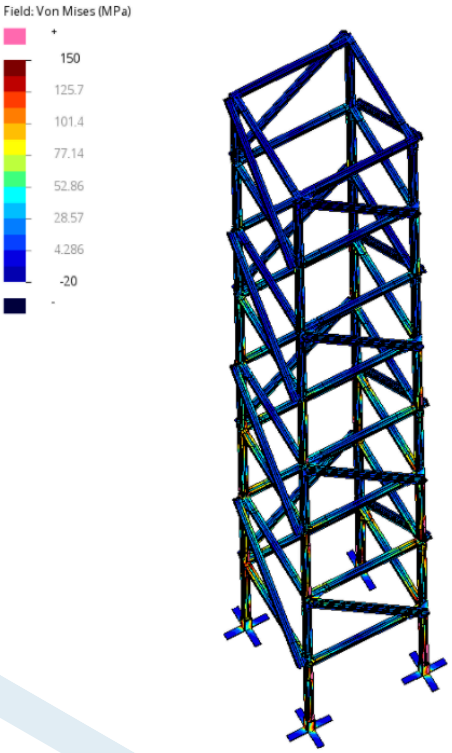
AKSELOS



DECISION SUPOPRT SYSTEM ON A BEAM SYSTEM

Application note

Objectives:

- Run an automatic analysis using the Decision Support System (DSS) in Akselos Modeler
- Visualize analysis results.

The Decision Support System (DSS):

Decision Support System is a powerful tool that provides engineers with a holistic view of the model's strength and stability, at the same time, for different loading conditions. In this exercise, we will use the DSS to identify components that may fail at a specified tolerance limit for a particular result field and a particular load combination.

Model Description:

The model below is a finite element representation of a beam system. The model is built by prefabricated aluminum beams made by the company Makerbeam, the sectional properties can be found in Figure 1.

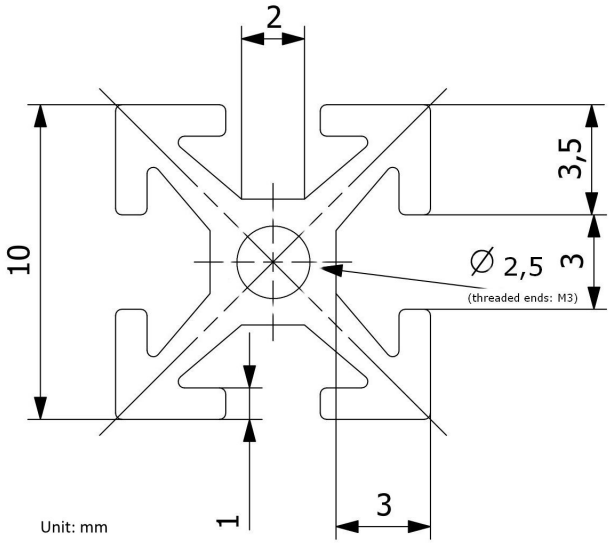


Figure 1: Makerbeam Cross-section (unit: mm)

The dimensions of the structure are:

X: 200mm
Y: 200mm
Z: 2 x 210mm = 420mm

Stiffeners/braces are located at every other level as corner diagonals. The structure-ground connection are fixed at the bottom of each of the 4 legs. Each leg will be connected to “ground” through 4 L-brackets.

Figure 1: Model schematic

Model properties:

Elastic Modulus	200 GPa
Poisson Ratio	0.30
Density	7850 kg/m^3

Table 1: Model Properties

Suggested Exercise Steps:

- Assemble in Akselos Modeler. *(Refer to the Component Editor tutorials in the Akselos User Manuals to learn how to create components in the Component Editor.)*
- Set up load cases and load combinations.
- Set up tolerances for some fields of concern.
- Perform DSS with Akselos Cloud.
- Review the results.

