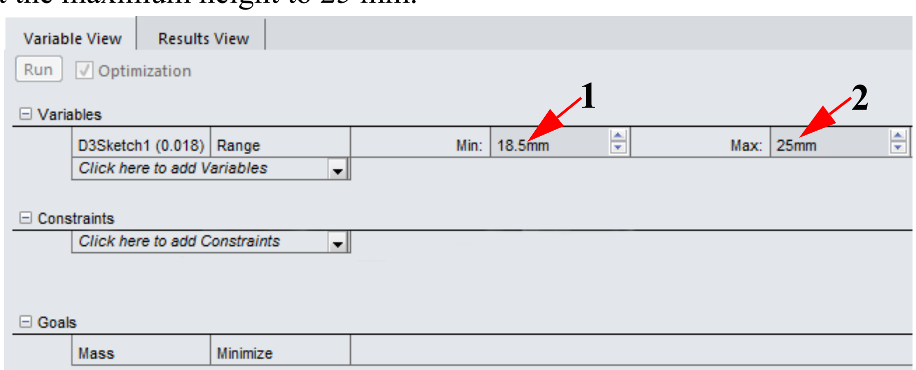
59 All dimensions are visible now.

1. Select the dimension of 25 mm that indicates the height of the model. Make sure to select the right dimension! In the **Name** field in the **Parameters** window you can see the selected dimension is extracted from sketch1 (the first sketch you made in this part).
2. Click **OK** in the **Parameters** window.



60 For the range of values set the minimum height to 18.5 mm.

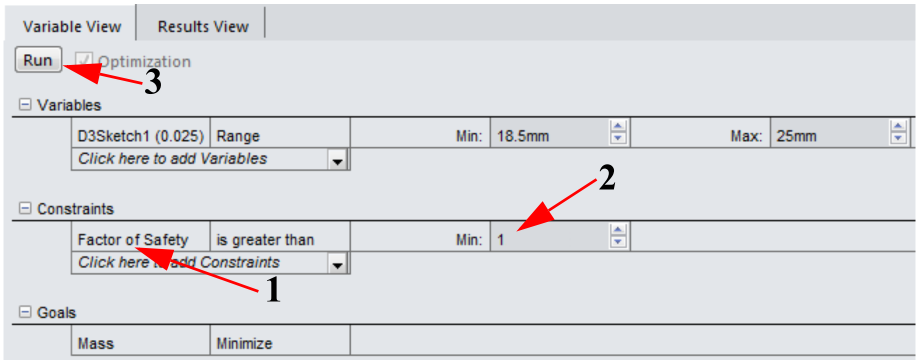
Set the maximum height to 25 mm.



61 Click the arrow under **Constraints** and select **Factor of Safety**.

Set the minimum value to **1**.

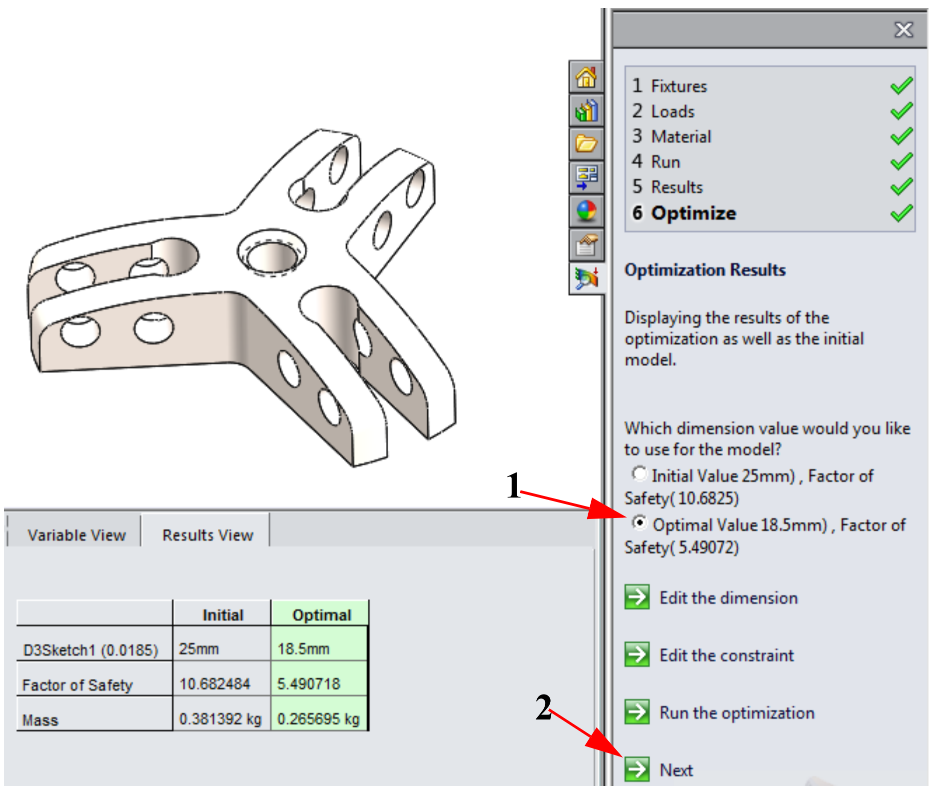
Click **Run**.



62 SimulationXpress has calculated that the model height can be reduced to 18.5 mm and still have a FOS of 5.49. The weight has been reduced by 30%, from 381 grams to 265 grams.

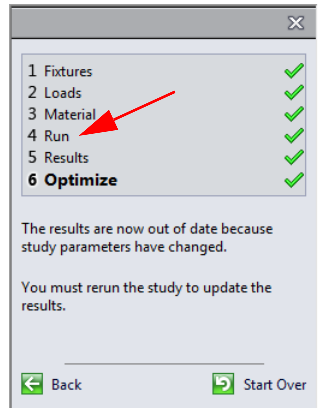
Choose **Optimal Value** for the **Which dimension value would you like to use for the model**.

Click **Next**.

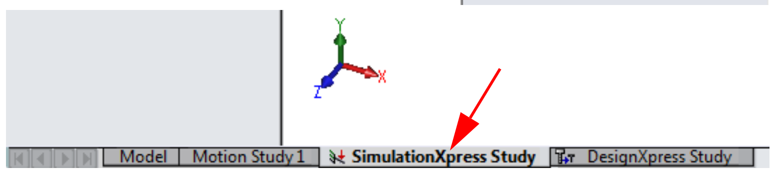


Note: The factor of safety is still above 1. However, the height of the part cannot be reduced without changing the hole locations. Other dimensions can be changed in order to reduce the factor of safety even more.

63 Click on the **Run** heading in the **SimulationXpress Wizard**.

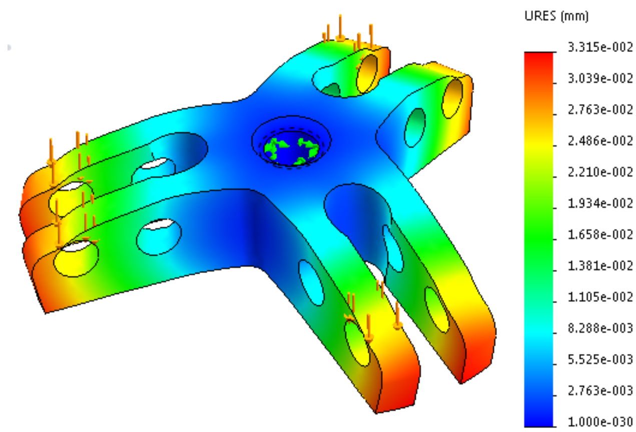


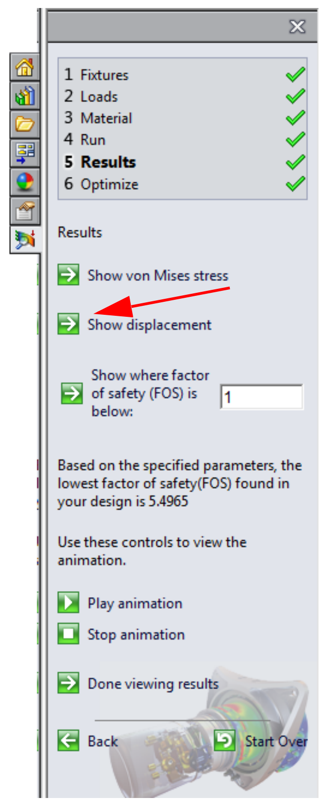
64 Select the **SimulationXpress Study** tab at the bottom of the window.



65 Next click on **Run Simulation** in the **SimulationXpress Wizard**.

When the study is finished running click **Show displacement**.





66 You can now see how the model distorts (exaggerated display) under the influence of the force.

Click on **Play** to see an animation of the distortion.

Click on **Stop** to stop the animation.

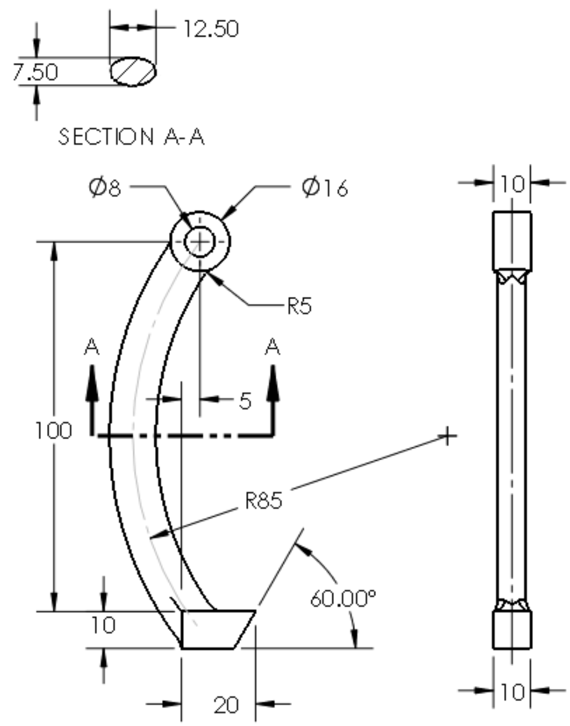
67 Save the changes to the file.

