

Laboratory 7-8

Topological Optimization of Vehicle Chassis Components

Scope:

The paper presents a detailed description of the steps that must be followed for CAD modeling and topological optimization of a support from the structural component of a vehicle chassis.

Introductory Concepts

The vehicle chassis is the part that forms the basic structure of the vehicle, intended for attaching the engine, transmission, suspension, axles, body, and any other assemblies (main and/or auxiliary).

A vehicle chassis must meet the following minimum conditions:

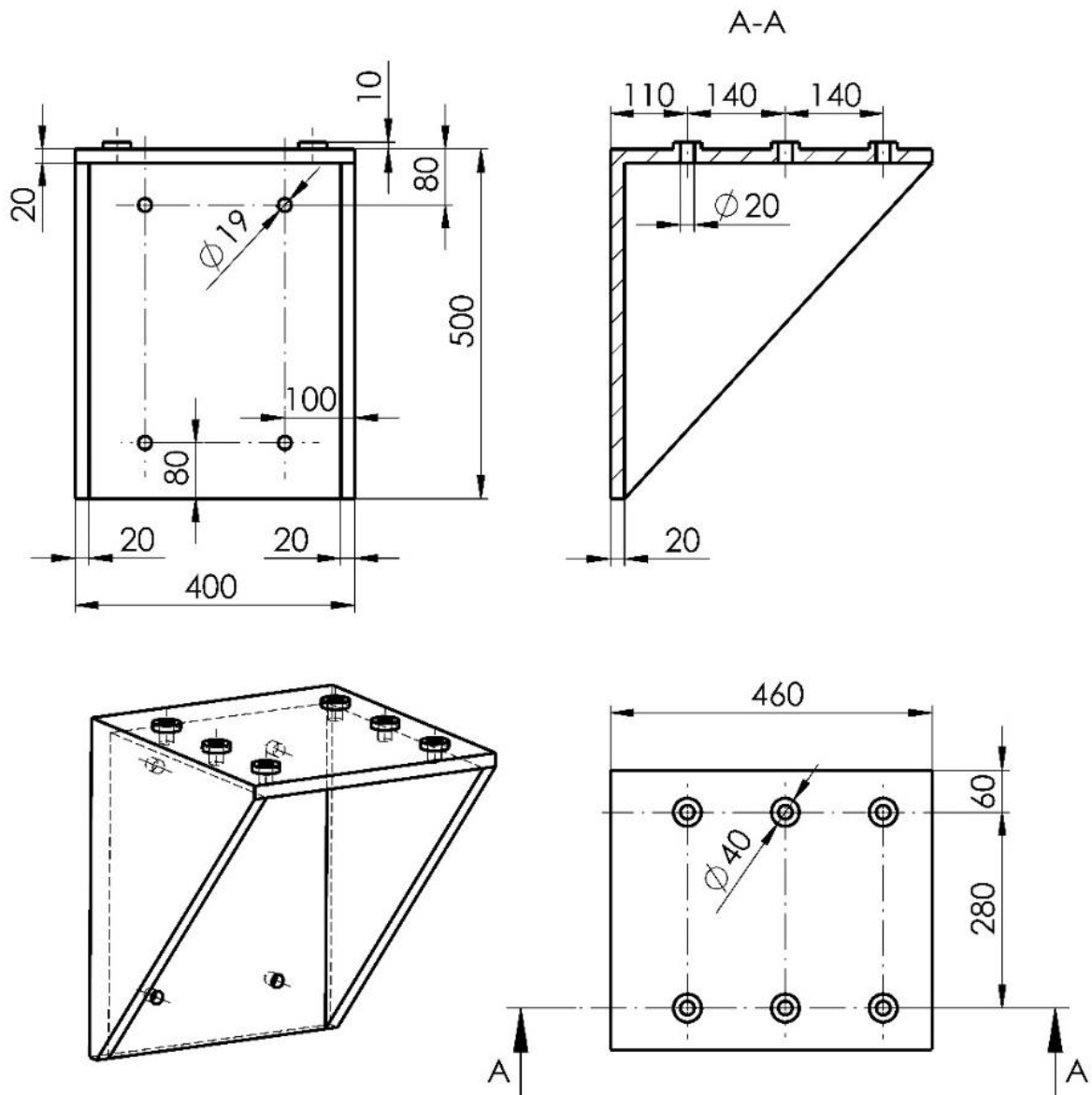
- a sufficiently good weight-to-stiffness ratio for the intended use of the frame support.
- simple construction.
- easy mounting and fixing of the body and auxiliary assemblies.
- reduced manufacturing costs.
- low maintenance.
- to allow the placement of vehicle components to achieve a center of gravity position as close to the ground as possible.

