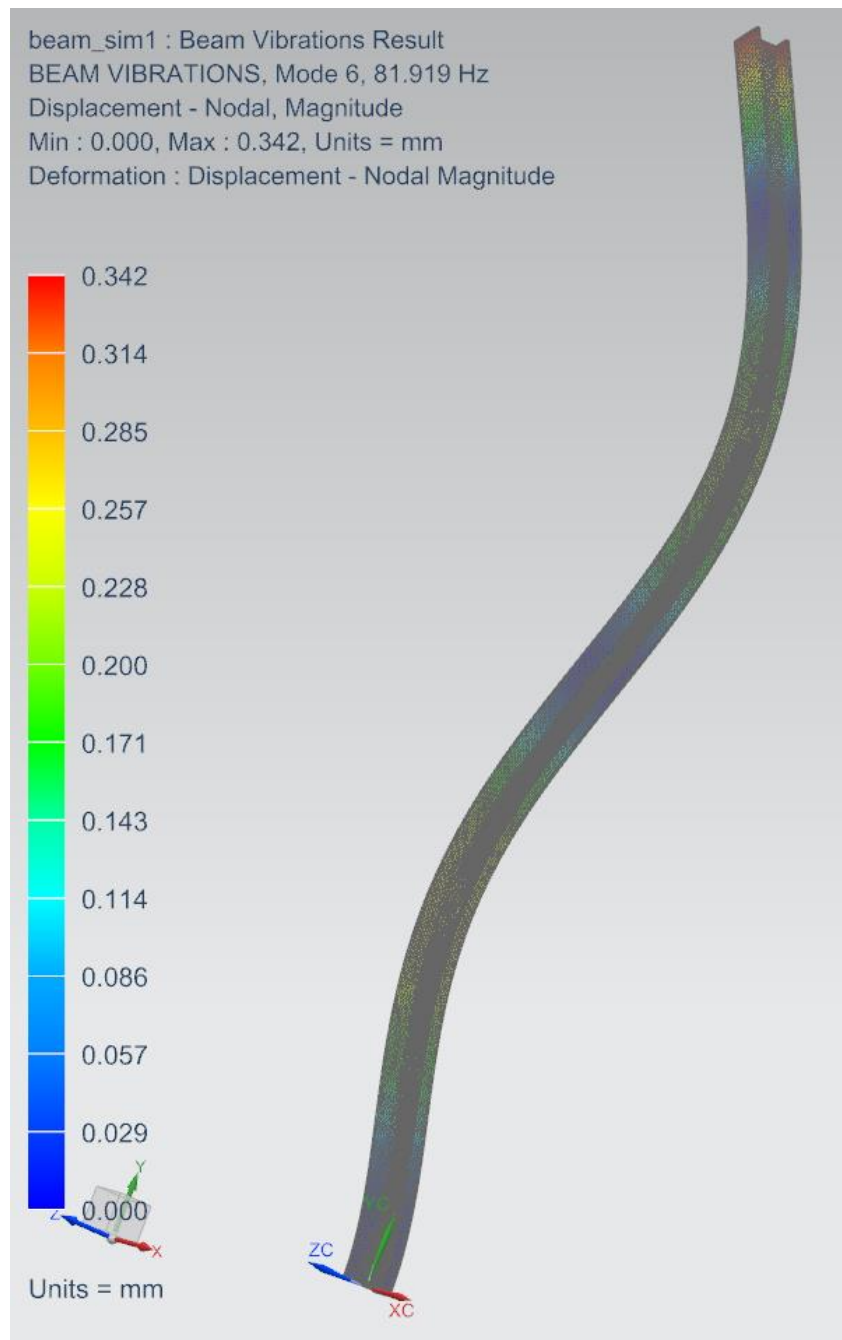


## NX.2 – BASIC OF FEM



**Figure 1: A vibration analysis of an I-beam.**

## Learning Targets

In this exercise you will learn:

- ✓ to create simulation model
- ✓ to define constraints, loads and meshes
- ✓ to run static analysis
- ✓ to run vibration analysis

Used program version is Siemens NX 10.0.2.

## Getting Started

Create an I-beam part (named for ex. **beam**) as seen in Figure 2. This part will be used as an example part in this exercise.