Setup of the DSS:

Before setting up the DSS analysis tool, you should load the model to Akselos Modeler and configure the necessary load cases and load combinations. In this example, a section of the Beam system model with 86 components is used. The load combinations in this example are all gravity loads applied to all components of the model. There are 5 load combinations that will be analyzed in this example, each one models the gravitational force of a different configuration of wind loads and wave load. *(Refer to the Shiploader tutorials in the Akselos User Manuals to learn how to create Load Cases and Load Combinations in Akselos Modeler.)*

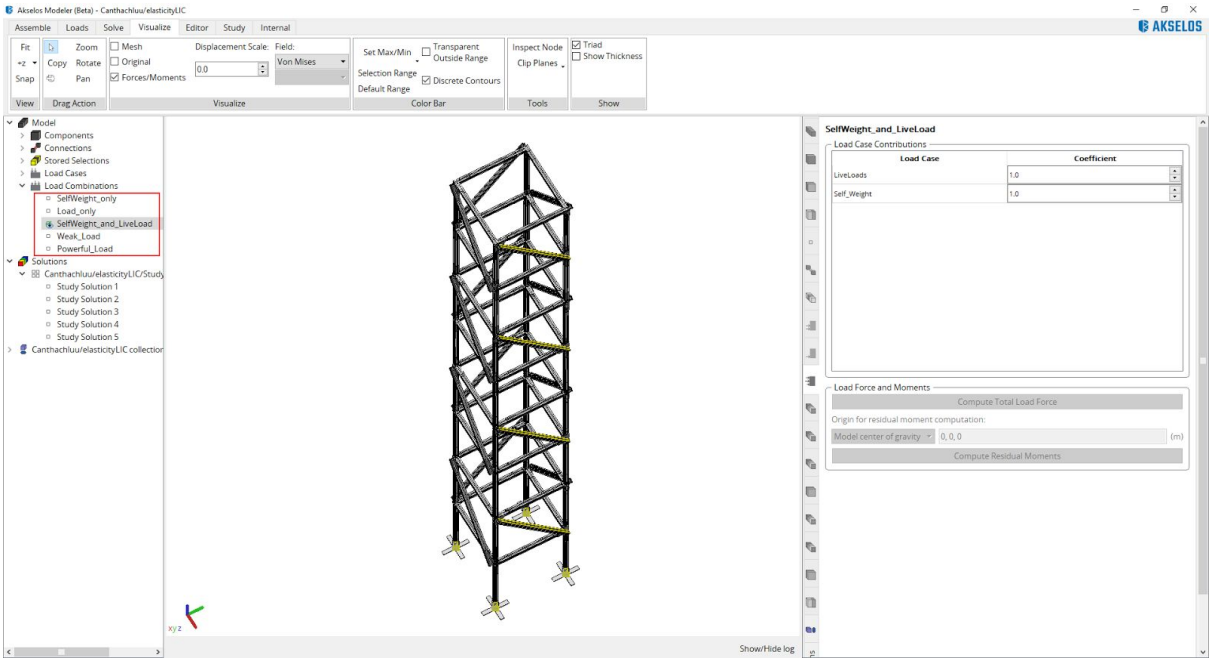


Figure 2: Load Cases and Load Combinations

- Tolerance Limits Configuration: To detect possible failures in the components of the model with the analysis tool, it is necessary first to define tolerance limits for each of the result fields that are relevant for the analysis (e.g. displacements, stresses, strains, etc.). Choose all components, then click on *Study* tab in the properties panel.

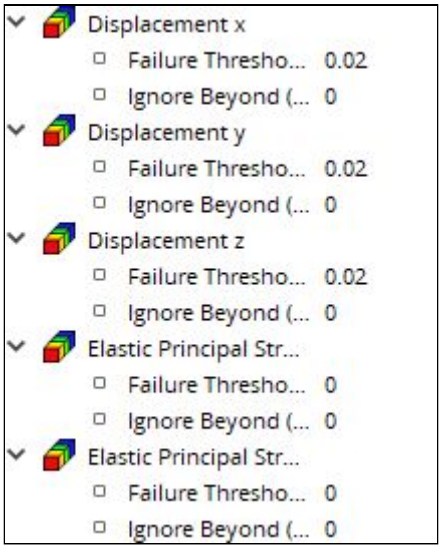


Figure 3: Tolerance Limits Configuration For Volumes

- The tolerance limits table is used to input the limits that are defined at the component or volume level. For each result field, two values can be set: a Lower Bound and an Upper Bound. A component (or volume) will be considered a failed case if its result for that field falls between the Lower Bound value and the Upper

Bound value. Initially all the bound values start with the ‘None’ value, indicating that they are not set. If the currently selected components/volumes have different bounds set, the ‘Multi’ word will appear instead of the values. It is possible to reset the limits to ‘None’ for a specific field by clicking on the broom icon.

