FEM Phase

MEC-E1060

Aalto University
School of Engineering

Kaur Jaakma

28.9.2020

Aim of the Phase

To give “machine designer” level view for FEM

- How the strength simulation process works?
- What you need to know to perform analysis?
- How to present results?
- How simulations can improve your design?

Strength Analyzes

Why we do them?

What is needed to perform an analysis?

- Case
- Geometry
- Constraints (placement)
- Forces

Strength Analyzes

Displacements

Stresses

Fatigue

Buckling

Analytical Calculations

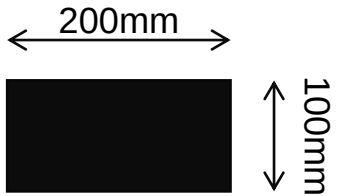
Elastic curve equation

$$w''(x) = -\frac{M(x)}{E \times I(x)}$$

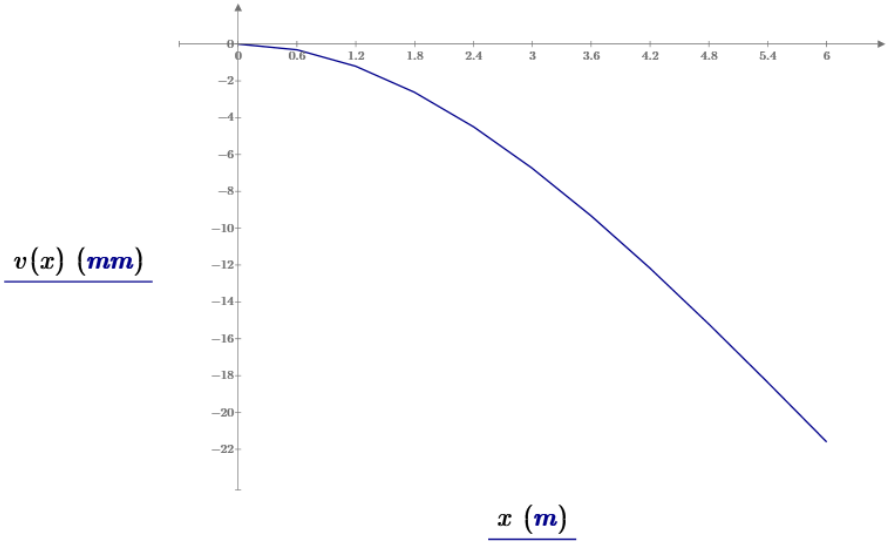
Mathcad

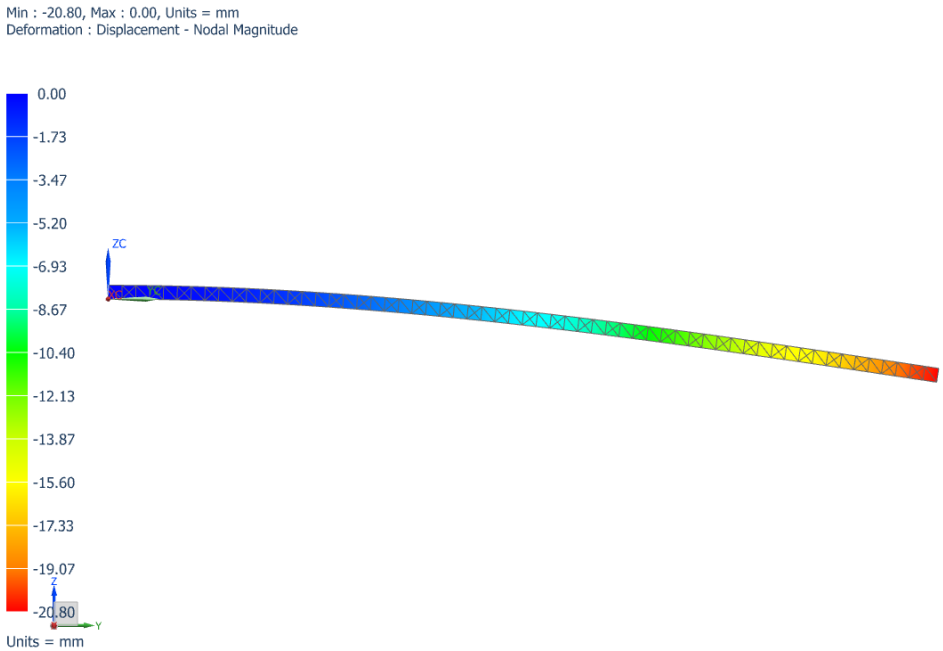
$$v(x) := \int\int w(x)\,\mathrm{d}x + C_1\,\mathrm{d}x + C_2 \xrightarrow{simplify} \frac{x^2 \cdot F \cdot (x - 3 \cdot L)}{6 \cdot E \cdot I}$$

Analytical vs. FEM



$$v(6\ m) = -21.6\ mm$$





Aalto University
School of Engineering

Why FEM?

Analyses are key to effective design

- If you know what you are doing

Faster than physical prototype

- Although real world results are needed

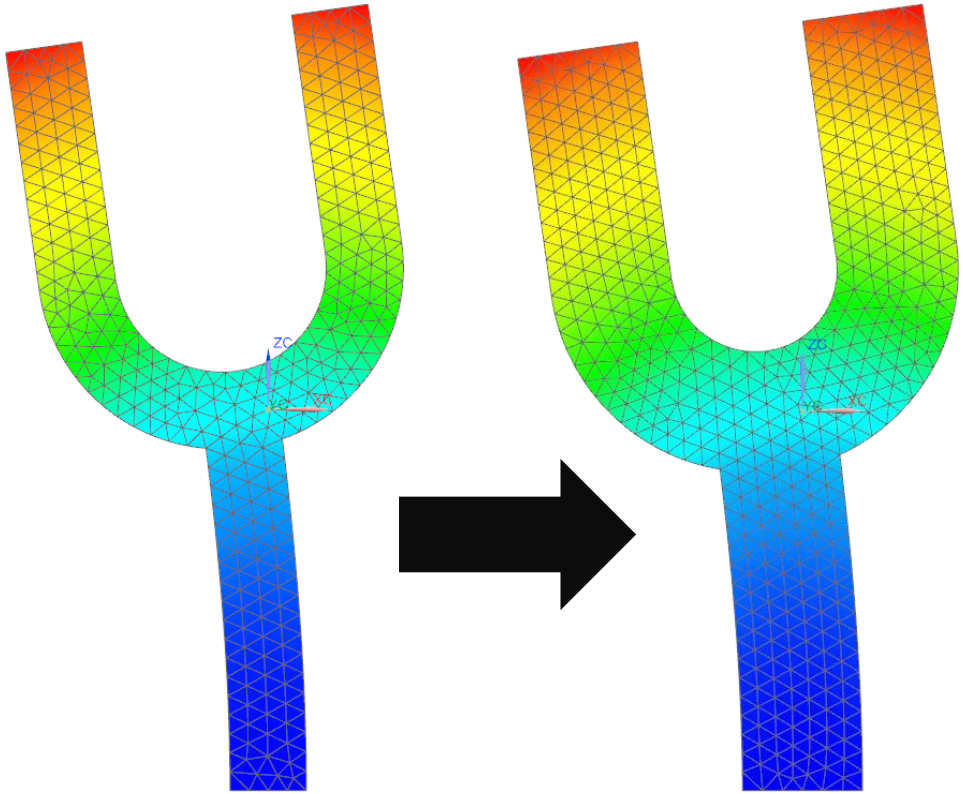
Can use optimization to find the best solution

- Constraints may cause problems
- Validating needed

Optimization

First modal mode to 440 Hz

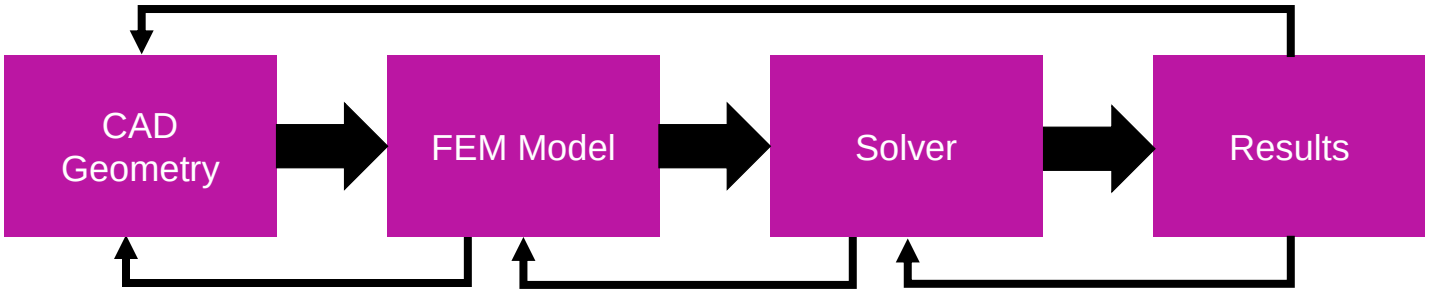
- Starting value 279,3 Hz
- Cross-section 20 → 32 mm



