

Laboratory 1÷2

Modeling and static analysis of a vehicle body frame

This laboratory presents the modeling of a frame component of a vehicle chassis. The first part of the paper presents a detailed modeling of this car frame using SolidWorks software, while the second part presents the structural analysis of the frame using the finite element method.

I. Body frame modeling in SolidWorks design software

To meet the deadlines for two laboratory assignments, the framework for their creation and simulation is presented in a simplified form.

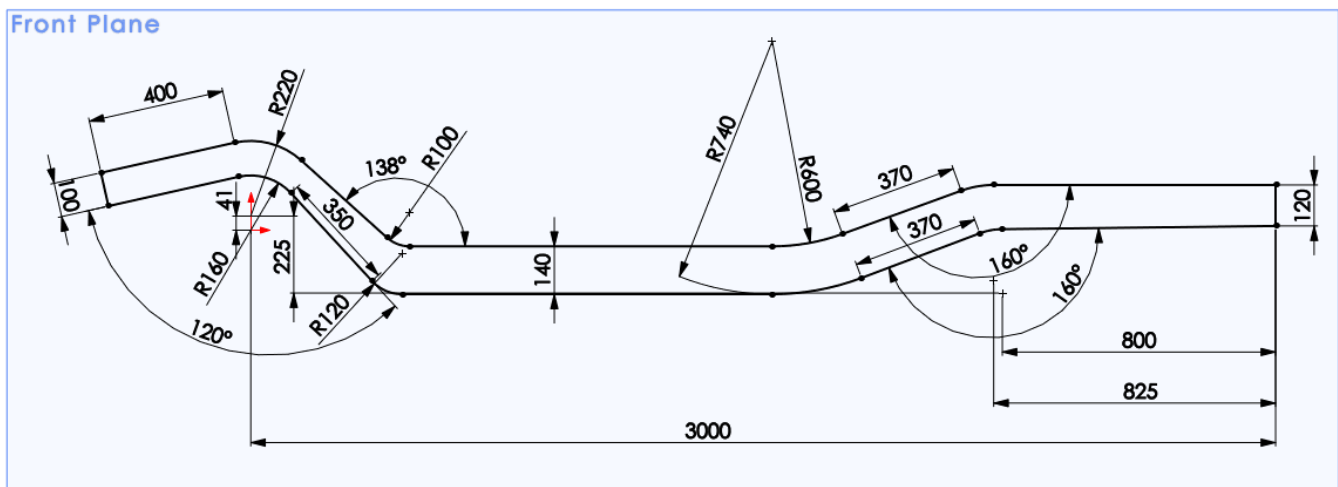
The steps required for modeling the frame are presented as follows:

1. Rail sketching

From the **Feature Manager** tab, select the **Front Plane** on which the sketch will be drawn.

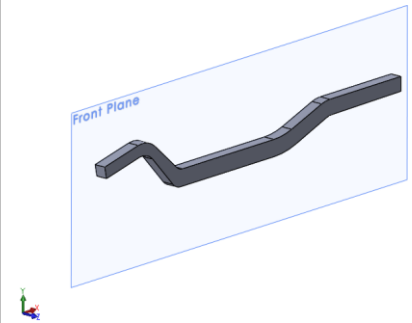
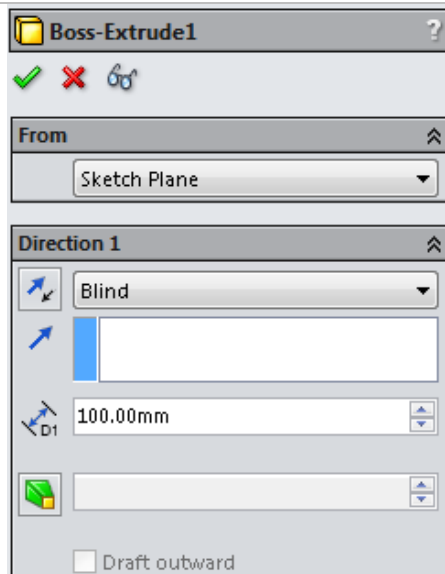
From the toolbar, select the Sketch command , from which the Line , Sketch Fillet

 and Smart Dimension  commands will be used to create the following sketch according to the dimensions.



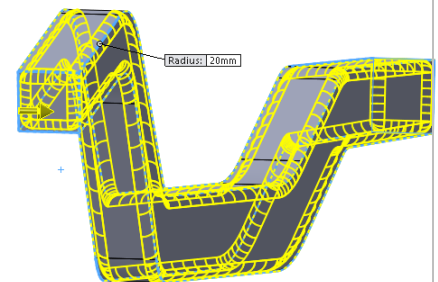
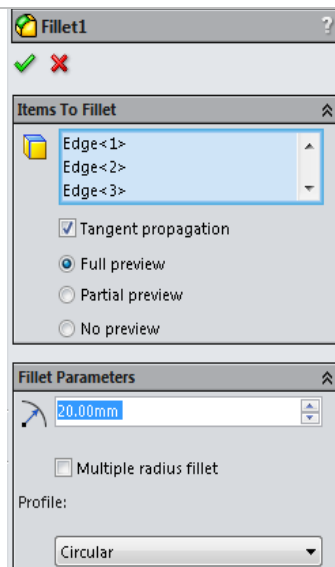
2. Using the **Extruded**

Boss/Base  command from the toolbar, the sketch is extruded to a distance of **100 mm**.



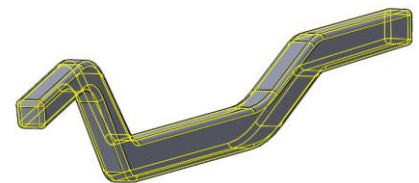
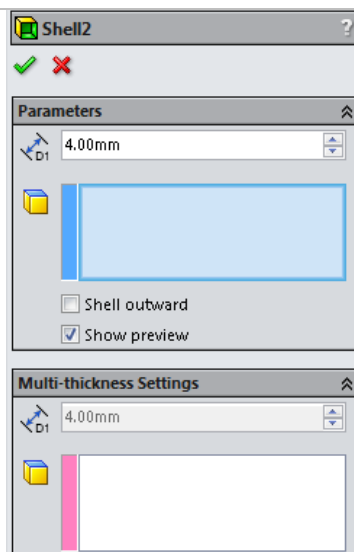
3. To round the edges of the instrument panel beam, select the **Fillet** command

, select all the edges, and enter a rounding radius of 20 mm, as can be seen in the adjacent image.