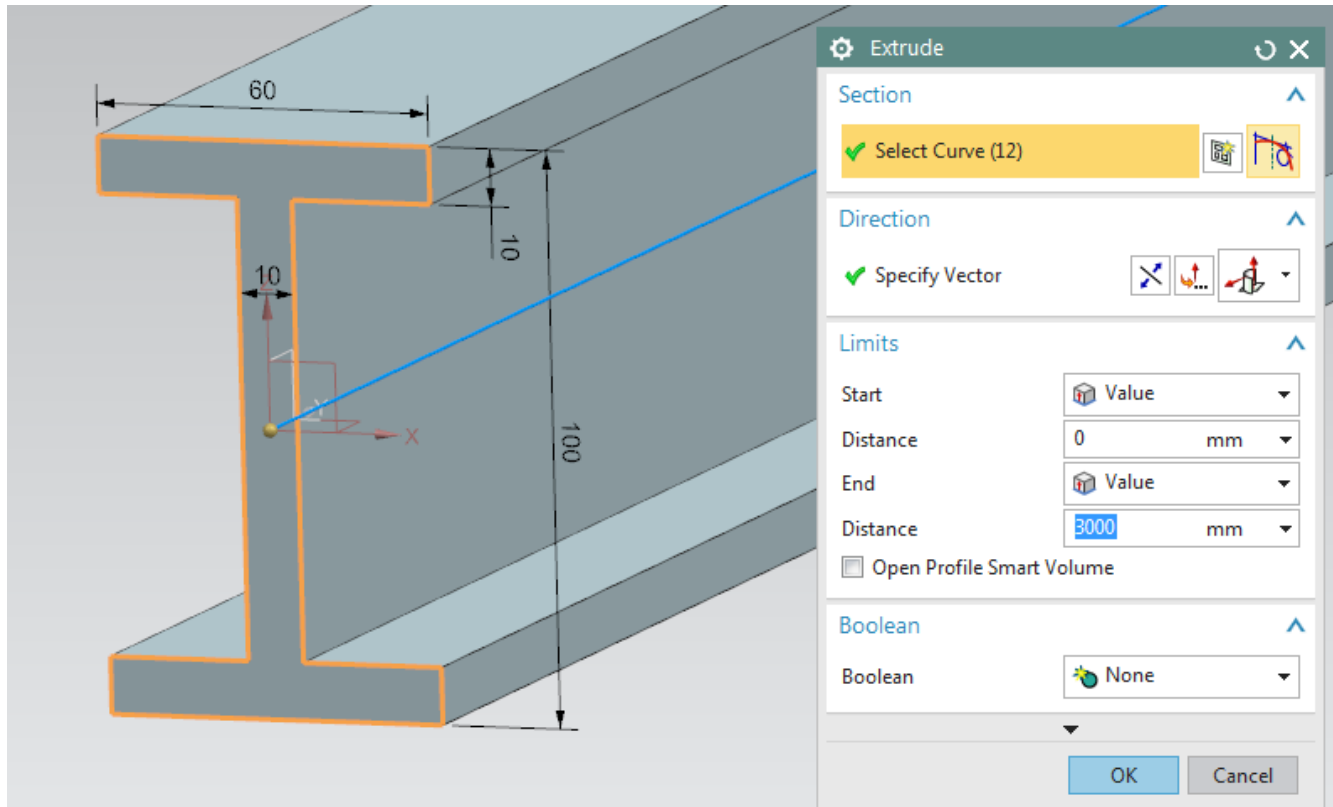


Figure 2: I-beam created on XY plane.

## Simulation Process

To perform a FEM analysis, several steps are needed:

- Create a simulation model (\*.sim)
- Create a simulation case (solution)
- Define used material
- Create a mesh file (\*.fem)
- Define loads and constraints
- Run simulation

## Design Simulation Mode

To enter FE analysis mode, select **Application** tab and **Design** (🔧) from *Simulation* group. A *New FEM and Simulation* window will open to create a \*.sim and \*.fem models. Default settings are fine for structural simulation (**OK**). When entered to Design Simulation mode, the program creates \*.sim and \*.fem models.

### Static analysis

A Solution window opens and asks what kind of simulation will be performed. One simulation model (\*.sim) can have several *Solutions*, i.e. simulation cases. First we perform a static analysis, so select **Linear Statics - Single Constraint** from the *Solution Type* list. Name it for ex. **Static Beam** (it's a good practice to give names that make sense) and click **OK** to accept.

The GUI in the Design Simulation mode shows the process to perform a simulation (Figure 3). First material needs to be defined, then mesh created, then set loads and constraints and finally simulation needs to be solved.

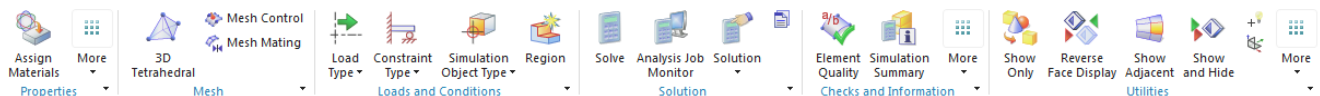


Figure 3: Home tab in the Design Simulation mode.

### Material

Select **Assign Materials** (🔧) from *Properties* group, select current part as a *Body* and select **Steel** as a material (**OK**).

### Mesh

Select **3D Tetrahedral** (🔧) from *Mesh* group. Select current part as a *Body* and give **10** mm as *Element Size*. Other settings are fine, select **Apply** to see the mesh (it may take several seconds to be created). Click **Cancel** to close the window (Figure 4). Mesh will be saved on \*.fem part.

