

EXERCISE 5.2 - CREO SIMULATE: A SUPPORTING PLATE CASE

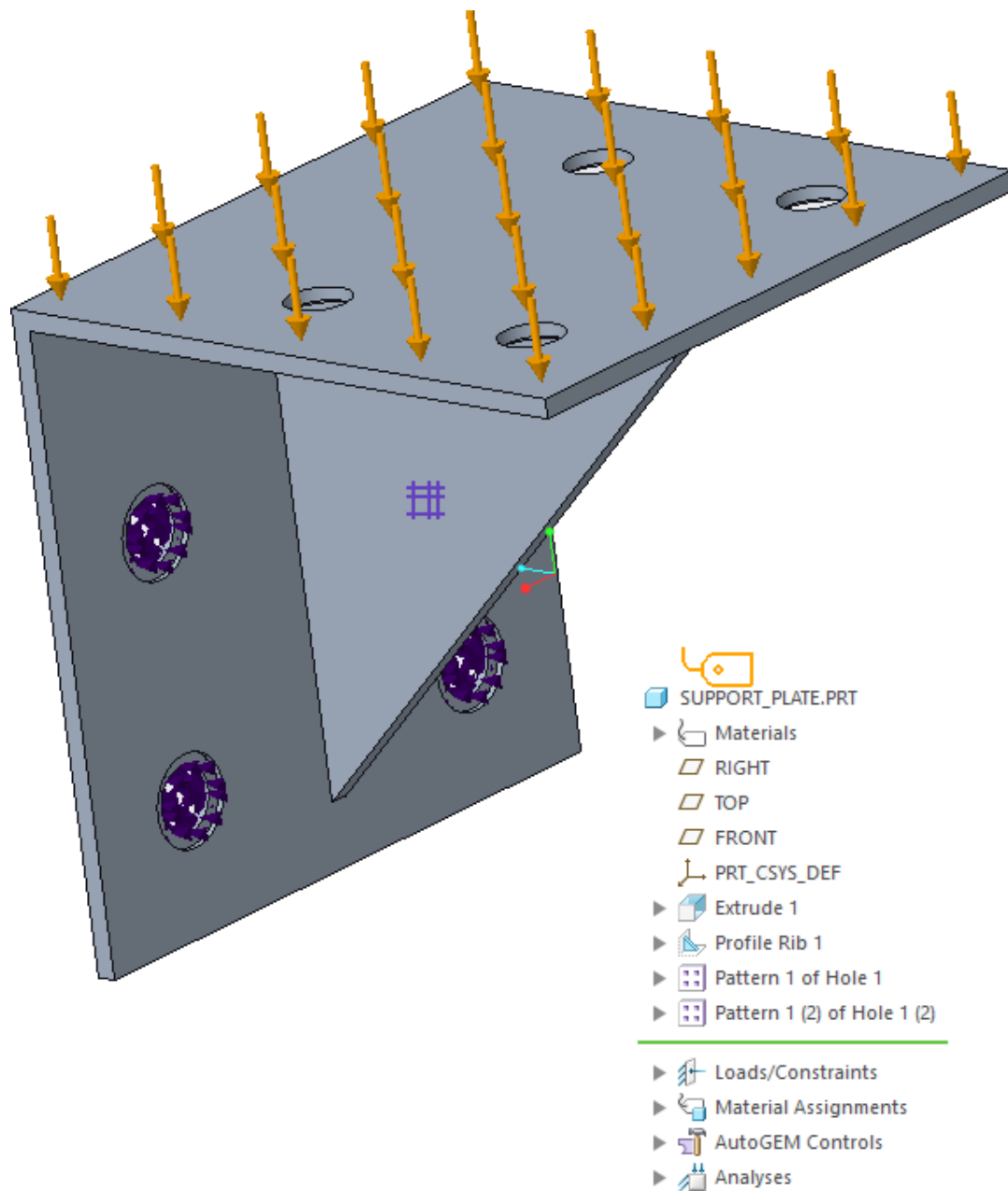


Figure 1: Creo Simulate model of the supporting plate and its model tree.

Learning Targets

In this exercise you will learn:

- ✓ Start Creo Simulate mode
- ✓ Set Material values
- ✓ Set Constraints
- ✓ Set Loads
- ✓ Perform an analysis
- ✓ Define mesh
- ✓ Viewing results
- ✓ Perform an optimization study

Utilized program is Creo 6.0.2.0, but the procedure described will also work with the older versions of Creo.

About This Exercise

In this exercise, a simple FEM analysis is performed. In addition, optimization tool is used to help to design a part. The idea of this exercise is to demonstrate the simulation capacities of Creo by performing a static stress analysis. There are also other simulation options available.

Getting Started

Copy support_plate.prt.1 from MyCourses) to your working directory and open it.

About the Model

As you notice, our support plate contains one L-shaped extrude, one rib and two four-hole sets. The thickness of extrude and rib is 3 mm.

Building the Creo Simulate Model

Open **Simulate** (🧩) from **Applications** tab (Figure 2). Notice that a new set of tabs in the ribbon.

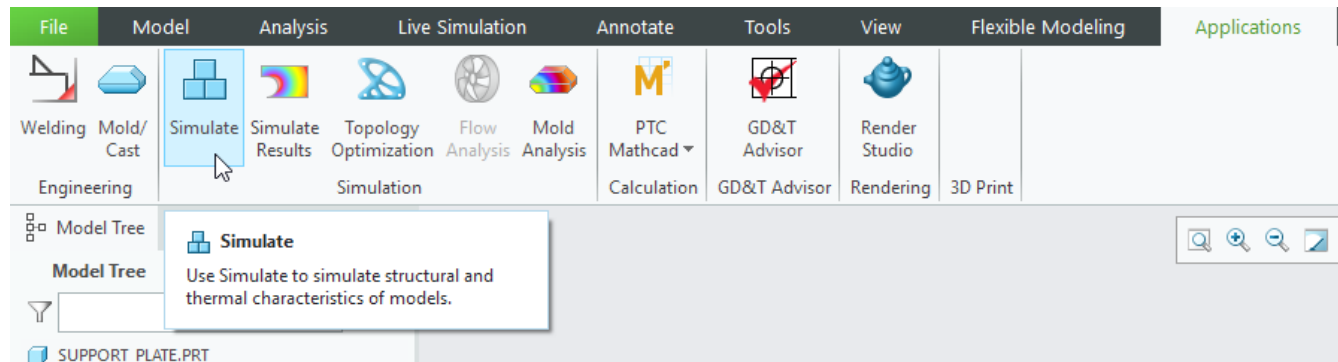


Figure 2: Selecting *Simulate* from *Simulation* group in the *Applications* tab.