Perform the DSS study:

- Click on Study tab in the properties panel.
- On *Solution Fields*, choose the result fields that are relevant for the analysis.
- Choose load combinations that you configured before.
- Finally, press *Perform Study* to run the DSS.

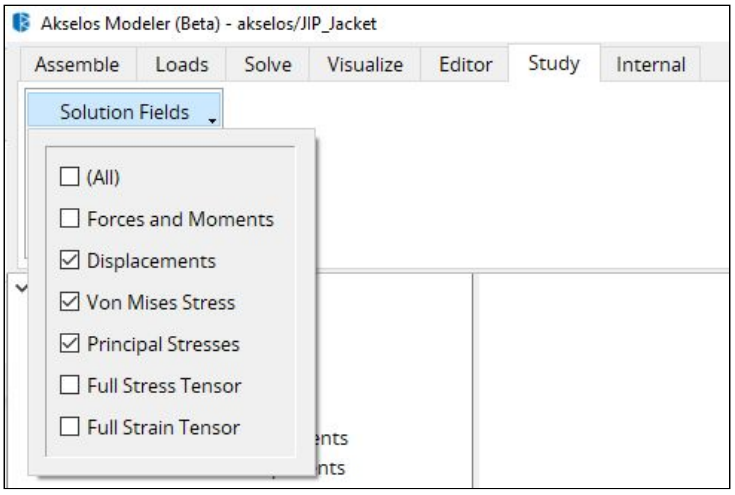


Figure 4: Select Solution Fields

Results:

After performing the DSS study, the results will be displayed as shown below. Components shown in red color are the components that failed at least one tolerance limit for the analysis of any of the defined load combinations. The more saturated the red color is, the more failed tolerance limits the component exhibited over the defined load combinations.

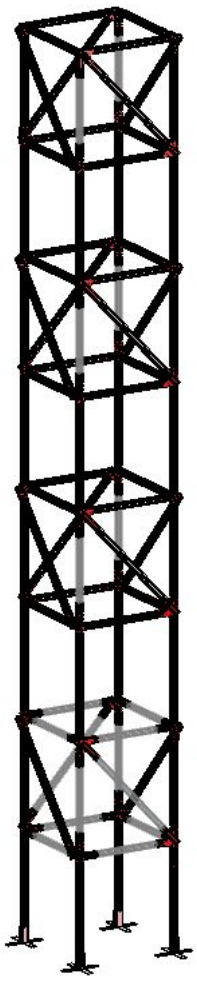


Figure 5: Analysis Overview

A summary of the analysis overview is displayed in the properties panel as a scorecard matrix. This matrix shows the number of components that have failed for each load combination. The higher the number of failed components the more saturated the red color is. With cells that show blank, it means there is no components that violates tolerances we set.

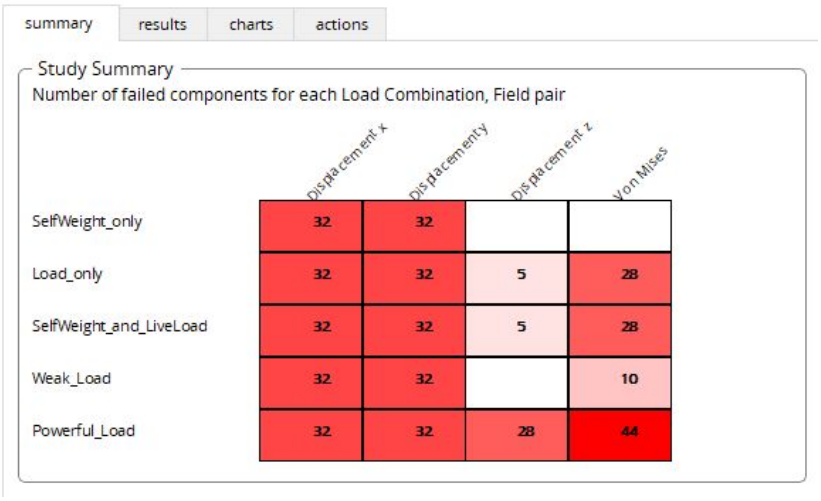


Figure 6: Analysis Summary

Clicking on one cell of the matrix will automatically select the corresponding components that failed for that specific load combination (Figure 7).