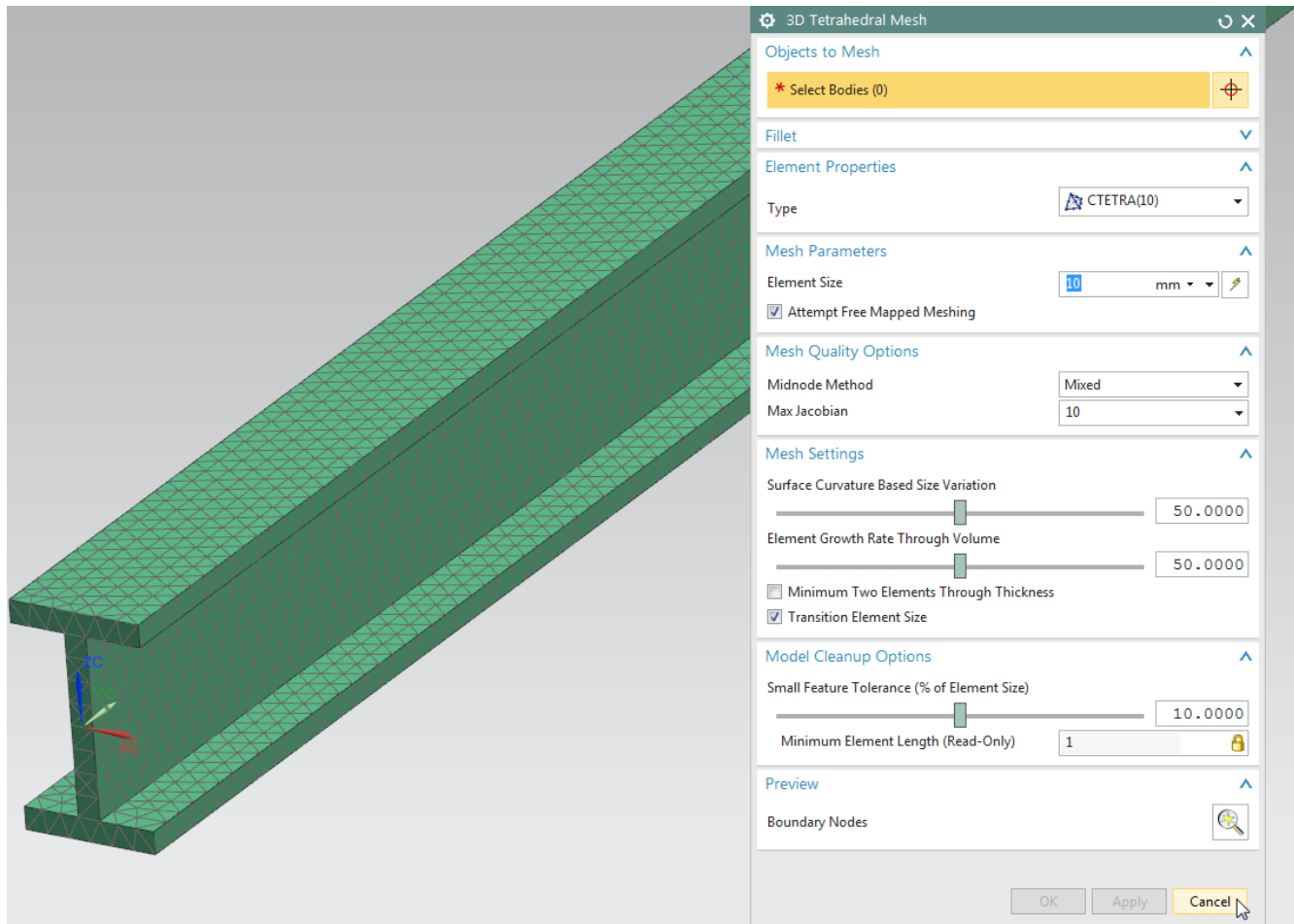
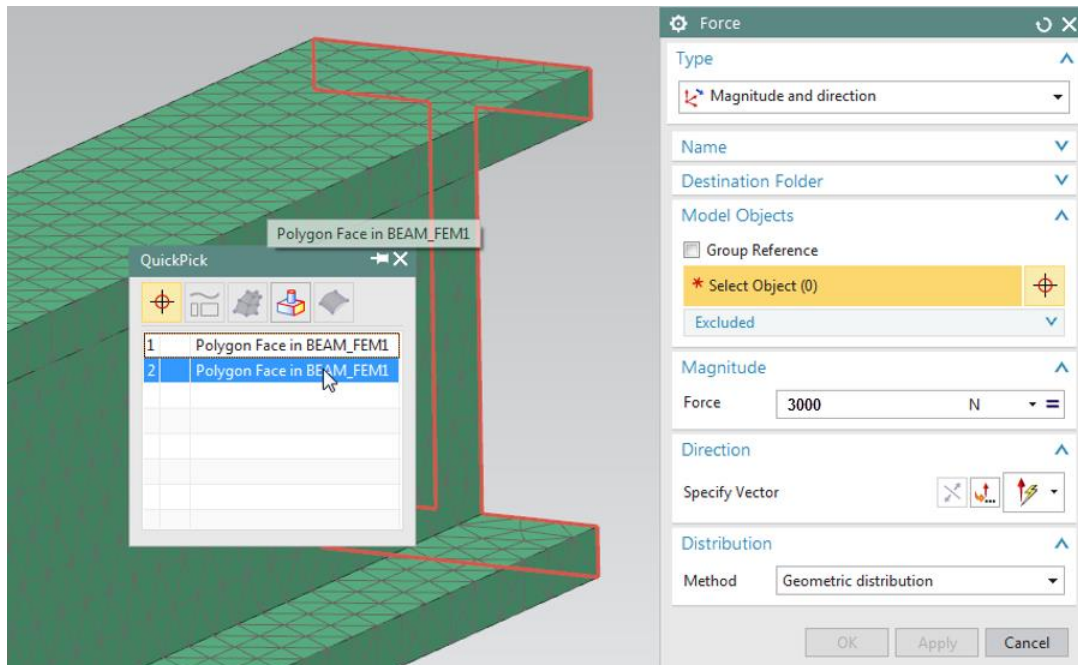


Figure 4: Mesh created, closing the meshing tool.

