

Lesson: Lightweighting of a Robot Gripper Arm

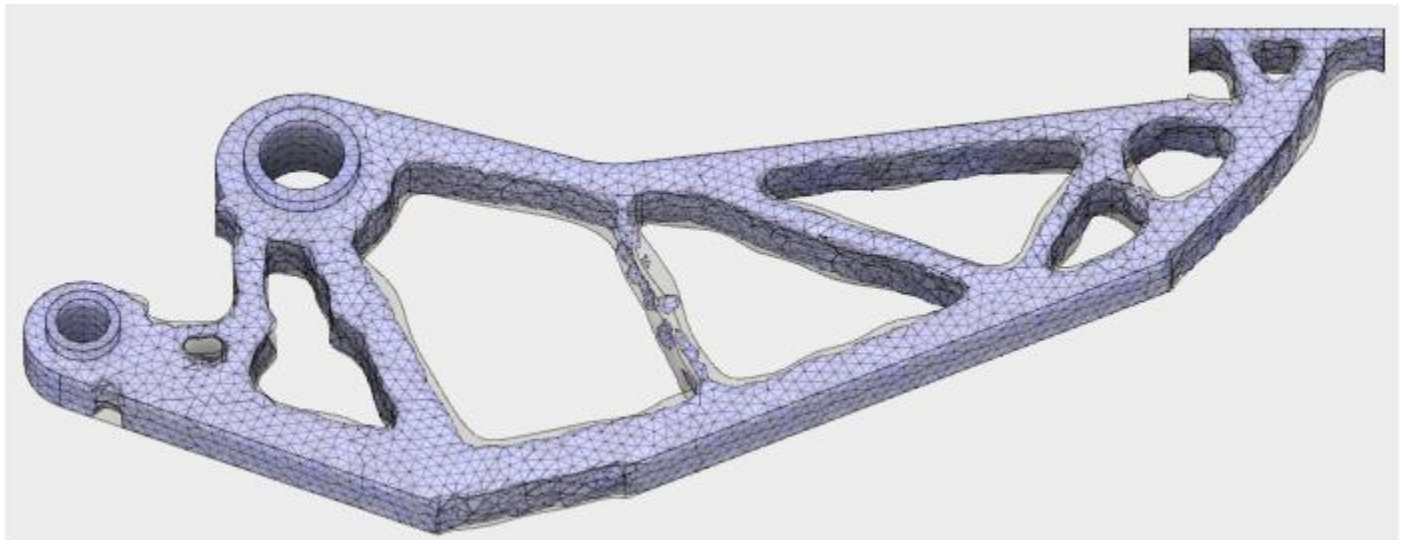
Note: This functionality is only available in Fusion 360 Ultimate.

In this exercise we'll perform a Shape Optimization study to reduce the weight of a robot gripper arm.

Objectives:

- Reduce the weight to approximately 40% of the original design.
- Maintaining a Safety Factor of at least 2.0 for the applied load.

The gripper arm, made of steel, is designed to withstand compressive loads on the gripping surface.



Open the Model

Open the geometry and save a copy.

Open the Gripper Arm Model

In the **Samples** section of your Data Panel, browse to:

Basic Training ➤ **11 - Simulation** ➤ **GripperArm**



1. Click the **Show Data Panel** icon (3x3 grid) at the top of the screen, if the Data Panel is not currently shown. The Data Panel appears at the left side of the program window.
2. The top level (home view) of the Data Panel is divided into two subsections - PROJECTS and SAMPLES. Scroll to the bottom of the PROJECTS list if necessary to see the SAMPLES list.
3. Locate the **Basic Training** entry under SAMPLES and double-click it. The Data Panel now displays a list of the folders containing the training lesson models.
4. Click the **11 - Simulation** folder.
5. Select the **GripperArm** model.

Save the Model

When opening a sample model for the first time the active workspace in Fusion is the Modeling Workspace. The model is in Read only and needs to be saved to a personal project.

