

Laboratory 9-10

Modeling and Structural Stress Analysis of a Towing Device

The paper presents a detailed description of the steps to be followed for CAD modeling and static structural stress analysis of a towing device that can be attached to a vehicle chassis.

Objectives:

- To develop skills in designing and developing an auxiliary mechanical component for the structure of a vehicle chassis.
- Understanding the general concepts of CAD modeling of a mechanical part.
- Understanding the general concepts of structural stress analysis of a mechanical part.

Introductory Concepts:

The efficiency of the goods transport process is determined by numerous factors related to the dynamic and structural performance of a transport vehicle. The structural performance of a goods transport vehicle can be defined by three major technical indicators:

- loading capacity.
- loading volume.
- The floor surface in the vehicle body.

If the loading capacity of a road goods vehicle refers to the maximum mass of goods permitted by the competent authority in the vehicle's country of registration, the loading volume represents the maximum volume available in the vehicle (measured in cubic meters) for transporting goods. It can thus be observed that there is a difference in terms of the optimal use of the energy source's power for freight transport, considering the loading capacity index and the loading volume.

For this purpose, the load-bearing structure of road freight transport vehicles includes systems for towing an additional trailer, so that the resulting road train (vehicle + trailer) effectively utilizes all the power available at the energy source (engine) that equips the road transport vehicle. Its design, dimensioning, and construction must meet the minimum requirements related to structural integrity under all operating conditions, regardless of the road infrastructure used and at all travel speeds. The installation and integration of a towing device can be done in a fixed or removable manner to the chassis/frame of the road vehicle. It should be noted that a towing device can also be used for towing special road vehicles, such as tourist caravans, as well as special construction equipment.

Case Study

I. Towing Device Modeling

The towing device is modeled by modeling the component elements that make up the towing device proposed for analysis. The necessary steps (algorithm) for device modeling are presented as follows:

Modeling the first part of the hook assembly – the Tow Hook

1. From the **Feature Manager** toolbar, select the **Top Plane** on which to

draw the first sketch. From the toolbar, select the **Sketch**



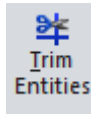
command, where the **Circle**



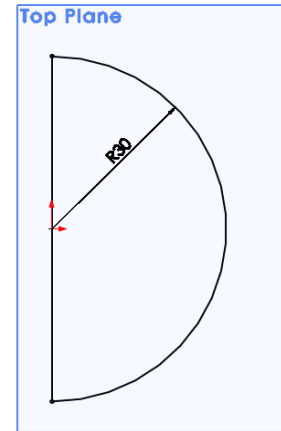
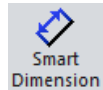
, **Line**



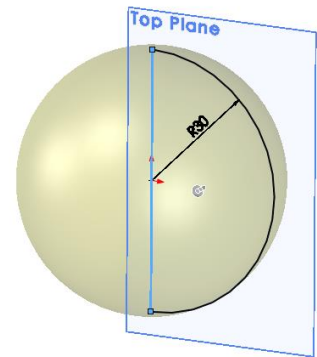
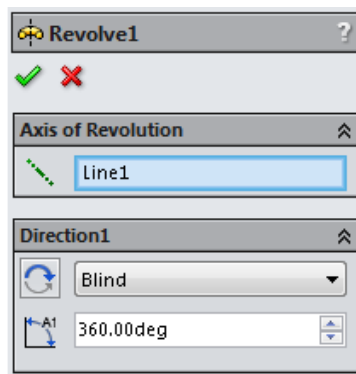
, **Trim Entities**



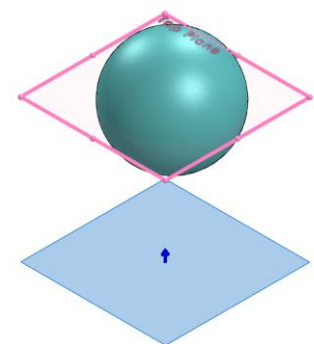
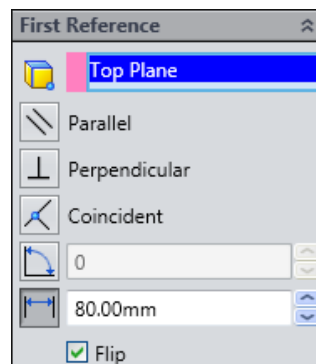
, and **Smart Dimension** commands will be used to create the following sketch according to the dimensions. A semicircle with a radius of **R=30** mm and a line that closes the semicircle are constructed.



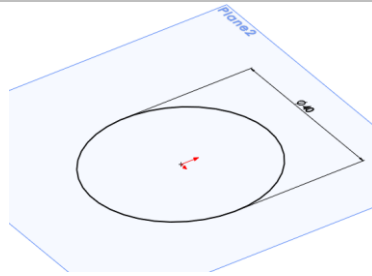
2. Using the **Revolve** command from the toolbar, the sketch is rotated around the axis, resulting in a sphere.



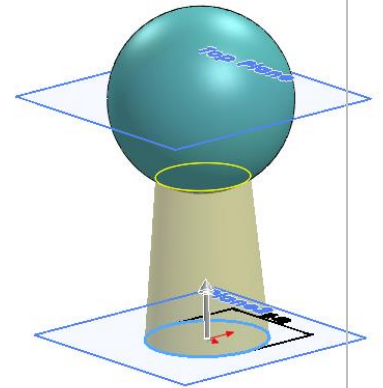
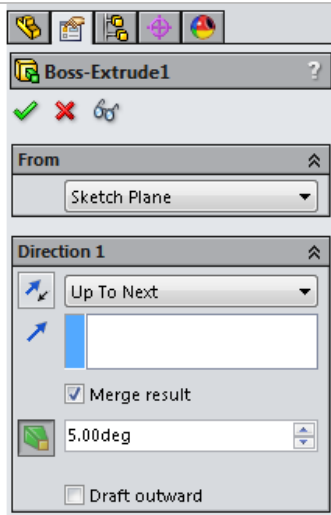
3. A new plane is drawn parallel to the **Top Plane** at **80 mm**.