## Loads

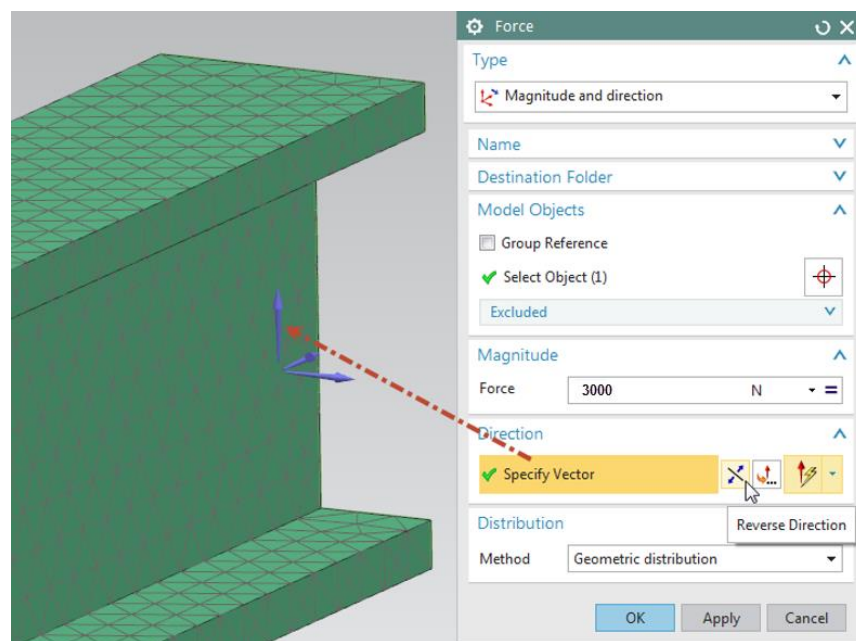
The most difficult part is to find the right constraints (i.e. boundary conditions) to the simulation model. These will affect the results greatly. In this case we assume that the beam is fixed from the beginning (where coordination is displayed) and all forces are applied to the other side.

Select **Load Type** (📏) and then **Force** (📌) from the *Loads and Conditions* group. Select the end surface of the beam and give **3000** as a *Magnitude* (Figure 5). If beam is showing poorly, press **CTRL + F**.



**Figure 5: Selecting the surface to apply the force.**

Next the *Direction* needs to be defined. Click on **Specify Vector** and select the one pointing upwards (Z-axis). Click on **Reverse Direction** (↕) once to change the force pointing downwards (Figure 6). Click **OK** to accept the force. The red arrows pointing downwards should appear. If the direction is wrong, double-click on the arrows to change them.



**Figure 6: Selecting Reverse Direction.**

Our beam's mass is not taken account. Create a new load, **Gravity** (🔧) from *Load and Conditions group*. Default settings are fine (gravity towards negative Z axis and value of  $9810 \text{ mm/s}^2$ ), click OK to grate the load (Figure 7).

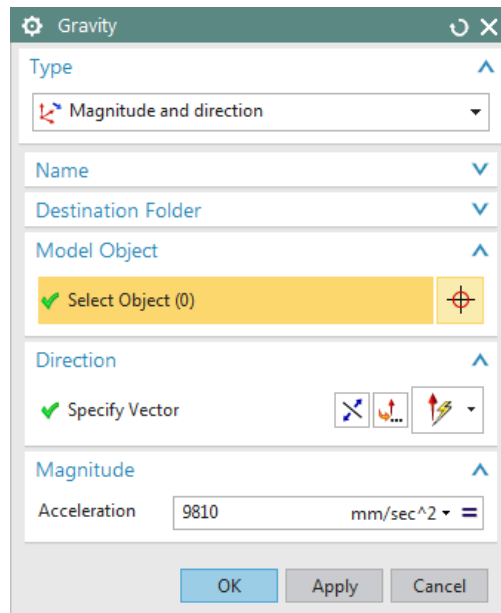


Figure 7: Gravity load.

## Constraints

Next we attach our model to the ground. Select **Constraint Type** (🔧) and then **Fixed Constraint** (🔧) from the *Loads and Constraints group*. Select the front surface of the beam (Figure 8) and click **OK** to place a constraint. This will fix all six degrees of freedom.

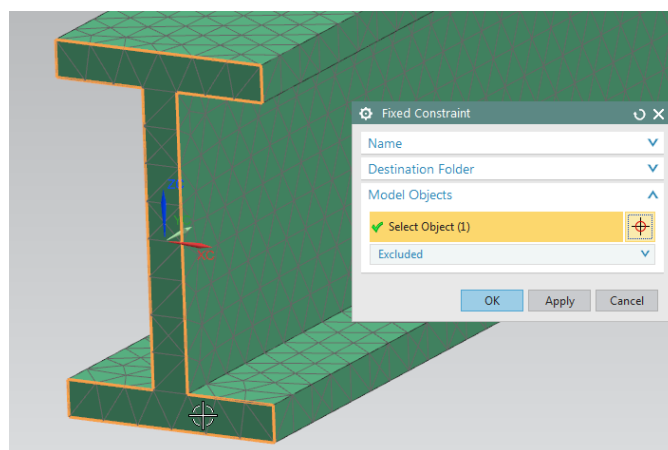



Figure 8: Front surface of the beam selected.

## Run the simulation

All needed information is given, time to run the simulation. Select **Solve** (  ) from the *Solution* group. Click **OK** to run the simulation (may take some time). The solver first checks that model has all needed values defined and then it solves the case. You can close the open windows.

## Results

To see results, double-click **Structural** from the model tree (Figure 9). You may notice, that this result set is a part of *Static Beam* simulation that was created in the beginning.