Material

Default Creo parts may not have any material values (Aalto installation uses steel as a default material), but to simulate the behavior of the model we need to define it. Select **Material Assignment** (🔧) from *Material* group. A Material Assignment window opens. As you can see, STEEL is selected as material and it affects the whole part (Figure 3). If need to change material in the simulation, you can select More from the window. Click **OK** to accept the

material assignment. Now in the model tree, Materials and Material Assignment features are presented.

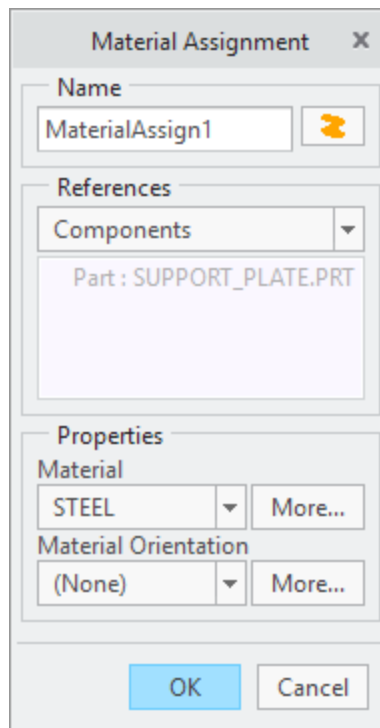


Figure 3: Steel assigned as the material in the simulation.

Constraints

Next task is to constraint our part. If there are now constraints, the part hovers freely in the space and the information value of the simulations are minimal. Select **Displacement** (📐) from the *Constraints* group. Select the inner surfaces of the four holes as seen in Figure 4 as references (remember CTRL!). There are two of them, select the deeper one (smaller diameter). The default constraint options are fine (translations fixed); accept the *Constraint* (OK). Notice that Loads/Constraints feature has appeared to the model tree.

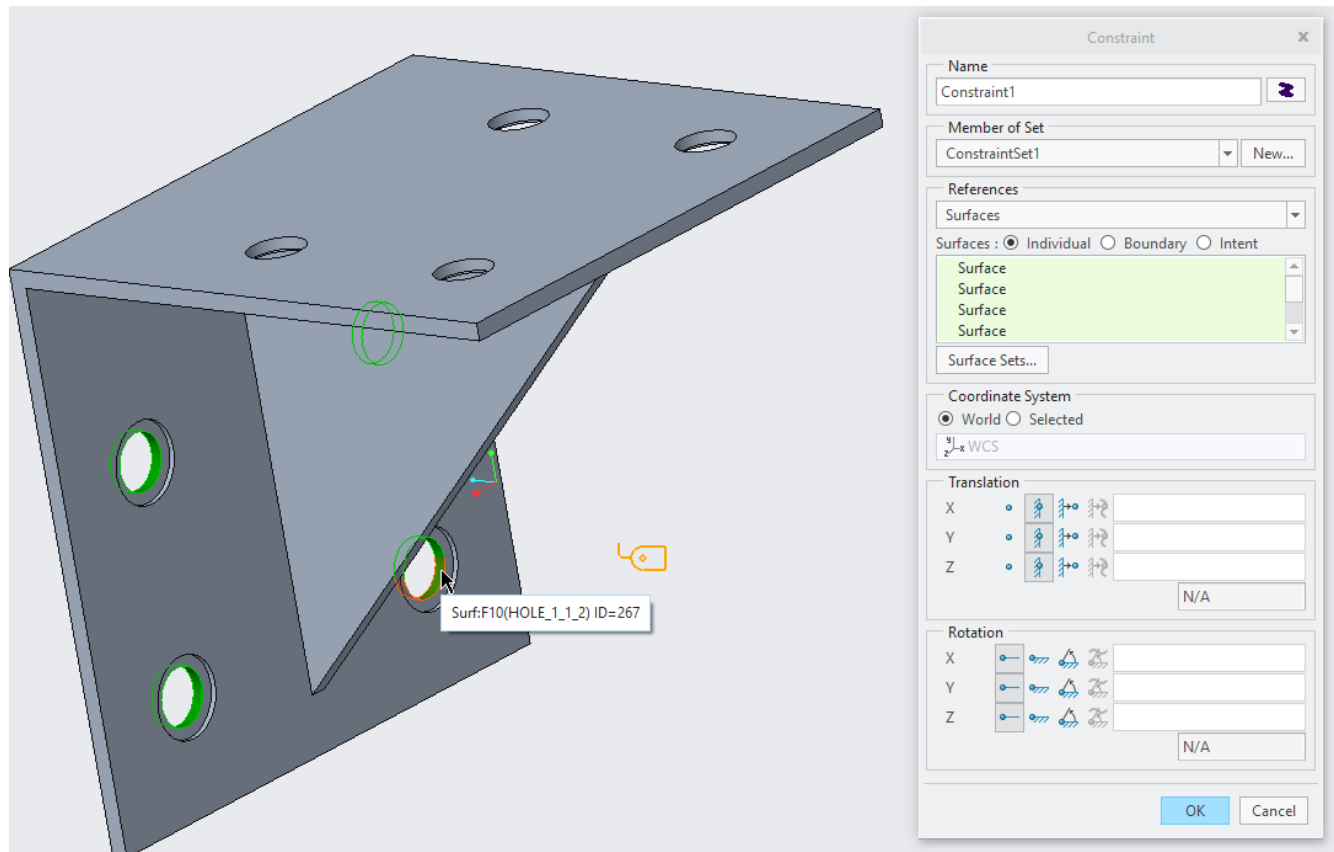



Figure 4: Four surfaces (highlighted in green) selected as references.