1. Click **File** ➤ **Save As**.
2. Optionally, create a Project to store your training models.
 - a. Click **New Project**
 - b. Specify the project name
 - c. Press **Enter**.
3. Optionally, create a folder within the project to store your training models.
 - a. Click **New Folder**
 - b. Specify a folder name
 - c. Press **Enter**.
 - d. Double-click the new folder to make it the current file saving location.
4. Click **Save**.

Create Study

Create a Shape Optimization study.

Access the SIMULATION Workspace and create a Study

1. Click the workspace selection in the top left corner.
2. Select the **SIMULATION** workspace from the drop-down menu.

Note: Notice that the *New Study* dialog opens automatically.

3. Select the **Shape Optimization** study type.
4. Click **OK**.

Choose the Units for the Simulation

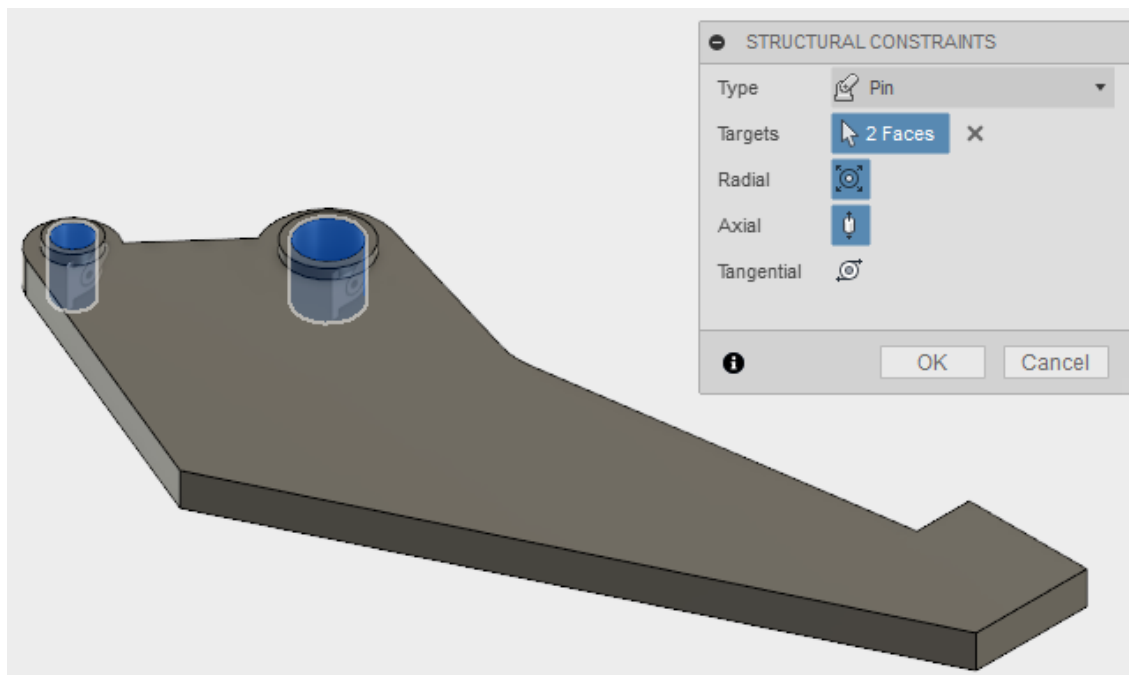
You may have set different default units than are initially defined when Fusion 360 is installed. Also, the simulation units are independent from the units specified in the MODEL workspace. So, the units system can change when you switch to the SIMULATION workspace. Therefore, we now verify that the proper units are specified to be consistent with this tutorial.

1. Click the **Edit** icon () that appears while the cursor is pointing at the **Units** node of the browser.
2. Choose **Metric (SI)** from the **Default Unit Set** drop-down list.
3. Click **OK**.

Apply Constraints

The gripper arm is held in place by two pins. As a result, we will use pin constraints to constrain the model for simulation.

1. Click **CONSTRAINT** ➤  **Structural Constraint**.
2. Switch the Type to **Pin**.
3. Select the surface of each bolt hole as shown in the image below.
4. Make sure the **Radial** and **Axial** directions are constrained.
5. Click **OK**.