Click on **Save** in the Standard toolbar.

Arm

Work plan

The next part we will make is one of the arms. In the drawing below the part is already completed.

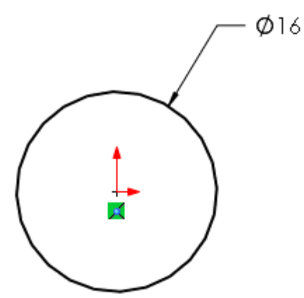


We will build this model by shaping the upper circle and lower part of the finger and will add the arm as a **sweep** later.

68 Open a new part and set the units as **MMGS**.

Start a sketch on the **Front Plane**.

Draw a circle with a diameter of 16 mm, with the midpoint on the **origin**.

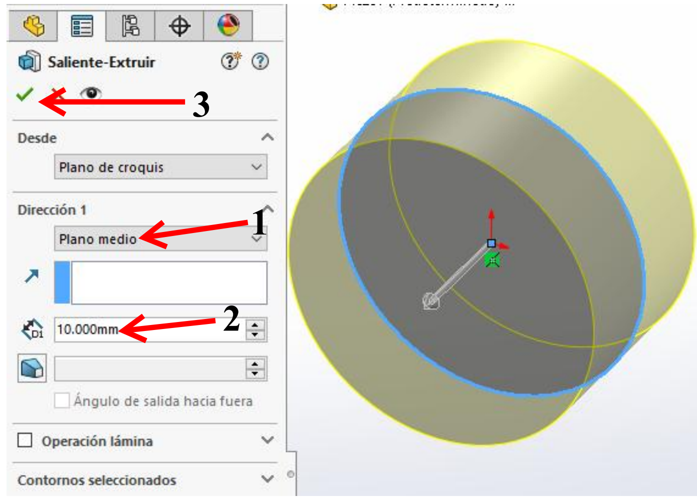


69 Make an extrusion from this circle:

1 Select the option **Mid Plane** in the **PropertyManager**.

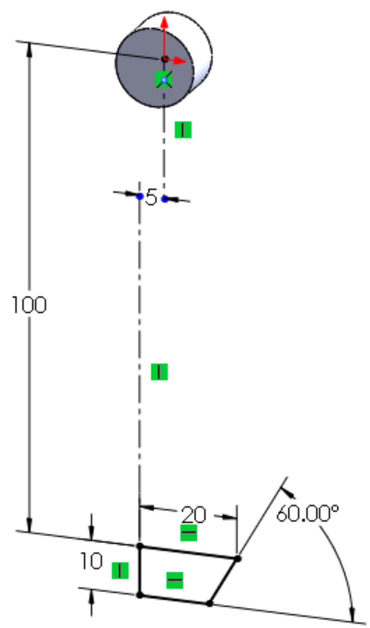
2 Set the thickness to 10 mm.

3 Click on **OK**.



Tip: We have not use the **Mid Plane** option before. This tool is very convenient when you want to build a symmetrical model. The sketch will extruded equally wide in two directions.

70 Select the **Front Plane** again and make the sketch similar to the drawing on the right.

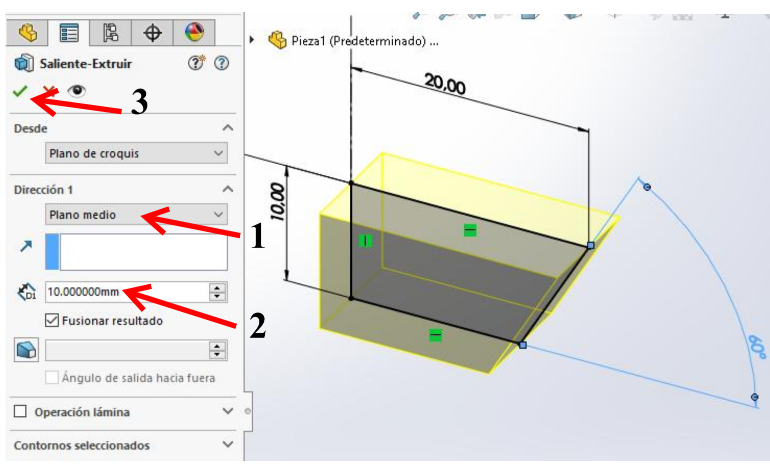


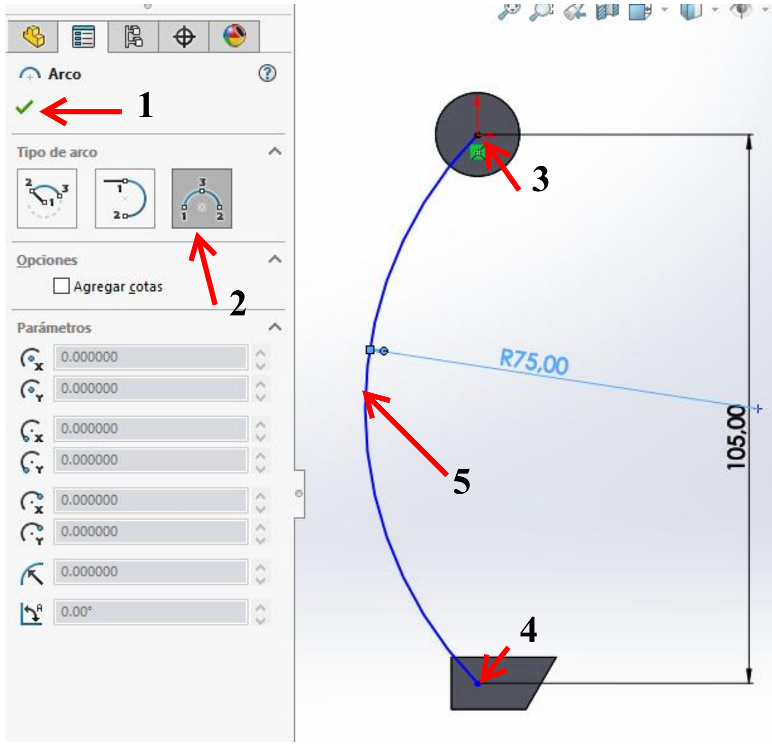
71 Make an extrusion from this sketch.

1 Use the option **Mid Plane** again.

2 Set the thickness to **10 mm**.

3 Click on **OK**.





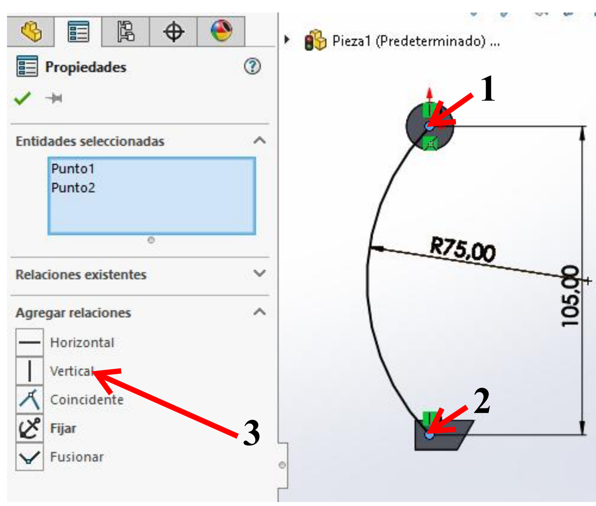
72 We will create a sweep now. A sweep is a feature in which you extrude a sketch following another sketch. So we have to make two sketches first.

Select the **Front Plane** and make a new sketch on it.

1. Click on **Arc** in the **CommandManager**.
2. Select **3-Point Arc** in the **PropertyManager**.
3. Click on the **origin** to set the starting point.
4. Click at the point as illustrated here to set the end of the arc. Its position does not have to be accurate at this point.
5. Click at the third point as illustrated here. Again, accuracy is not required.

Add two dimensions as illustrated.

It does not matter if the arc is not properly aligned at this point.



73 Select the upper end of the arc.

Select the bottom end of the arc too (use the <Ctrl> key).

Click on **Vertical** in the **PropertyManager**.

74 We will use this sketch later on.

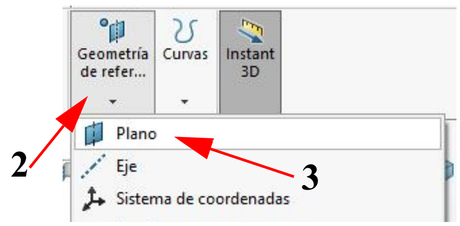
Click on **Exit Sketch** in the **CommandManager** to close the sketch.

75 The second sketch is made at a right angle to the end of the first sketch. For this we need to create an auxiliary plane first.

1 Click on the **Features** tab in the **CommandManager**.

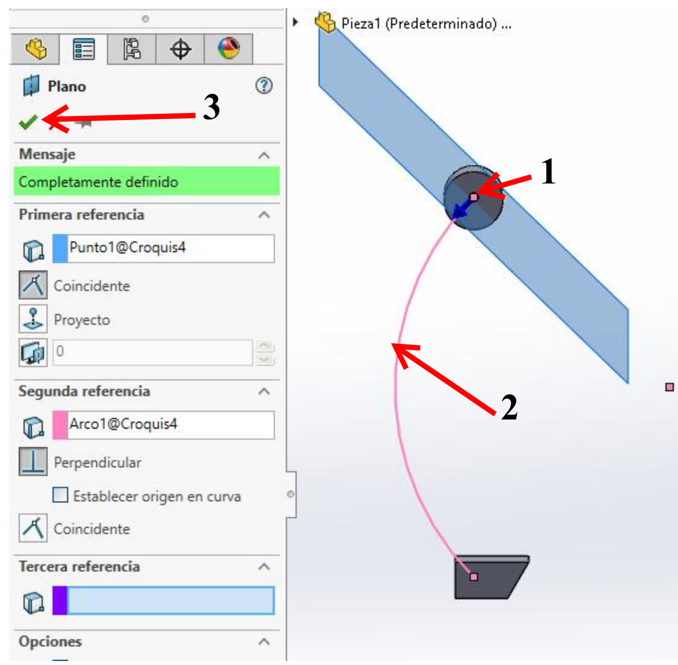
2 Click on **Reference Geometry**.