Loads

We have material and constraints defined; next we need to define loads. Select **Force/Moment** () from *Loads* group. A *Force/Moment Load* window opens. Select the upper surface of the L-profile as reference. As you can see from the bottom right corner, the Y-axis in Creo points upwards. Set value of **-1000** to Y Force. Click **Preview** to see that the force arrows points downwards (Figure 5) and click **OK** to accept the force.

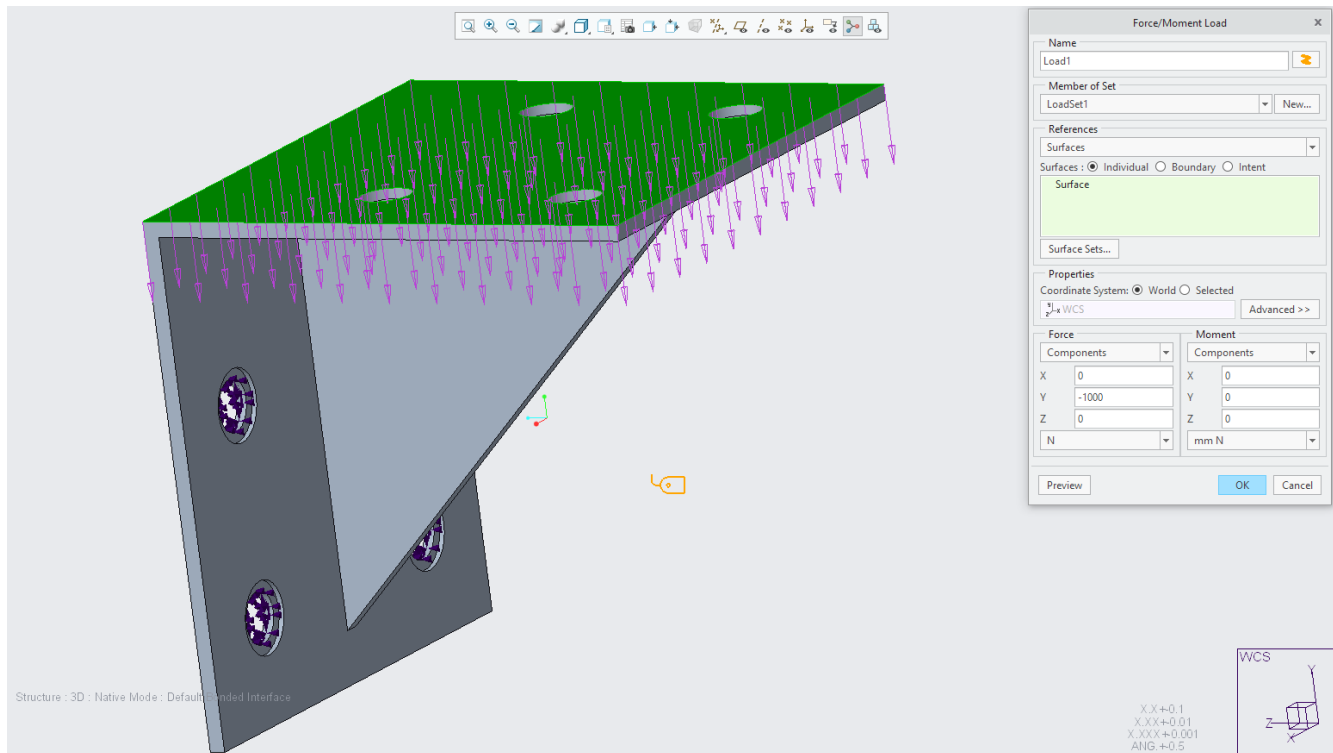


Figure 5: Values set, viewing preview.

Performing a simulation

Now the material, constraints, and loads are defined, time to perform a simulation. Select **Analyses and Studies** (🔧) from *Run* group. A new window opens. Select **File** and then **New Static** (Figure 6).

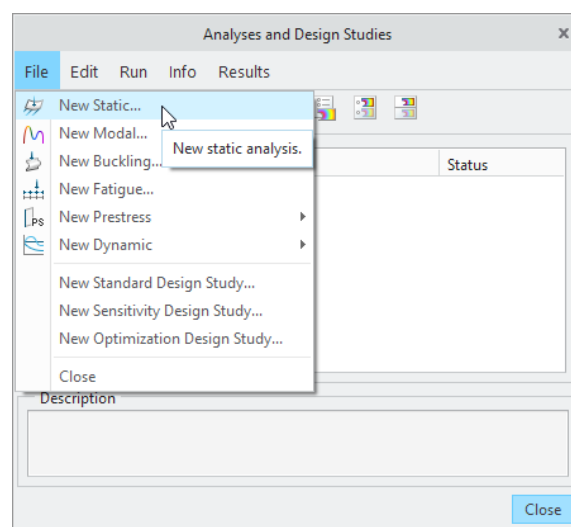


Figure 6: Selecting *New Static* under *File*.

A *Static Analysis Definition* window opens. Notice that constraints and loads are already selected to the simulation. Name analysis to *Static1* and click **OK**. Now this analysis is listed in *Analyses and Design Studies* list. Click **Configure run settings** (📄) or **Run → Settings**). A *Run Settings* window opens. Change *Memory Allocation* to **2048** and if needed change the output directories (default is the working directory). Click **OK** to accept the settings. Click **Display study status** (📄) or **Info → Status**) to open a log window. Click on the green flag (🚩) to start a simulation and select **Yes** to question about interactive diagnostics. The status of the study can be seen from log window.

When the simulation is ready, check that everything is in blue or yellow in *Diagnostics* window and click **Close**. Notice, that *Static1* status in the list is *Completed*.