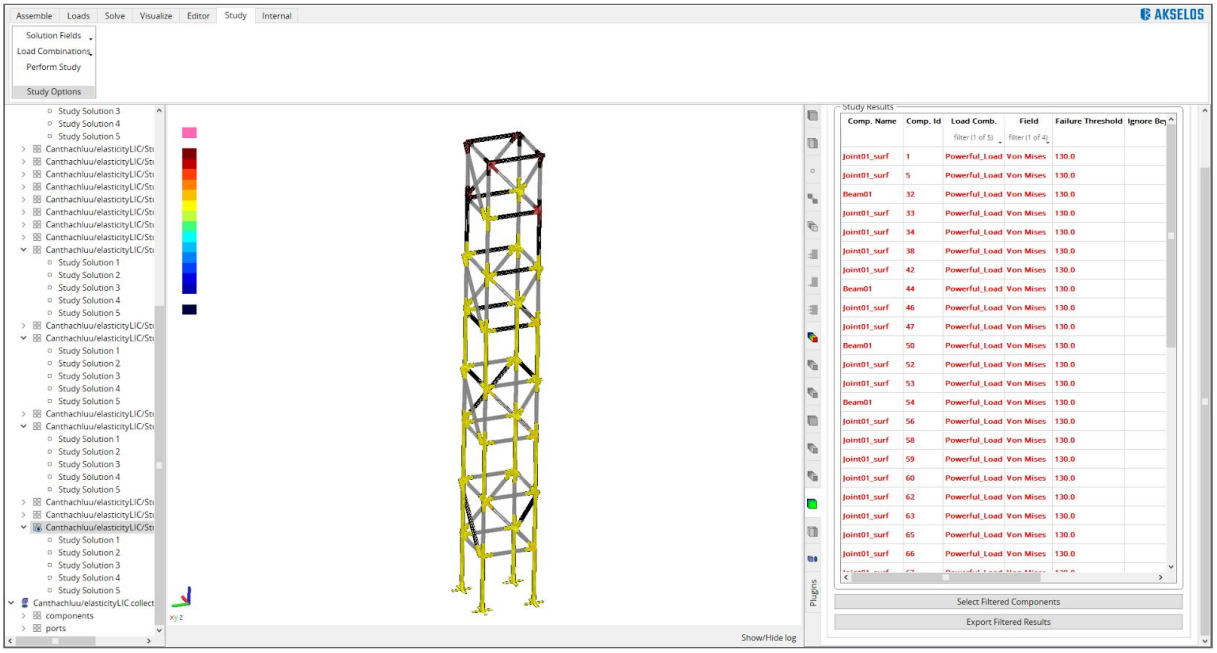


Figure 7: Corresponding components that failed for that load combination/field result pair

In the properties panel, you can click on the ‘results’ tab to get additional information about each result such as the ‘Worst Result’ and ‘Worst Component’ for each load combination.

akselos/JIP_Jacket/Study 12

Study with 5 load combinations and 7 fields.

summary | results | charts | actions

Study Results

Load Comb.	Field	Num Failed	Worst Result	Worst Component
SelfWeight_Only	Displacement x	20	0.0102492645383 m	JIPfloor01left03
SelfWeight_Only	Principal Stress 1	21	215.002786485 MPa	JIPfloor02right01
SelfWeight_Only	Principal Stress 2	29	74.1585693033 MPa	JIPfloor02right01
SelfWeight_Only	Principal Stress 3	18	214.155741215 MPa	JIPfloor02right01
SelfWeight_Only	Von Mises	39	212.155210225 MPa	JIPfloor02right01
Wave_Wind	Displacement x	20	0.0109898839146 m	JIPfloor01left03
Wave_Wind	Displacement y	118	0.0380297414958 m	JIPfloor01midleft01
Wave_Wind	Principal Stress 1	18	244.738067421 MPa	JIPfloor03right01
Wave_Wind	Principal Stress 2	25	74.5157502127 MPa	JIPfloor02right01
Wave_Wind	Principal Stress 3	16	238.356701827 MPa	JIPfloor02right01
Wave_Wind	Von Mises	43	233.404894408 MPa	JIPfloor02right01
Wave Only	Displacement x	20	0.0101324440911 m	JIPfloor01left03
Wave Only	Displacement y	12	0.0271475687623 m	O711_20500

Figure 8: Analysis Summary Table

In the ‘charts’ tab, a histogram will be generated showing the distribution of components for the selected result field and load combination (Figure 9). Clicking on a bar in the histogram will select the corresponding components.