3 Click on **Plane**.



76 Click on the upper end point of the arc and then on the arc itself. The auxiliary plane will be positioned at a right angle to the end of the arc.

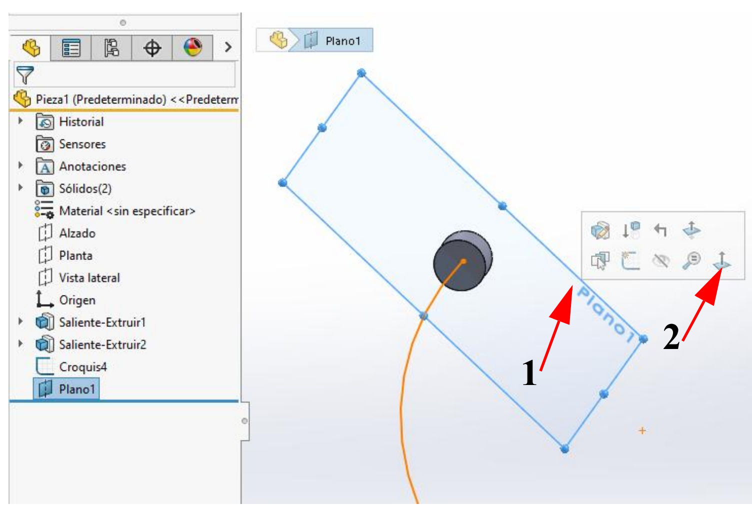
Click on **OK**.



77 Rotate the model so you will have a clear view of the plane you just created.

1 Click on the last mentioned plane.

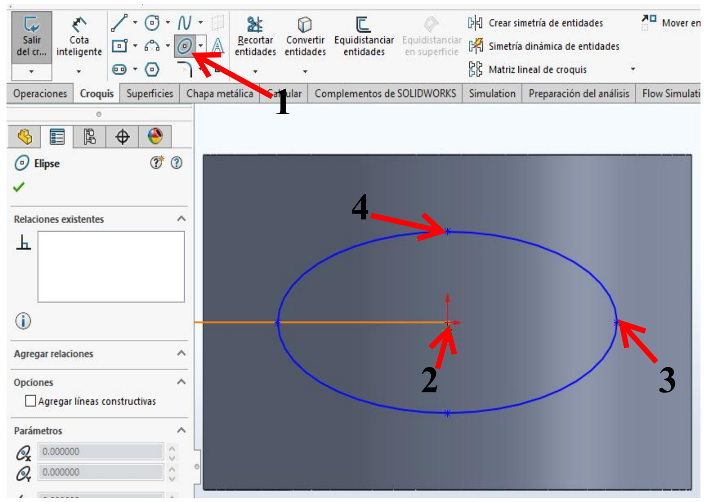
2 Click on **Normal To** in the menu that appears.



78 Zoom in on the **origin**, and draw an **ellipse**:

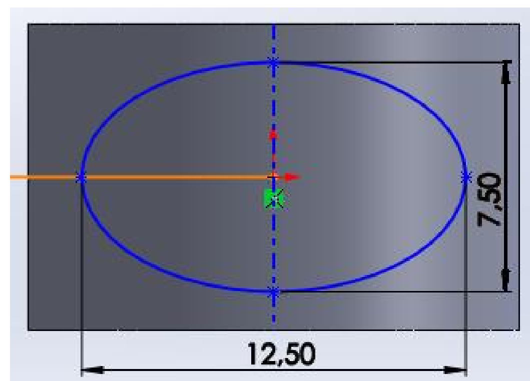
1. Click on **Ellipse** in the **CommandManager**.
2. Click on the **origin**.
3. Click on a horizontal position besides the **origin** to set the long axis of the **ellipse**.
4. Click straight above the **origin** to set the short axis.

The exact dimensions do not matter yet.

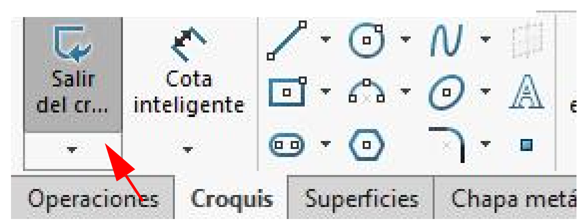


79 Set the dimensions of the two axes as illustrated on the right with **Smart Dimension**.

Add a horizontal relation between the two points on the long axis.

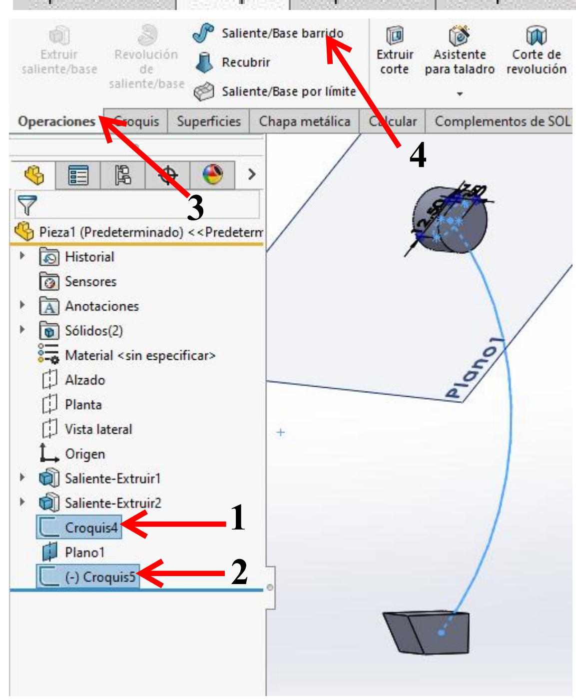


80 This sketch is now done, so click on **Exit Sketch** in the **CommandManager**.



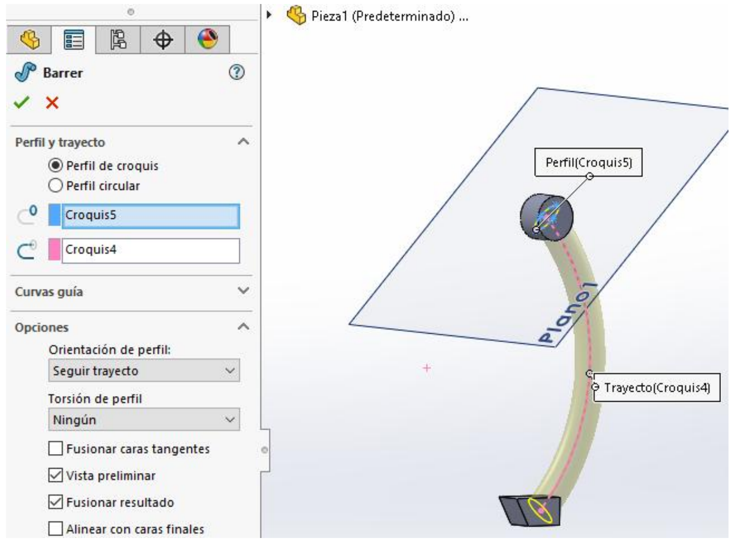
81 We will combine the two sketches into a sweep.

1. Select the sketch with the arc in the **FeatureManager**.
2. Select the sketch with the ellipse too (use the <Ctrl> key).
3. Click on the **Features** tab in the **CommandManager**.
4. Click on **Swept Boss/Base**.



82 You do not have to set any other features in the **PropertyManager**.

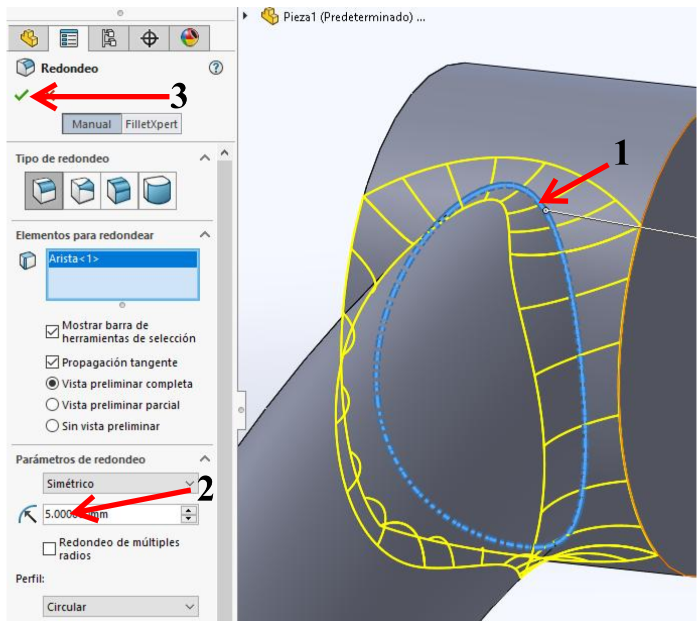
Click on **OK**.