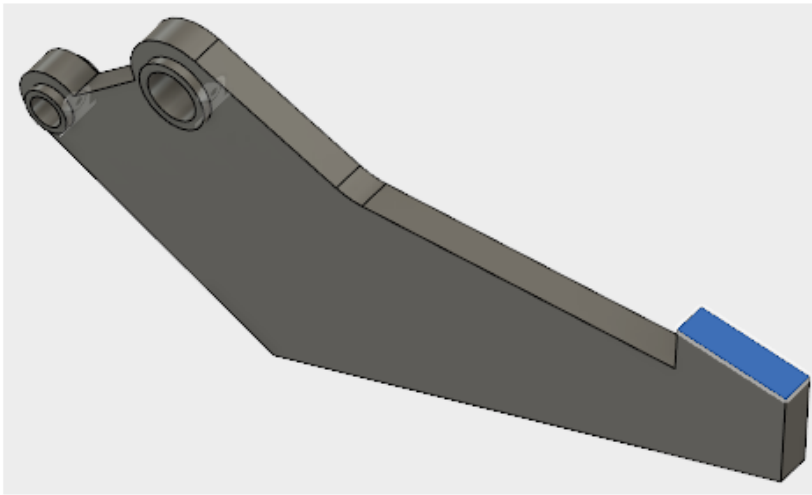


Apply Loads

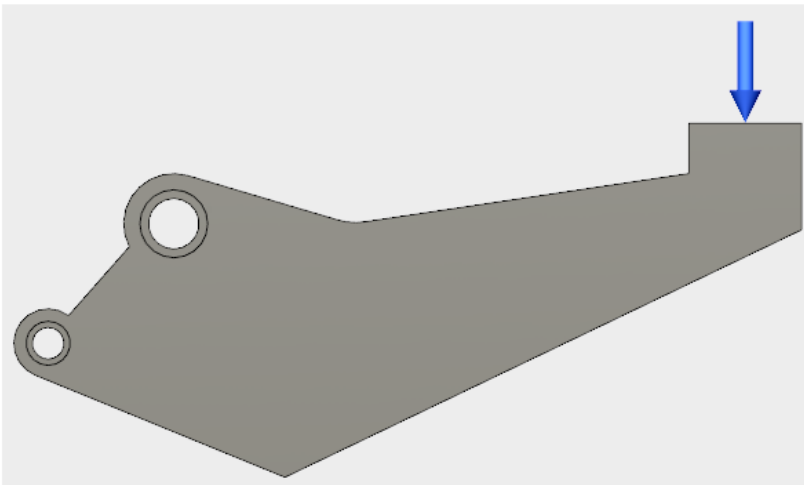
The gripper arm, as the name suggests, is used by the robot with one or more other arms to grab objects. As a result, we want to generate a shape that is designed to handle compressive loads on the gripping surface.



1. Click **LOADS** ➤ **Structural Loads**.
2. Set the Type to **Force**.
3. Select the gripping surface as shown in the image below.