The installation of auxiliary systems and components is done using support, designed and built according to their characteristics. The construction of a vehicle chassis is primarily done using rolled metal profiles (U, H, or I shape) and contributes significantly to the total weight of the load-bearing structure (chassis + body) of a vehicle. For this reason, the need for topological optimization of structures, elements, and components in the construction of a vehicle chassis is of direct importance in relation to reducing/optimizing weight (without losing structural properties), which influences the vehicle's fuel consumption and pollutant emissions. For this purpose, various methods of optimizing the geometric shape of a part are used (topological optimization), with the aim of reducing the mass (weight) of that part, under the conditions imposed by maintaining the initial mechanical strength capabilities. The structural and functional form of a vehicle's frame is determined by the arrangement of the vehicle's main components: the engine, the type of suspension, the position of the drive axle, the shape of the body, etc.

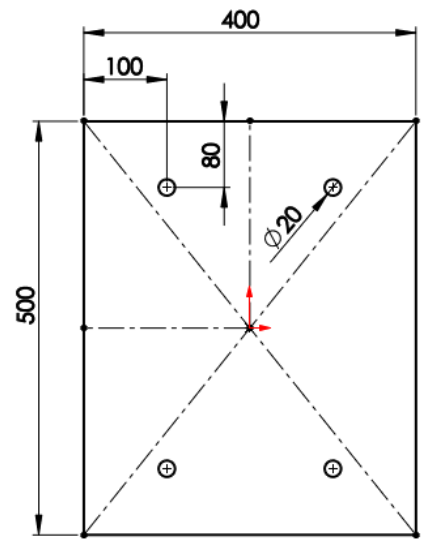
A special case is busses with a distinct frame, which have a curved shape of the longitudinal members above the two axles. This allows for a low floor for buses, making it easy for passengers to board. It also lowers the position of the center of gravity, which has a direct influence on the bus's stability.

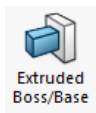
Example

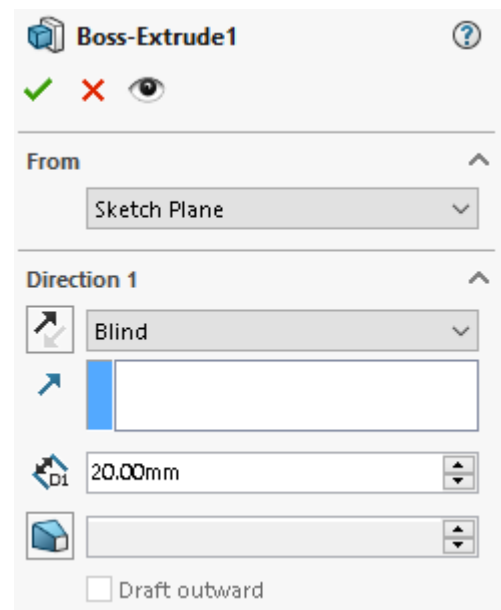
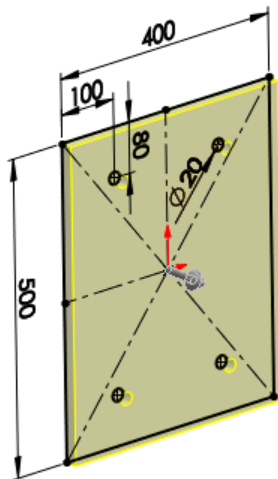
The algorithm of the steps to be followed for CAD modeling and topological optimization of support from the structural component of a vehicle chassis is presented below, based on the model proposed in the following sketch.



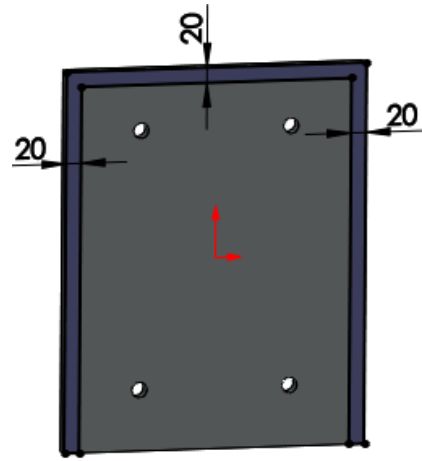
- The initial sketch of the part model is drawn on the **Right Plane**  using the **Center Rectangle**, **Centerline** , **Circle** , and **Smart Dimension**  commands from the **Sketch**  toolbar. The rectangle will be drawn with its origin at the intersection of the diagonals. This sketch will have the shape and dimensions shown in the adjacent figure.