4. A circle with a diameter of **40 mm** is drawn on the new plane.

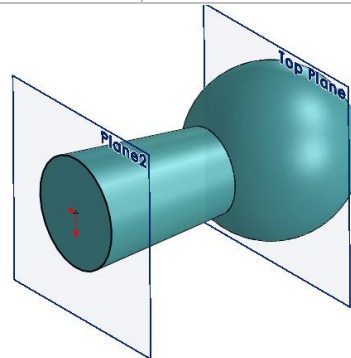


5. The sketch on the new plane will be extruded using the **Extruded Boss/Base** command  at a 5° angle up to the constructed sphere.

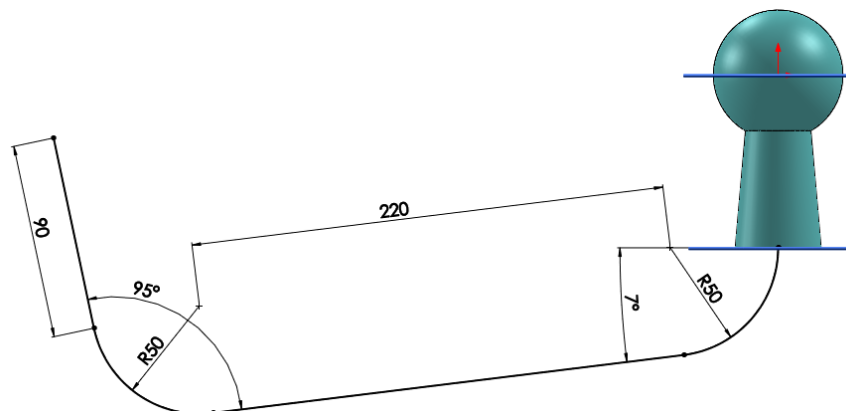


6. On the face of plane 2, convert the outer diameter of the previously extruded sketch using the **Convert Entities**

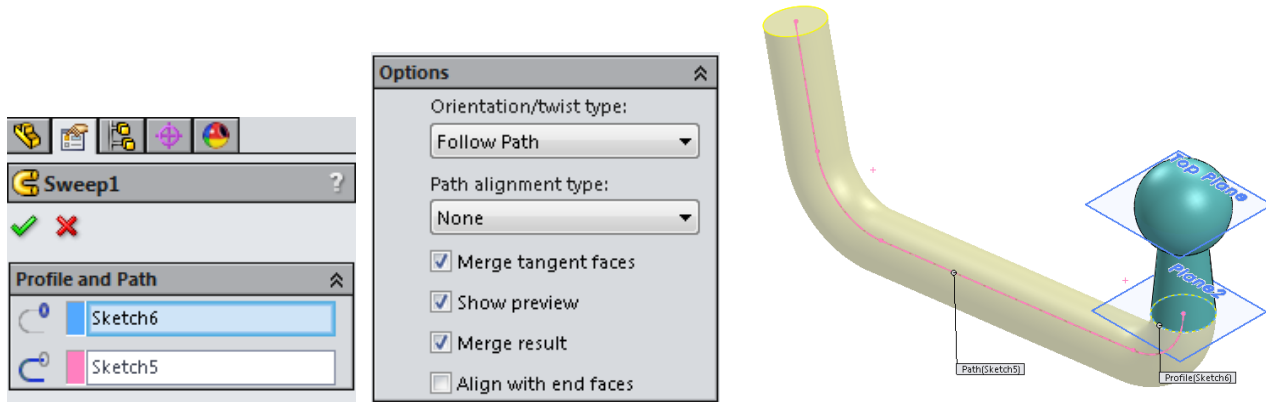
command .



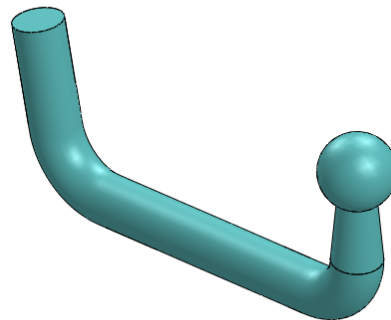
7. Draw the adjacent sketch on the **Front Plane**.



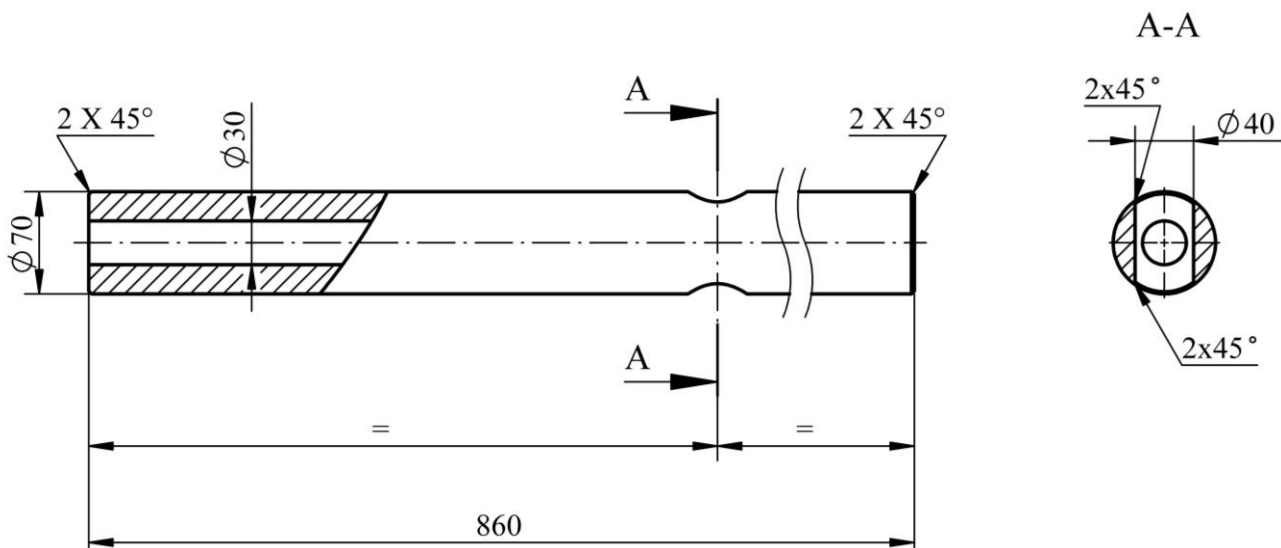
8. By selecting the **Sweep** command  and using the previous sketch as the guide path, the 40 mm diameter circle chosen as the profile will generate the tow hook.