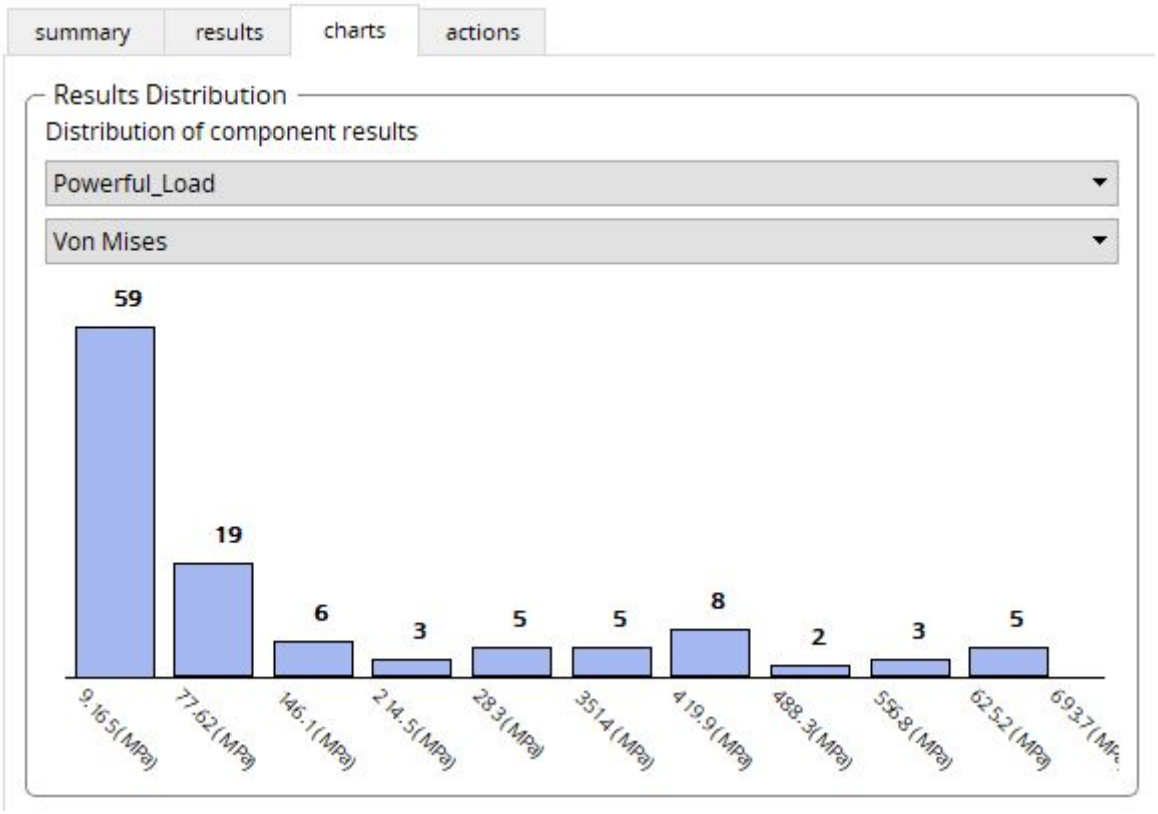


Figure 9: Analysis Charts

Finally, the 'actions' tab provides actions that allow further detailed exploration of the analysis results for a subset of components (Figure 7).