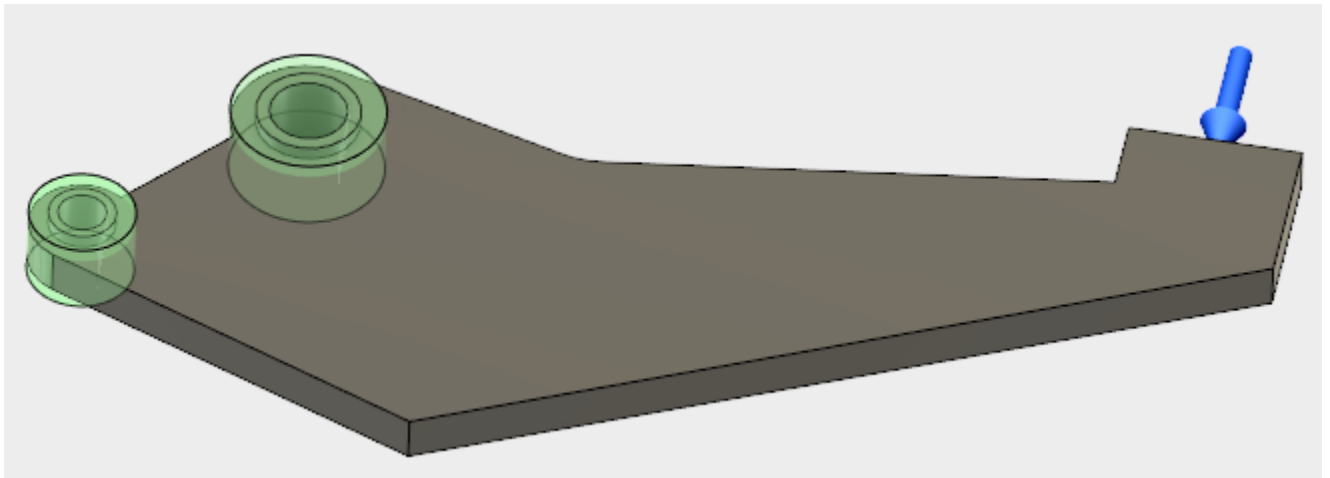
4. Enter a load of **500 N**.
5. Make sure the load is acting in compression (pointing toward the surface) and click **OK**.



Preserve Regions

Since the gripper arm is held in place by the two pins, we want to be sure to preserve the material surrounding the bolt holes, so the pin can function as expected.

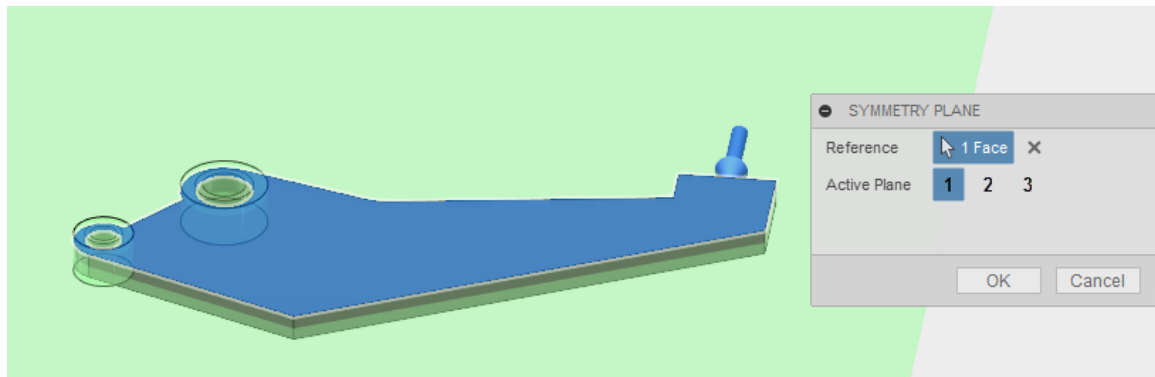
1. Click **SHAPE OPTIMIZATION** ➤  **Preserve Region**.
2. Select the surface of the large bolt hole.
3. Adjust the diameter to **8 mm**.
4. Repeat steps 1 through 3 for the small bolt hole, adjusting the diameter to **5.5 mm**.
5. Click **OK**.



Create Symmetry Plane

The gripper arm is symmetric in the thickness direction. As a result, we can add a symmetry plane through the thickness so the optimized shape is also symmetric through the thickness.