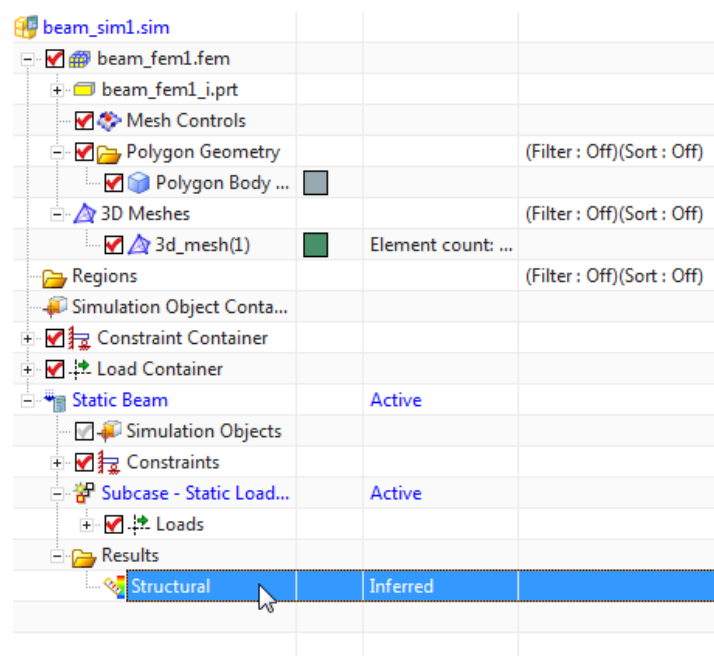


Figure 9: Selecting Structural results of Static Beam simulation.

The result mode opens. Expand the **Structural** to see what the program has solved (Figure 10), just double-click to see the results. For ex. we can see the Elemental von Mises stresses in the model (Figure 11). As you may notice, the stresses are within the limits of a steel part (for ex. steel S235 yield strength is 235 MPa).



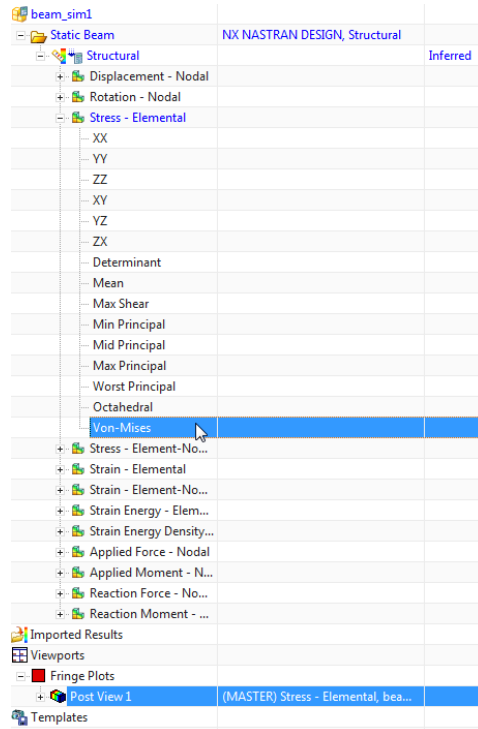


Figure 10: Selecting von Mises stresses.

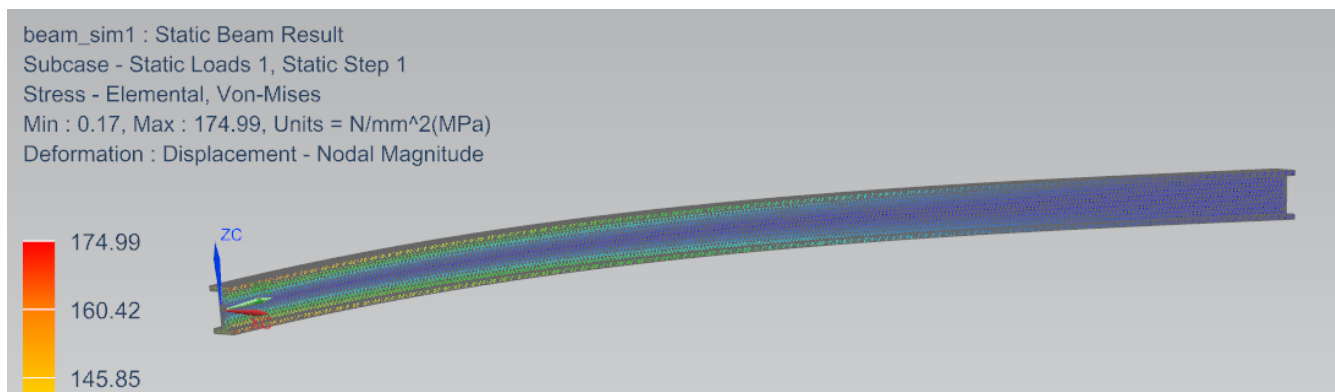


Figure 11: Elemental von Mises stresses. Notice the maximum value.

To get back to simulation model, select **Return to Home** (🏠) from *Context* group. Now it's a good time to save the model (**Ctrl + S**). This saves both \*.sim and \*.fem models.