FEM Process

Iterative process

- All can be done in one program
- Different program for each step can also be used



Geometry

Native geometry

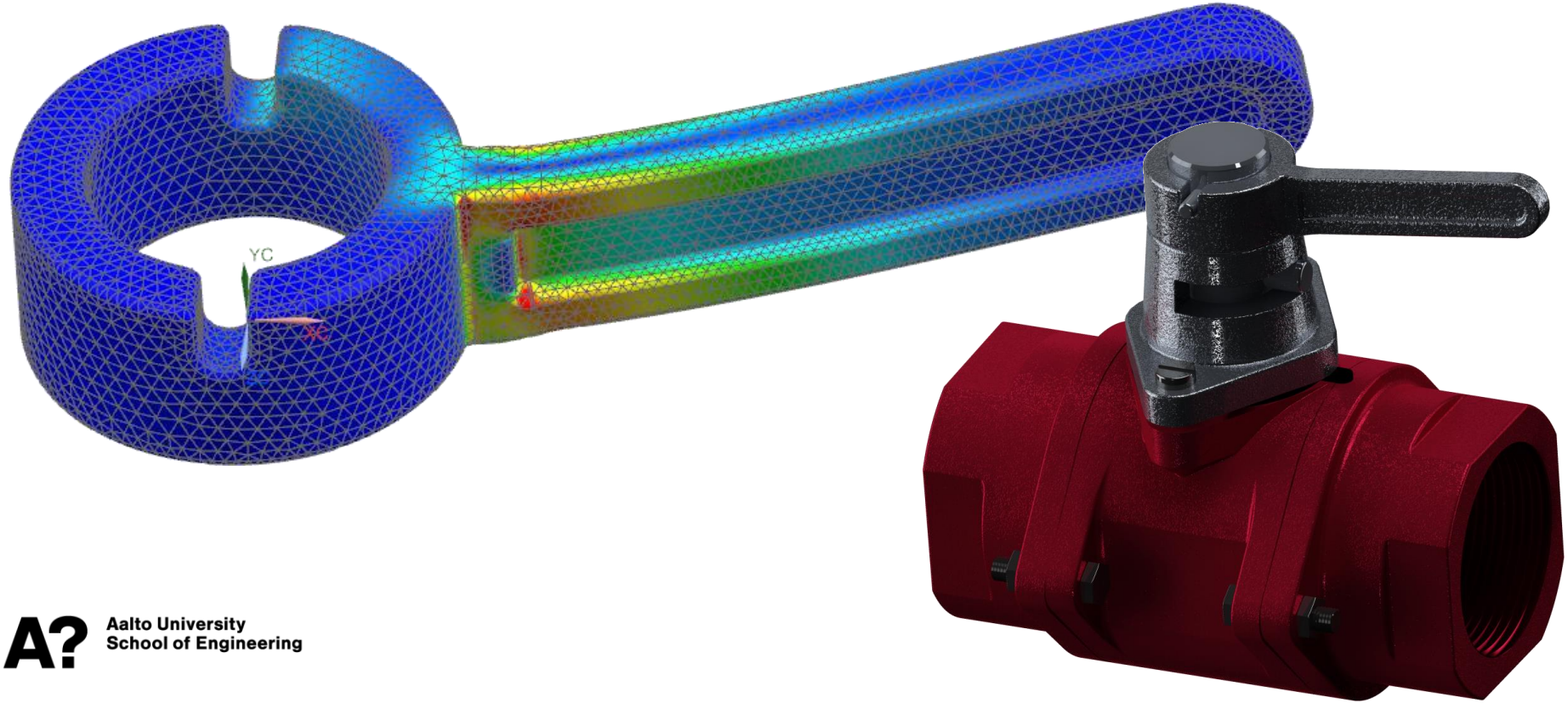
- Created within the analysis program

Imported geometry

- Neutral formats (for ex. STEP, IGES)
- Some programs may have importers for CAD programs