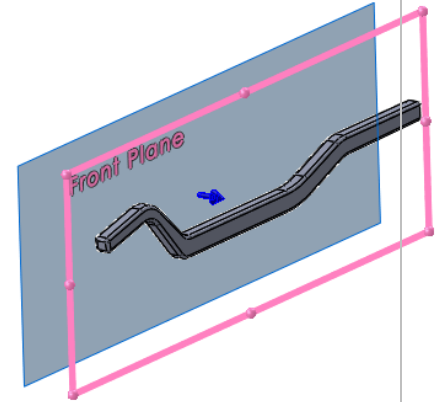
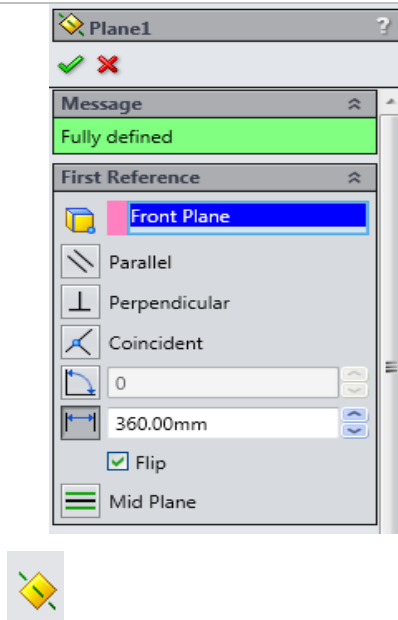


4. The tubular section is created using the **Shell**

command . From the tool palette, select the Shell command with a section thickness of 4 mm.



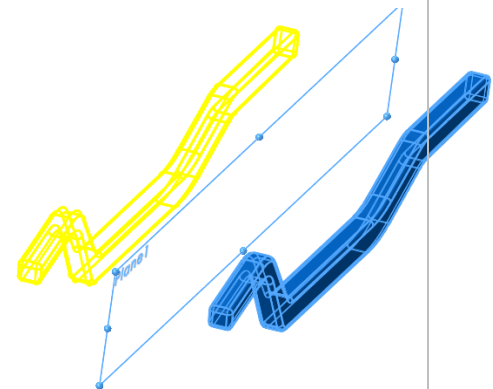
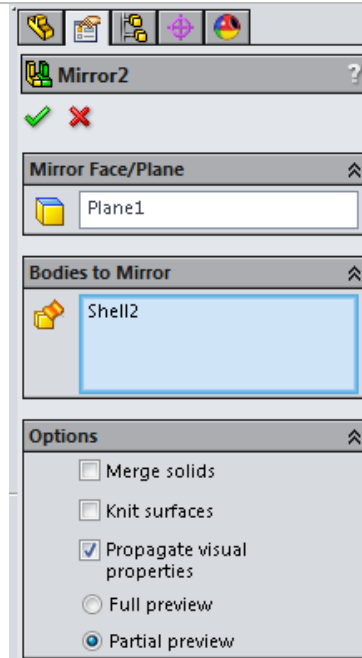
5. To maintain the frame's symmetry and shorten working time, the second longitudinal member will be mirrored. The first step is to draw a new plane parallel to the **Frontal plane** at a distance of 360 mm. The command to create a new plan is located in the toolbar, **Reference Geometry, Plane**.



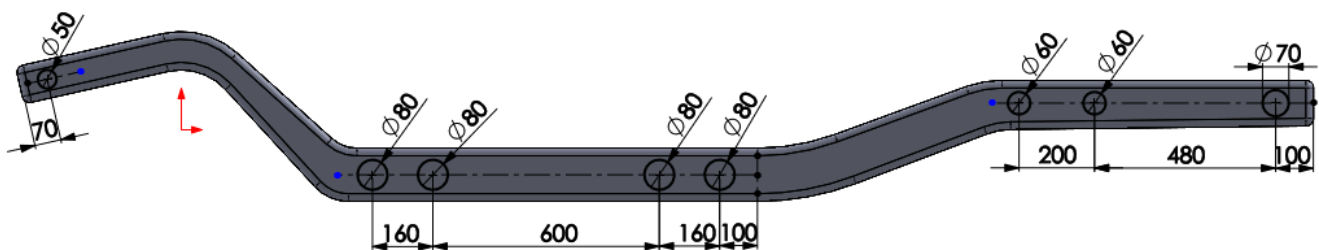
6. To mirror the longitudinal rail, select the **Mirror** command from the toolbar...



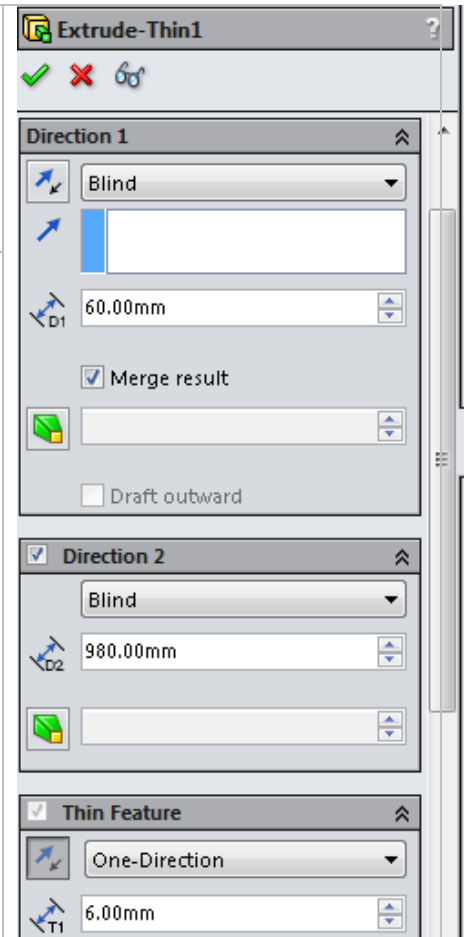
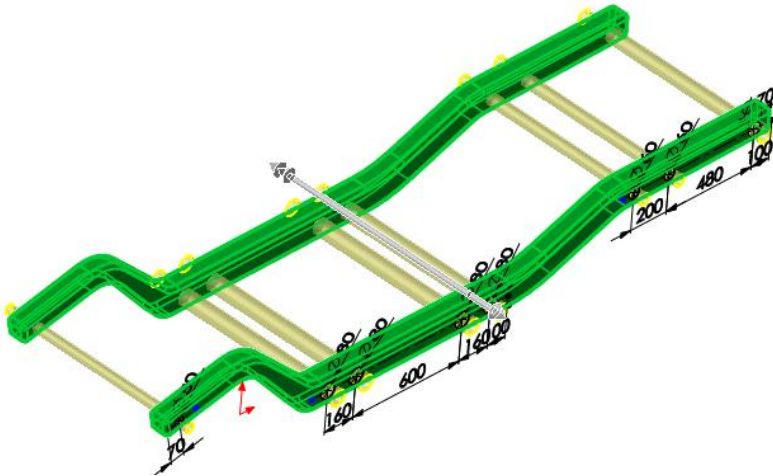
The plane against which the mirroring will be performed and the body to be mirrored will be selected.