9. The final model of the tow hook is shown in the adjacent image.



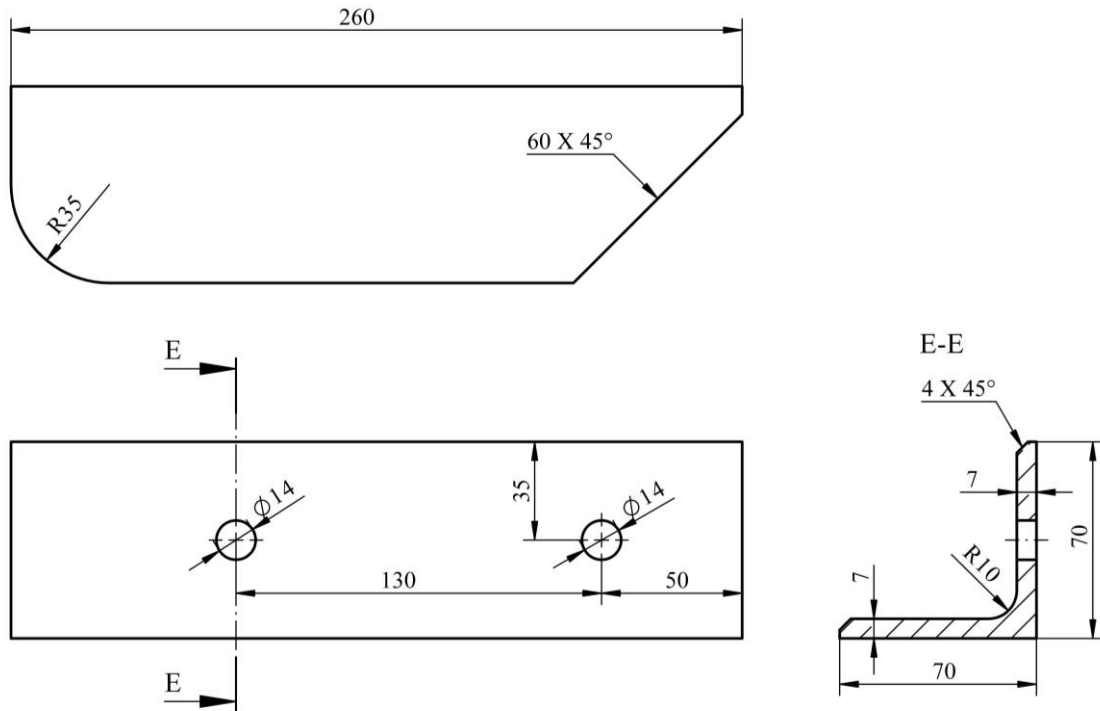
Modeling the second part of the hook assembly - the Hook Support Element



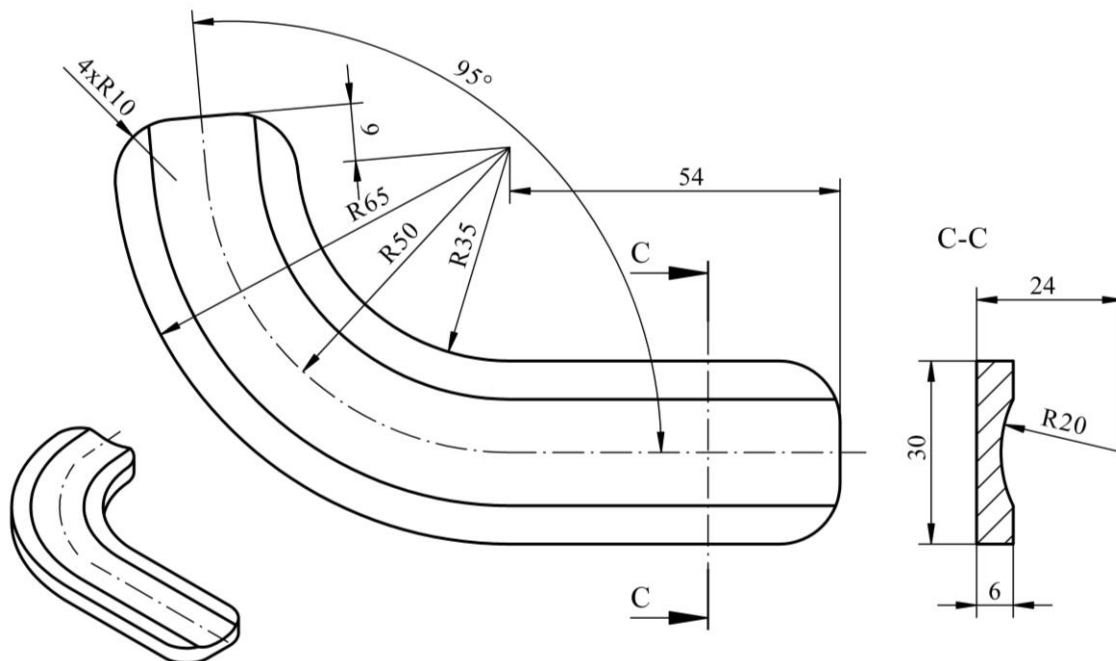
Modeling the fourth component element – Support Element 1



Modeling the fifth component – Support 2



Modeling the sixth component – Left Reinforcement

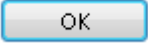
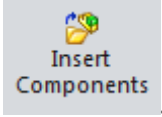


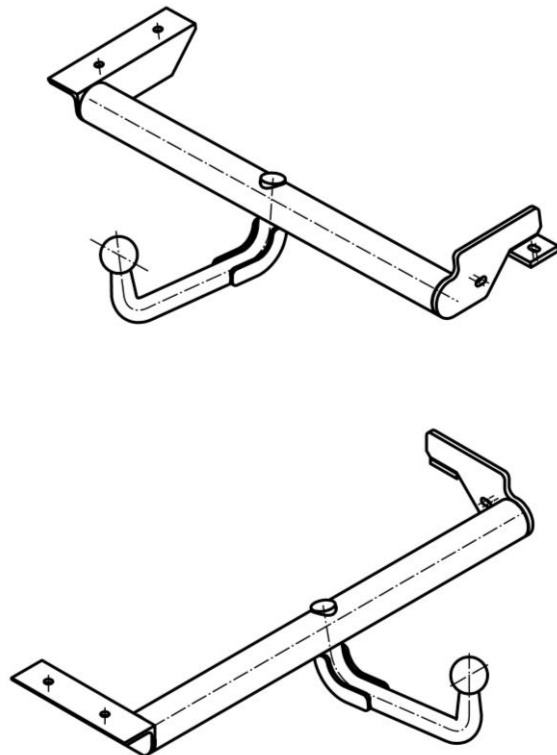
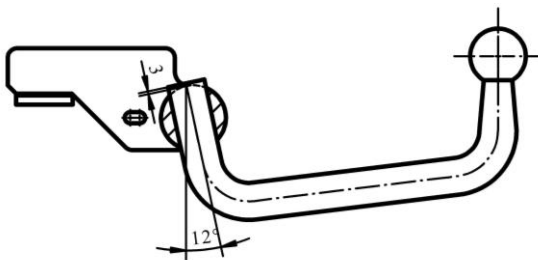
Important!