### Optimizing geometry

Our beam can hold the loads of 3 kN and gravity. There is a possibility that we could gain from optimizing the mass of the beam so that it could still hold the 3 kN load. To optimize the part mass

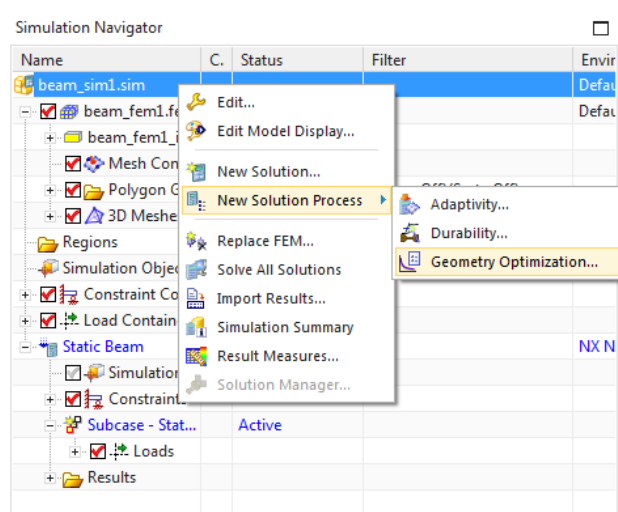
wise, we can manually change the dimensions of the part and check how it affects the simulation results. Other way is to use *Geometry Optimization* and let the program to find the optimal shape.

The optimization process consists of following things:

- New solution process for geometry optimization.
- Goal for optimization, for ex. minimize the mass.
- Limits for optimization, for ex. von Mises stresses less than 200 MPa.
- Design parameters, i.e. dimensions that the optimization process can change in your model, for ex. thicknesses of the beam.
- General setting for optimization (accuracy etc.).
- Waiting, the optimization will take some time.

## Solution Process

Select the \*.sim model from the *Simulation Navigator*, **RMB** and select **Geometry Optimization** (Figure 12).



**Figure 12: Selecting Geometry Optimization.**

Give a name that makes sense, for ex. Min\_Weight\_Stress (we want to find the lightest beam that can hold the load without high stresses) and click **OK**. *Geometry Optimization* window appears. Select **Next**.

## Goal

As *Define Objective*, select **Weight**, **Minimize** and **N** as *Unit* (Figure 13). Click **Next**.

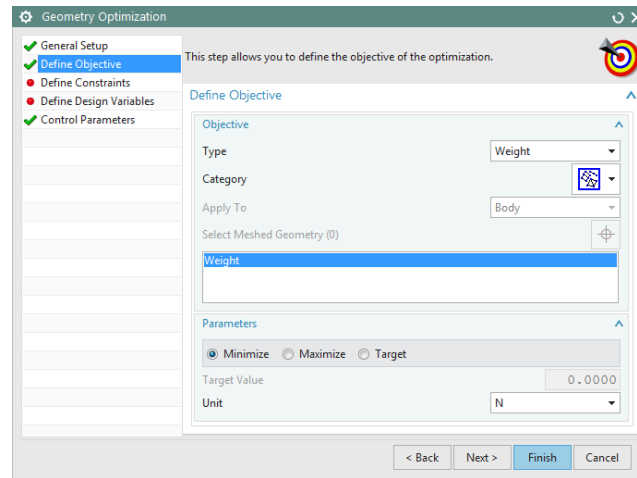


Figure 13: Geometry Optimization, Define Objective.

## Limits

In *Define Constraints*, select **Create constraints** (🔧). As *Type*, select **Result Measure** from the drop-down menu (Figure 14).

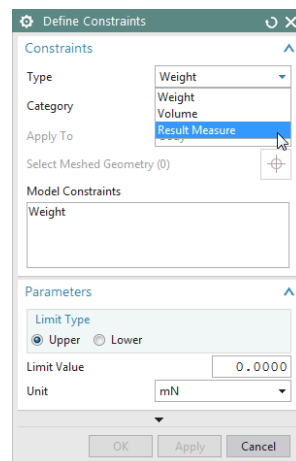


Figure 14: Selecting Result Measure as *Type*.

Then select **Result Measures** (🔧). Select **New** (🔧), then previous simulation from the list (in this case named as *Static Beam*), **Stress - Elemental** as *Result Type* and **von Mises** as *Component*. Select *Operation* as **Maximum** and name it (for ex. stress\_von\_mises) as seen in Figure 15.

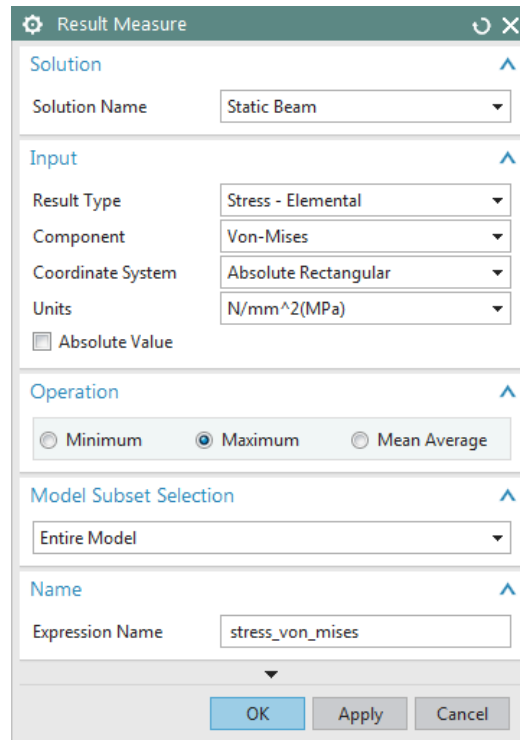


Figure 15: Result Measure.