7. On the outer face of the first rail, the outline for creating the connecting elements between the two rails is drawn. The sketch will retain the parameters shown in the following image.



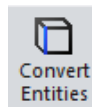
8. The sketch is extruded in **Direction 1** by 60 mm and in **Direction 2** by 980 mm with a 6 mm tubular section, as can be seen in the adjacent image.



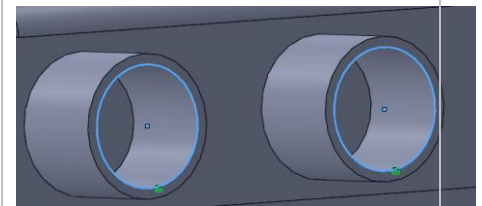
9. The removal of material remaining inside the longitudinal connection elements is achieved by introducing a new sketch



on the face of the elements. The inner diameter of the elements will be selected and converted into a sketch for each

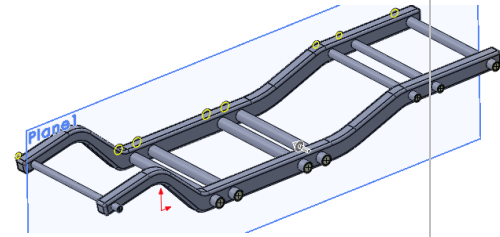
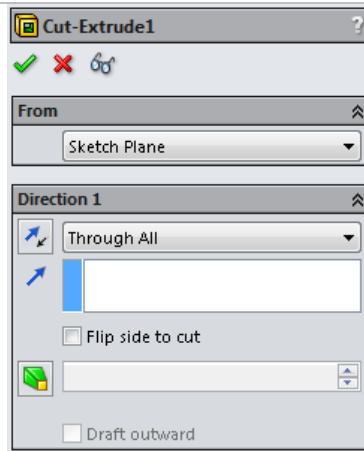
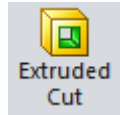


element using the **Convert Entities** command.



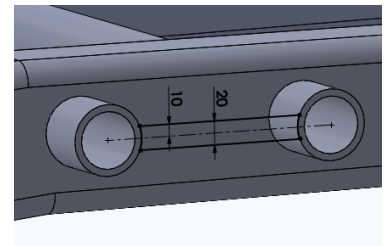
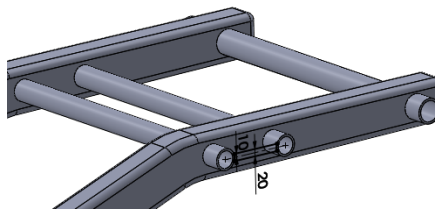
10. The cut will be made using

the **Extruded Cut** command from the toolbar



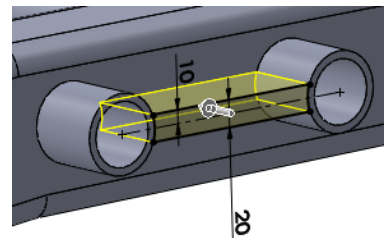
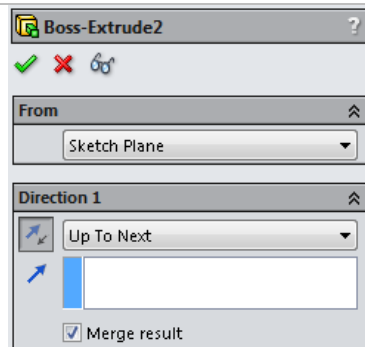
11. In the rear part of the

chassis, a sketch  is drawn on the front of the connecting elements as shown in the adjacent images.



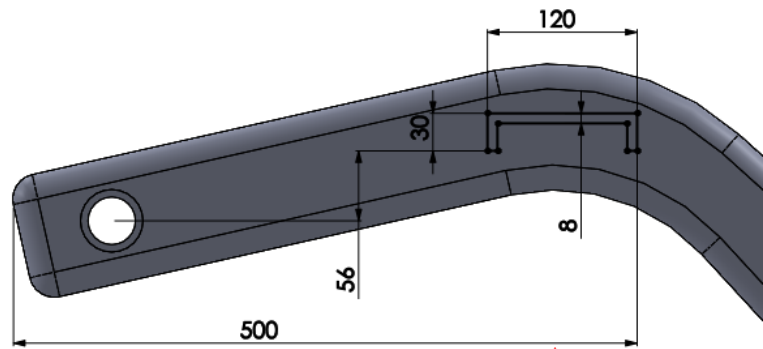
12. The created sketch will be

extruded  up to the rail.



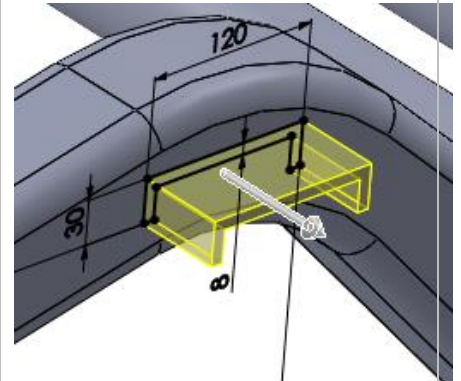
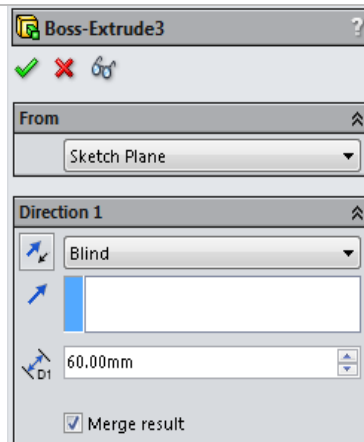
13. A new sketch  is

drawn on the front of the first rail, as can be seen in the adjacent image.