It will be considered that there are two symmetrical elements (left-right). Another symmetrical element will be built to match this one.

II. Creating the Towing Device Assembly

To create the assembly, the following steps will be followed:

1. New file , Assembly , Ok .
2. Insert components .
3. The component elements are introduced within the assembly and connection relationships are created between them using the **Mate**  command.

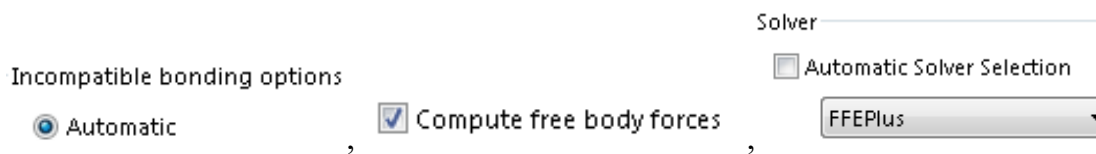


III. Static Analysis of the Towing Device

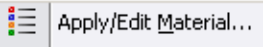
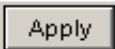
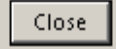
The steps for conducting this static study are presented as follows:

1. From the **Tools** menu, select **Add-Ins...**, where you will check the **SolidWorks Simulation** button.

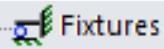
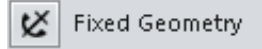
2. From the **Simulation** menu located in the menu bar, choose a new analysis study by clicking the **Study...** button. This button can also be accessed from the program's toolbar, in the **Simulation** tab, under the **Study Advisor** submenu, by clicking on **New Study**.
3. On the screen, in the history area of performed operations, the **Study** dialog box will appear, from which a **Static** study named "**Static Case**" by the user will be selected. Right-clicking on the created study **Properties...** brings up the following options:

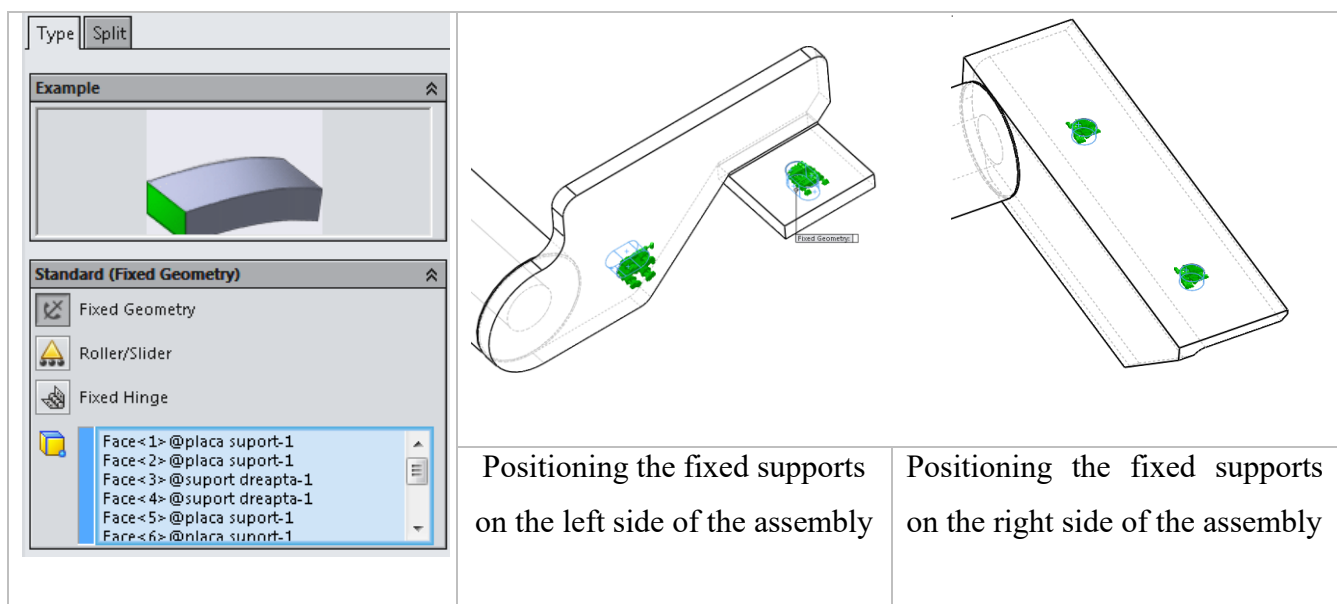


1. Material selection and assignment

The material is assigned to the assembly by right-clicking on the model, selecting , and then choosing an alloy steel  from the dialog box containing the existing material types in SolidWorks. After that, this material is assigned by clicking  and then closing the dialog box .

2. Applying restrictions to the entire tow hook assembly

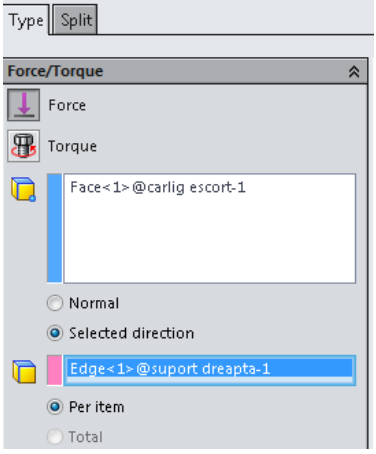
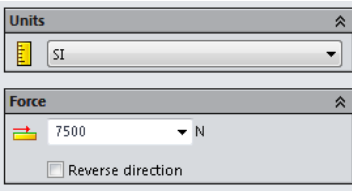
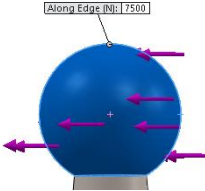
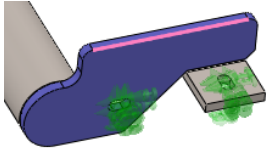
This is done by right-clicking on , **Fixed Geometry...** , and then left-clicking on the lower surfaces indicated in the image below.