Stresses

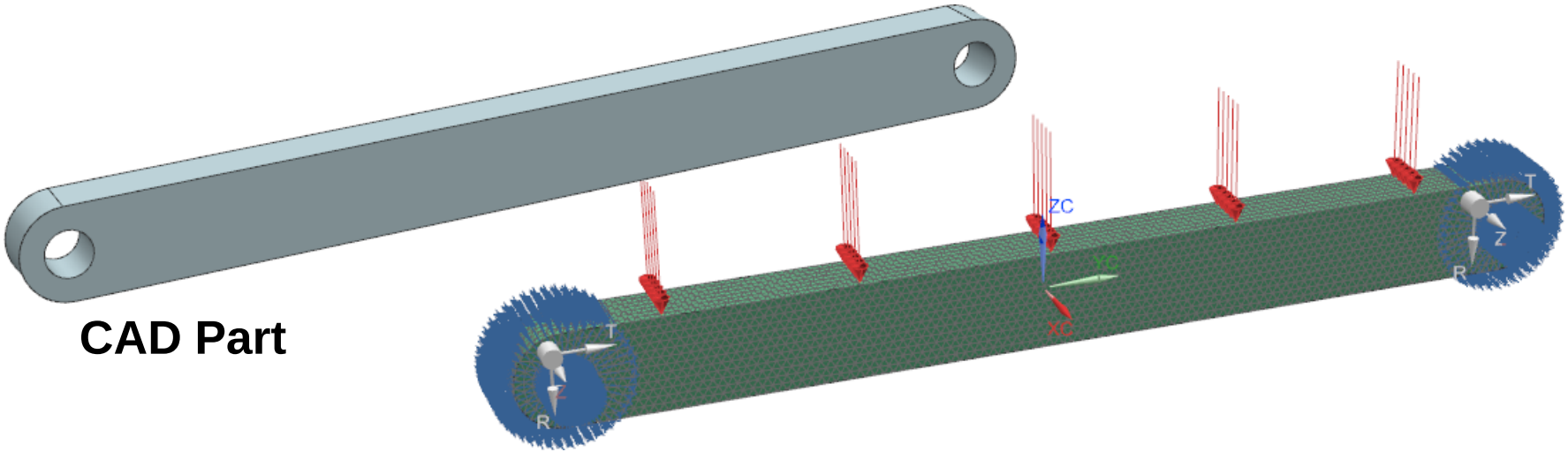
Imported geometry



Constraints

Several ways to define boundary conditions

- i.e. constraints, forces



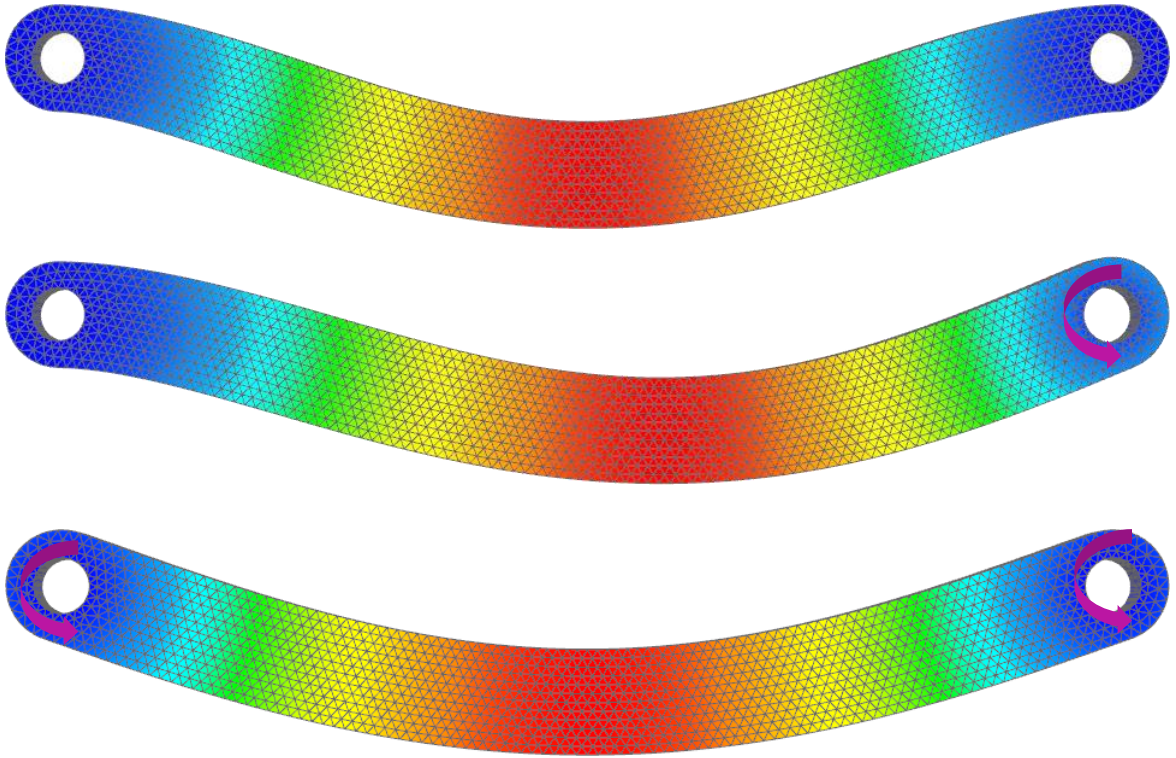
CAD Part

FEM Model

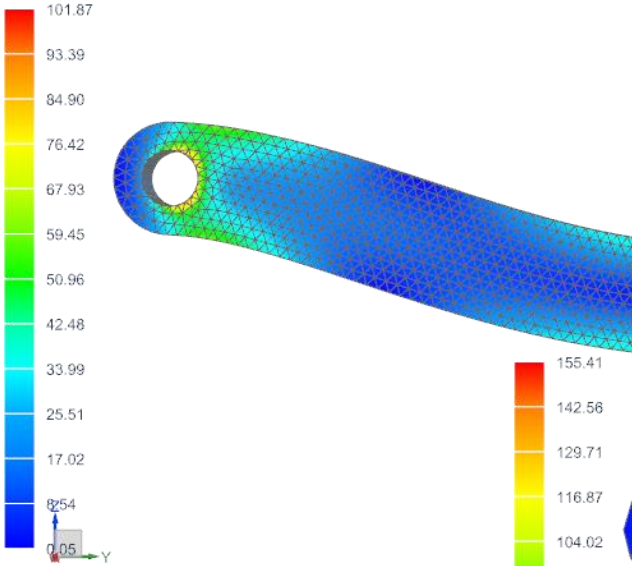
Constraints

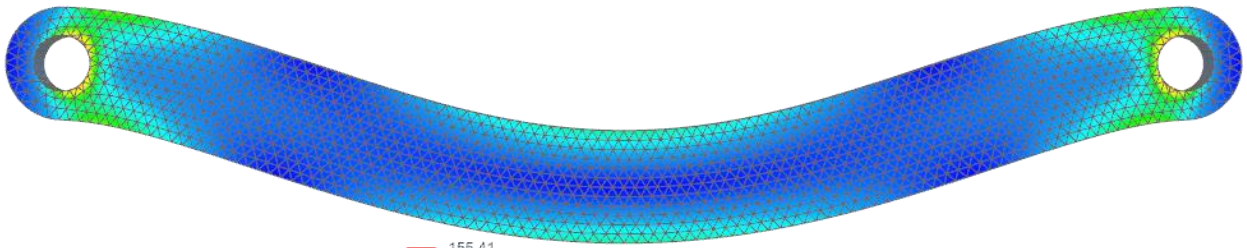
Constraints matters

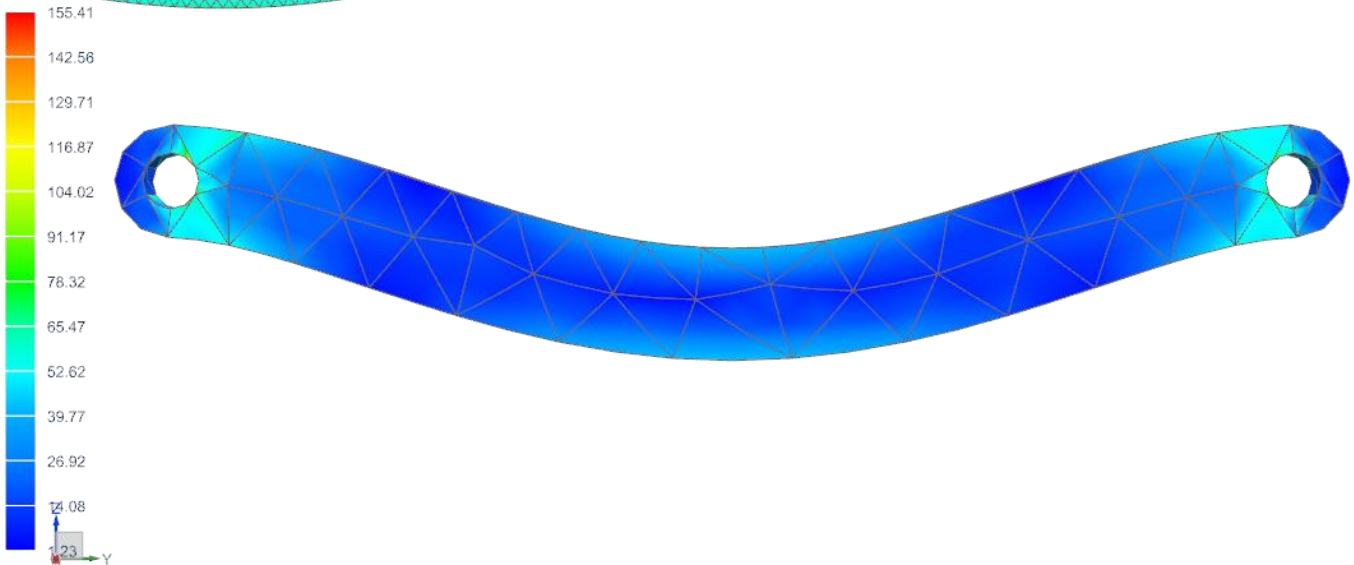
- Fixed or
- Revolute



Mesh Size







Nodal stresses (MPa)

Aalto University
School of Engineering

Example Stand-alone FEM Programs

Courses about FEM

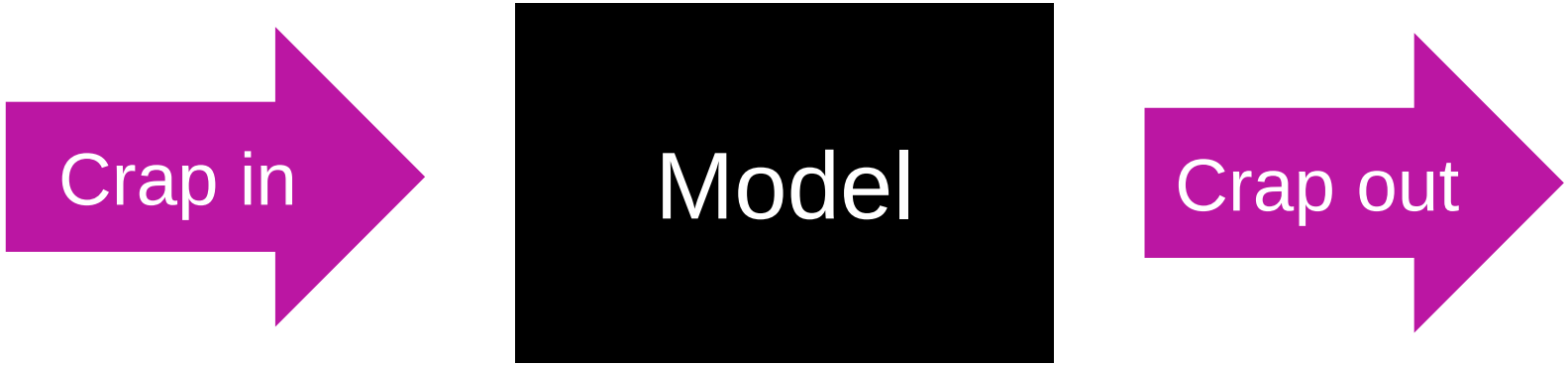
MEC-E1050 Finite Element Method in Solids