83 The connection between the arm and the top and bottom parts has to be finished.

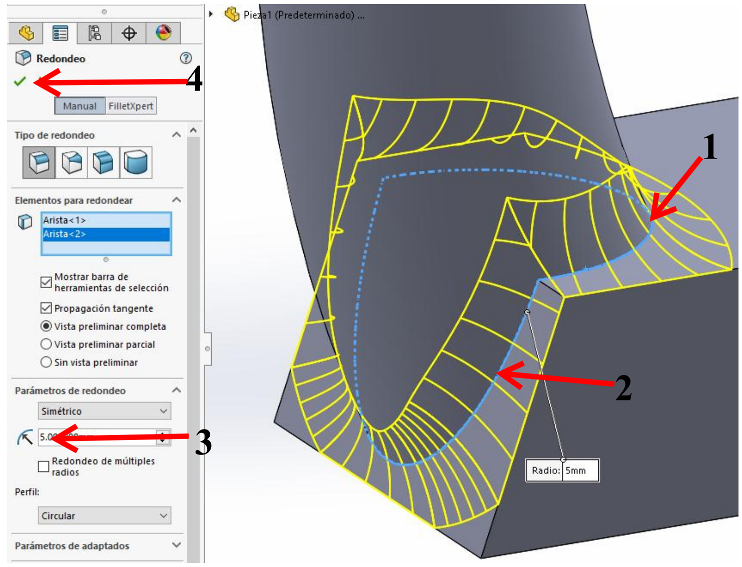
Click on **Fillet** in the **CommandManager**.

1. Select the cutting edge between the arm and the upper circle.
2. Set the radius to **5 mm** in the **PropertyManager**.
3. Click on **OK**.



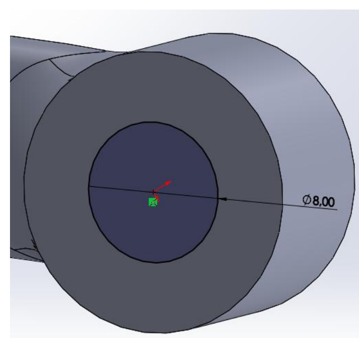
84 Next, round off the connection at the bottom. Click on **Fillet** in the **CommandManager**.

Select both cutting lines now. The radius is also set to **5 mm**.



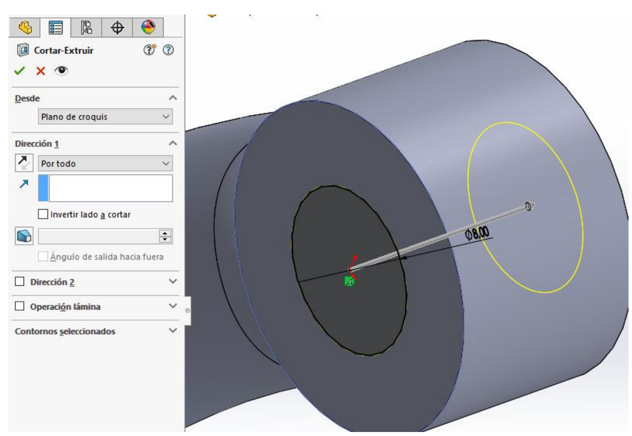
85 Finally, we have to put a hole in the upper circle to accommodate a bolt.

Make the sketch as shown on the right.

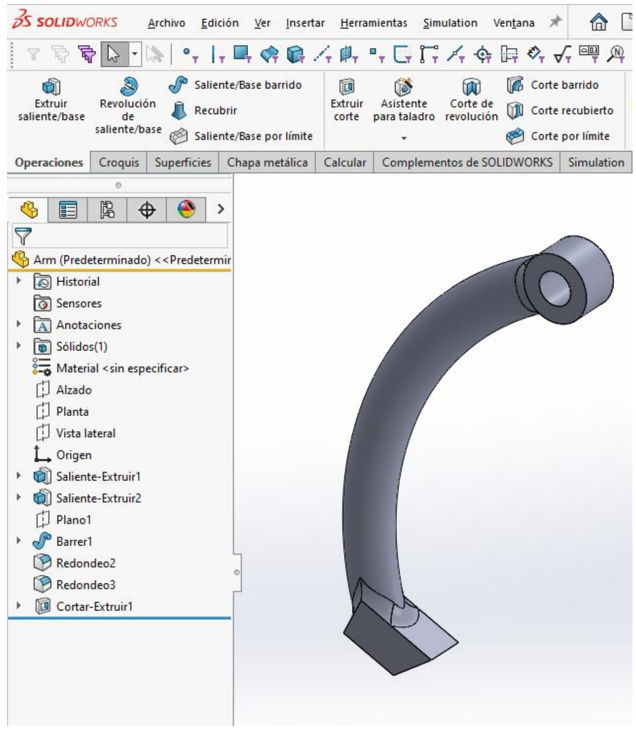


86 Make an **Extruded Cut** from this sketch.

1. Set the option **Through All** to go all the way through the material.
2. Click on **OK**.



87 Save the file as: Arm.SLDPRT.



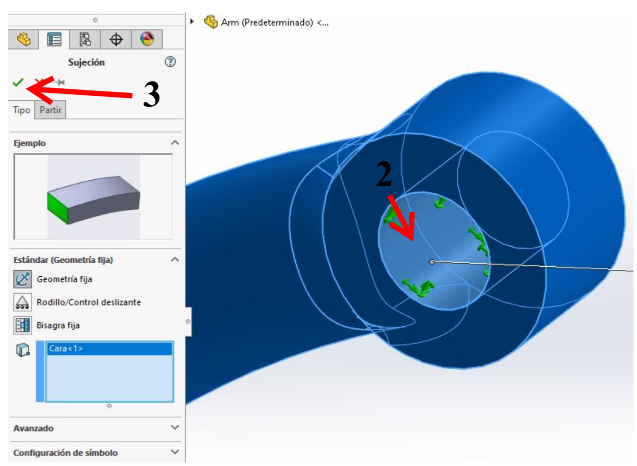
88 Of course, we also want to know if the arm is strong enough for our purpose. The complete tool should be able to pull 600 kg, or about 200 kg (2000 N) per arm.

1. Click on the tab **Evaluate** in the **CommandManager**.
2. Click on **SimulationXpress Analysis Wizard**.

Run the wizard by clicking Next every time. We will only display and describe the steps that need input.

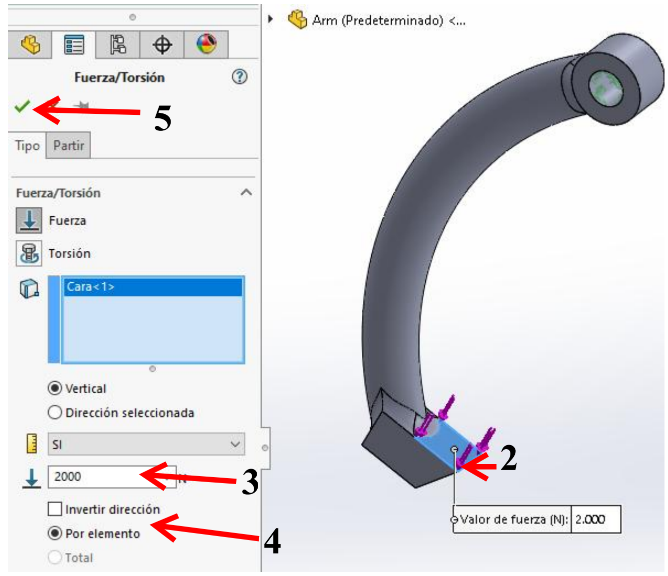
89 Define the Fixture:

1. Click on **Add a fixture**.
2. Select the hole where the bolt goes though.
3. Click on **OK**.
4. Click on **Next**.



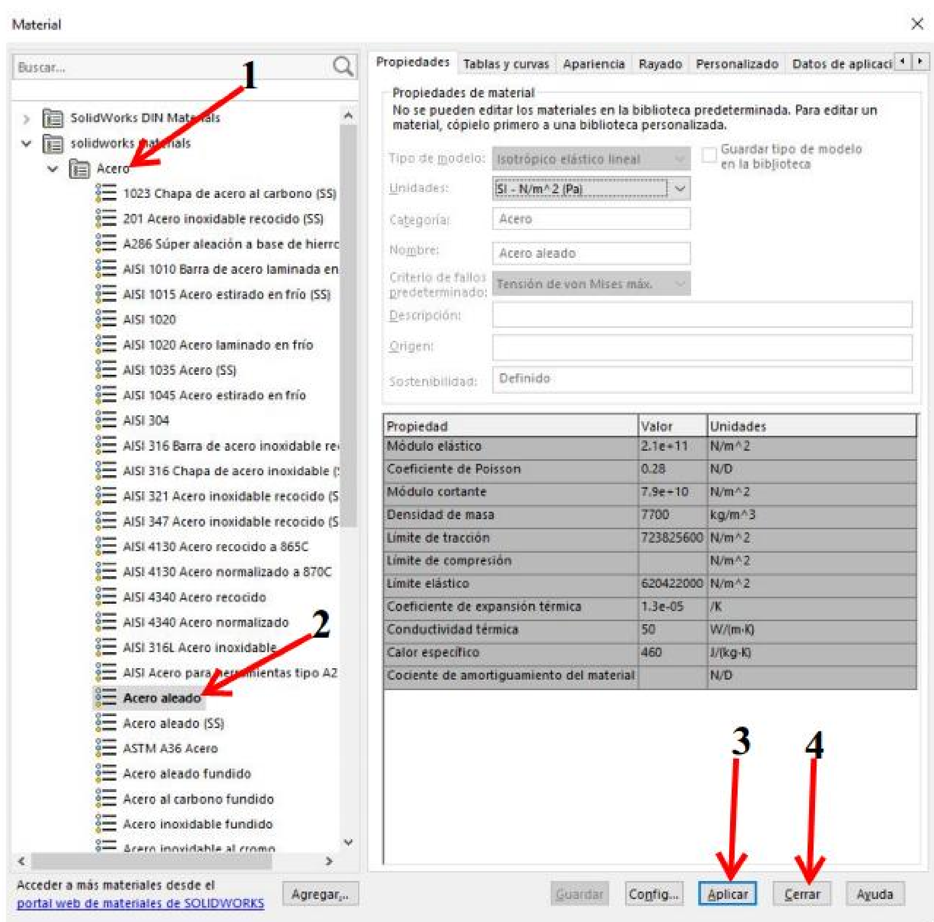
90 Set the **Load**:

1. Click **Add a force**.
2. Select the plane in the model as illustrated on the right.
3. Set the force to **2000 N**.
4. Make sure the force is downward. When they do not, click on **Reverse direction**.
5. Click **OK**.
6. Click on **Next**.