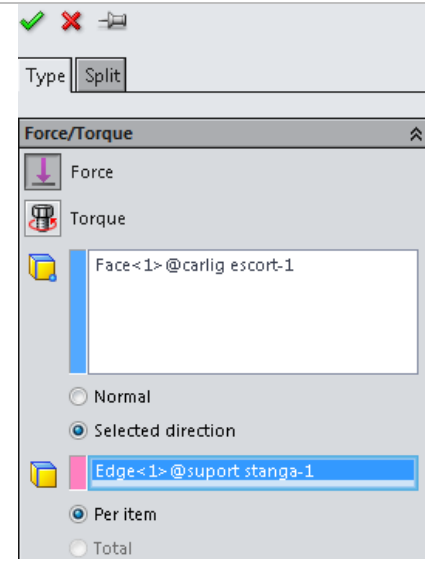
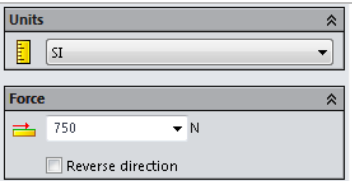
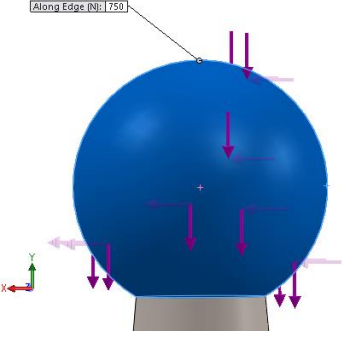
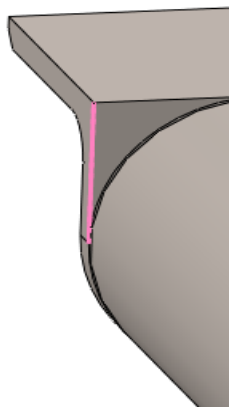
3. Creating the loading case

Right-click on , and the dialog box below will open  **Force...**. Two forces are applied to approximate the total mass of the trailer. A horizontally oriented force and a vertically oriented force.

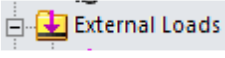
- The first force has a magnitude of **7500 N** and is positioned horizontally in the area of the spherical joint. The images below show how this force is applied in the case of the studied assembly.

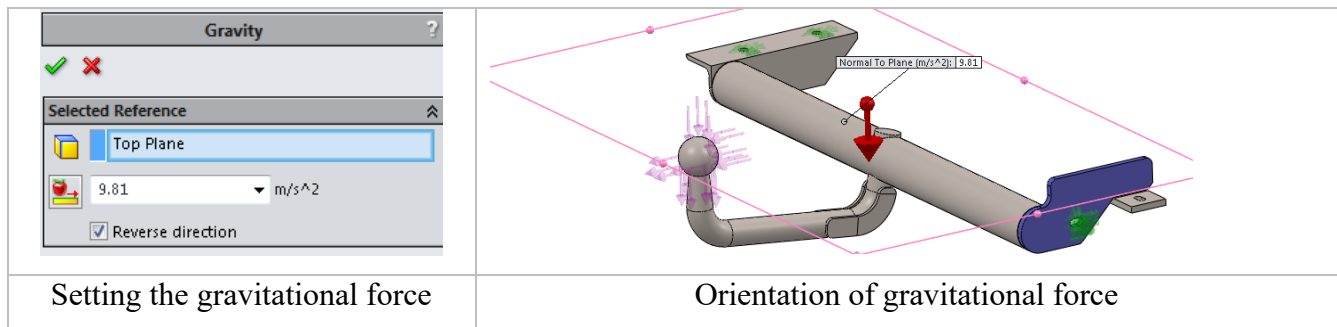
			
Selecting the charging type	Choosing the unit of measurement	Application point	Choosing the direction of the force

- The second force applied has a magnitude of **750 N** and is positioned vertically in the joint area between the trailer and the hitch. This force is positioned as follows:

	 	
---	---	---

4. Applying gravitational force

This force is applied by clicking on  **External Loads** and selecting  **Gravity...** from which the **Top Plane** will be chosen, relative to which the gravitational force will be positioned.



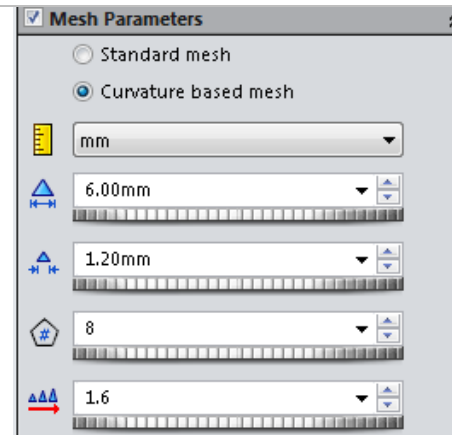
8. Discretization of device geometry into finite elements

To discretize the assembly, right-click, select  **Mesh** "Create Mesh...", and check "Mesh Parameters". It is chosen:

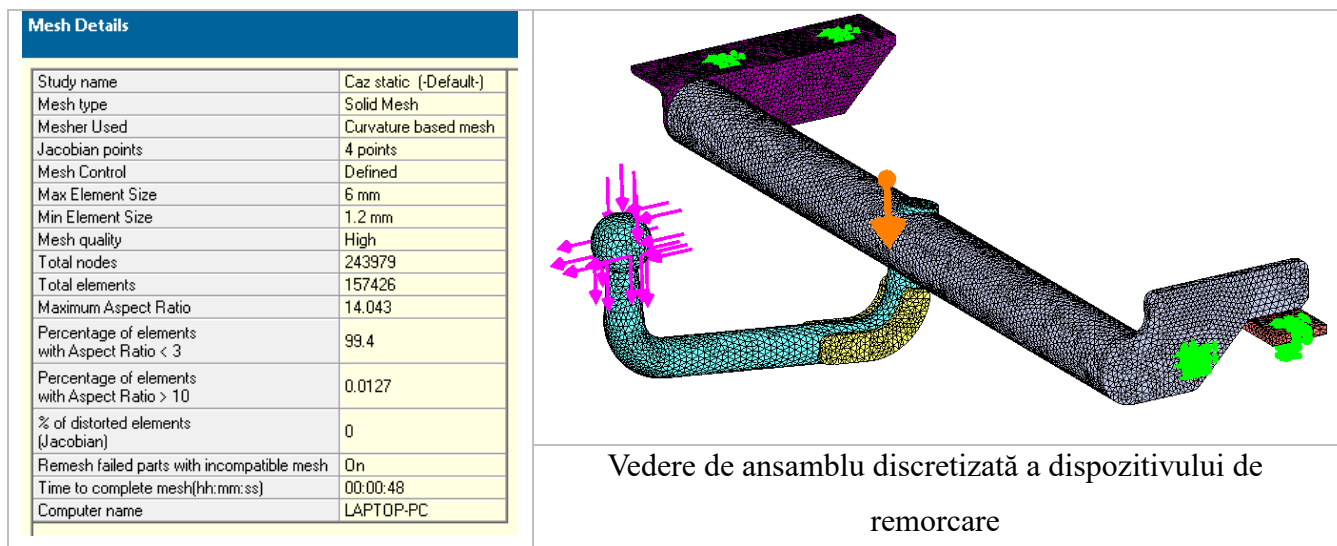
- maximum element size of 6 mm,
- minimum element size of 1.2 mm,
- The element growth ratio is 1.6

Jacobian points 4 points

☒ Remesh failed parts with incompatible mesh

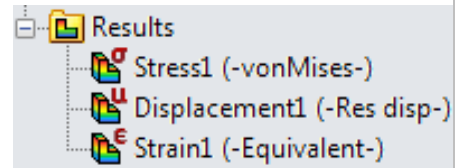


The quality criteria chosen for discretization, the total number of elements, and other details regarding the discretization operation can be found in **Mesh details...** by right-clicking on  **Mesh** and selecting **Details...**. In the studied case, the details are presented in the following figure.

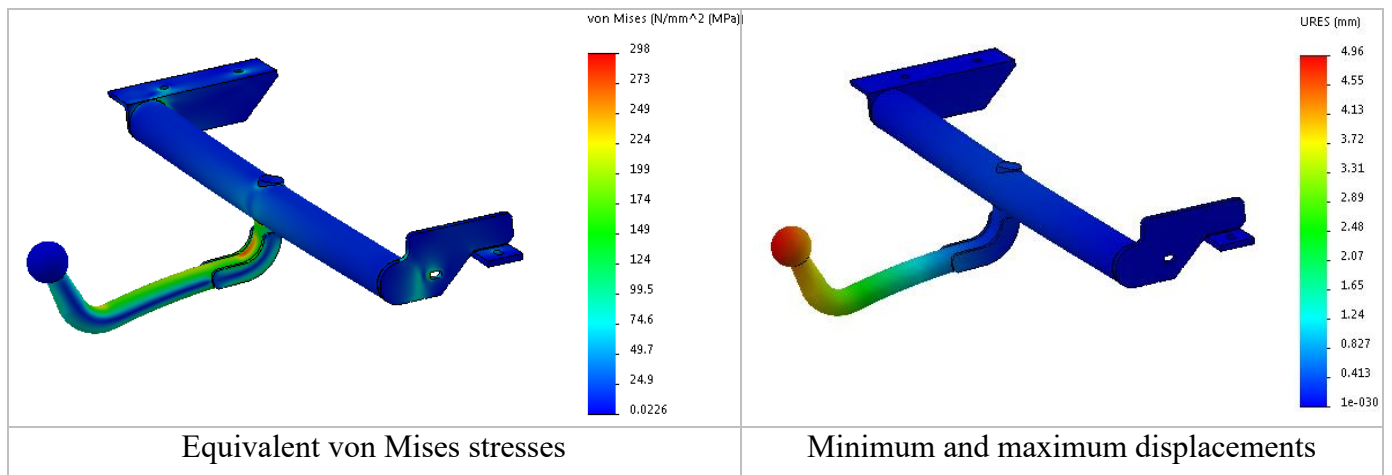
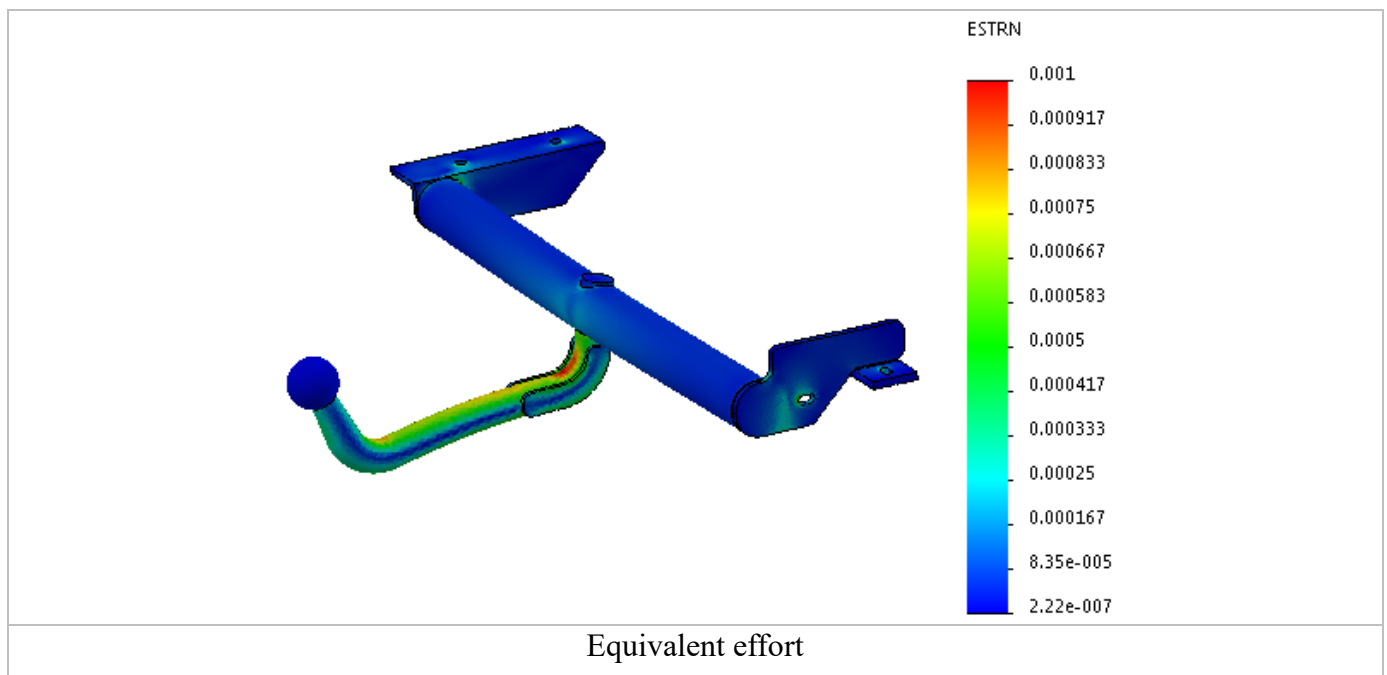