- 5 cr, starts on 2nd period

MEC-E8001 Finite Element Analysis L

- 5 cr, starts on 3rd period

User’s Responsibility



FEM Process in NX

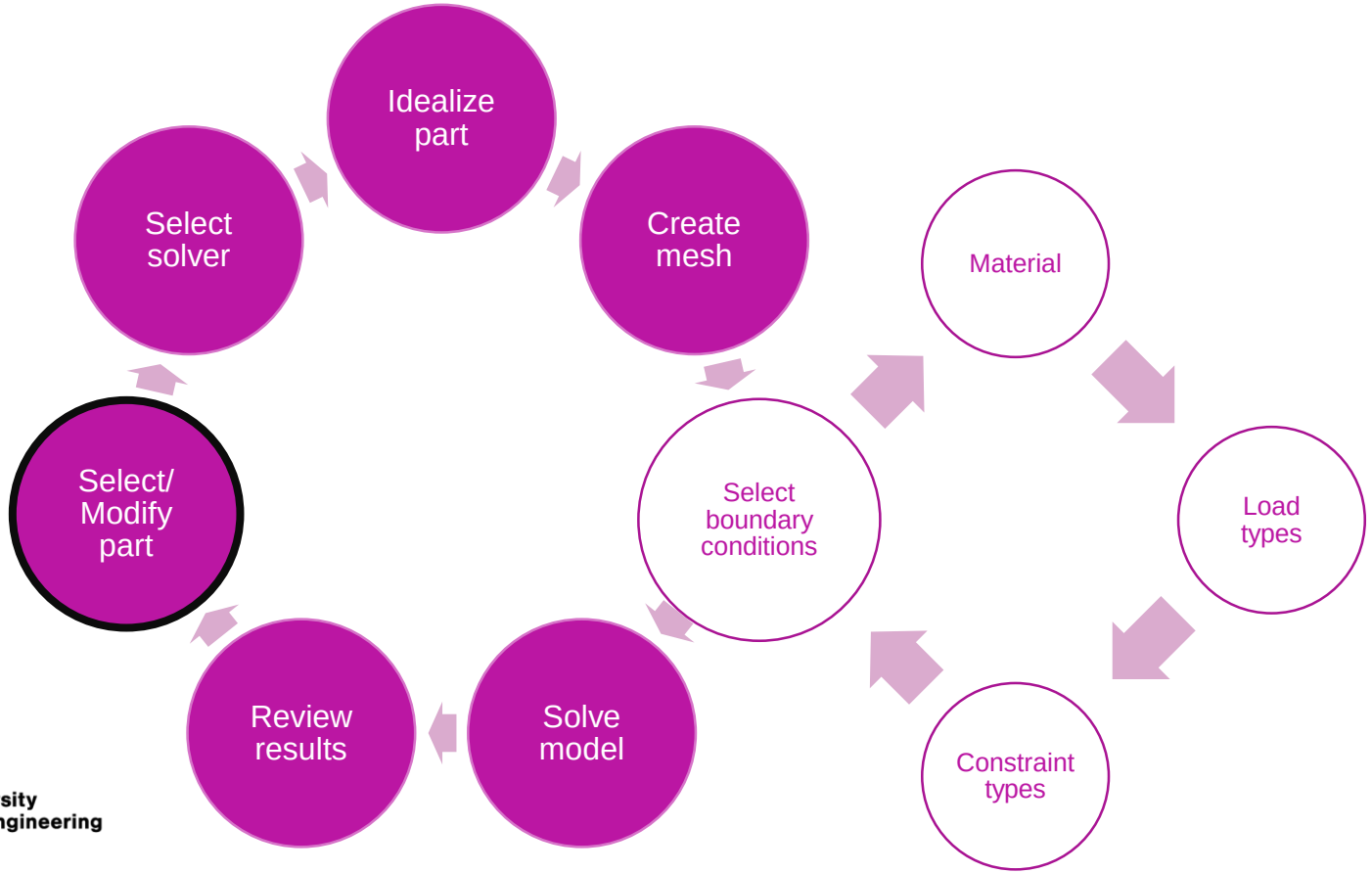
NX Simulate

Integrated into Siemens NX

Uses NASTRAN solver

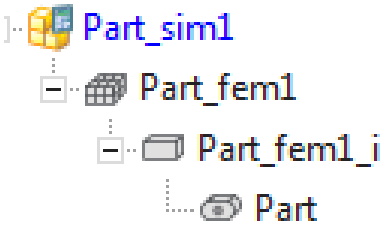
Export to other solvers also possible

FEA Process


Idealize part
Create mesh
Select boundary conditions
Material
Load types
Constraint types
Solve model
Review results
Select/ Modify part
Select solver

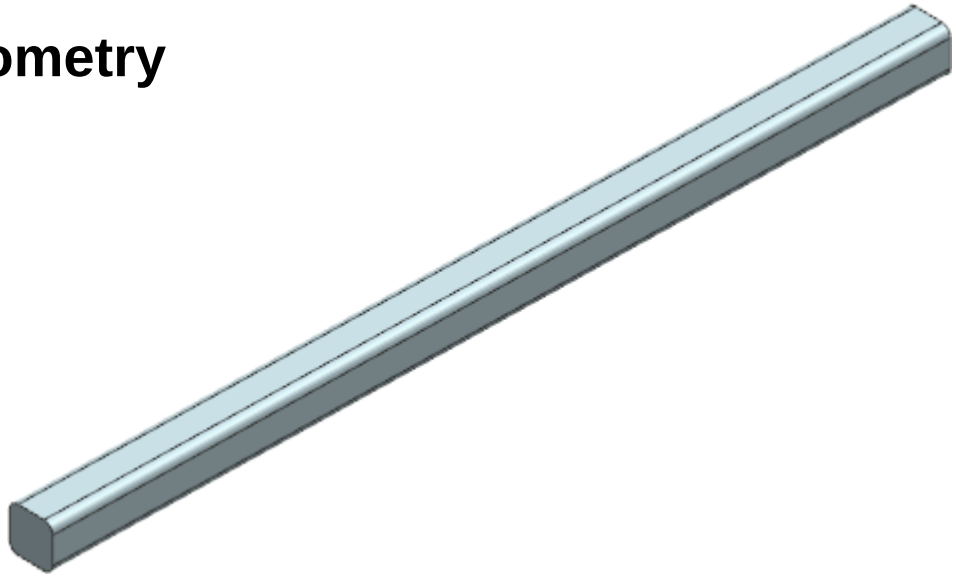
Different Parts in NX

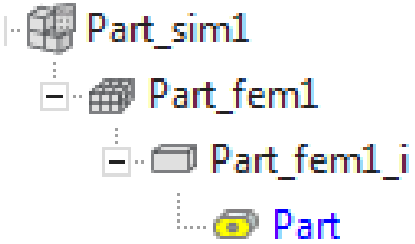
Part name and type	Description
Part.prt	Original CAD geometry
Part_fem1_i.prt	Idealized geometry copied from original CAD file (if selected when creating a new simulation)
Part_fem1.fem	Stored mesh file
Part_sim1.sim	Simulation file, defines constraints, material information and what is being calculated