91 Define the desired material:

1. Click **Choose Material**.
2. Select **Alloy Steel**.
3. Click on **Apply**.
4. Click on **Close**.
5. Click **Next**.



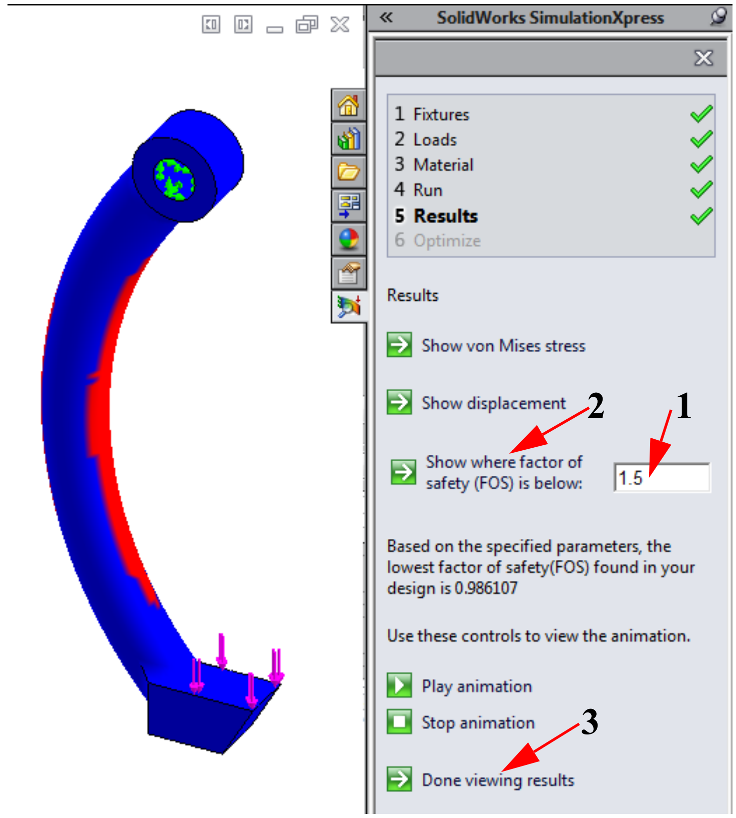
92 After the analysis is done, the FOS value turns out to be 0.98. This is not strong enough!

1 Fill in **1.5** in the menu.

2 Click on **Show where factor of safety (FOS) is below:**

You can now see clearly where the strain is the highest: on the inside of the arm.

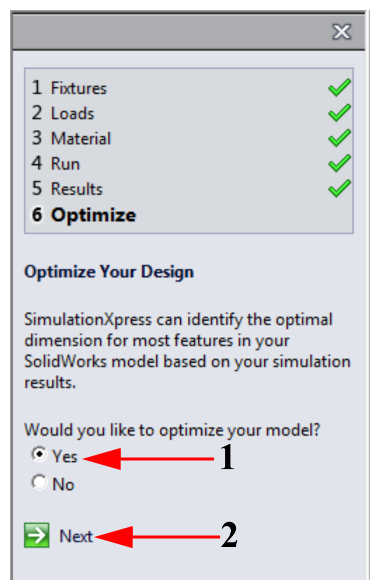
3 Click on **Done viewing results**.



93 We can strengthen the part by decreasing the curve of the arm, so the radius will increase. We improve the model to get a FOS value of 1.

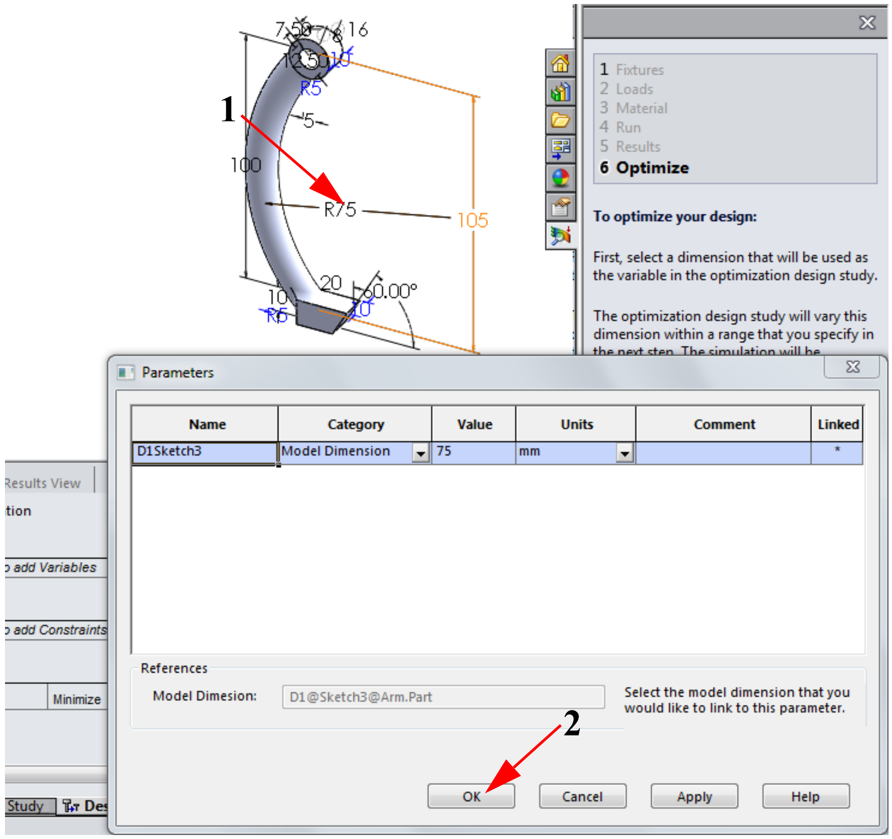
1 Click on **Yes**.

2 Click on **Next**.



94 All dimensions are visible now.

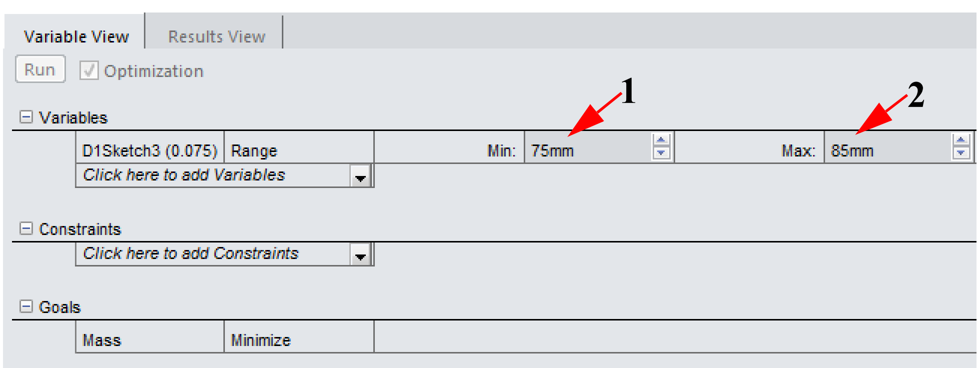
1. Select the dimension of R75 in the model.We will change this radius to optimize the model. In the **Name** field in the **Parameters** window you can see the selected dimension is extracted from sketch3 (the third sketch you made in this part).
2. Click **OK** in the **Parameters** window.



95 For the range of values set the minimum value to 75 mm.

Set the maximum value to 85 mm.

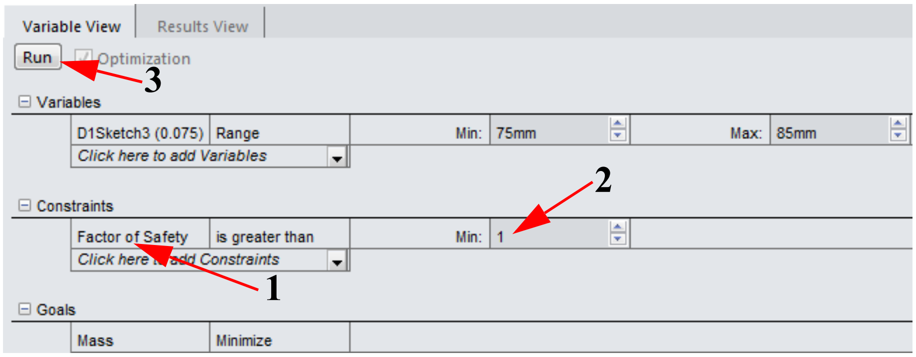
Pay attention: the minimum and maximum values are values that should be within a certain range. When you change a value that leads to an error, **SimulationXpress** cannot use that value.



96 Click the arrow under **Constraints** and select **Factor of Safety**.

Set the minimum value to **1**.