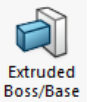


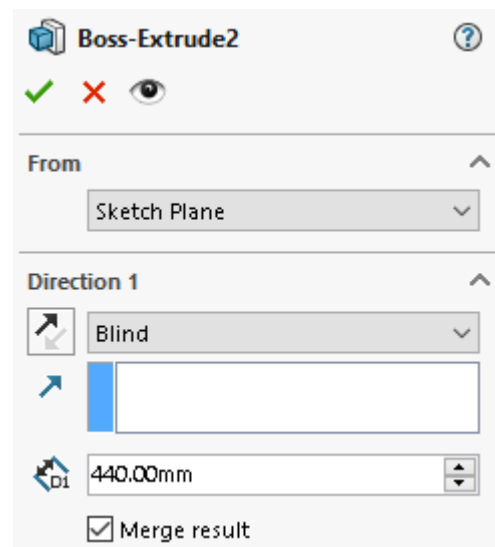
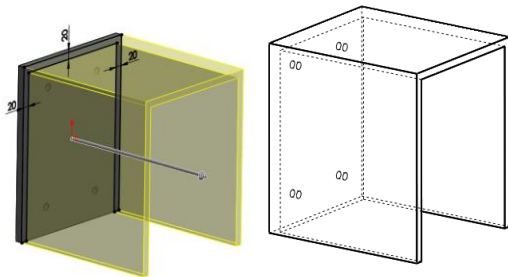
- From the **Features** toolbar, select the **Extruded Boss/Base**  command. The sketch is extruded to a distance of 20 mm, as shown in the adjacent figure.



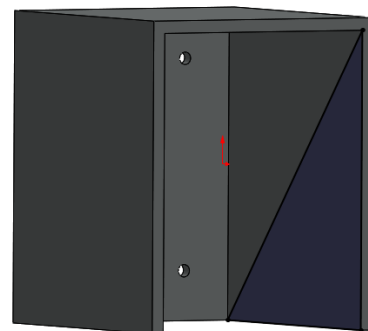
3. On the surface of the extruded model, a sketch is introduced using the **Line**  command from the Sketch toolbar . The sketch closes at 20 mm, as can be seen in the adjacent figure.



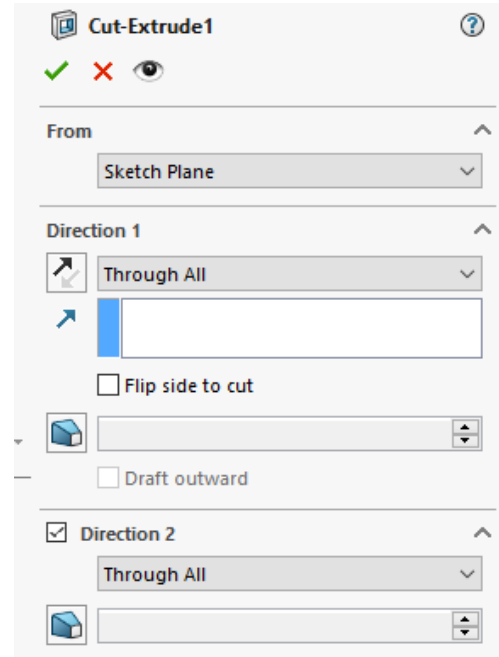
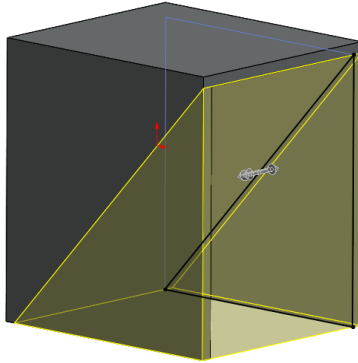
4. The sketch is extruded to a distance of 440 mm using the **Extruded Boss/Base**  command from the **Features** toolbar.



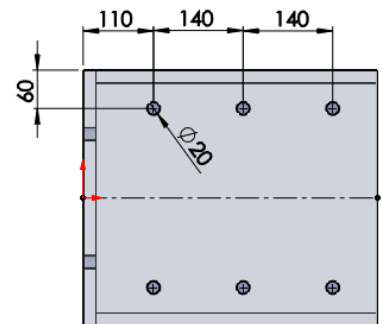
5. On the side surface of the model, a triangular sketch is introduced using the **Line**  command. This outline is drawn with the intention of removing the material inside it. din interiorul ei.