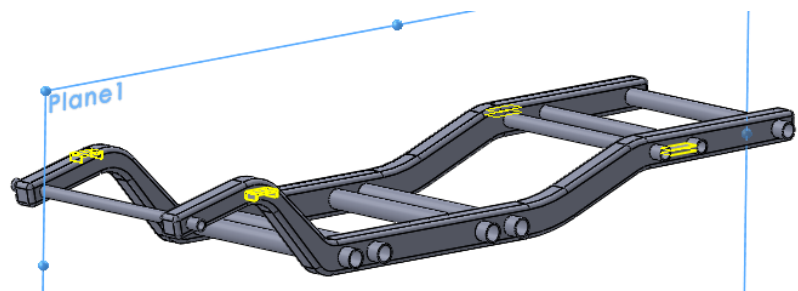
14. The sketch drawn above

will be extruded  to a distance of 60 mm.

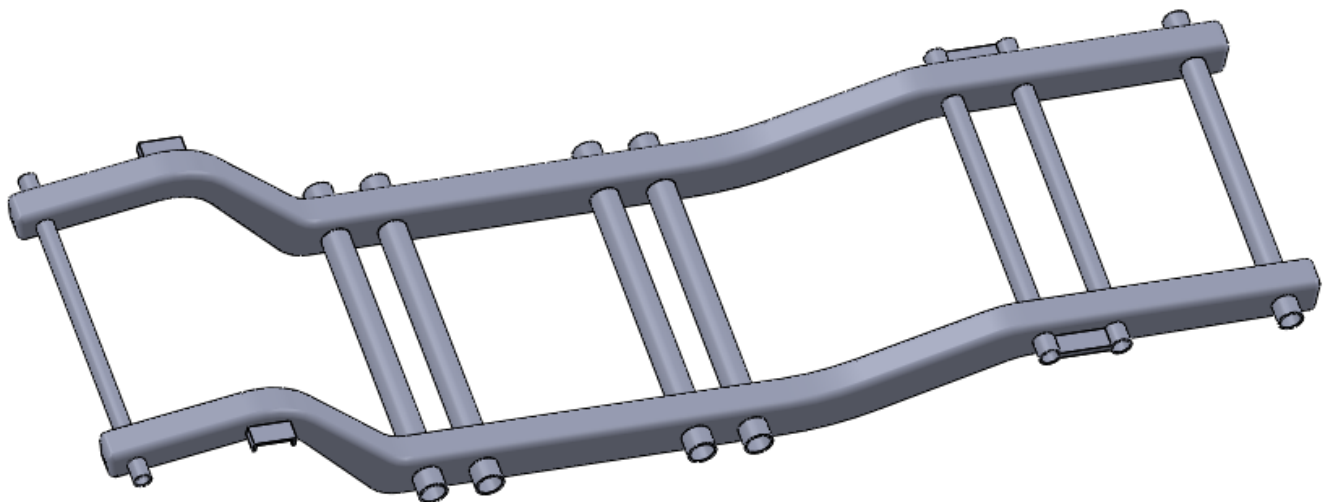


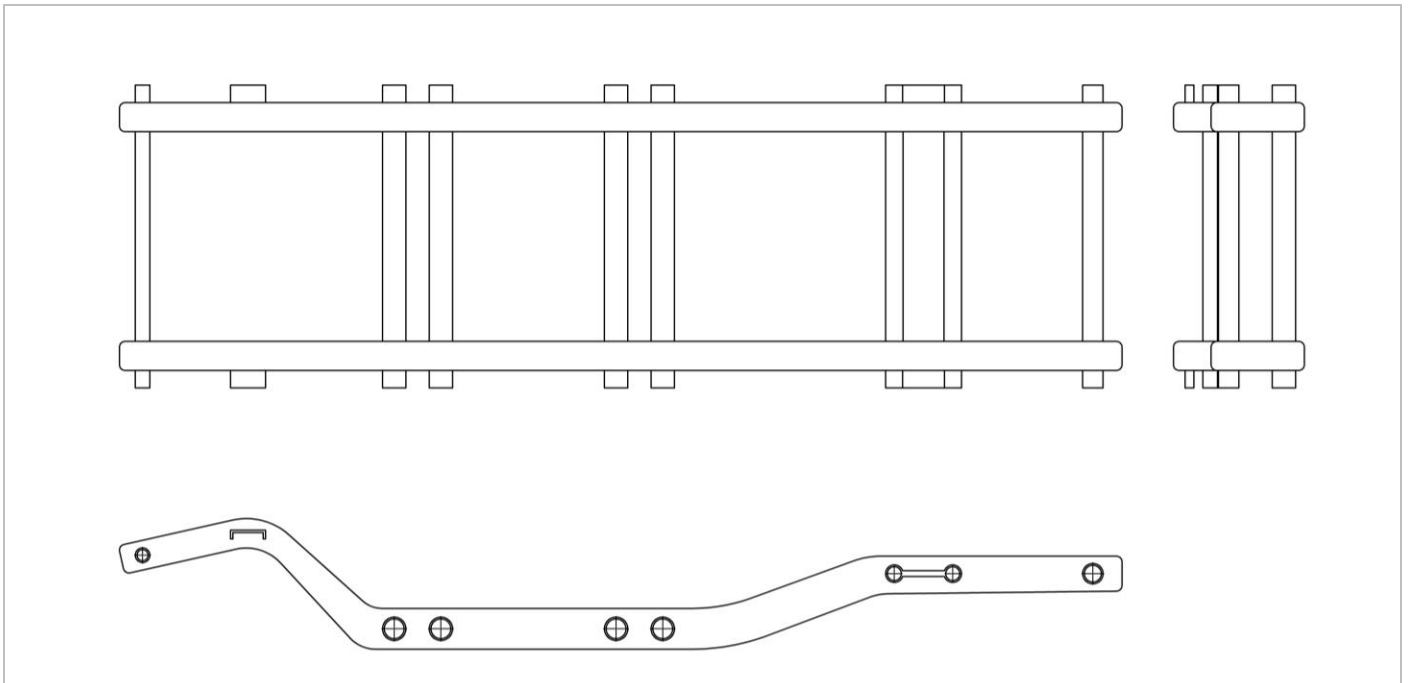
15. The extrusions created in steps 12 and 14 will be mirrored

mirrored  across **Plane 1** to be on the other side of the longitudinal member as well.