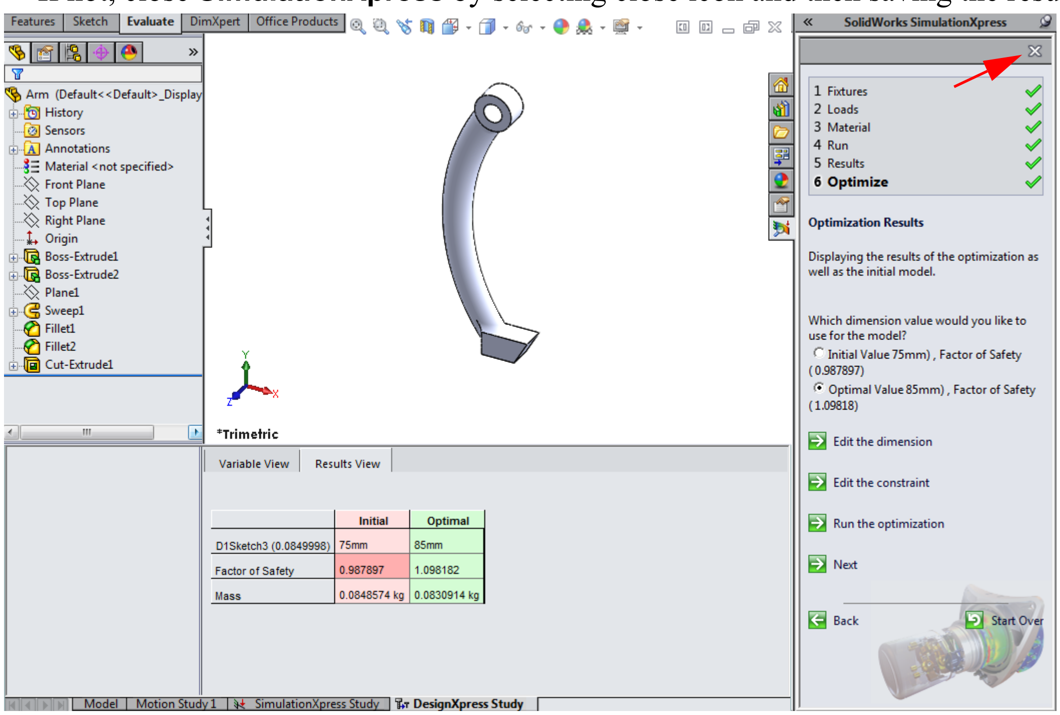
Click **Run**.



97 **SimulationXpress** has calculated a new radius.

If you would like to see more data (e.g.,the distortion), choose **Optimal Value** for the option **Which dimension value would you like to use for the model**. and click **Next**.

If not, close **SimulationXpress** by selecting close icon and then saving the results.

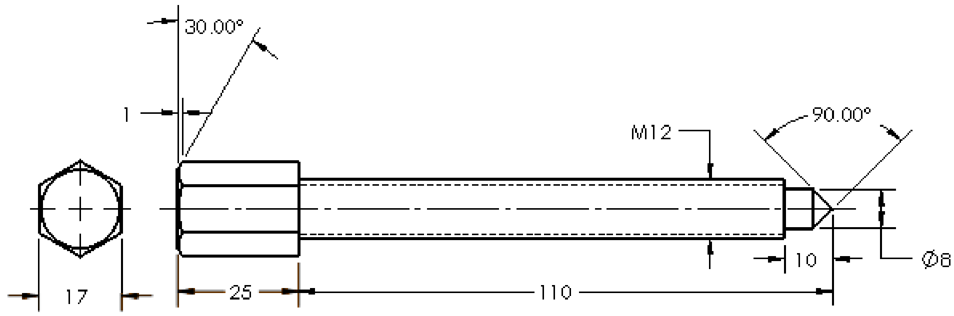


98 Save the changes to the file.

Bolt

Work plan

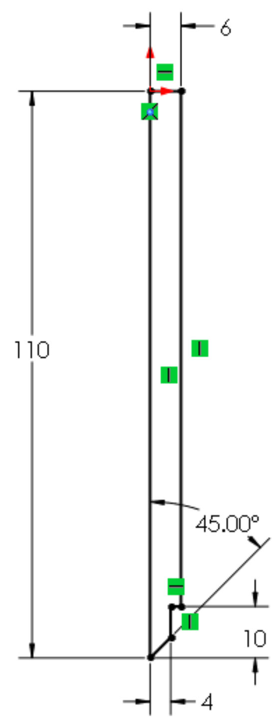
The third and last part of this product is relatively simple: an extended bolt with an M12 thread. In the drawing below you can see how this part looks.



We will create the rod with the thread and the pointed end as a rotation form. The hexagonal part will be added to this as an extrusion.

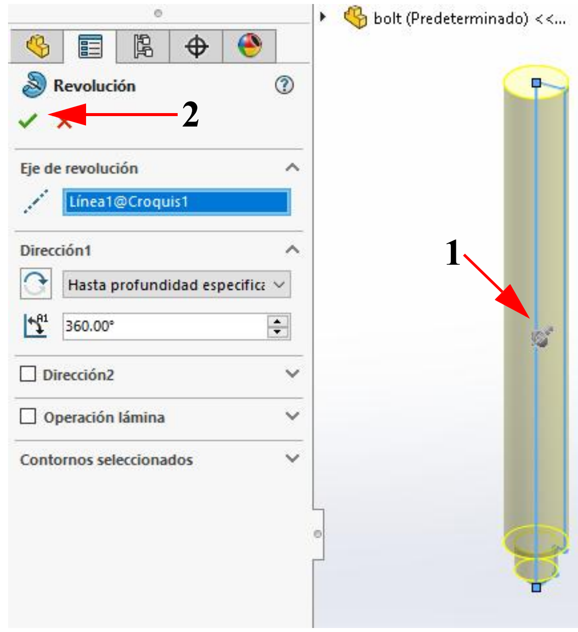
99 Open a new part and set the unit to MMGS.

Make the sketch on the **Front Plane** as you can see on the right.



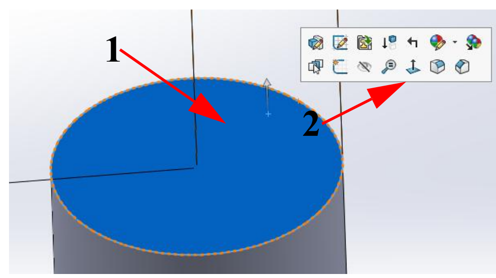
100 Make a **Revolved Boss/Base** from this sketch.

1. Select the line which you want to use as a rotation axis.
2. Click on **OK**.



101 Select the top face to the model. We will make the next sketch on this.

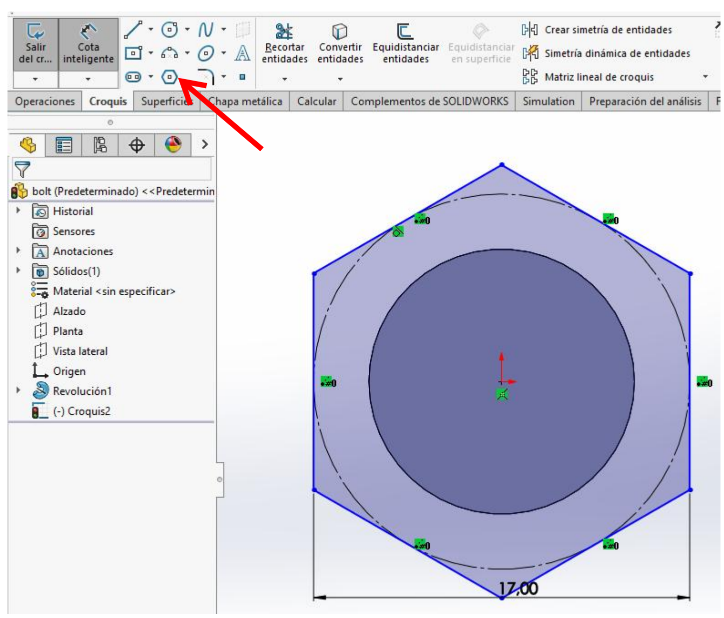
Rotate the model to **Normal To**.



102 Click on **Polygon** in the **CommandManager**.

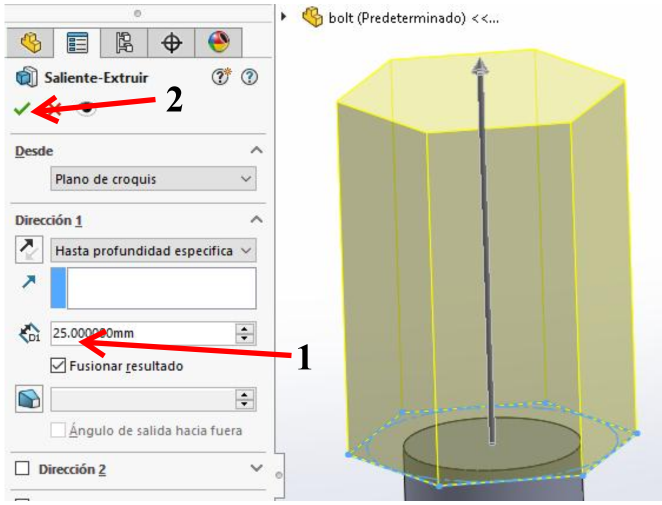
Draw a hexagon, and set the dimension according to the illustration on the right.

Make sure that one of the vertices of the hexagon is vertically aligned directly above the **origin**.



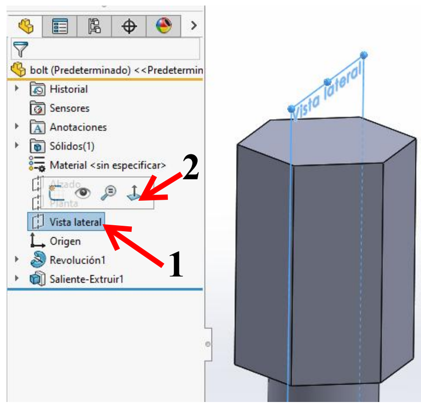
103 Make an extrusion from this sketch.

1. Set the height to **25 mm**.
2. Click on **OK**.



104 We have to create a sloped edge at the top of the hexagon head.

Select the **Right Plane** in **FeatureManager**, and rotate the model **Normal To**.

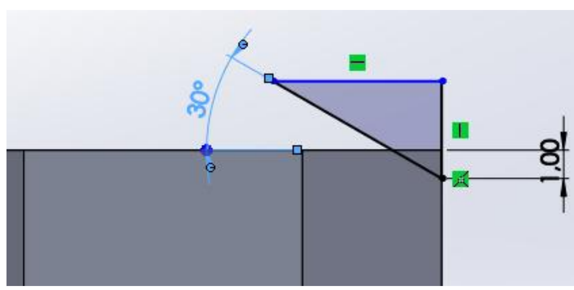


105 Make the sketch as in the illustration:

Draw the centerline from the **origin** vertically upward.