Viewing results

When simulation is completed, click **Review results** (📄) to see the results. Background will change and a *Result Window Definition* window opens. Check that *Stress*, *von Mises* and *MPa* are selected. Click on **Display Options** tab and check **Deformed** and **Show Element Edges** options (Figure 7). Click **OK and Show** to see the results (Figure 8). *Note! The Result window may be under the current simulation model window!*

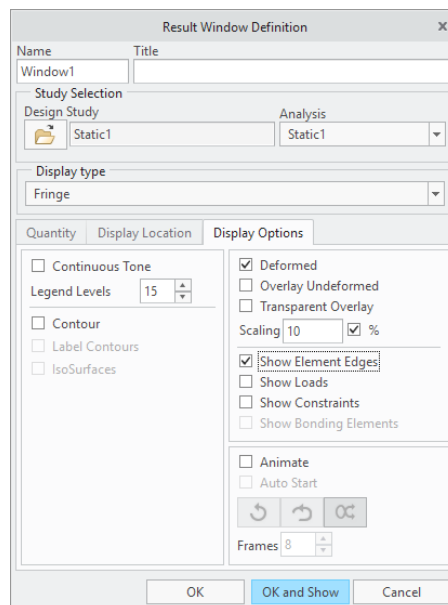


Figure 7: Result Window and options there.

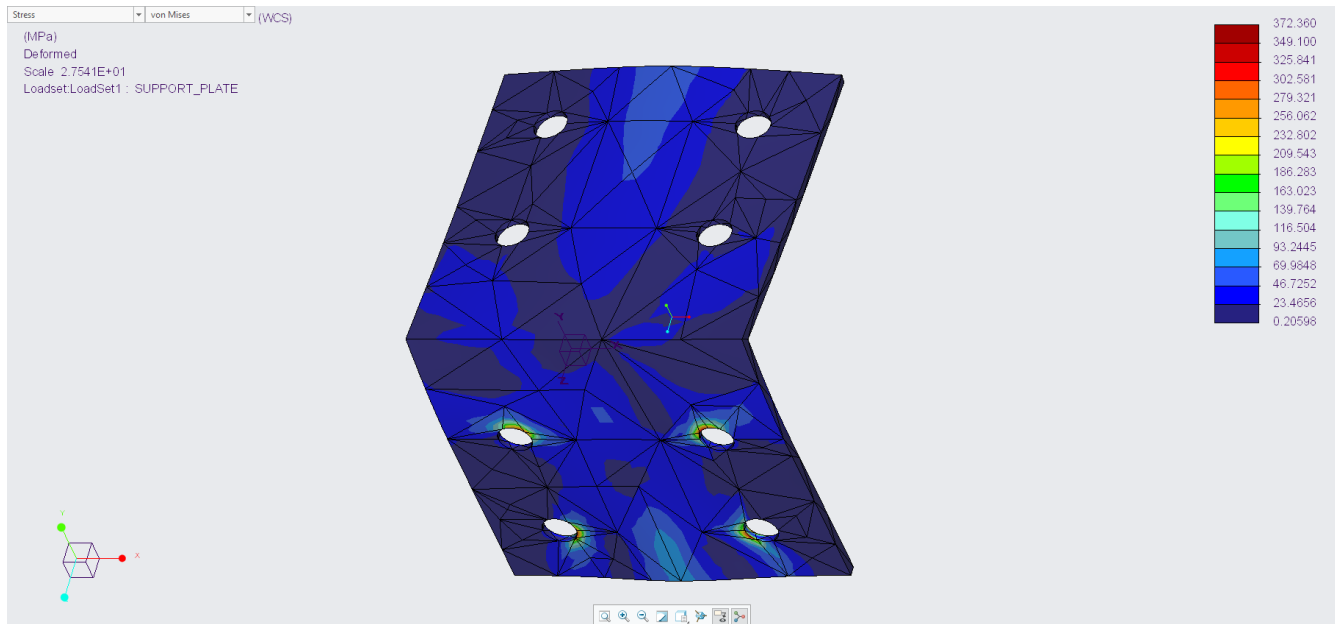


Figure 8: Creo Simulate results window and deformed plate.

The result window shows where the highest stress (Stress von Mises) is and how the part will deform under the stress. Please note, that the deformation is scaled up. Also, the scale of colors is relative to min and max values. If you want to see for ex. where the stress is higher than 250, a scale is needed to change. Select **Format** tab and **Edit** (🌈) from *Legend* group. Give **0** as *Min* and **250** as *Max*. Click **OK** to see the changes (Figure 9).

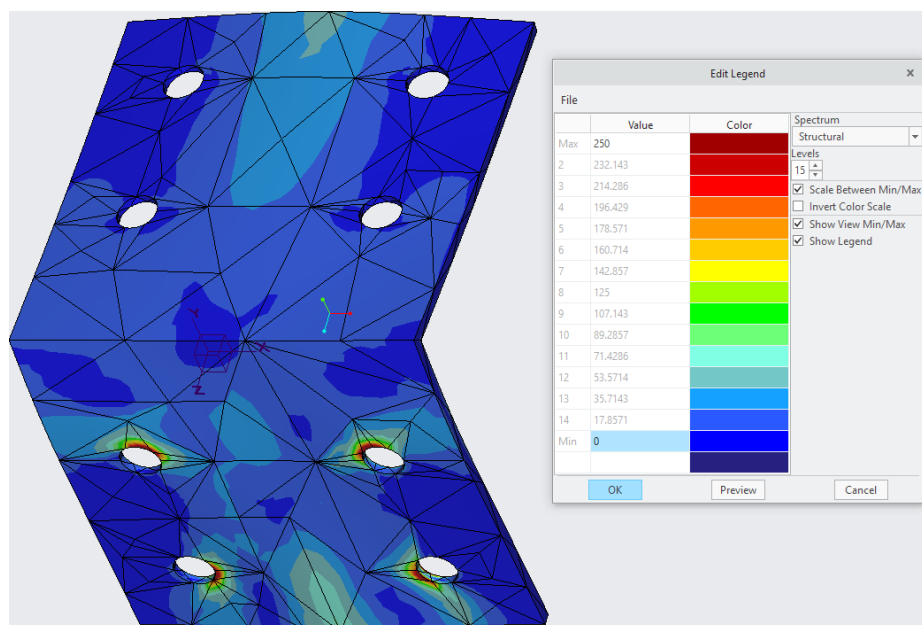


Figure 9: Color scale changed.