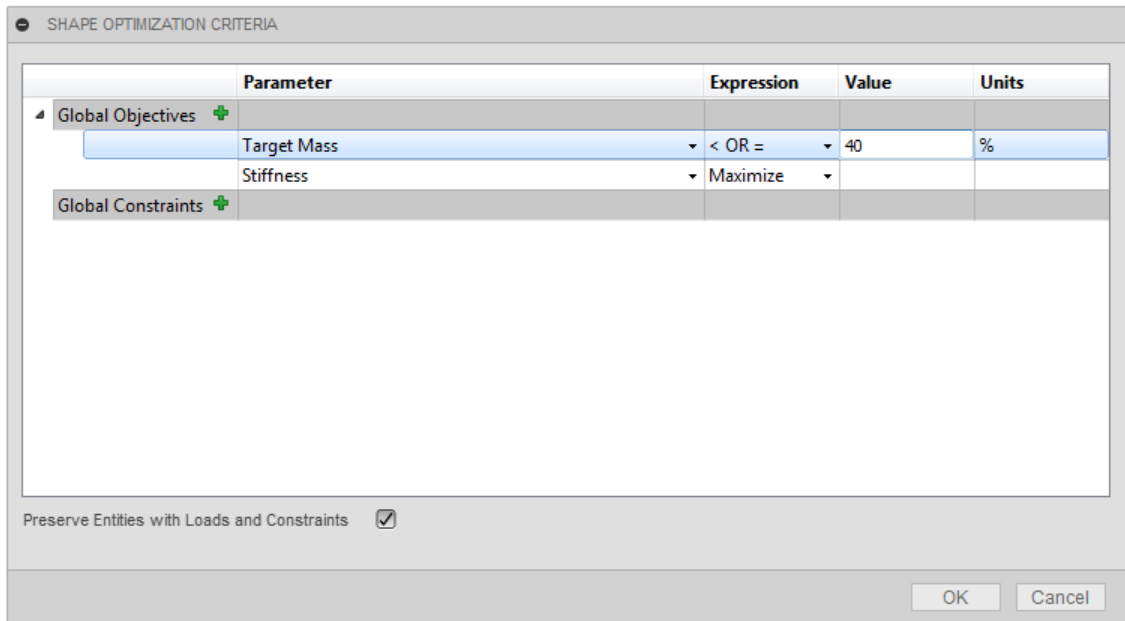
1. Click **SHAPE OPTIMIZATION** ➤  **Symmetry Plane**.
2. Select the top surface of the gripper arm.
3. Turn on **Active Plane 1** and click **OK**.



Shape Optimization Criteria

The objective of this problem is to produce a final geometry that is approximately 40% of the original mass. To help achieve this goal accurately, we'll reduce the default mesh size.

1. Click **SHAPE OPTIMIZATION** ➤  **Shape Optimization Criteria**.
2. Set the Target Mass to **40%**.



The dialog box titled "SHAPE OPTIMIZATION CRITERIA" contains a table with the following data:

	Parameter	Expression	Value	Units
Global Objectives +	Target Mass	< OR =	40	%
	Stiffness	Maximize		
Global Constraints +				

At the bottom, there is a checkbox labeled "Preserve Entities with Loads and Constraints" which is checked. The "OK" and "Cancel" buttons are at the bottom right.

3. Click **OK**.
4. Click **MANAGE** ➤  **Settings**.
5. Switch to the **Mesh** panel.
6. Select the **Absolute Size** radio button.
7. Enter **1 mm** for the mesh size.
8. Click **OK**.