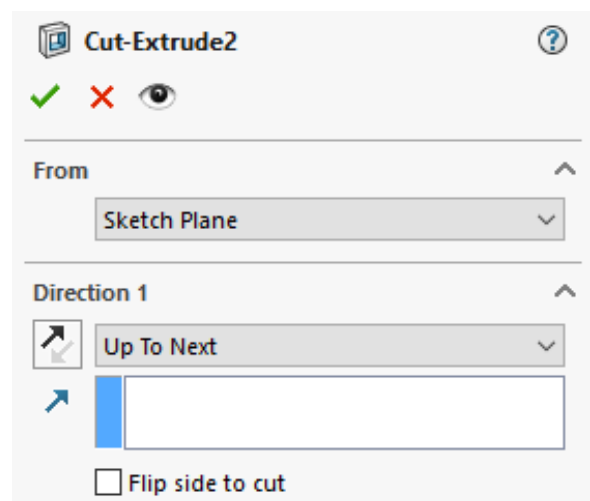
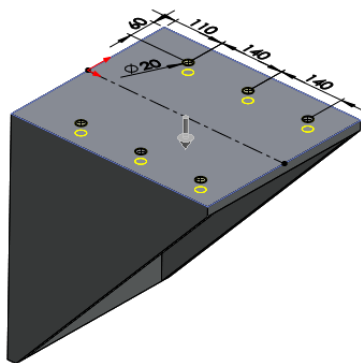
6. From the **Features** toolbar, select the **Extruded Cut** command. The resulting sketch will completely remove the material from the inside by selecting the **"Through All"** option from **Direction 1**.



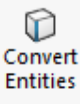
7. A sketch consisting of six circles is introduced at the top of the model. In the first step, draw three circles using the **Circle**  command, each with a diameter of 20 mm, and dimension them according to the adjacent figure, using the command  **Mirror Entities**. These circles are mirrored across the **centerline** , drawn in the middle of the surface.



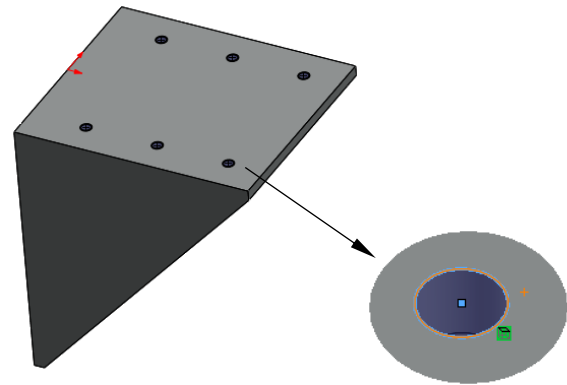
8. To create holes in the **Features** toolbar, select the **Extruded-Cut** command. The previously drawn sketch will remove the material inside the circles by selecting the **"Up To Next"** option from **Direction 1**.



9. To create the bosses around the mounting holes, a new sketch is introduced on the top

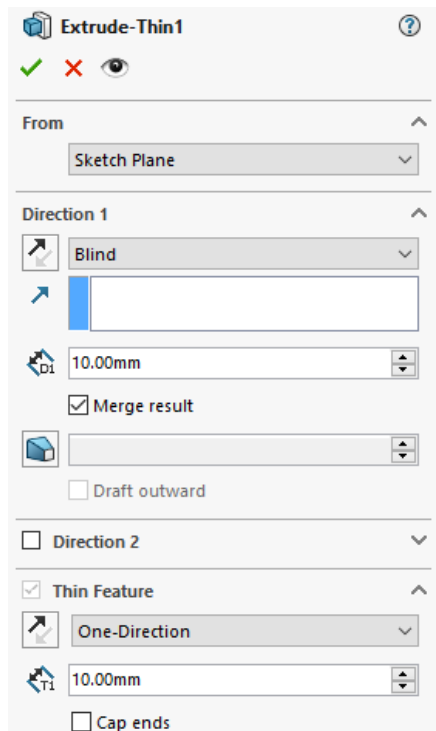
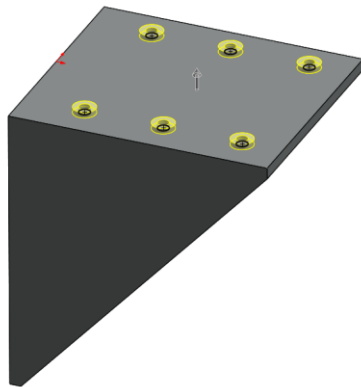
surface of the model. The  command is selected from the  toolbar. Select the circles that define the six holes and confirm .

The figure below shows how the hole edges are converted into a sketch.



10. From the **Features** toolbar, select the **Extruded**

 **Boss/Base** command. The sketch is extruded to 10 mm, applying the **Thin Feature** option as shown in the adjacent figure.