To see other simulation results, from **Home** tab, click **Edit** (✎). Same *Result Window Definition* window will open. In the **Quantity** tab, change *Stress* to **Displacement** and as *Component* select **Y**. Click **OK and Show** to see the displacement of the part in Y-axis direction (Figure 10). Please note that positive side of the Y-axis is upwards.

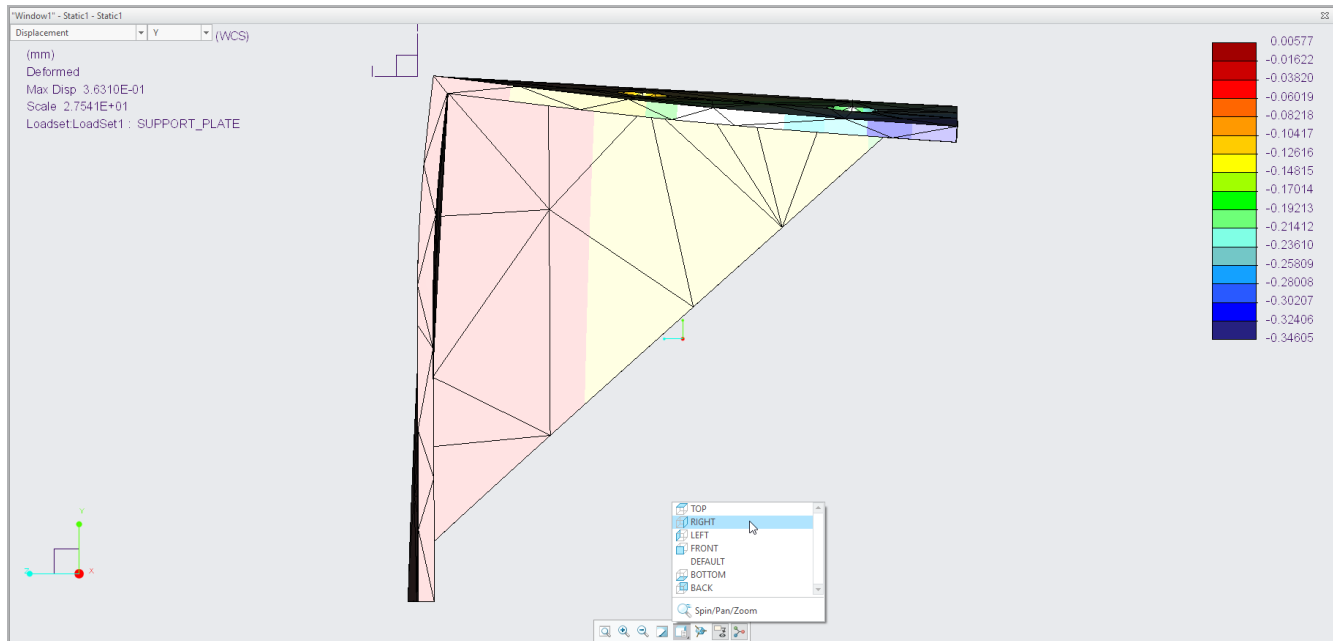


Figure 10: Displacement in the Y-axis direction.

This is how basic static simulations are done. There are lots of other things to simulate. Select **File, Close** (⌵) and click **Don't save** to close the result window. You are back in the *Analysis and Design Studies* window. **Close** both windows.

Changing meshing options

As you can see from previous pictures (Figure 9 and Figure 10), the mesh quality is a little bit bad. The FEM method divides geometries to smaller elements (triangles for ex.) and then calculates those. To get more accurate results, a tighter mesh (more elements) is needed.

Select **Refine Model** tab. Select **Maximum Element Size** (📏) under *Control* from *AutoGEM* group. A window appears. Select **Components** as *References* and give **6** as *Element Size* (Figure 11). Click **OK** to accept changes.

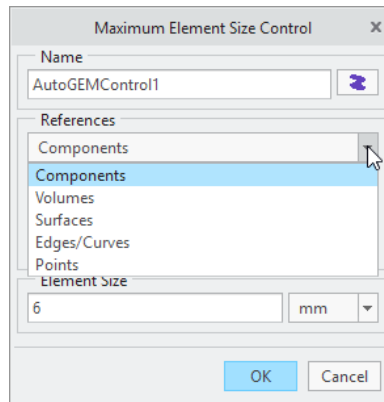


Figure 11: Maximum element size defined.

What did we just do? We said to the mesher, that the maximum size of elements, which part is divided into, is 6. To see the mesh, select **AutoGEM** (🔍) from *AutoGEM* group and select **Create**. This will take some time. The result will be similar as seen in Figure 12. In the example case, a mesh containing 7087 elements is created (you may have different, but close value).