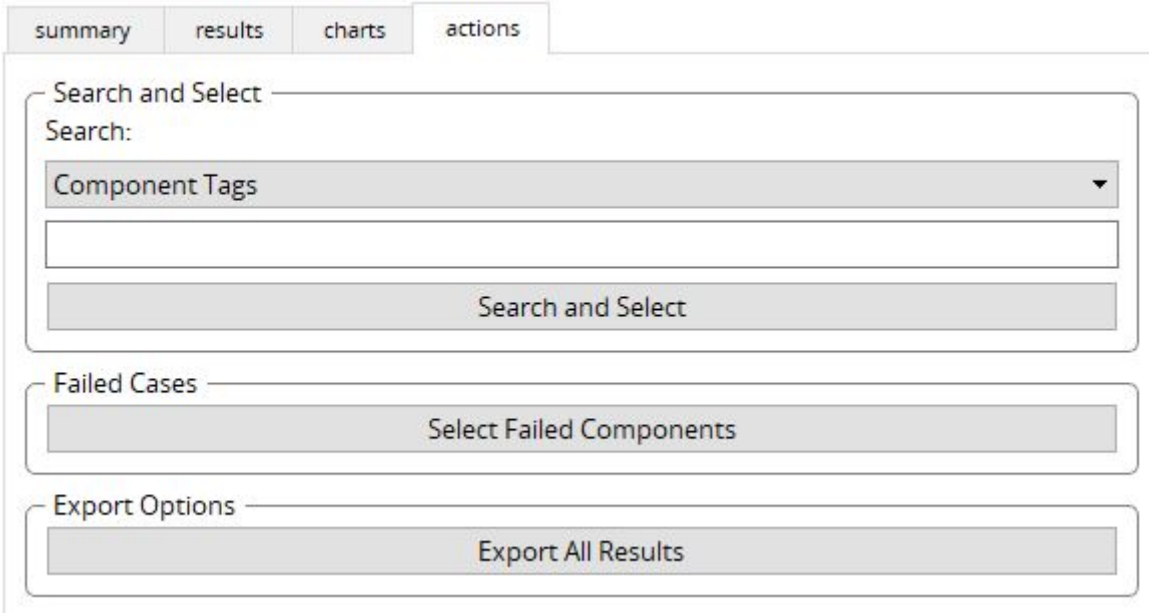


Figure 10: Analysis Actions

In addition to exploring the analysis results through the analysis overview matrix described in the previous sections, it is also possible to review in detail the results for each of the solutions generated by selecting the solution from the object tree to the left.

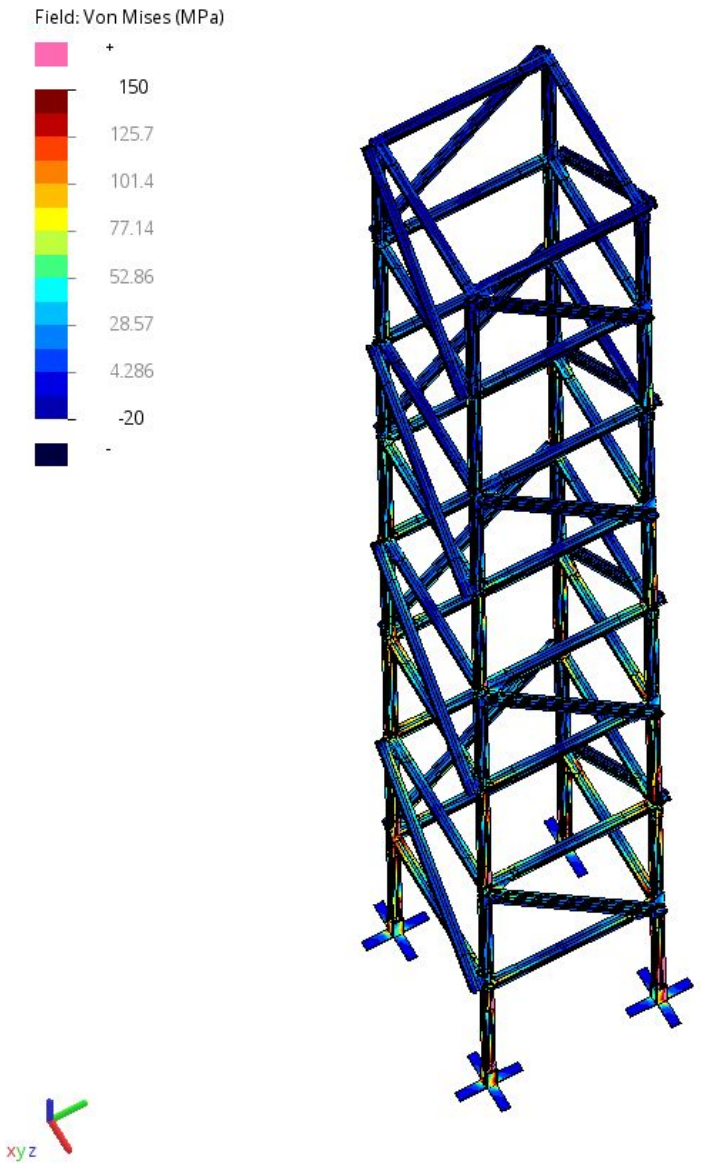


Figure 11: Von Mises stress of full model in a load combination