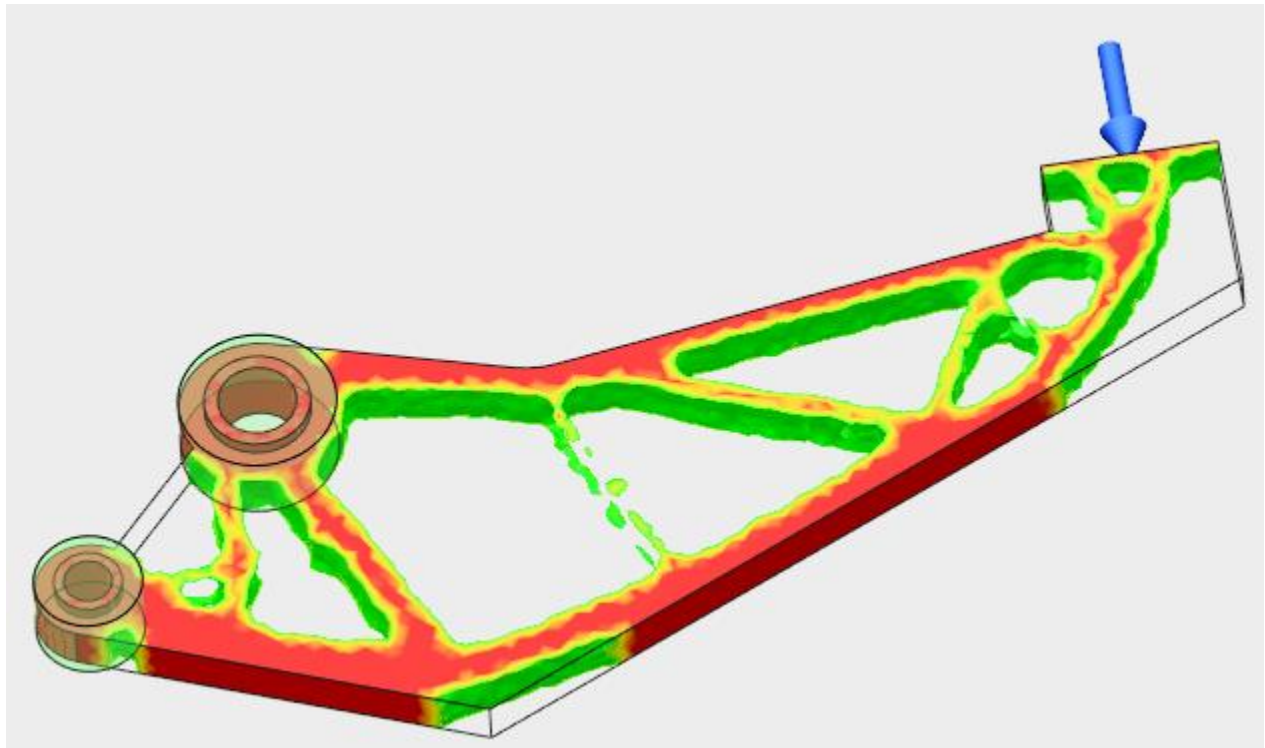
Solve the Study

Now we are ready to solve the study and produce an optimized shape.

1. Click **SOLVE** >  **Solve**.
2. Make sure **On Cloud** is selected and click **Solve 1 Study**.

Note: Shape Optimization studies are only available for cloud solves.

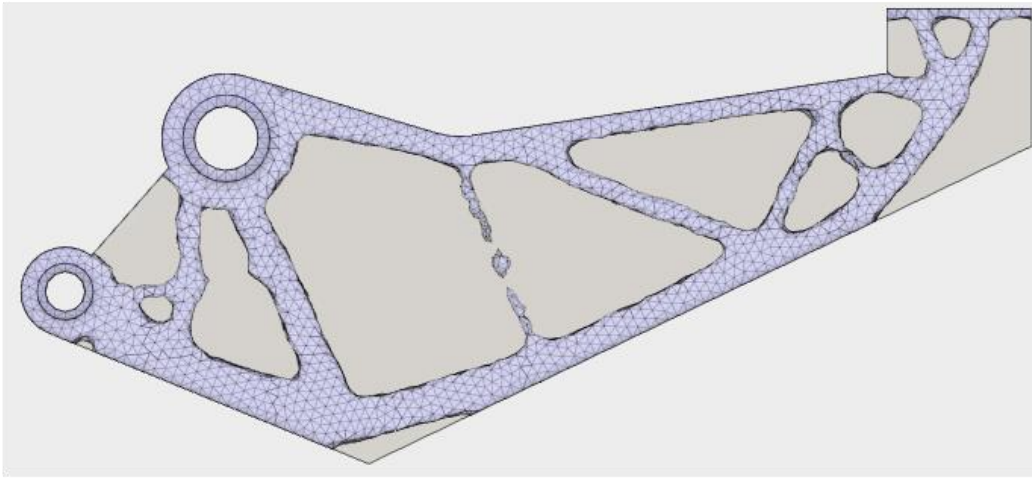
3. After the study is complete, review the results for the Load Path Criticality. Notice how the regions deemed less critical have been removed from the optimized shape.



