

# 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

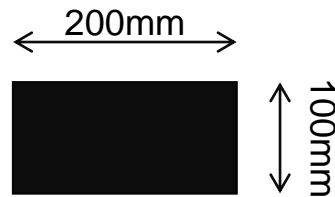
Elastic curve equation

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

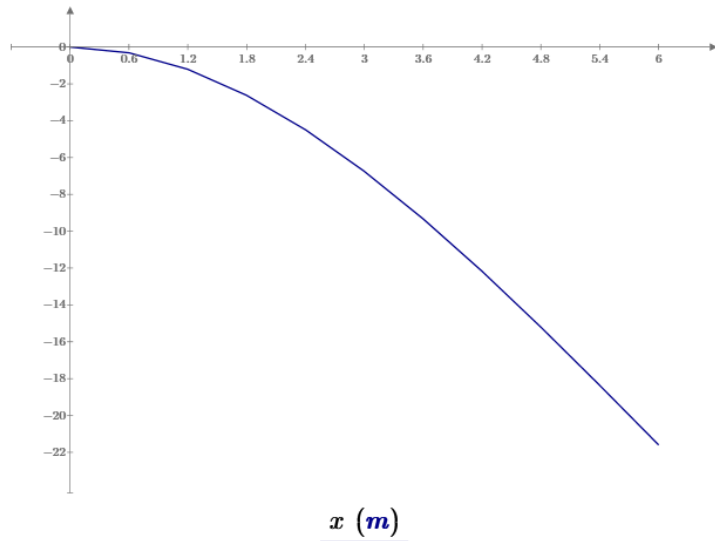
Mathcad

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

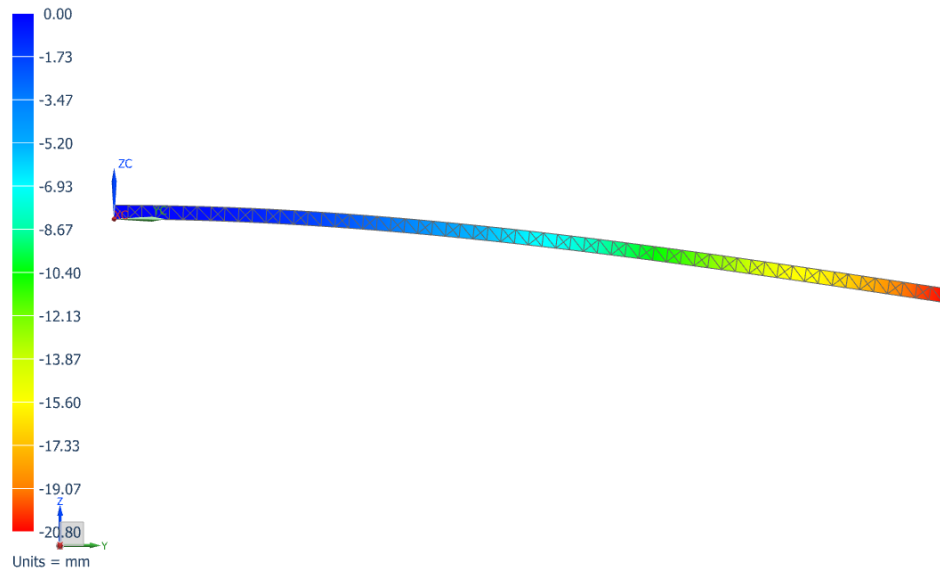
# Analytical vs. FEM



$$v(6\text{ m}) = -21.6\text{ mm}$$



Min : -20.80, Max : 0.00, Units = mm  
Deformation : Displacement - Nodal Magnitude



# 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

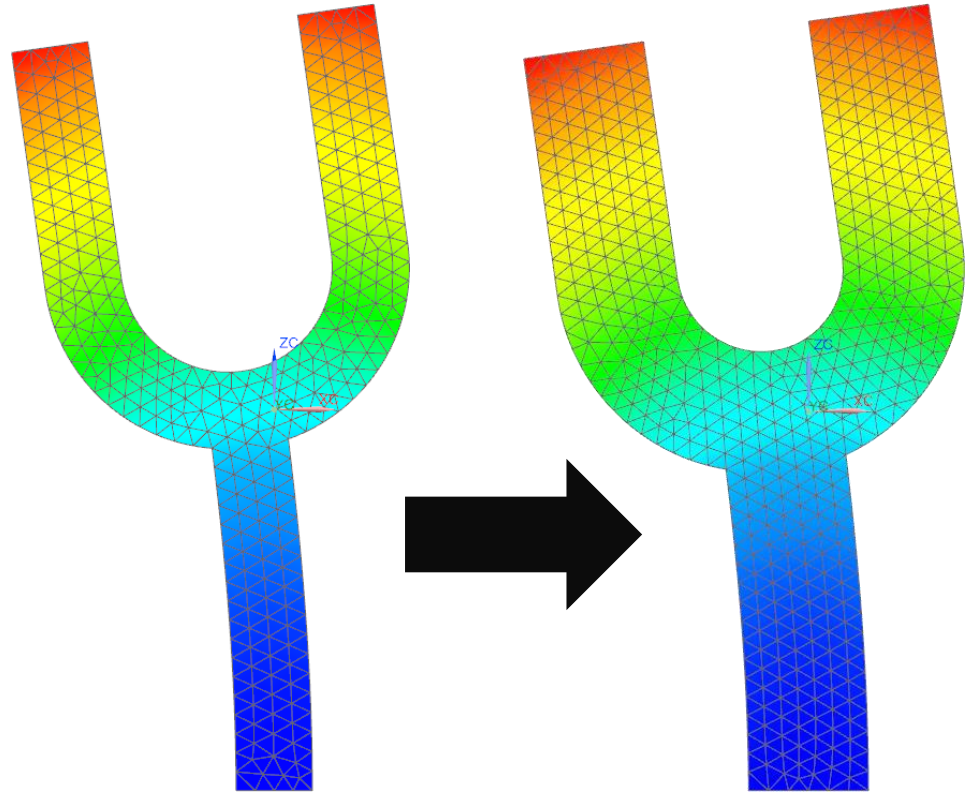
## **Can use optimization to find the best solution**