Click **OK** to create a measure and **Close** to close the listing of measures. As *Limit Value*, give for ex. **200** and ensure that *Limit Type* is **Upper** (Figure 16). **OK** and select **Next**.

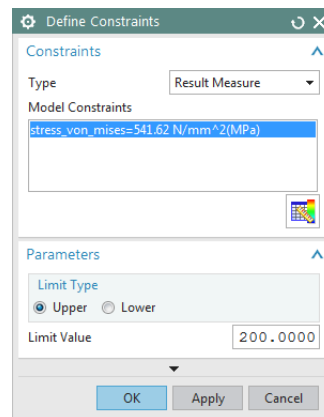
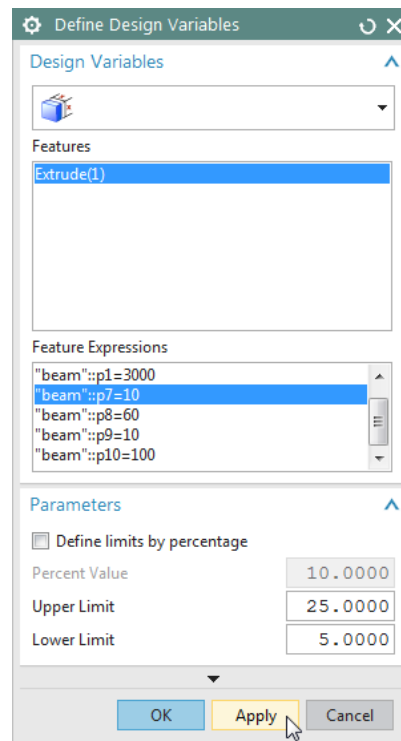


Figure 16: Constraints.

## Design parameters

In *Define Design Variables* tab, click **Create Design Variables** (🎁). Select the only feature from the list named as **Extrude(1)** and select the dimension with value **10** (there are two of them, but we use both

of them) and give range values as seen in Figure 17. Click **Apply** and give another **10** valued dimension with same limits (5...25) and click **OK**.



**Figure 17: First Design Variable defined. This case dimension named as p7.**

In the *Define Design Variables* tab you should have two different *Design Variables* (both have initial value as 10). If not, fix the problem.


## General settings

Select *Control Parameters* tab. Here you can change the optimization parameters, for ex. error percent can be set to 1 % (*Max Constraint Violation*, default 2.5 %). Click **Finish** to create a simulation process.

## Solving

Select *Min\_Wight\_Stress* from the bottom of the *Simulation Navigator*, click **RMB** and select **Solve**. The program optimizes the case and a taking Excel graph appears. This may take several minutes. The optimization is ready when Excel gives error. If a problem occurs, ask for help. Possibly it is not worth to run the optimization again.

The result can be seen in the Excel sheet (Figure 18). In this case, the optimal solution was a beam with thicknesses of 5 and 8.7. The overall weight was reduced from 460 N to 335 N (47 kg to 34 kg).

(If Excel sheet is not available, happens sometimes, result can be seen by selecting  *Part Navigator*, *User Expressions* and selecting changed dimensions with *RMB* and *Browse*.)

Optimization History									
Based on NX Optimizer									
Design Objective Function Results									
Minimum Weight [N]	0	1	2	3	4	5	6	7	8
	460,8149	534,5453	552,9779	279,4201	406,3195	327,8245	338,6567	335,0096	334,638
Design Variable Results									
Name	0	1	2	3	4	5	6	7	8
"beam"::p7=10	10	14	10	5,00056	7,072654	5,002208	5,001735	5,000182	5,003288
"beam"::p9=10	10	10	14	6,478846	9,977987	8,38745	8,815217	8,672477	8,655482
Design Constraint Results									
Result Measure	0	1	2	3	4	5	6	7	8
Upper Limit = 200.000000 [N/mm^2(MPa)]	174,99	167,53	150,95	249,8	175,89	208,21	194,39	199,1	198,53
Small change in design, run converged.									

Figure 18: The result of the optimization study.

To get back to simulation mode, Excel worksheet needs to be closed. Double-clicking **Results** (Figure 19) under Min\_Weight\_Stress allows you to see the results from different design cycles.

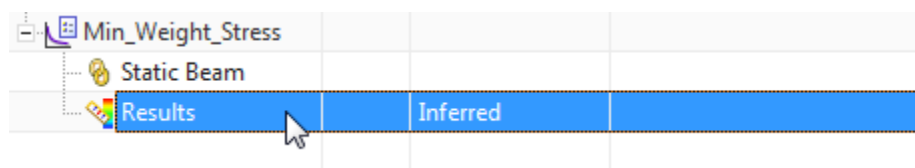
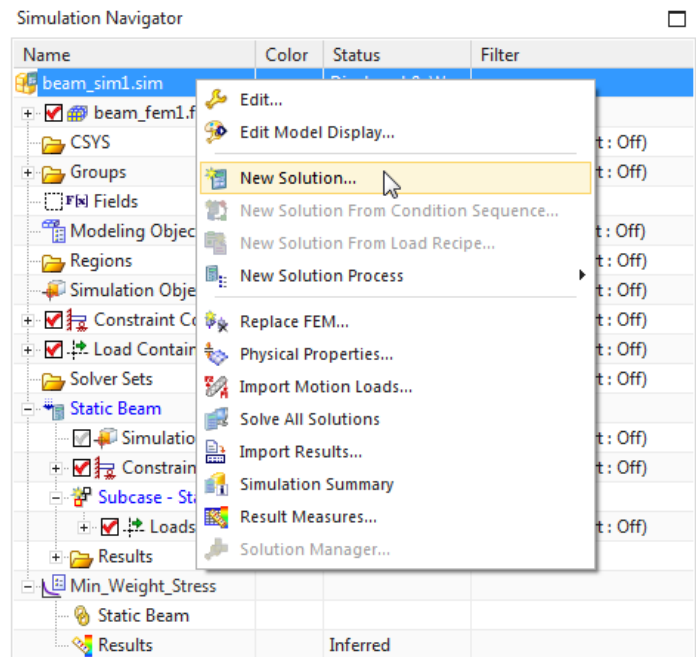


Figure 19: Selecting Results from optimization study.