Promote Mesh & Modify Original Geometry

Now that the generated shape for the gripper arm has been produced, we can export the mesh to the Modeling workspace to use as a template for making modifications to the original geometry.

1. Click **RESULTS** >  **Promote**.

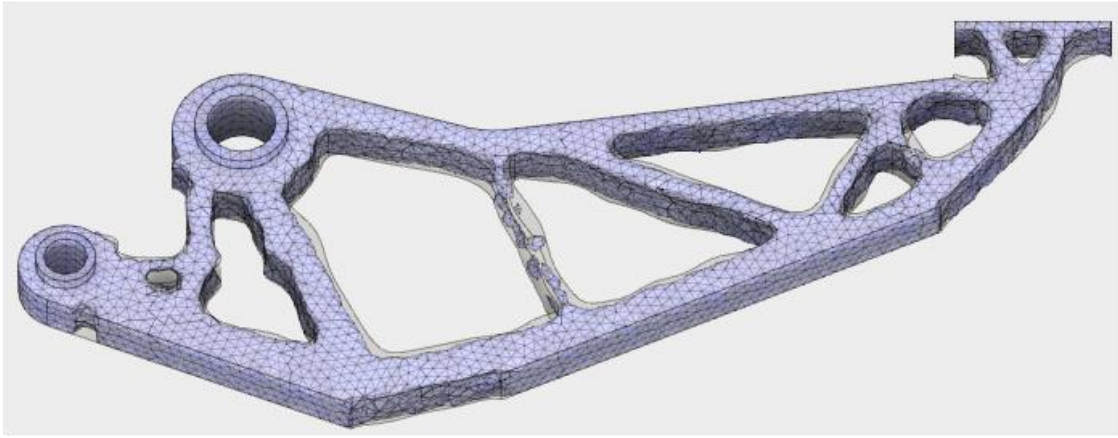


2. Click **SKETCH** >  **Create Sketch**.
3. Select the top surface of the gripper arm as the sketch plane.
4. Use the spline command (**SKETCH** >  **Spline**) to create closed sketch objects around the material to be removed. Click along the cutout regions drawing splines point-to-point. Clicking on the starting point closes the loop after which you can start on a new closed spline.

Tip: You do not need to follow the meshed body contour too closely. In fact, it is advantageous to smooth out the roughness of the raw shape optimization output resulting from the irregular element shapes and locations. Optionally, you may wish to draw straight lines and arcs instead of splines to make the cutouts look more like a typical manufactured part. However, for this example, we are following the meshed body fairly closely to see what the stress level in the raw optimized shape would be.

5. Click **Stop Sketch** after all of the closed splines have been made.
6. Click **CREATE** >  **Extrude**.
7. In the **Operation** drop-down list of the EXTRUDE dialog, choose  **Cut**.
8. Select all of the enclosed sketch areas by clicking inside of the closed splines.

9. Click and drag the extrusion direction/distance manipulator arrow. Pull the cut through the full thickness of the body.
10. Click **OK** to complete the operation.



Static Stress Study

The objective of the shape optimization study was to reduce the weight of the part to approximately 40% of the original weight and to maximize the stiffness. A stress analysis is required to verify that the stress and displacement of the modified part shape is acceptable.

Now that we have modified the geometry to represent the generated shape, we are ready to perform a Static Stress study. Recall that the generated shape must have a safety factor of at least 2.0.

1. Return to the **SIMULATION** workspace.
2. Right-click **Study 1 - Shape Optimization** in the browser and select **Clone Study**.
3. Click **MANAGE** ➤  **Settings** and switch to the **Static Stress** study type.
4. Click **OK**.
5. Click **SOLVE** ➤  **Solve**. You can choose to solve the static stress study either locally or on the cloud.
6. Ensure that Study 2 is selected and click **Solve 1 Study**.
7. Review the Safety Factor result to verify that the modified part meets the design requirement. Your results will likely vary somewhat from the following image, depending on the shape of the closed splines that you sketched.