Displacement of a Beam

CAD geometry

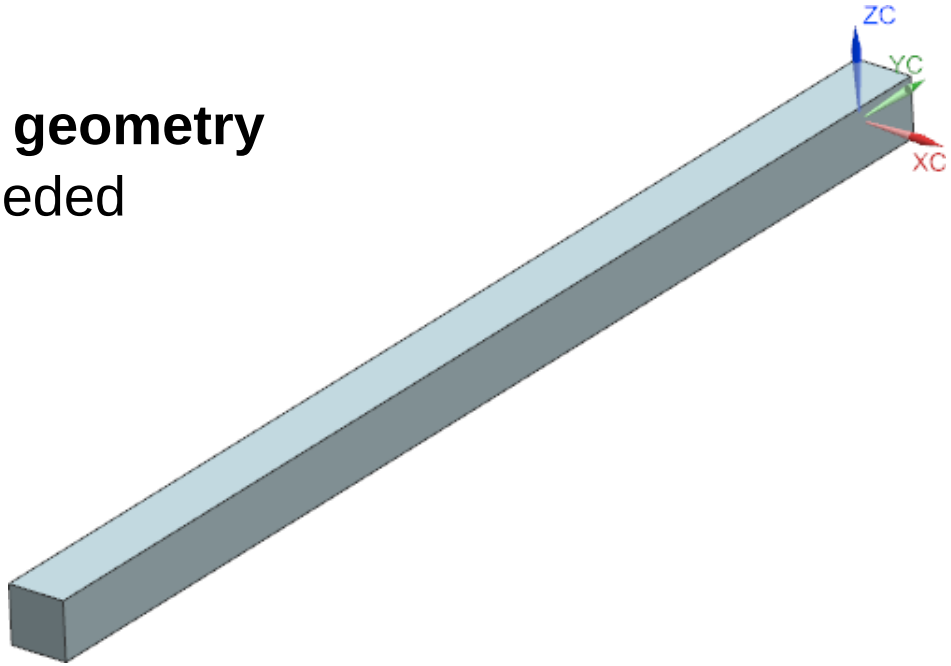


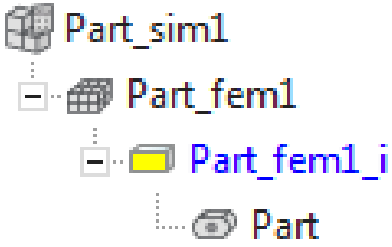


Displacement of a Beam

Idealized geometry

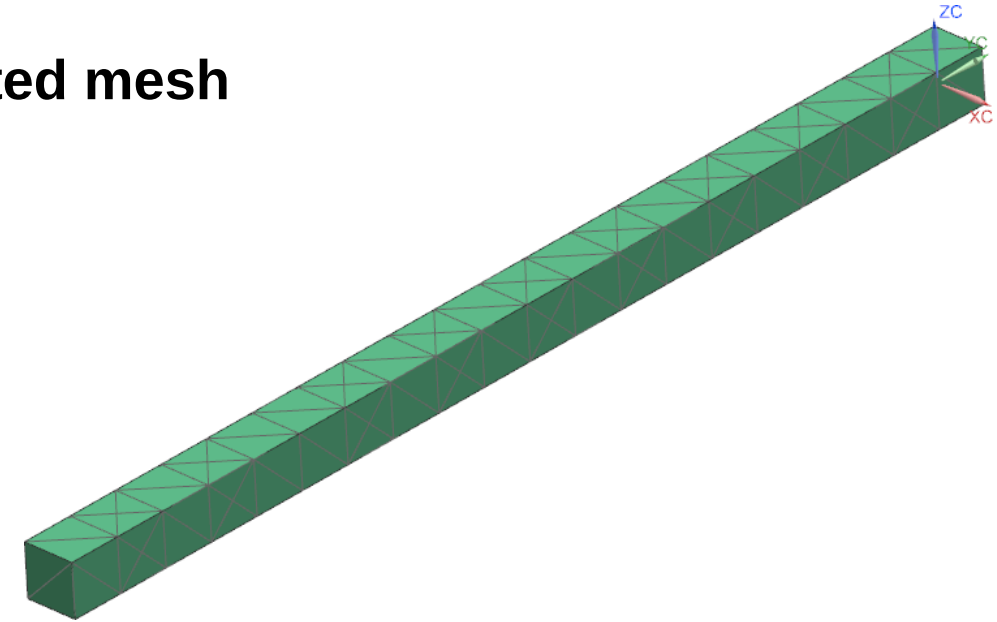
- If needed

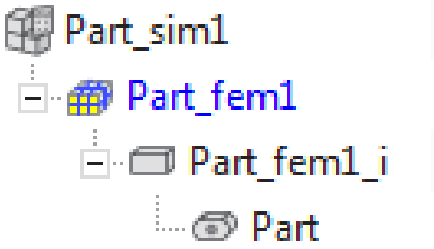




Displacement of a Beam

Generated mesh

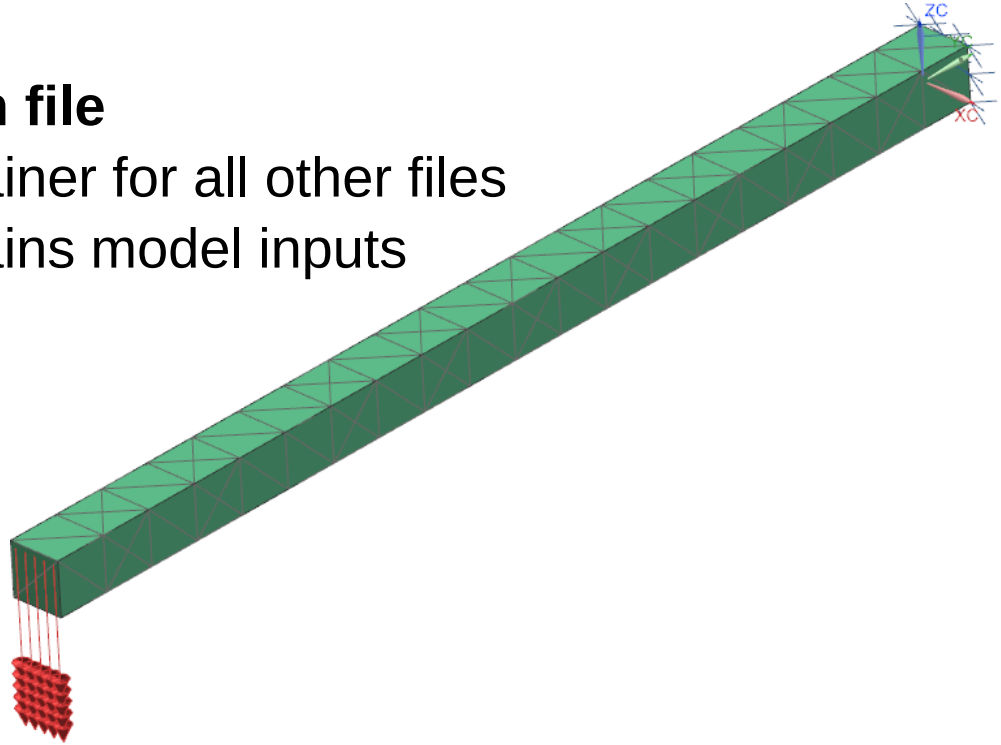


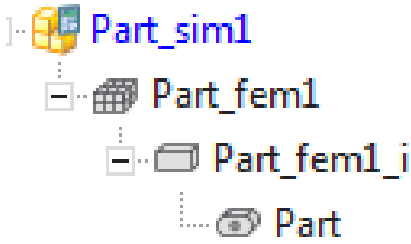


Displacement of a Beam

Simulation file

- Container for all other files
- Contains model inputs



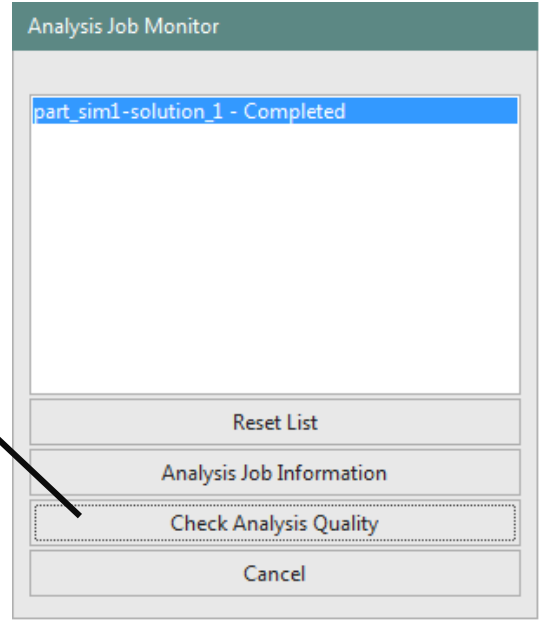


Model Quality

NASTRAN solver sees under 5% error acceptable

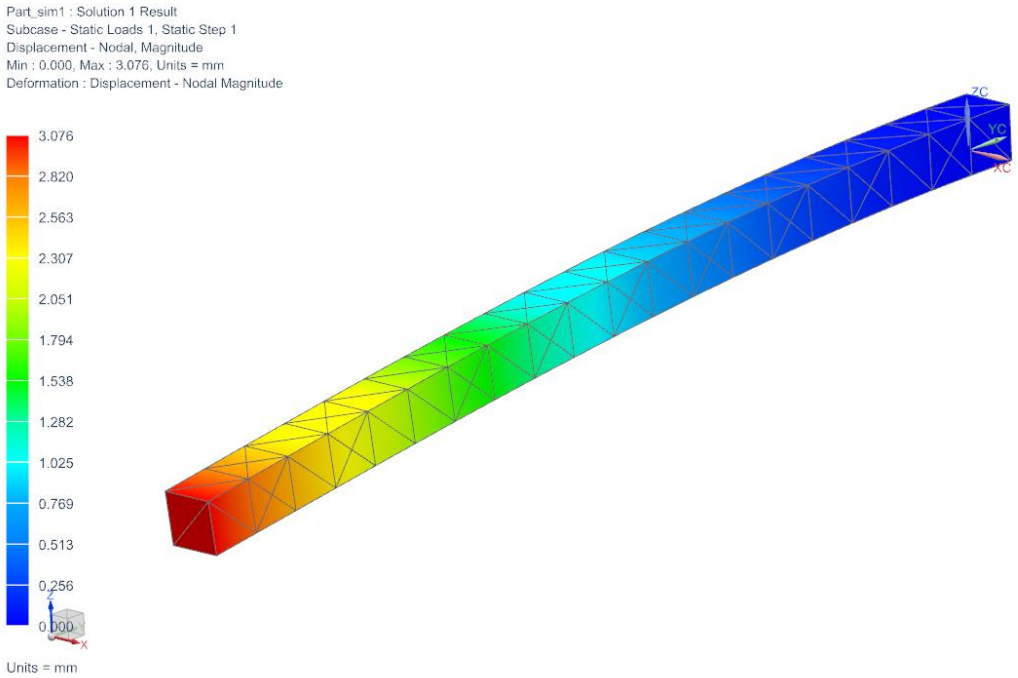
- Smaller mesh elements normally increase quality

```
Analysis Quality Report:

      Total Number of Elements: 322

      Total Number of Nodes: 709

      Strain Energy Error [ 3.937797% ]

      Steady Stress (Relative) Error [ 4.002476% ]

      Steady Stress (Absolute) Error [ 4.135979N/mm^2(MPa)]
```