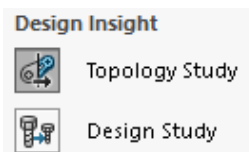


Performing the Topological Optimization Process

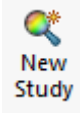
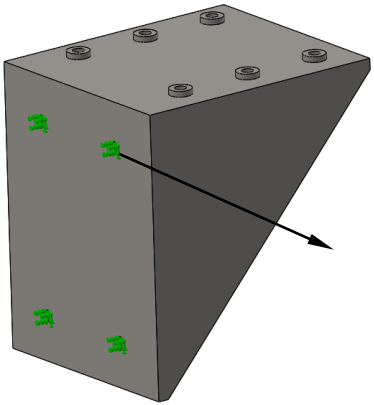
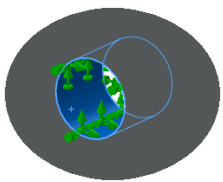
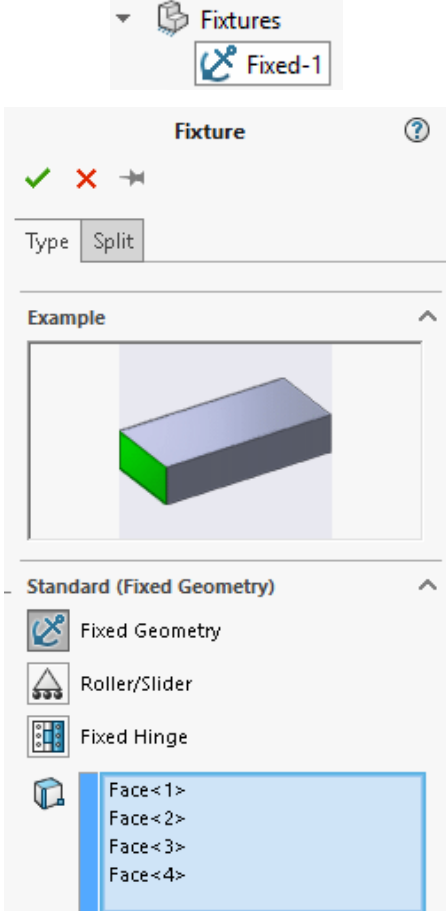
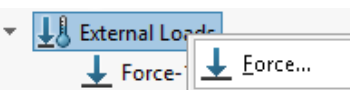
1. From the Tools tab in **Solidworks Add-**

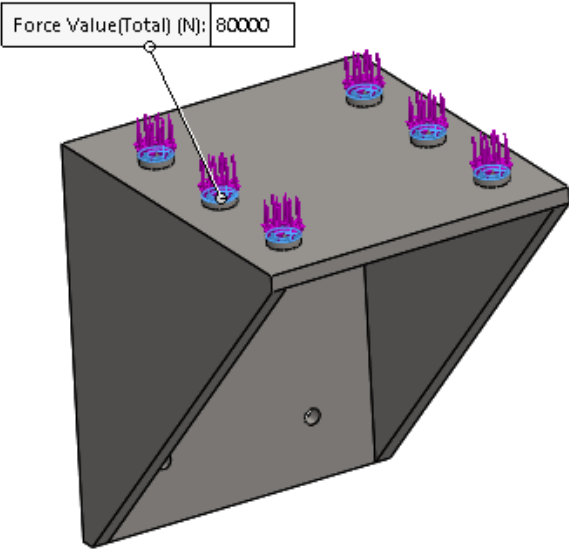
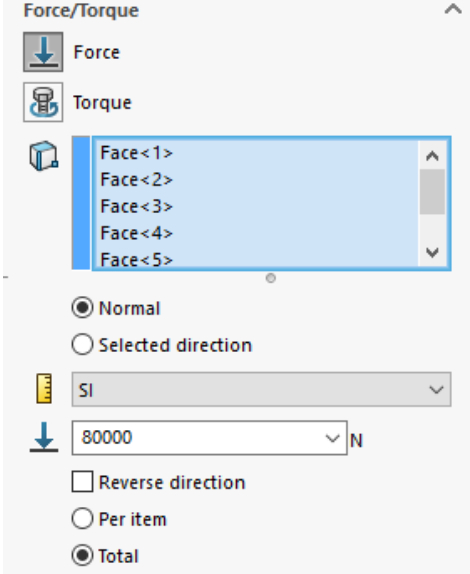
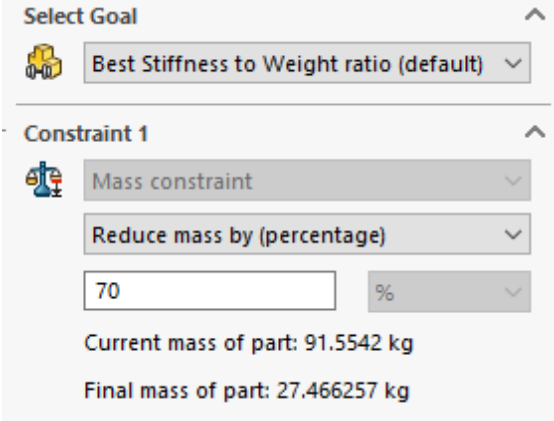
Ins... select , activating the finite element simulation module. A new