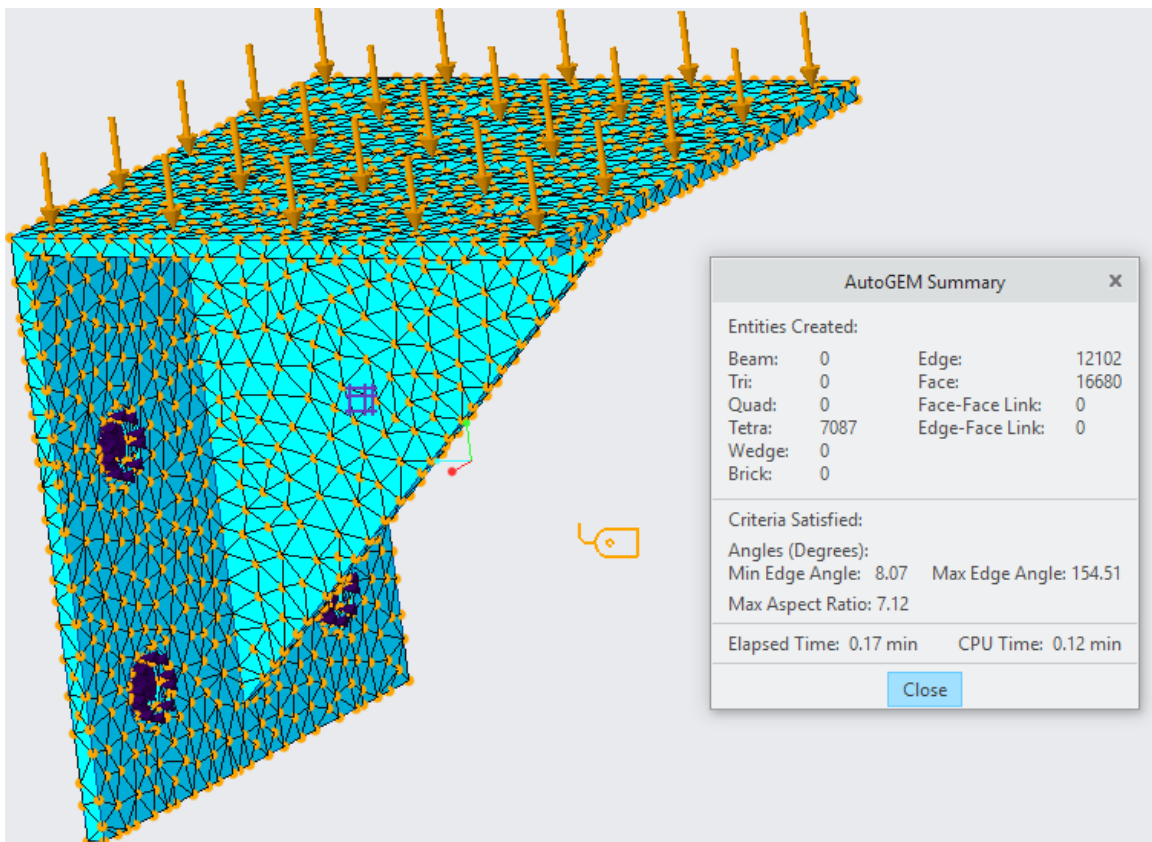


Figure 12: Created mesh.

Close the *AutoGEM Summary* window. **Close** also AutoGEM window and select **No** to not save the mesh to the current model.

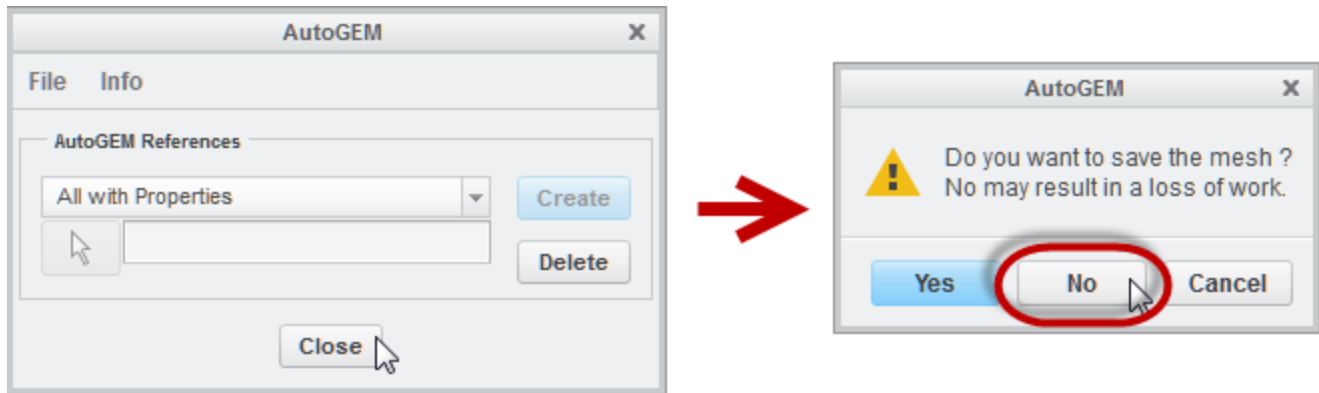


Figure 13: Closing *AutoGEM* without saving the mesh!

Optimization

Next we are using optimization to find out what is the smallest possible thickness of the plate that can hold 1000 N while stress stays under 250 MPa. Open **Analysis and Design Studies** (🔧) window from **Home** tab. Select **File** and **New Optimization Design Study** from the list. Give *optimization1* as a name. Check that in the *Goal* field *Minimize* *total_mass* is selected. To the *Design Limits* field, add a **new row** (🔼🔽). From the *Measures* list, select **max_stress_vm** and click **OK**. Give value of **250** to value field and notice, that stress should be smaller (<) than given value. From *Variables* field, click **Select dimensions from model** (🔍) to add dimensions that are changed during design study. Select the **Extrude 1** and select the **3 mm** dimension (d3). Click 🔍 again to add a new dimension. Select **Profile Rib 1** and then **3 mm** dimension from there (d4). Update the *Variables* field as seen in Figure 14.