- Constraints may cause problems
- Validating needed

# Optimization

## First modal mode to 440 Hz

- Starting value 279,3 Hz
- Cross-section 20  $\rightarrow$  32 mm

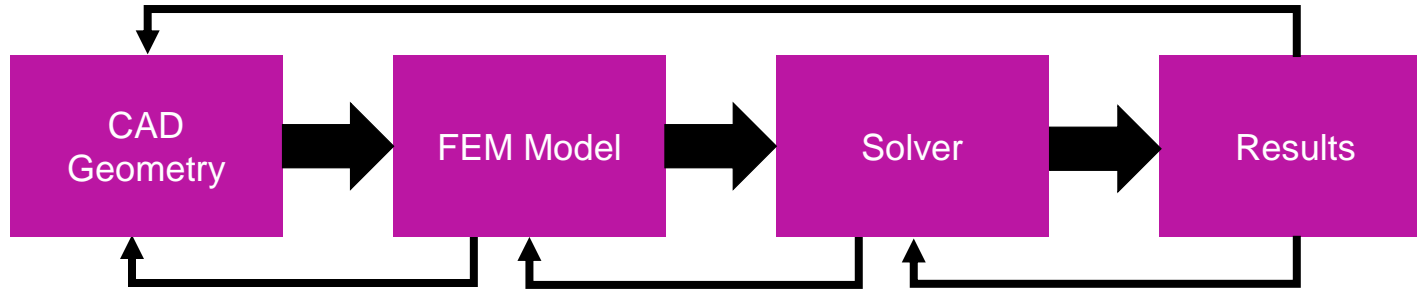




# FEM Process

## 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



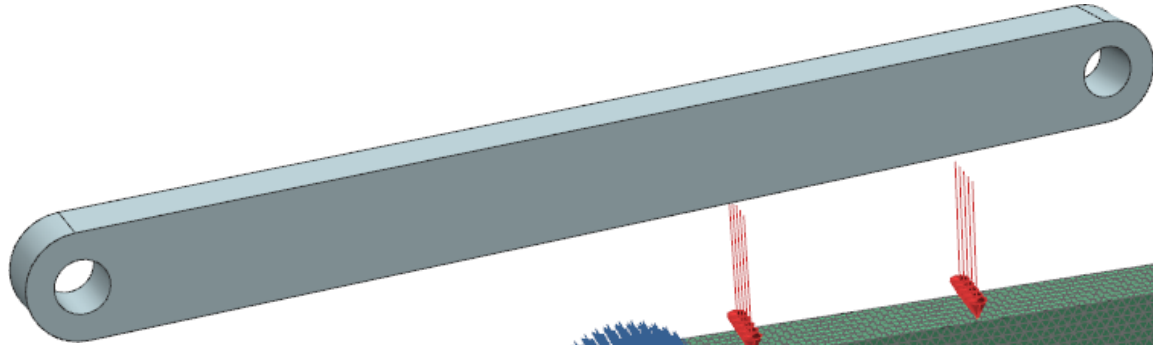