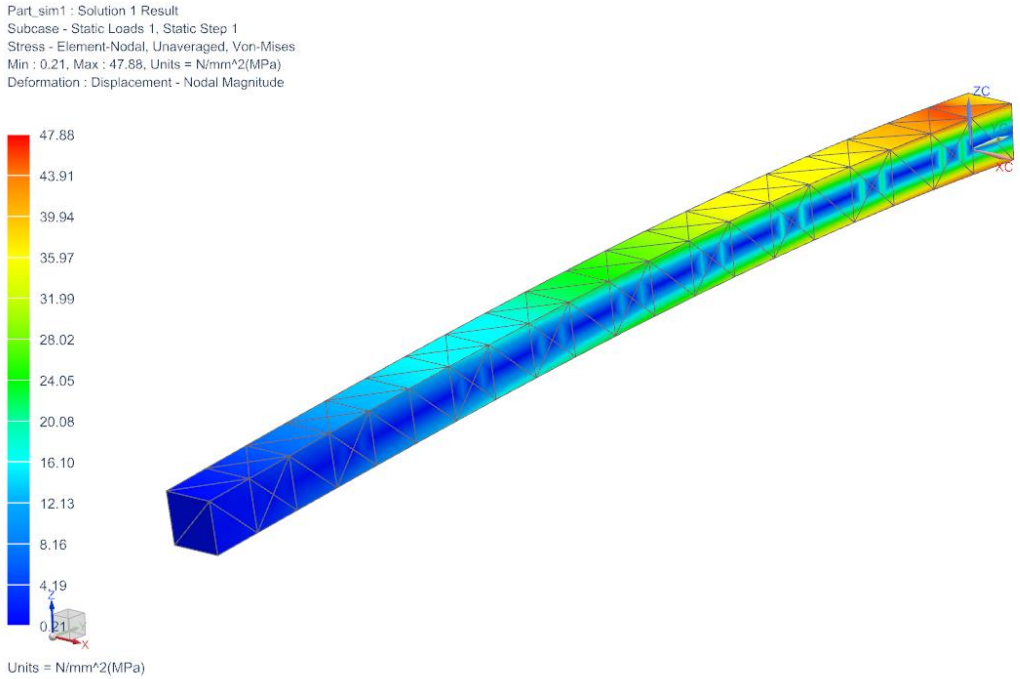
Results

Displacement



Results

Stress



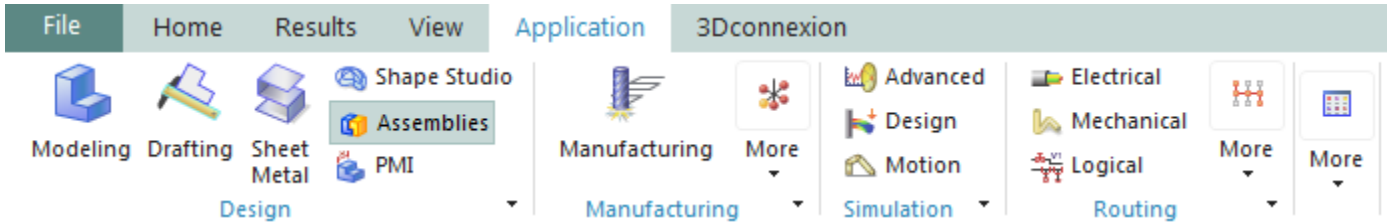
Different modes

To update CAD geometry

- When part file is active, Application → Modeling

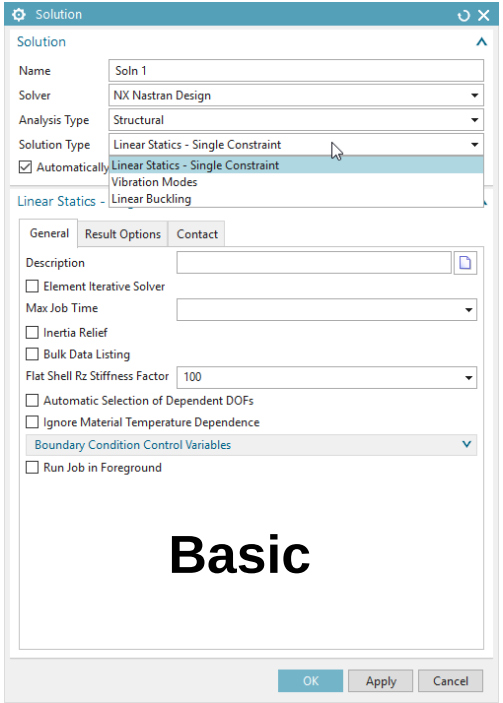
To access FEM model

- Application → Design

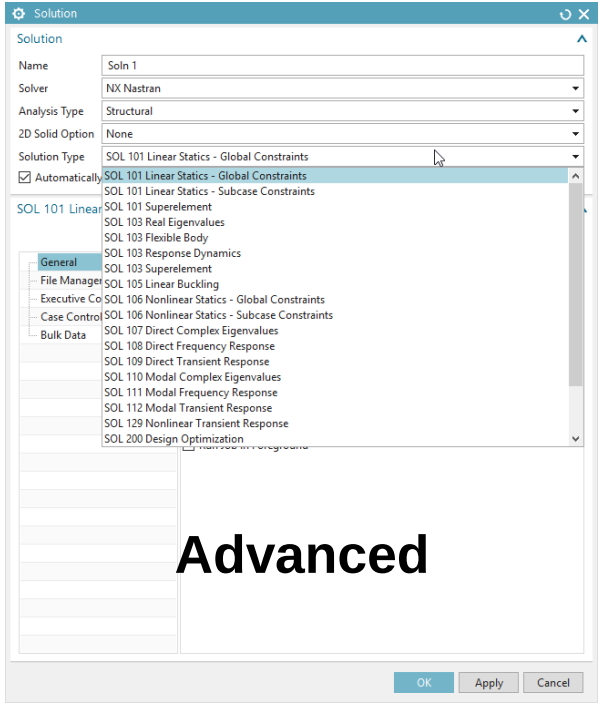


Different Solvers

Basic solver should be fine for this course



Basic



Advanced

Geometry Optimization

Goal

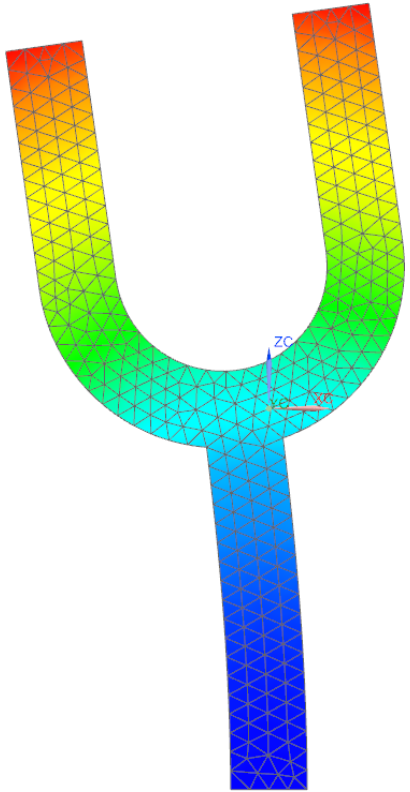
- First modal mode to 440 Hz

Vibration analyze result

- First mode 279,3 Hz

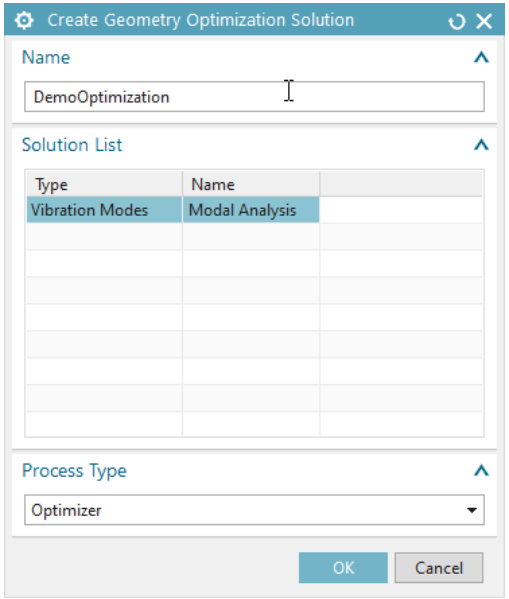
Starting geometry

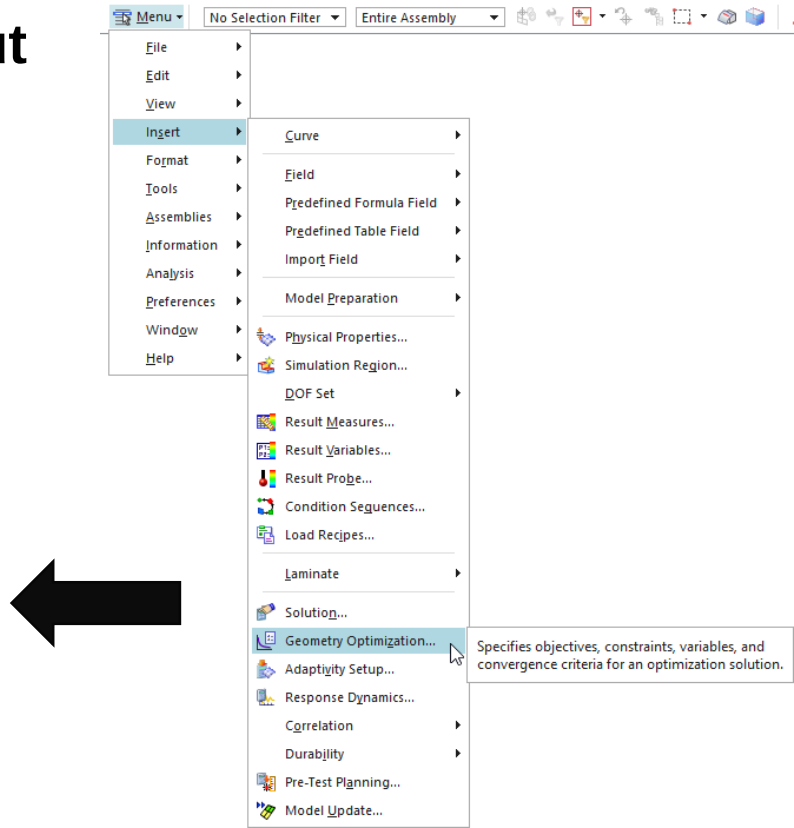
- Cross-section 20



Geometry Optimization