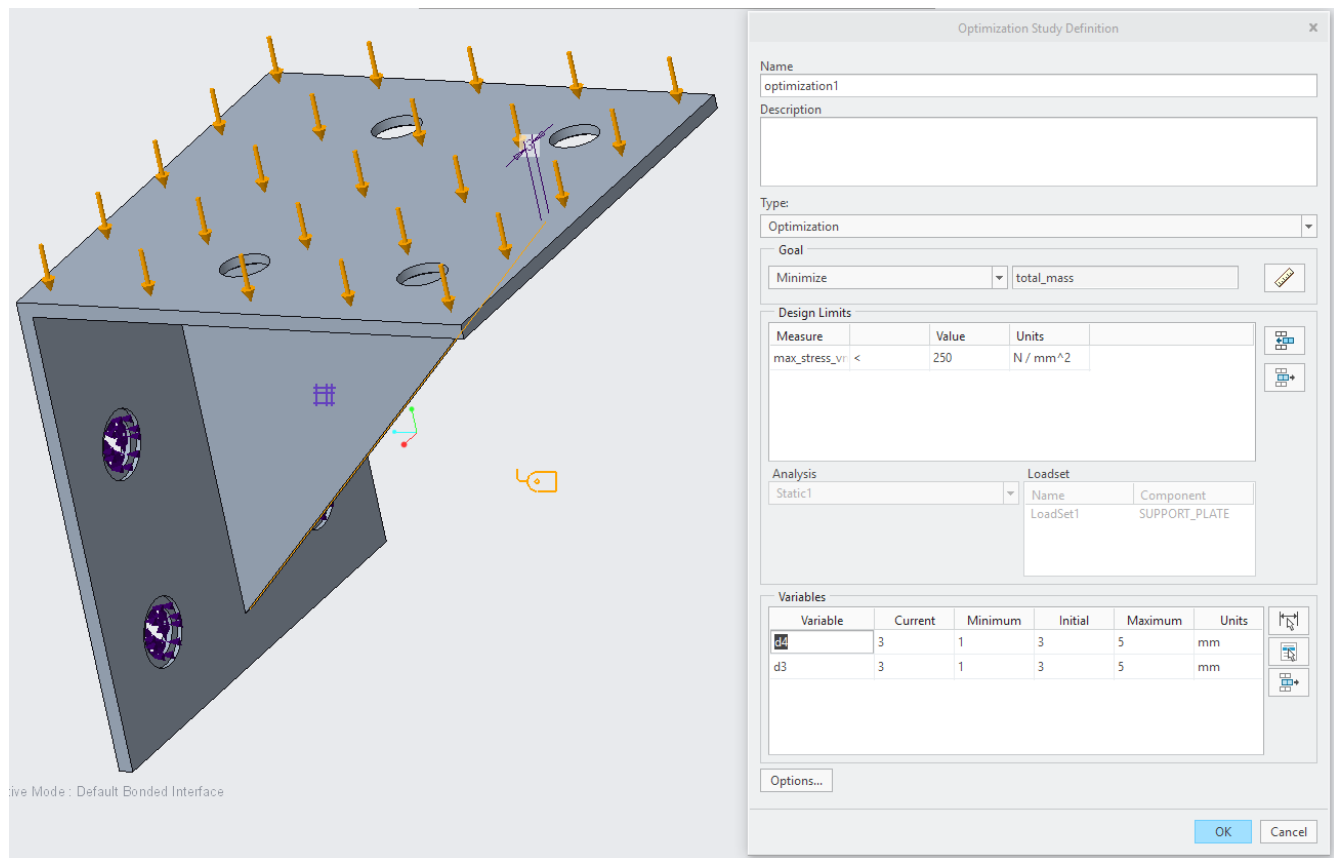


Figure 14: Bottom part of the Optimization Design Study Window.

Performing the optimization study


Click **OK** to accept the *Optimization Design Study*. Open the log window (📄) and start an analysis (🚦). Select **Yes** when prompted about interactive diagnostics. Wait until Creo performs the optimization study. It may take about 10-15 minutes (depends on hardware etc.).

Looking the results

When runs are completed, scroll in the log windows **Summary** tab to find out the last values of d3 and d4 dimensions. It may look like in Table 1. The final values of design variables are **bolded** in example.

Table 1: Part of the Run Status window (Summary tab).

Begin Optimization Iteration 5	(12:25:54)
Result of Optimization Iteration 5	
Parameters:	
d3	3.71227
d4	1.43209
Goal: 5.8909e-04	
Status of Optimization Limits:	
1. max_stress_vm 2.5000e+02 < 2.5000e+02 (satisfied within tolerance)	
Resource Check	(12:26:39)
Elapsed Time (sec):	562.24
CPU Time (sec):	457.31
Memory Usage (kb):	4156663
Wrk Dir Dsk Usage (kb):	20
Begin Optimization Iteration 6	(12:26:39)
Converged to optimum design.	
No improvement to the initial design was found that does not violate any limits.	
Optimization study statistics:	
Number of Base Analyses: 6	
Number of Perturbation Analyses: 13	

To see the results in action, select **Review results** () using *Stress, MPa* and *von Mises* as *Quantity* and **Deformed** and **Show Element Edges** as **Display Options**. Result may look like in Figure 15. Notice the thicker L-shape and thinner rib.