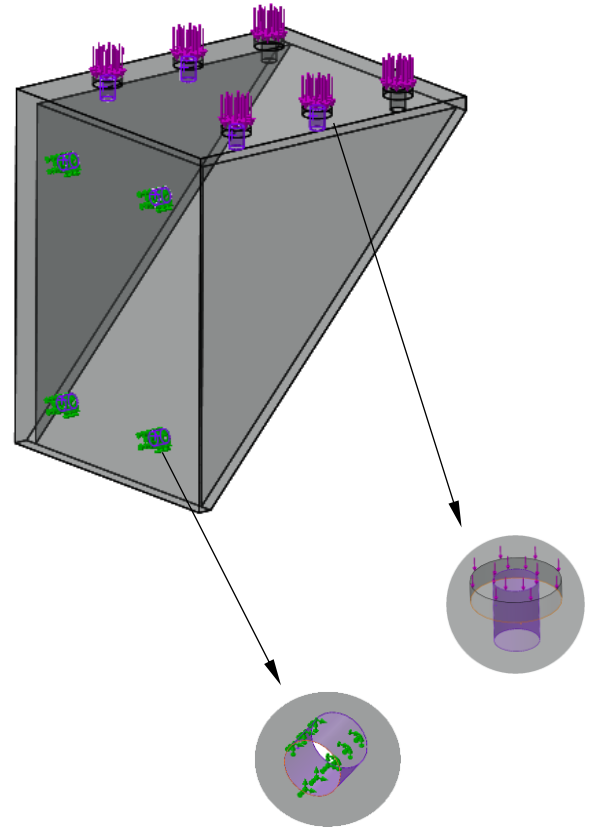
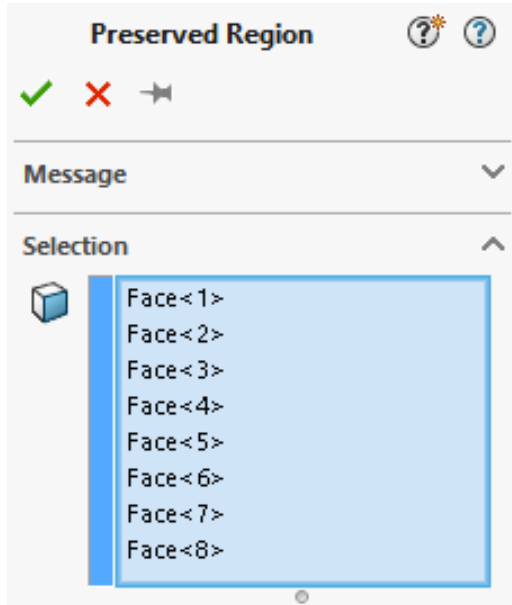
From the **Design Insight** section, a topology optimization study is selected.



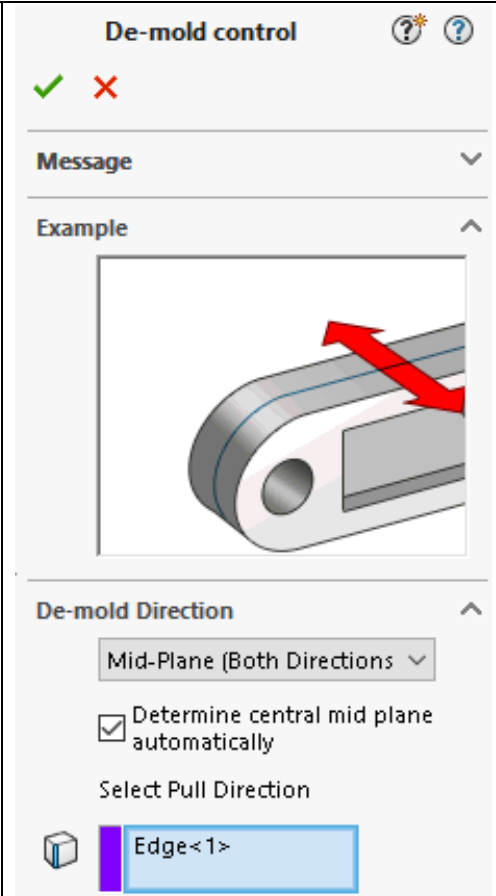
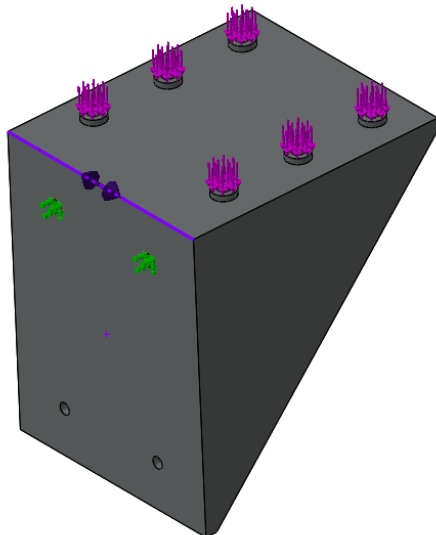
Then confirm the selection with .