Next, draw a triangle.

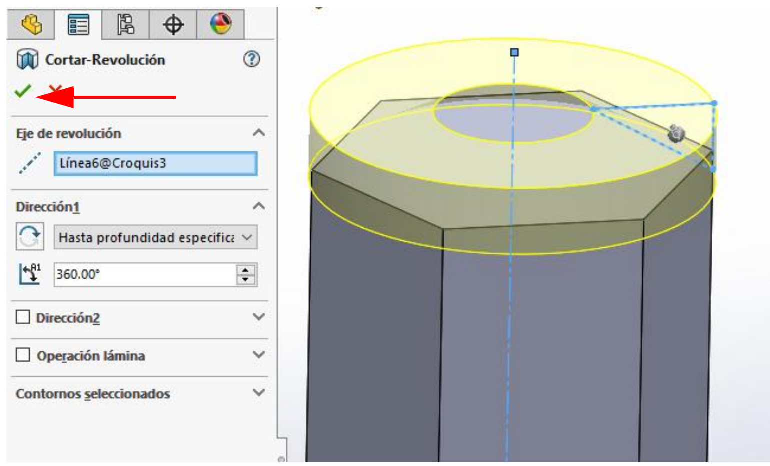
Add two dimensions to finish it.



106 Click on the **Features** tab in the **CommandManager**.

Click on **Revolved Cut**.

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



108 Finally, we will cut threads on bolt.

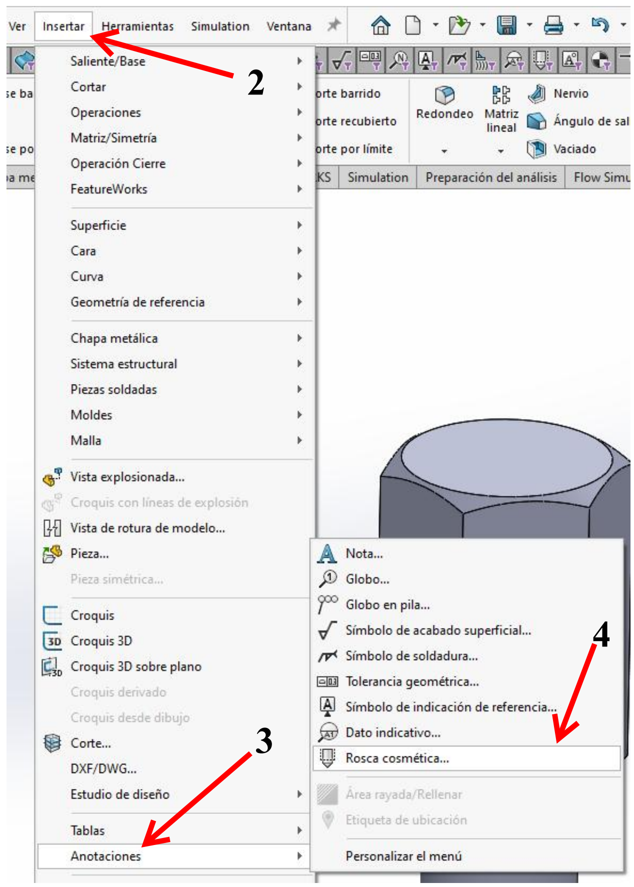
You will find the command for this in **Pull-down menu**:

1 Open the **Pull-down menu**.

2 **Insert**.

3 **Annotations**.

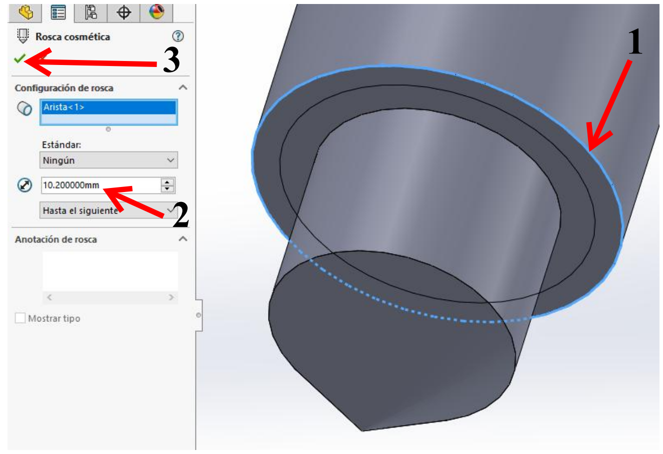
4 **Cosmetic Thread**.



109 Select the edge of the face you want to convert into threads.

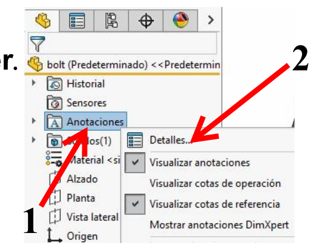
Set the diameter to **10.2 mm**.

Click on **OK**.



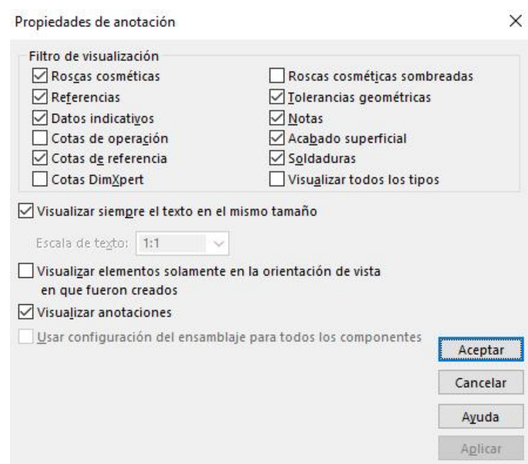
110 To display the thread you can:

1. Right-click on **Annotations** in the **FeatureManager**.
2. Click on **Details**.

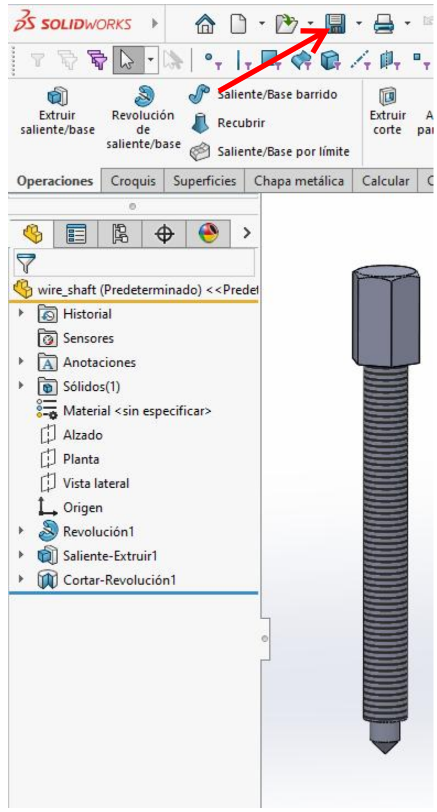


111 Check the option **Shaded cosmetic threads** in the menu that appears.

Click **OK**.



112 This part is also now done. Save it as: wire_shaft.SLDPRT.



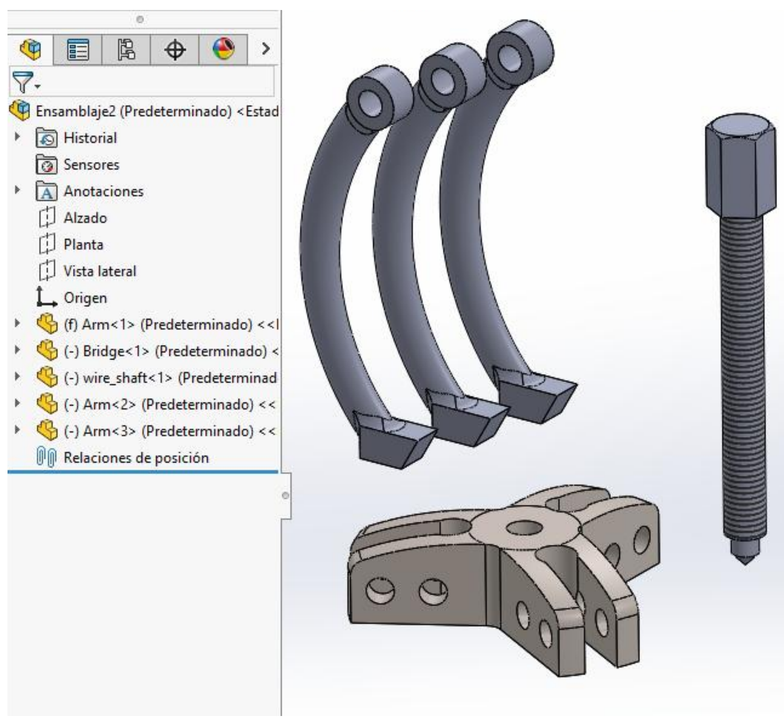
Bearing Puller Assembly

113 We will assemble all the parts to build the bearing puller.

Open a new **assembly**.

Put the bridge in the assembly first.

Next, add the arm three times and add the wire_shaft once. Place them at random positions in the assembly.