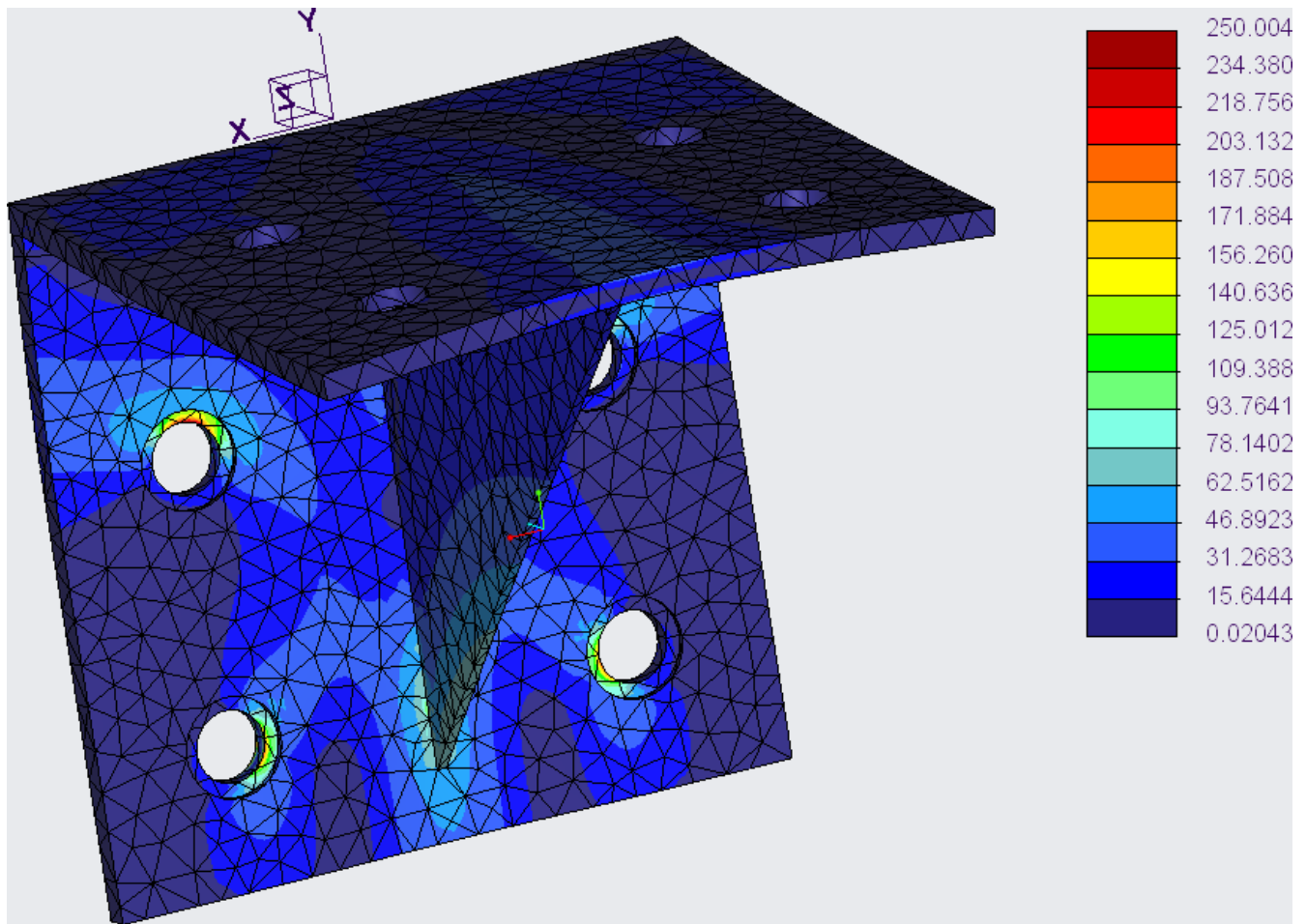


Figure 15: The result of optimization study. Notice the thickened L-shape and thinner rib.

Exporting Results

Select **File→Save As** and select **Creo View (*.pvs)** to save the visible model as a standalone Creo View model. You can name the file freely. (Figure 16)

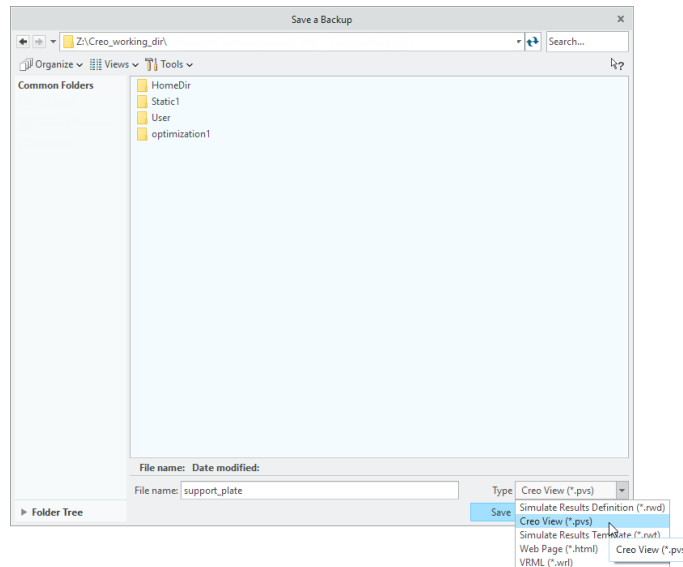


Figure 16: Saving results as a Creo View file.

To check that the export was a success, navigate to your saving folder (by default working directory) and open the ***.ol file** (.pvs file creation also creates an .ol file). Result may look like in Figure 17.

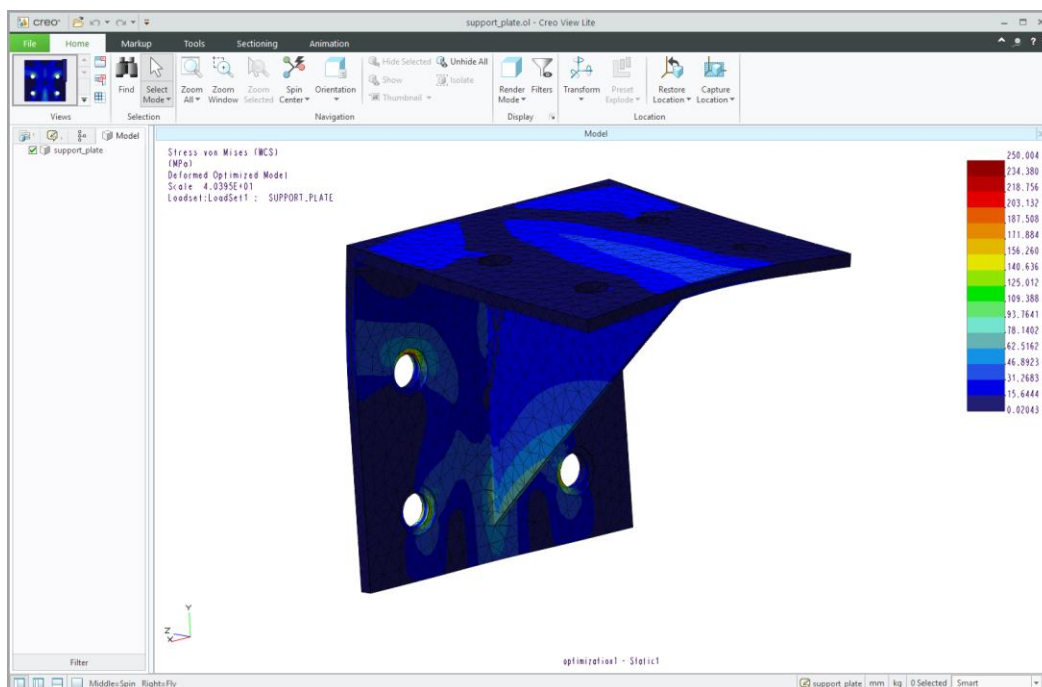


Figure 17: Viewing saved result as a *.ol file

This concludes this exercise. Return the created ***.ol file** to MyCourses.