Requires ready made simulation as input



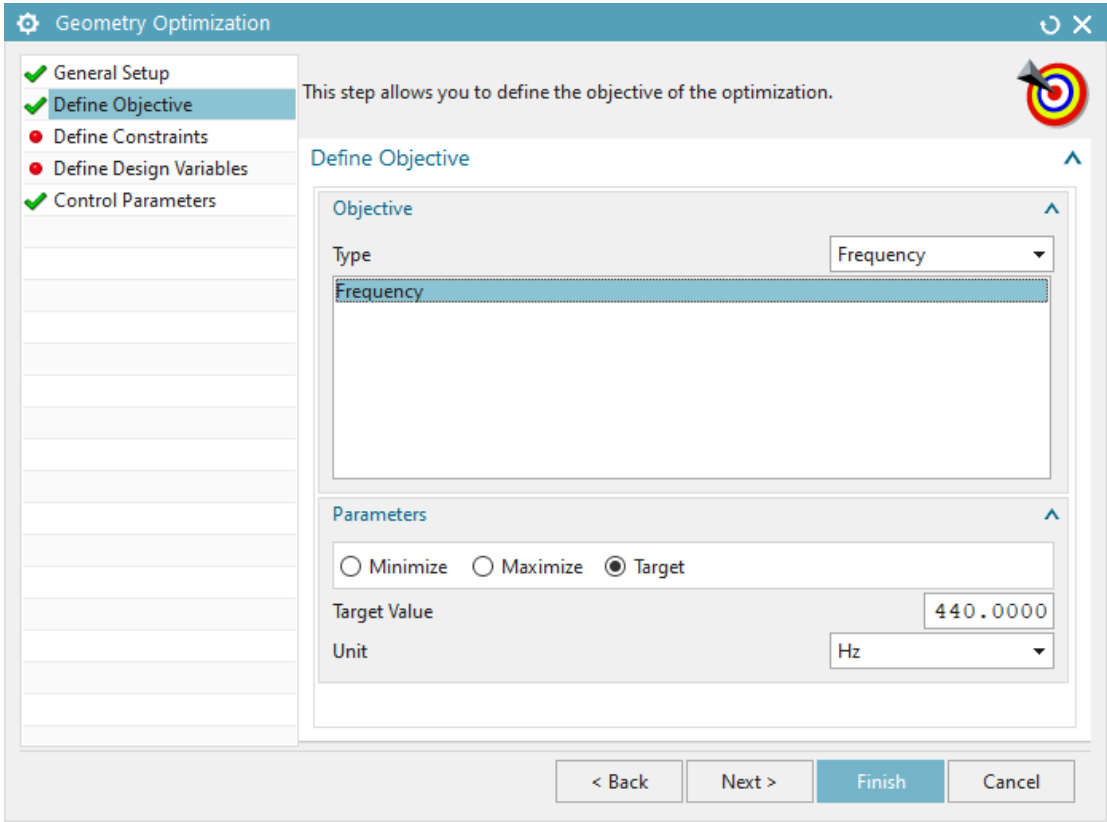


Aalto University
School of Engineering

Geometry Optimization

Objective

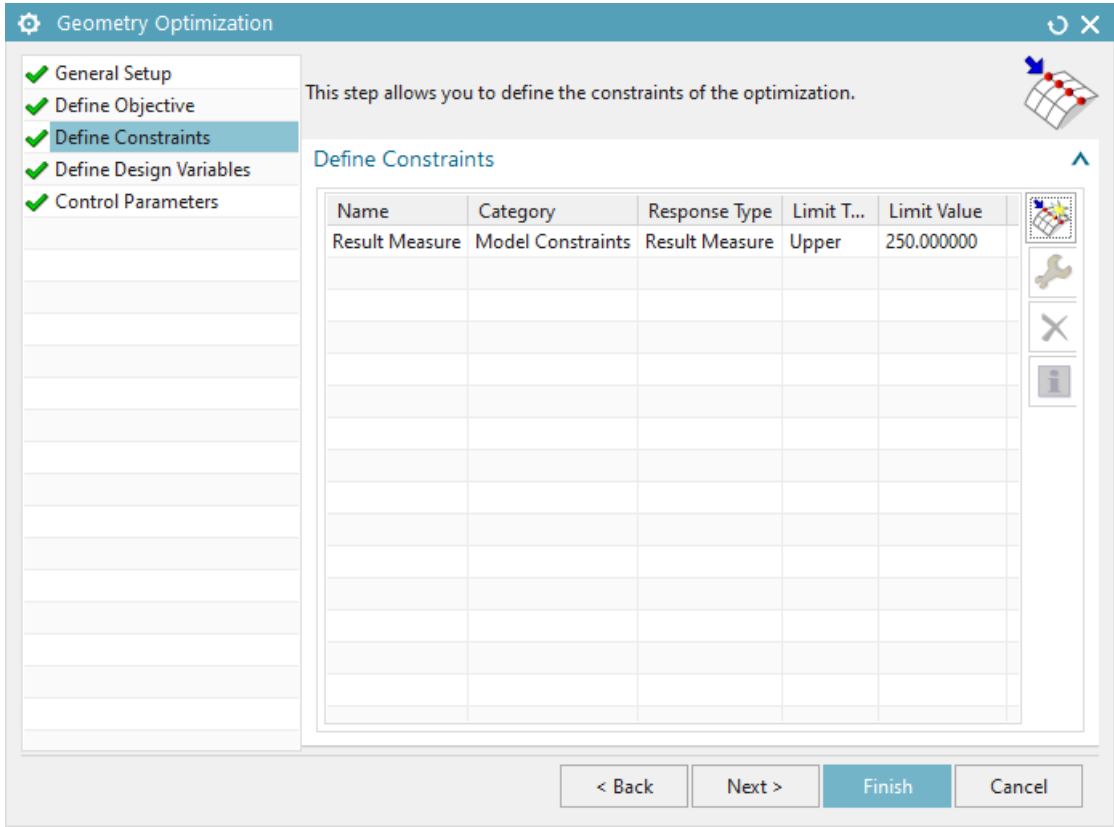
- First mode frequency to 440 Hz



Geometry Optimization

Constrains

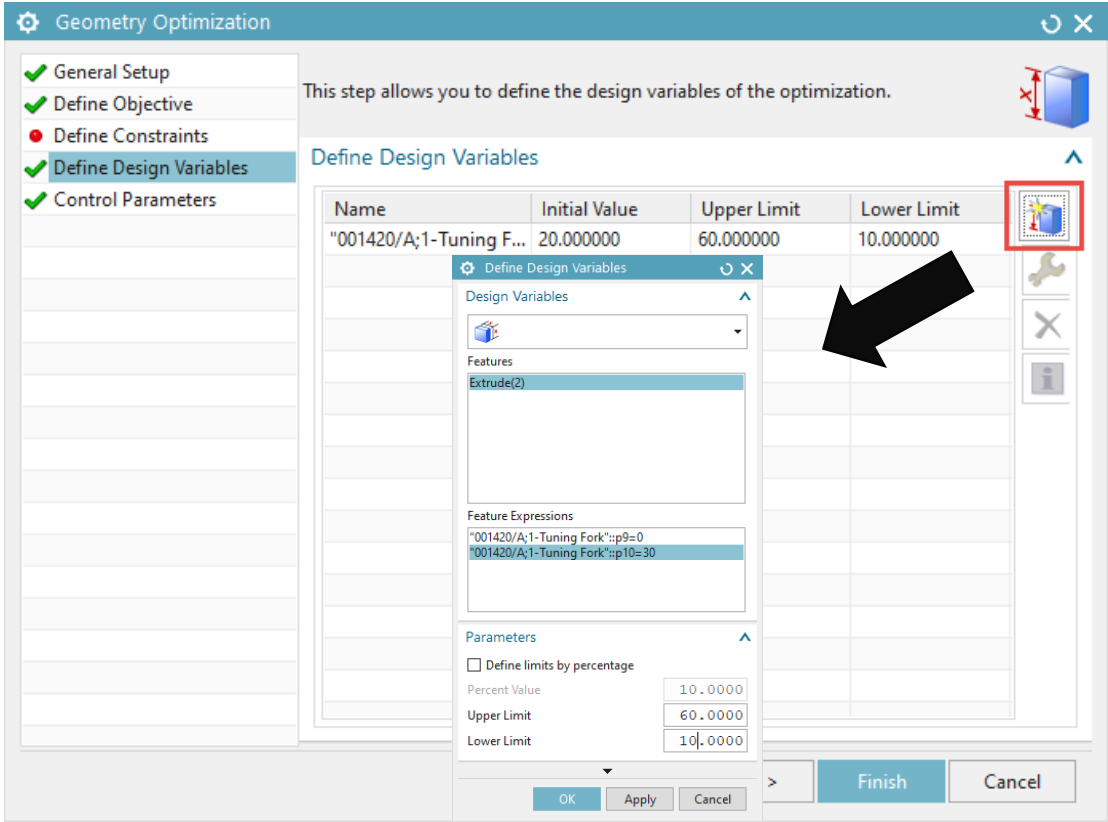
- For ex. Stress limit



Geometry Optimization

Dimensions that optimization tool can change

- Within limits

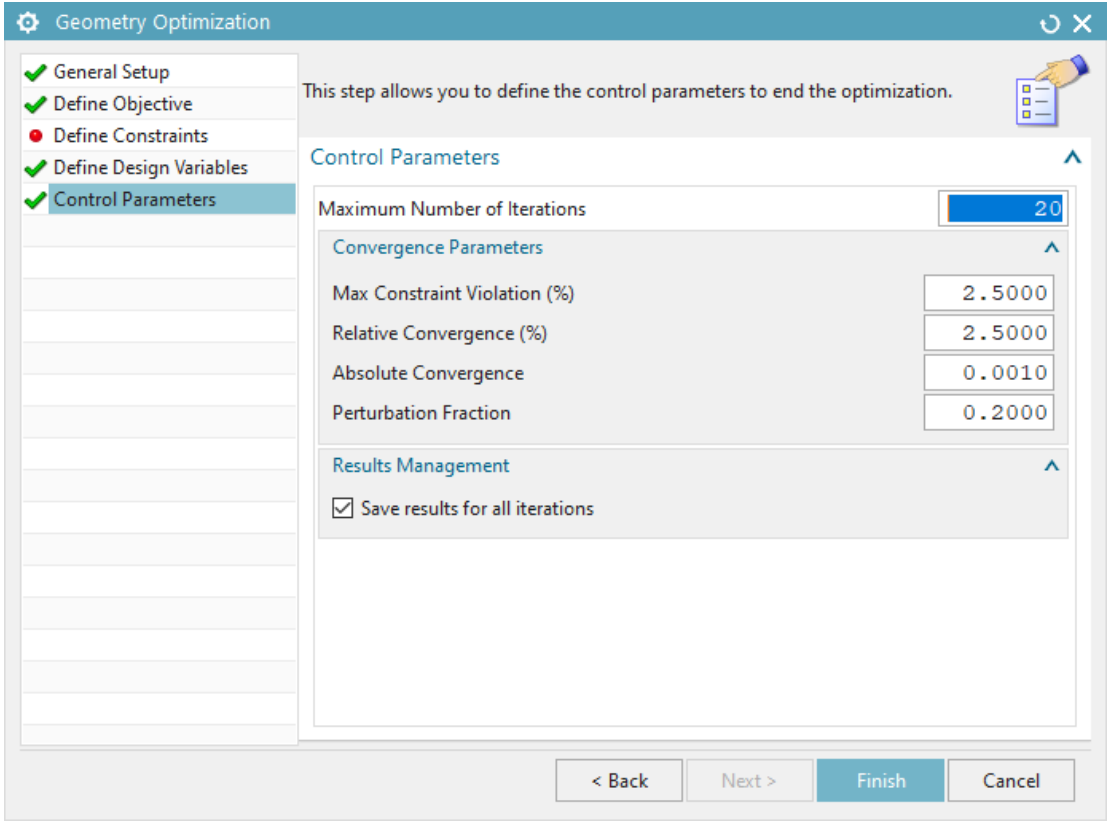


Aalto University
School of Engineering

Geometry Optimization

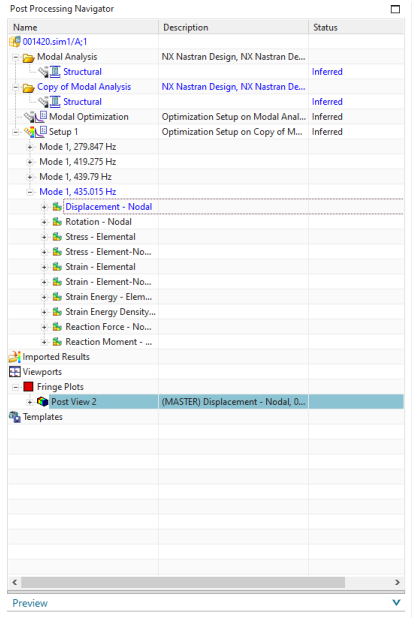
Control parameters for solver

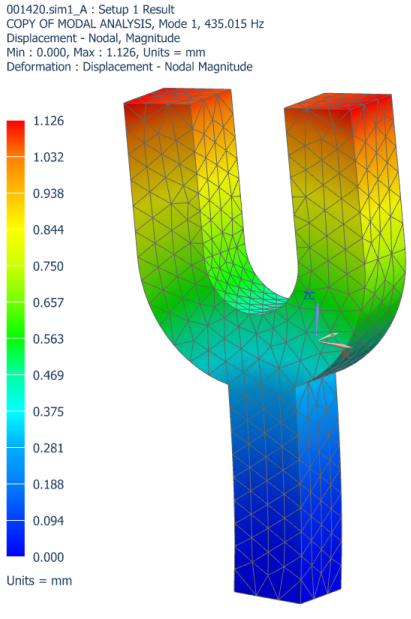
- Defaults mostly OK
- Sometime convergence margin % can be changed to stricter one
- Notice the amount of iterations



Geometry Optimization

Results (notice the 2.5% convergence margin)





Optimization History				
Based on Optimizer				
Design Objective Function Results				
Target Frequency (440.000000) [Hz]	0	1	2	3
	279,8468	419,2751	439,7899	435,0152
Design Variable Results				
Name	0	1	2	3
"001420/A;1-Tuning Fork"::p10=20	20	30	31,48642	31,14084
Design Constraint Results				
	0	1	2	3
Result Measure				
Upper Limit = 250.000000 [N/mm^2(MPa)]	229,66	244,48	251,94	249,69
Small change in design, run converged				

Aalto University
School of Engineering

Siemens Learning Advantage

Database for tutorials and videos related to software

- NX Design mode

Students have access

- Instructions in MyCourses

aalto.fi

Aalto University
School of Engineering