study is being conducted by clicking the  button.	
2. Material is assigned to the model by selecting Apply/Edit Material... from the Feature Manager Design Tree's material library, Apply/ Edit Material...	 3. The material is selected: Plain Carbon Steel if the command selection is confirmed with  .
4. The model is secured in the four holes on the side. From the FeatureManager Design Tree, right-click the Fixtures command and select the command from the open menu  Fixed Geometry.... To secure the part, the cylindrical surface of the four holes is selected according to the adjacent images.  	
5. A normal force of 80000 N is applied to the surface of the six bosses, at the top of the model, by left-clicking External Loads in the Feature Manager Design Tree.	

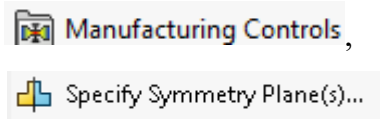
	
6. The topological optimization criterion is defined by right-clicking the command in the FeatureManager Design Tree and selecting the optimization criterion  Best Stiffness to Weight ratio (default). It is desired that the final mass of the model be reduced by 70% without affecting the mechanical characteristics of the part.	
7. The mounting and fixing holes are selected so as not to be included in the optimization process. In the FeatureManager Design Tree, right-click the command  Manufacturing Controls, select  Add Preserved Region..., select Add Preserved Region..., and then select the surfaces of the mounting and support holes.	



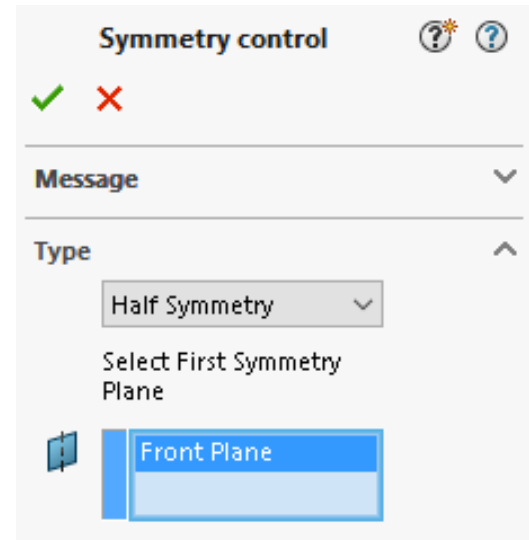
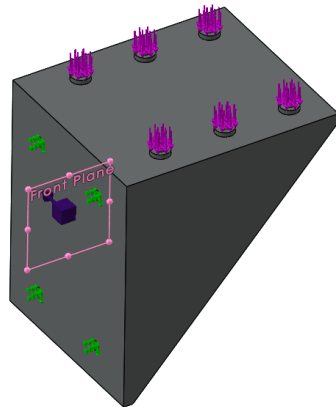
8. If the finished part is produced by die casting to achieve the optimized model, the part needs to satisfy certain shape constraints. In the **FeatureManager Design Tree**, right-click on the command and select the top edge of the part, as shown in the image below.



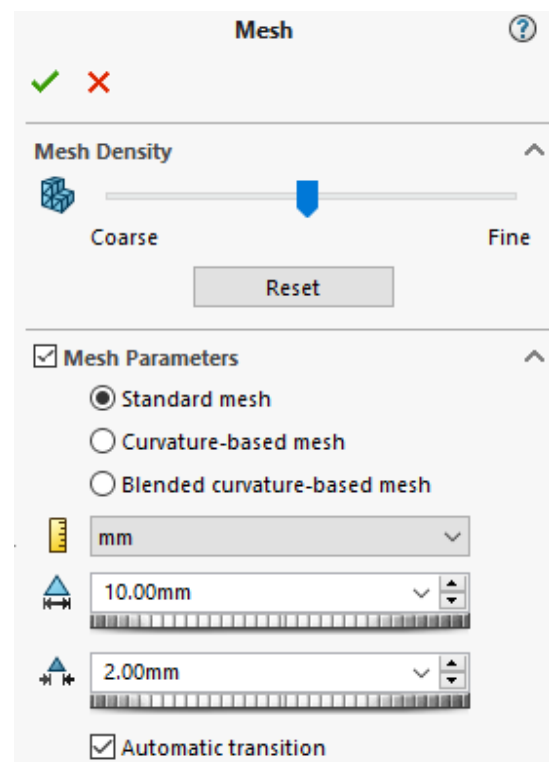
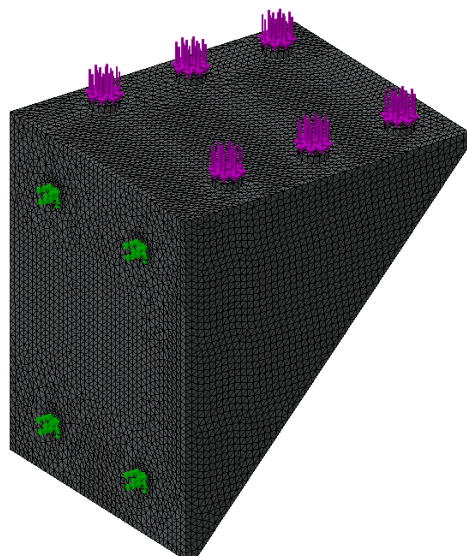
9. To achieve symmetry in the optimized model from the **FeatureManager Design Tree**, you select