## Modular analysis

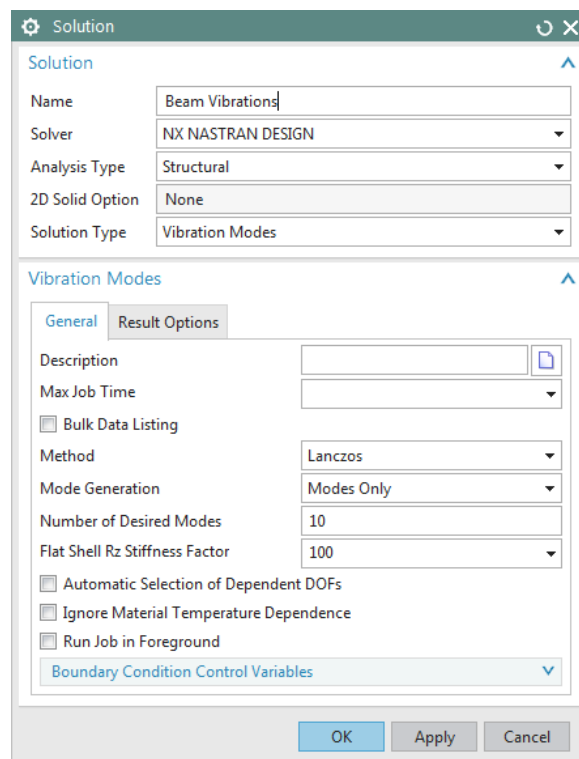
Our \*.sim file has one solution (Static Beam, i.e. linear static analysis). We can have several solutions using the same environment (constraints, loads). Next we create a vibration modes analysis to see how your optimized beam behaves.

Select **beam\_sim1.sim** from the model tree, **RMB** and select **New Solution** (Figure 20).



**Figure 20: Selecting *New Solution* from the *Simulation Navigator*.**

Select Vibration Modes as Solution Type and give a describing name (for ex. Beam Vibrations) as seen in Figure 21. **OK**.



**Figure 21: Settings for vibration analysis.**

## Constraints

To copy existing constraints from the previous solution (Static beam), select **Constraint Container** from the *Simulation Navigator*, **Fixed** constraint, **RMB** and select **Add to active solution or step** (Figure 22). The active solution is displayed in blue in the navigator.

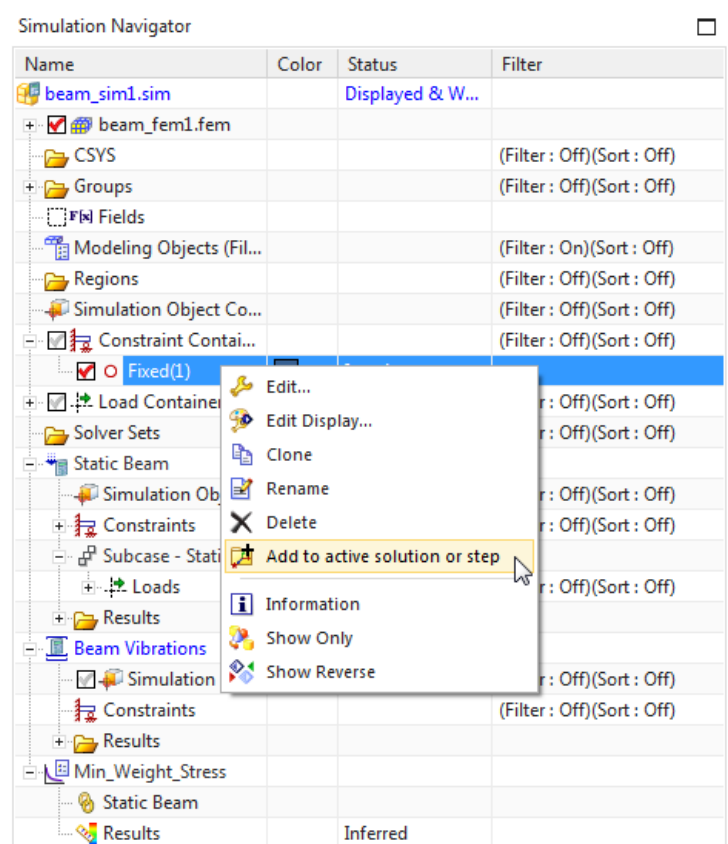


Figure 22: Copying Fixed constraint to the new solution.

## Solve and results

Vibration analysis is ready, select **Beam Vibrations** (seen in Figure 22) from the *Simulation Navigator*, **RMB** and select **Solve**. Click **OK** to run the simulation. You can access the results in the same way as with static analysis.

This concludes the exercise. Be free to test different options on your own.