The final shape of the frame will look like the following images:

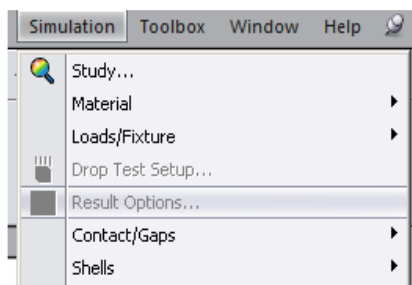




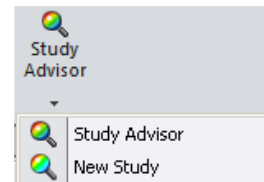
II. Creating a static analysis for the proposed model using the finite element method

The necessary steps for preparing the static analysis:

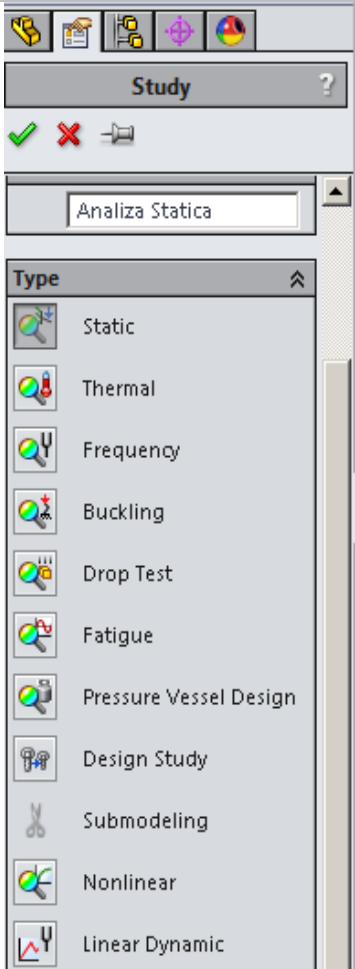
From the **Tools** menu, select **Add-Ins...**, where you will check the **SolidWorks Simulation** button.



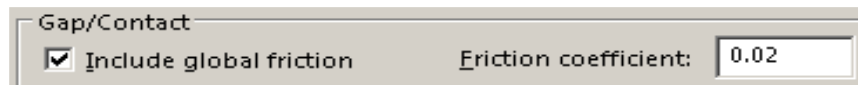
sau



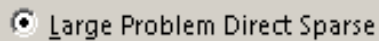
1. From the **Simulation** menu located in the menu bar, select a new analysis study by clicking the **Study...** button. This button can also be accessed from the program's toolbar in the **Simulation** tab, under the **Study Advisor** submenu, and by clicking on **New Study**.
2. On the screen, in the history area of the operations performed, the dialog box shown in the image below will appear, where you will choose the simulation type and name it **Static Analysis**.

	From this tab, we can choose the desired simulation type as follows:	
	Static – Static analysis of structures	
	Thermal – Thermal analysis	
	Frequency – Vibration analysis, natural frequencies of structures	
	Buckling – Buckling analysis of structures	
	Drop Test – Analysis of drop test	
	Fatigue – Fatigue analysis of structures	
	Pressure Vessel Design – Pressure analysis and design of tanks, cisterns, etc.	
	Design Study – Optimization study of structures and components	
	Submodeling – Allows for the remodeling of structures by transferring the conditions imposed on the main model.	
	Nonlinear – Allows for static or dynamic nonlinear analysis of models	
	Linear Dynamic – Linear dynamic analysis of models	

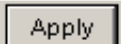
4. Right-clicking on the study  , **Properties...** is selected, a friction coefficient of 0.02 is chosen, and Include Friction is checked.



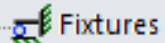
The solver is selected:

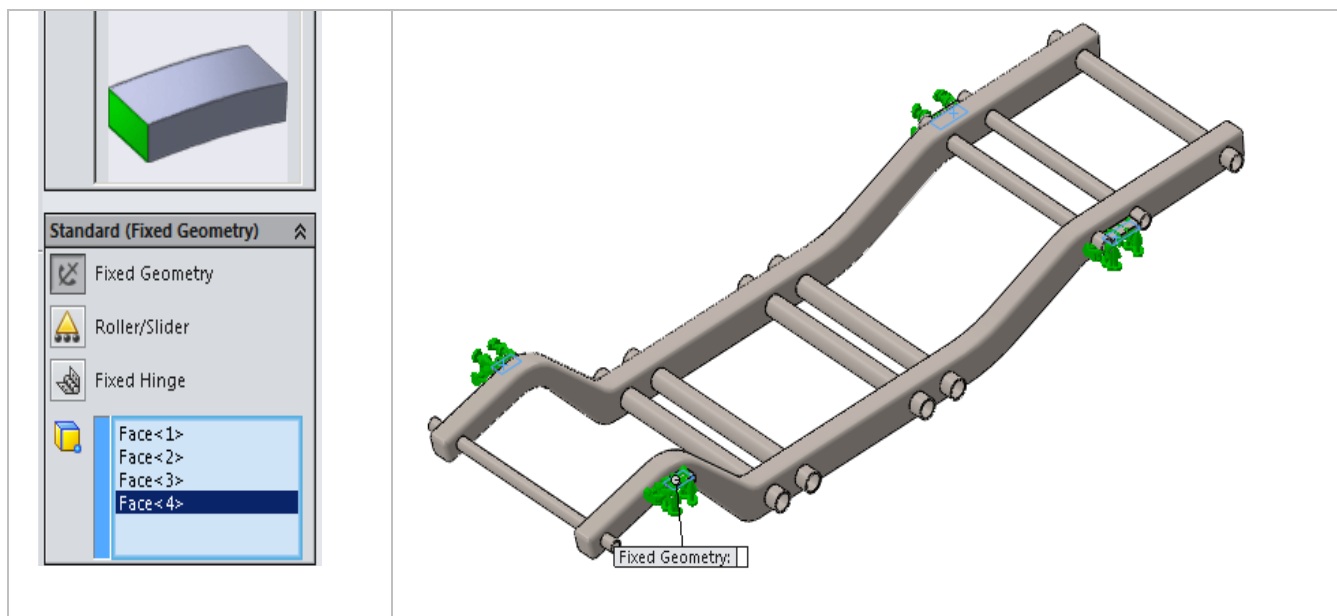


5. Selection and material assignment

The material is assigned to the frame by right-clicking on the model, selecting "Material"  Apply/Edit Material... from the context menu, then choosing an alloy steel  from the dialog box containing the existing material types in SolidWorks, and finally assigning this material by clicking  and then closing the dialog box .