9. Solve the static analysis of the towing device.

To run the study, click the button **(Run)** , located in the study run toolbar. After running the study in the **Feature Manager Design Tree tab...** the results appear as you can see in the adjacent image.



The following images present the results obtained after the applied loads.



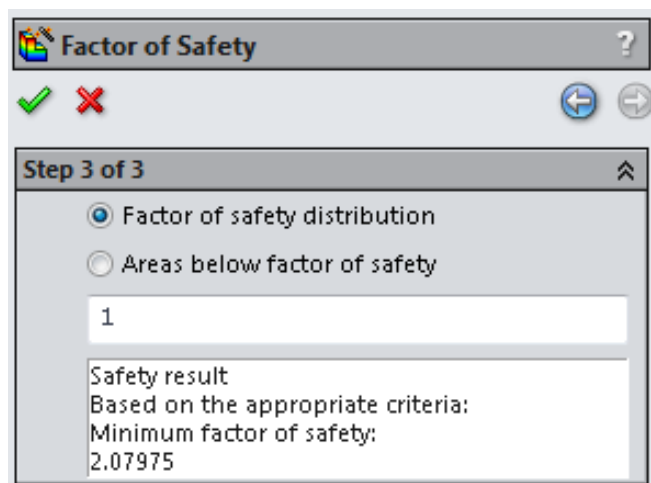
Fatigue Resistance Analysis

Fatigue resistance is characterized by a material's ability to withstand fracture under variable stresses applied over time. Due to the repeated absorption and release of deformation energy, the phenomenon of fatigue failure occurs. The variation of loads over time in direction and intensity has a decisive influence on the materials being stressed. In this case, fractures can occur at stresses lower than the yield strength or the tensile strength for static loading. It was found that the break starts with the formation of microcracks in the most stressed area, which, gradually developing, can lead to the piece breaking.

The following outlines the necessary steps for conducting a fatigue analysis.

- A safety factor is defined.

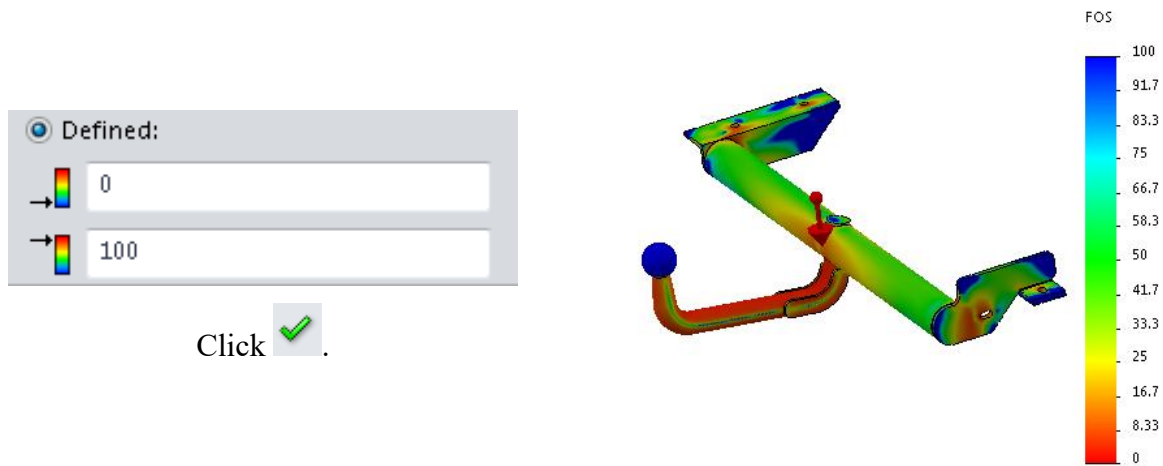
By right-clicking on  **Results**, you select  **Define Factor Of Safety Plot...**, after which the dialog box opens  **Factor of Safety**. You apply the  command twice, after which the dialog box opens where you can view the minimum safety factor.



Click .

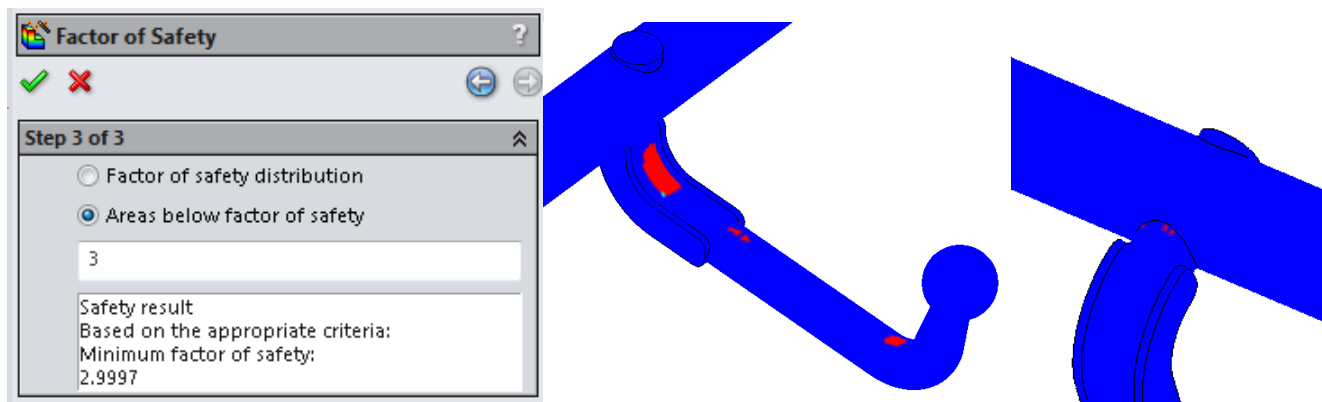
To better visualize the load distribution, a display scale ranging from 0 to 100 will be defined.