and the front plane is selected as the plane of symmetry.

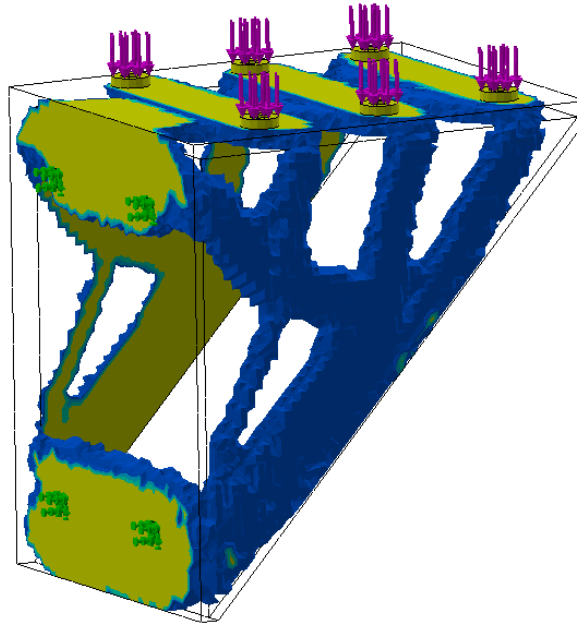


10. The model is discretized into tetrahedral solid elements with control from the **Feature Manager Design Tree**. The maximum side length of an element is 10 mm, and the minimum is 2 mm. The settings are made in accordance with the adjacent figure. The discretized finite element model is shown in the figure below.



11. Running the optimization solver is activated by right-clicking  **Topology Study 1** on
and selecting  **Run** from the **Feature Manager Design Tree**.

12. The geometry resulting from the topological optimization process can be seen from  **Results**. The adjacent image shows the resulting geometry in its raw form.



13. Optimized geometry can be generated by right-clicking on

 **Material Mass1 (-Material Mass-)**,

 **Export Smoothed Mesh...**

Export Smoothed Mesh 

✓ ✗

Message ▼

Save mesh into ▲

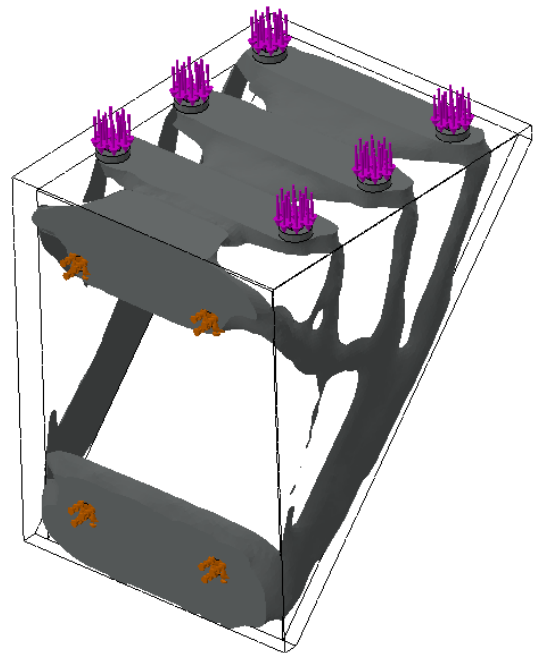
Save mesh into

☐ Current active configuration

☒ New configuration

☐ New part file

Configuration name:



III. Self- evaluation test

Answer the requirements below by completing them in a Word file:

1. In the example above, change the material to aluminum. What is observed?
Add a picture of the results you obtained to the file.
2. What is the minimum required thickness of the material to create the aluminum support that offers the same characteristics as steel as steel?
3. Using the algorithm presented earlier, perform topological optimization to reduce the support mass by 50%
Run again with the new settings and add the picture of the resulting geometry.