6. Constrains creation

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

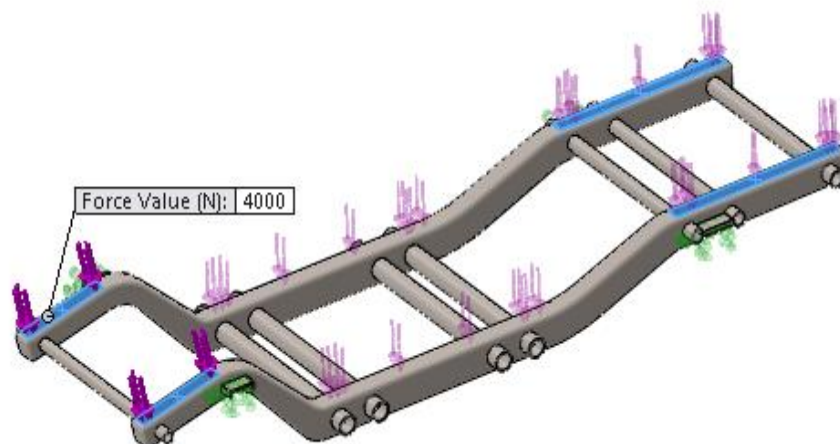
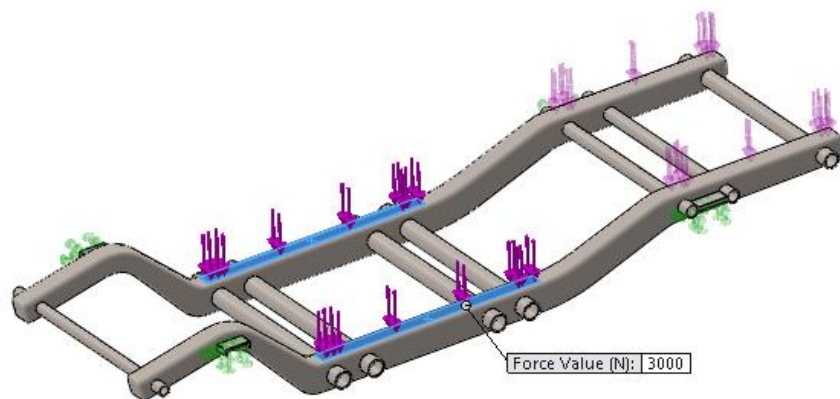
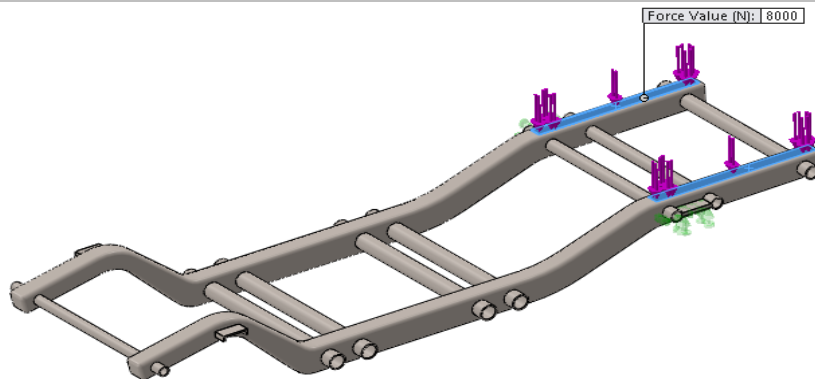
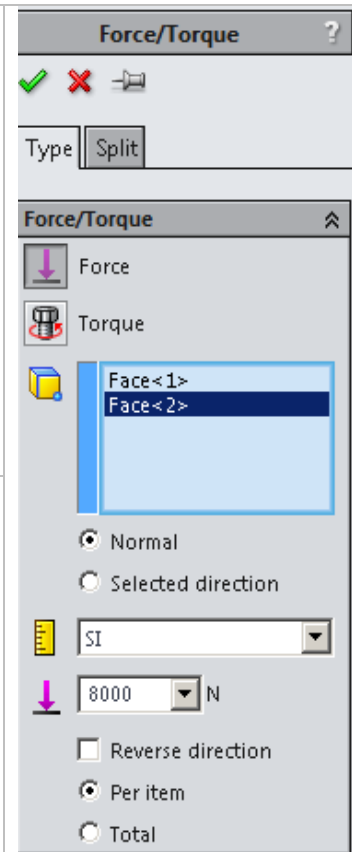


7. Load case creation

By right-clicking on  **External Loads**, you select  **Force...**, which opens the adjacent dialog box.

To estimate the mass of the vehicle and its load, the forces are applied as follows:

- A force of 8000 N is applied to the rear of the frame.
- A force of 3000 N is applied in the middle area.
- A force of 4000 N is applied to the front area.



7. Discretization of the frame into finite elements

The discretization of the geometric model determines the transition from the physical continuum of the material from which the structure is made to the conventional - geometric, discrete - model for which the finite element analysis will be performed. The discrete model developed through discretization will approximate the real structure as closely as possible, considering the application of forces, masses, support conditions, geometric shape, etc. To achieve the most optimal discretization from a simulation and result generation perspective, the mesh must have appropriate nodes for:

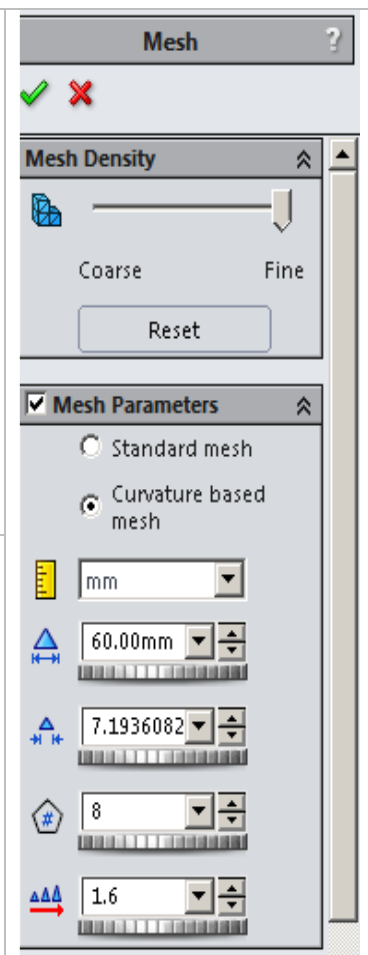
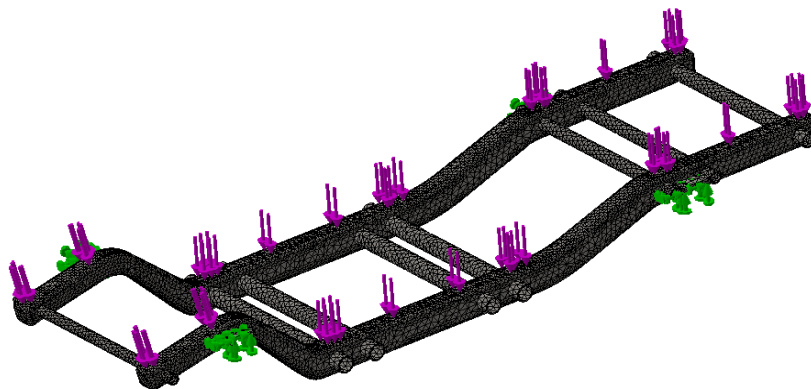
- Modeling the structure's outline
- Accurately determining the joints and intersections between the structural elements
- The discretized mesh should contain nodes for the structure's supports and applied forces
- Defining nodes for areas where the values are desired to be known.

In this case, the development of the discretized model is done through automatic discretization, taking into account only the parameters indicated below.

To discretize the frame, right-click  **Mesh**, select "**Create Mesh...**", and check "**Mesh Parameters.**"

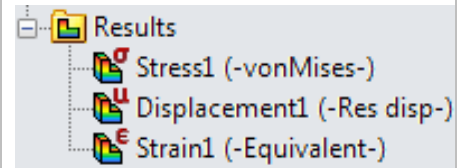
As can be seen in the adjacent figure, the following parameters will be taken into account:

- a maximum element size of 60 mm is chosen,
- the minimum element value is left at the default,
- the minimum number of elements arranged in a circle is chosen as 8,
- the element growth ratio is 1.6.

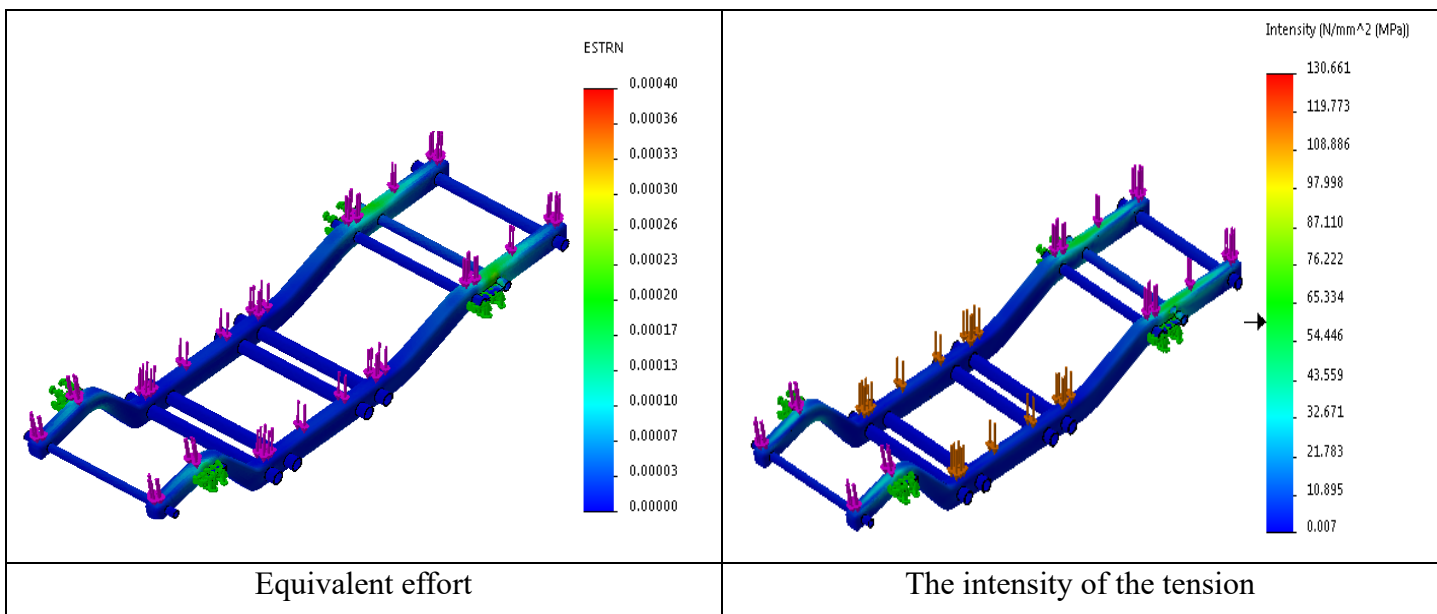
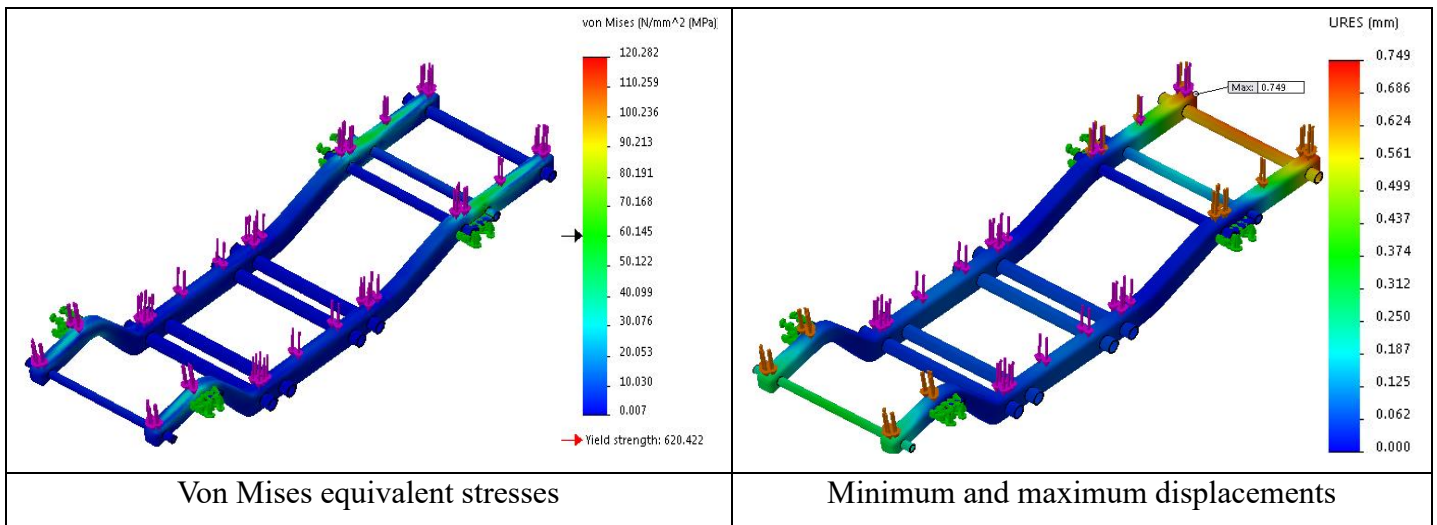