- Right-click on  **Factor of Safety1 (-FOS-)**,  **Chart Options...**,

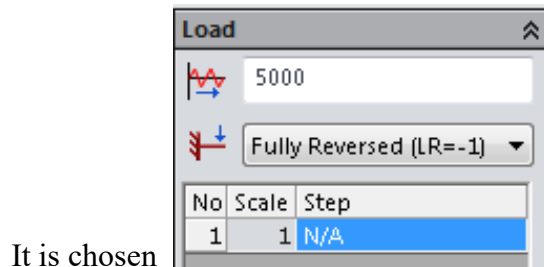


Graphical representation of the safety factor

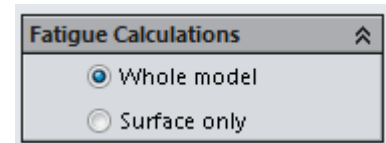
- Right-clicking on **Results**, choose **Define Factor Of Safety Plot...**, then the dialog box opens **Factor of Safety**, apply the **Apply** command twice, and then select a safety factor to a rounded value. The resulting critical zones are presented in the following images.



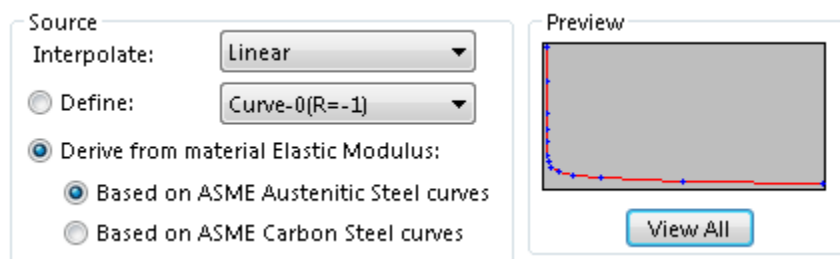
- A new fatigue test study **Study Advisor** is created, click **New Study**, select the study for resistance to **Fatigue**, with constant amplitude **Constant Amplitude**, click **OK**.
- Right-click on **Loading (-Constant Amplitude-)**, **Add Event...**.



- Right-click on **Result Options**, it is chosen **Define/Edit...**,
Click .



- Right-click on **Parts**, it is chosen **Apply Fatigue Data to All Bodies**. The dialog box opens with the material properties chosen for the assembly references.
If it is set:

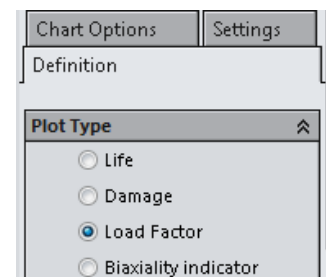


Click **Apply**, **Close**.

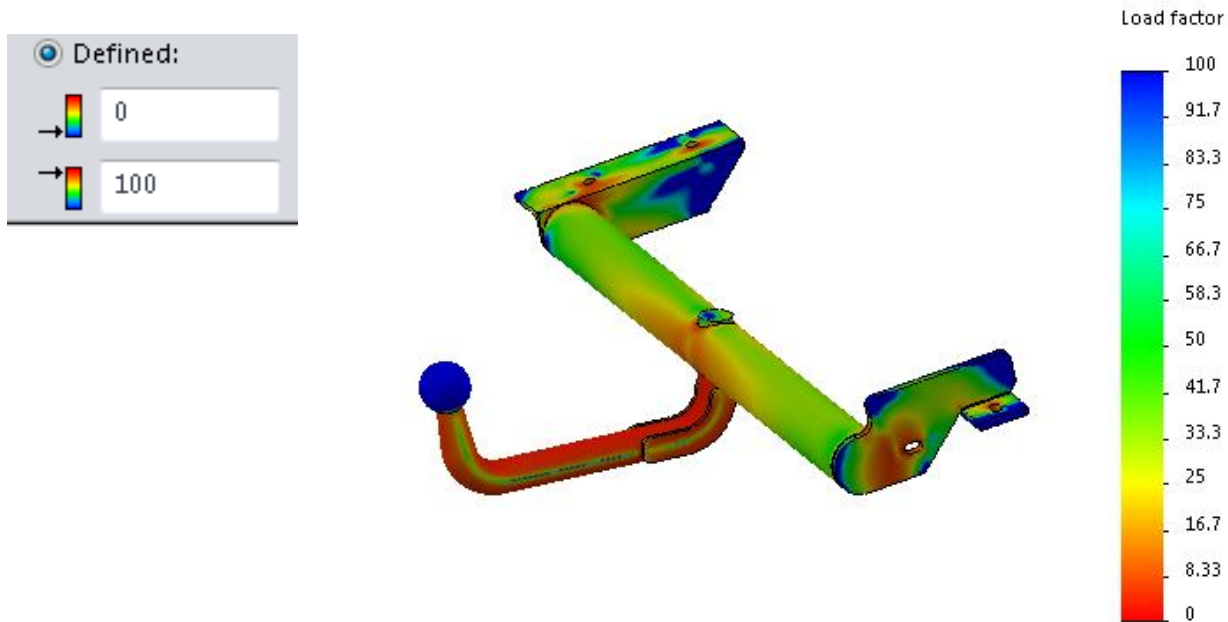
- This study is being conducted **Run This Study**.

- The following results are displayed:

- Right-click on **Results**, it is chosen **Define Fatigue Plot...**
Click .



- The display scale, ranging from 0 to 100, is also defined, **Chart Options...**.



Graphical representation of the load factor

IV. Self-assessment test

1. In the example above, change the material, using a different material (e.g., aluminum) for all components of the tow hook. What is observed?
2. Perform a static analysis of the tow hook with the geometric characteristics shown in the following figure under structural loads.