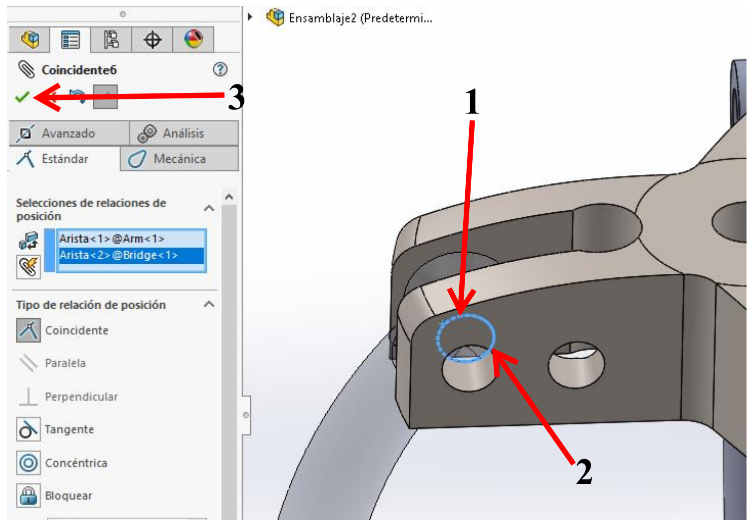
114 First, put the arms in the bridge.

Click on **Mates** in the **CommandManager**.

Select the two edges as illustrated to put the first arm in its place.

Next, set the two others in their positions in the same way.

Pay attention: use the Mate alignment command (**aligned** or **anti-aligned**) to turn the arm around when necessary.

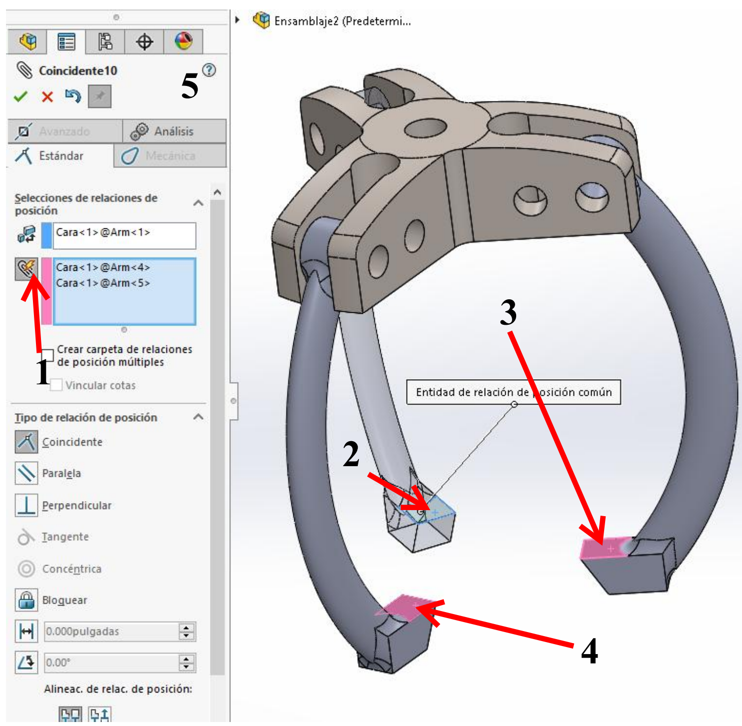


115 To set the arms straight, we will add a few extra mates.

1 Click on **Multiple Mate Mode** in the **PropertyManager**.

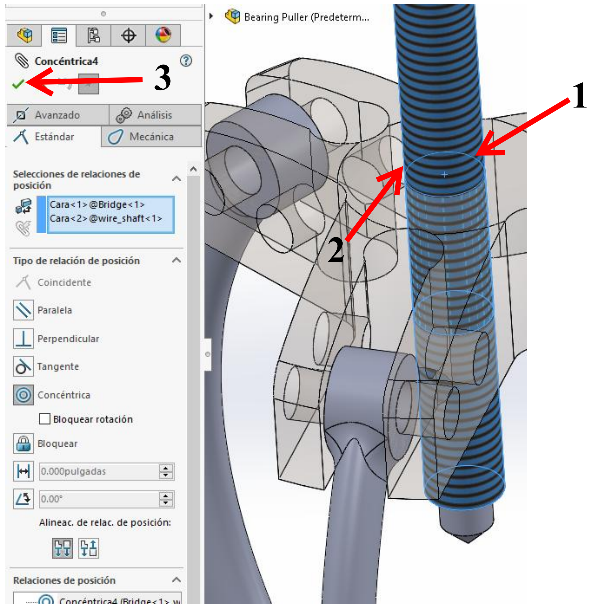
2-4 Select the three top planes at the end of each arm, one by one,

5 Click on **OK**.



116 Finally, we have to put the bolt in position. Create a mate between the surfaces as illustrated on the right.

How far to insert the shaft in the bridge is up to you.



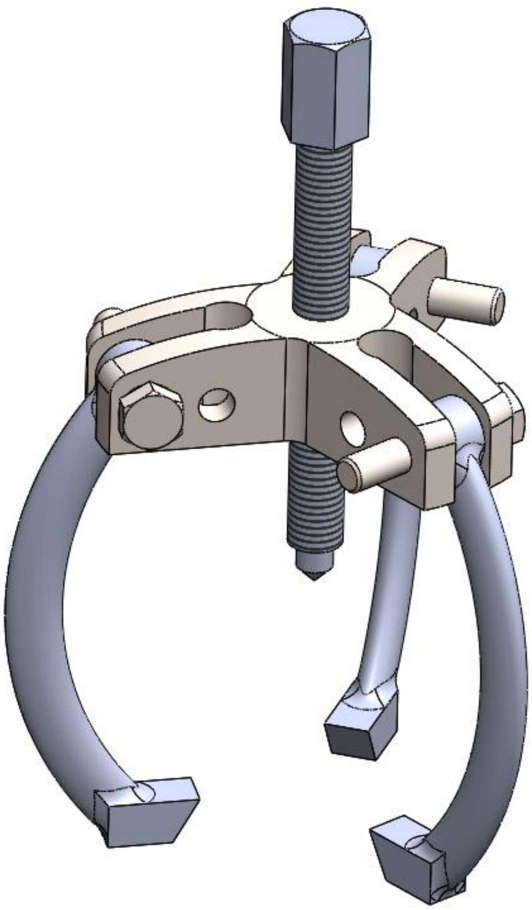
117 Add bolts, washers, and nuts to the assembly from the **Toolbox**.

Find the bolts in the **Toolbox** by looking for **DIN > Bolts and Screws > Hex Bolts and Screws**.

Select **Hex Screw Grade AB - DIN EN 24014**.

Set the size: **M8** with a length of **40**.

Add this bolt to the assembly three times.

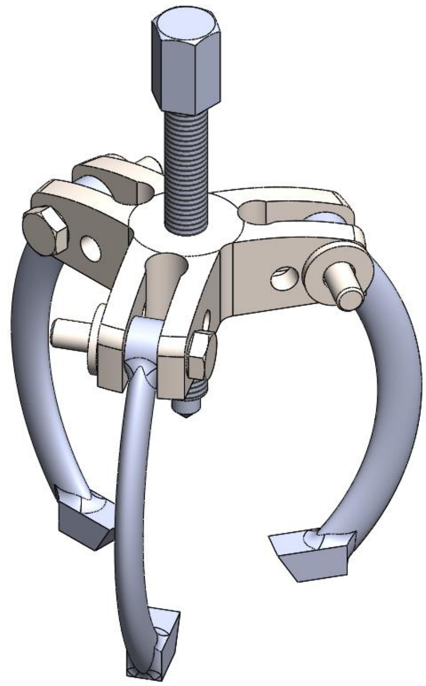


118 For the washers, find **DIN > Washers > Plain Washers** in the **Toolbox**.

Select **Washer - Grade A - DIN125 Part1**.

Select size: **8.4** (for thread **M8**).

Add this washer to the assembly three times too.



119 Finally, we need to place the nuts. Use **DIN > Nuts > Hex Nuts** from the **Toolbox**.

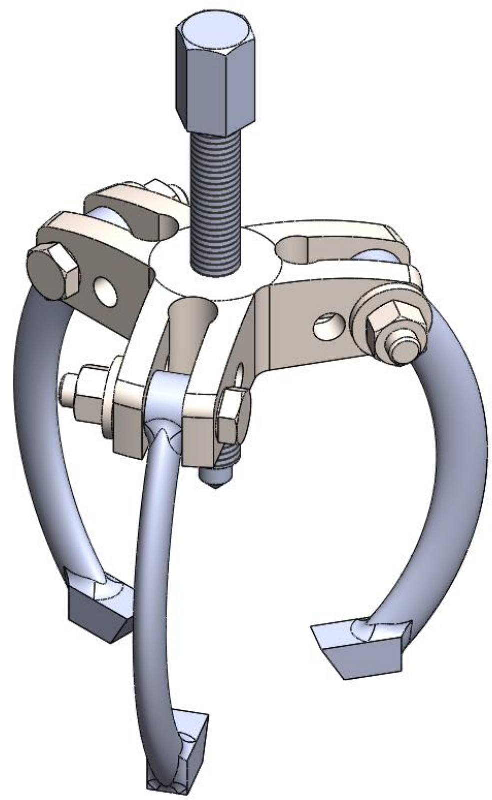
Select **Hex Nut Grade C- DIN EN 24034**.

Select size: **M8**.

Again, add this nut three times to the assembly.

120 We have finished the assembly.

Save the file as Bearing_puller.SLDASM.



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

The most important item you have seen in this tutorial is how to use **SimulationXpress** to find out if a model is strong enough to perform its designed purpose.

A number of other new items include:

- Creating a more complex model (the bridge) and using the **circular pattern** command.
- Using an **Axis** and learning another way to define an auxiliary plane.
- Creating a model using a real material.
- Determining the weight and volume from a part of from the model.
- Using the **sweep** feature.
- Learning it is very convenient to create outer parts first and building up the middle sections later, as in the modeling of the arms.
- Working with **Cosmetic Threads**.

After finishing this tutorial, you have learned a lot about SolidWorks. You probably understand much more about using the program now and are building real expertise in the use of SolidWorks. You can continue to grow your SolidWorks skills and learn even more by discovering the purpose of additional functions yourself. If you get stranded at any point, use the Help functions or refer to a book on SolidWorks where all of the functions are explained.