9. Solve and presenting the results.

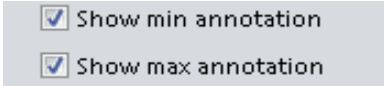
To run the study, click the button  (Run), located in the study run toolbar. After running the study in the **Feature Manager Design Tree**..., the results appear as shown in the adjacent image.

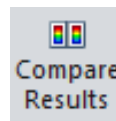
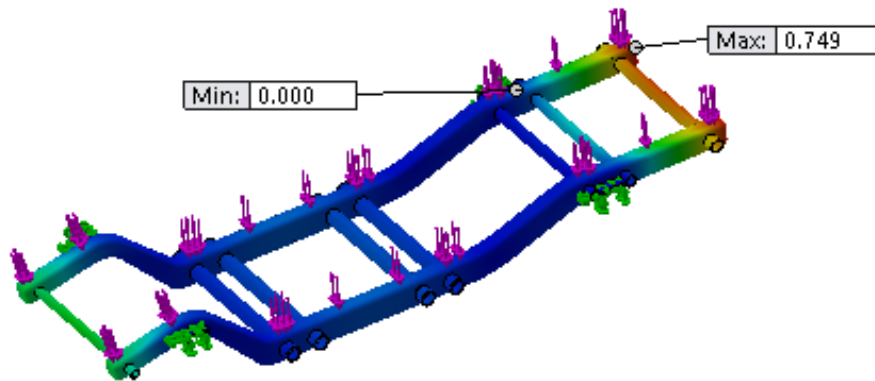


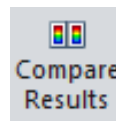
The following images show the results obtained from the applied loads.

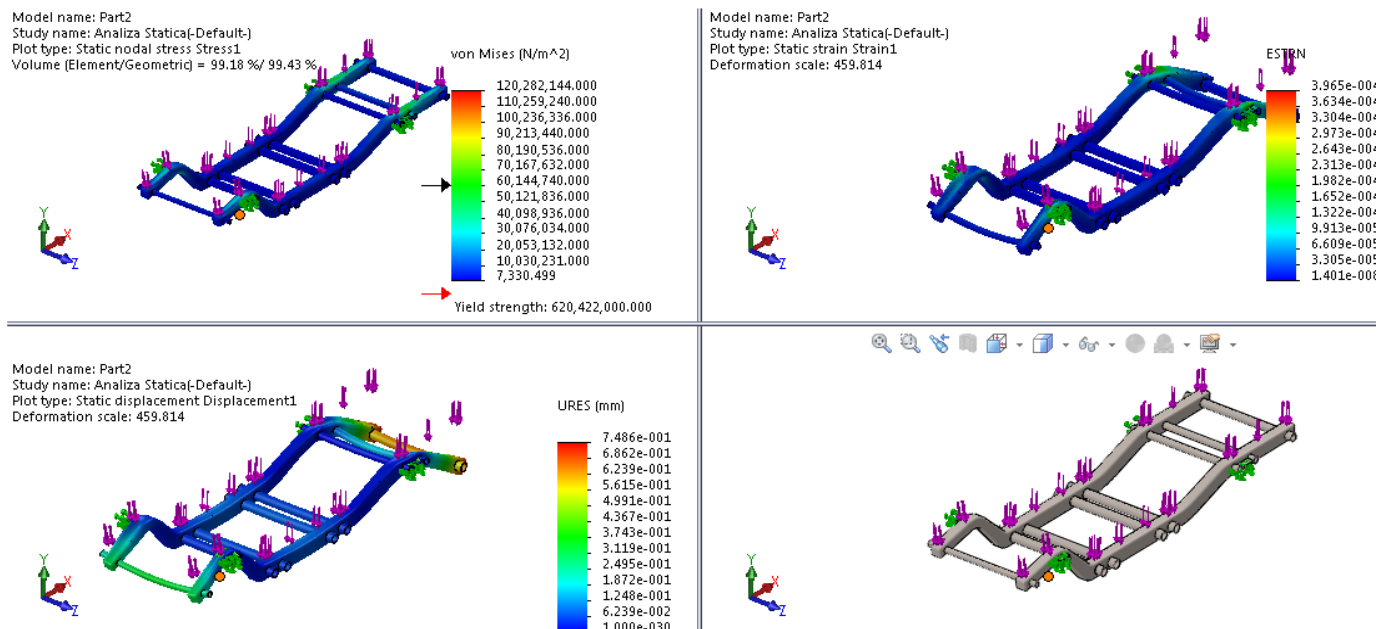


If you want to display the minimum or maximum value of the displacement, right-click on the result  **Displacement1 (-Res disp-)**, click – **Settings..., Chart Options**, where you check the display

of the minimum and maximum values  , the minimum and maximum values will be displayed on the analyzed frame as in the following image:



Comparison of studies can be done in full screen by clicking on the  icon in the toolbar. In the figure below, the comparison of results from the static simulation of the frame was chosen.



III. Self- evaluation test

1. In the example above, change the material, using aluminum for the frame crossbars. What is observed?
2. Propose constructive solutions to reduce stress in the most exposed areas.
3. Perform a static structural analysis of the X-braced frame, which has the same geometric characteristics as the one presented in the previous example. Compare the modeled and simulated frames in terms of structural performance.