## Imported geometry



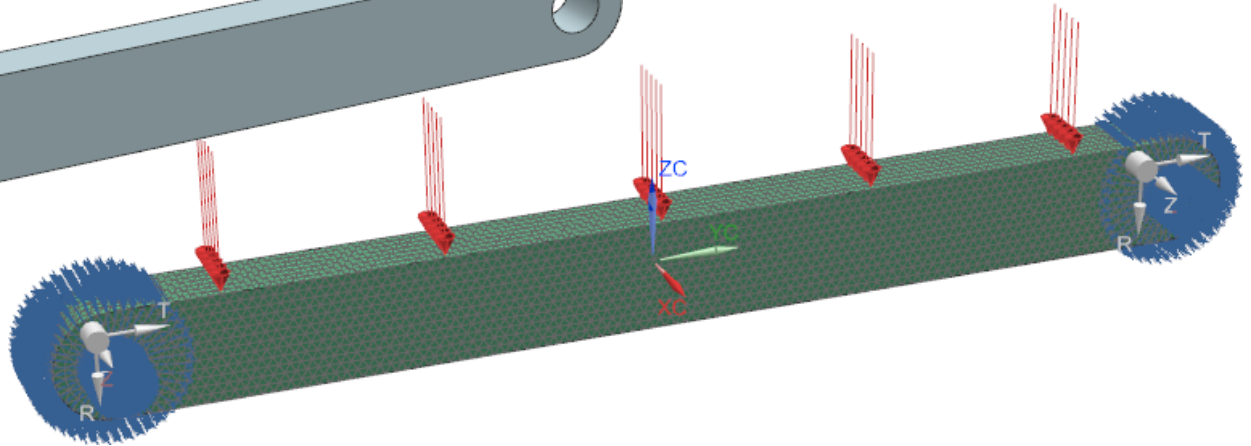
# Constraints

## Several ways to define boundary conditions

- i.e. constraints, forces



**CAD Part**

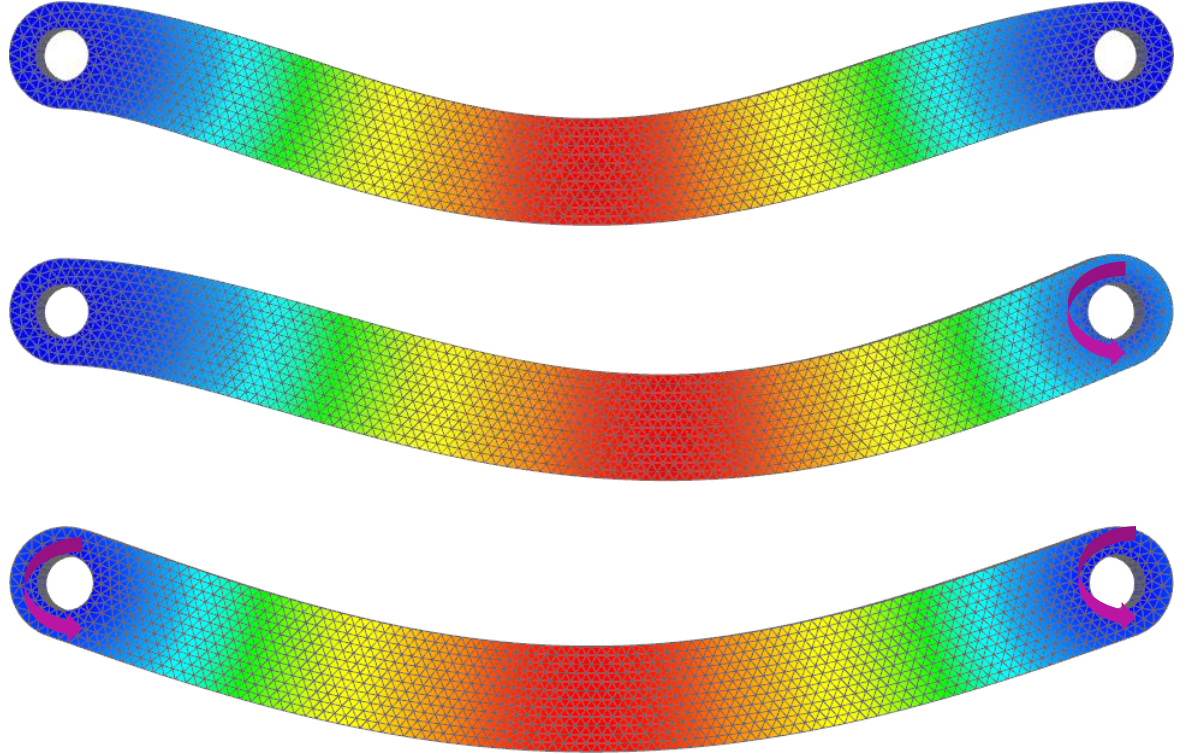


**FEM Model**

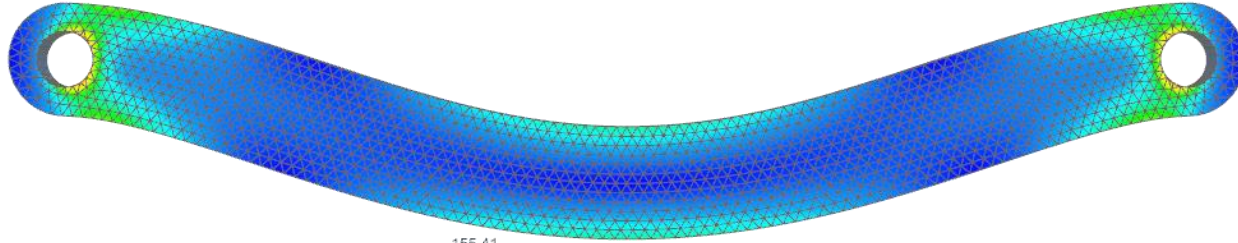
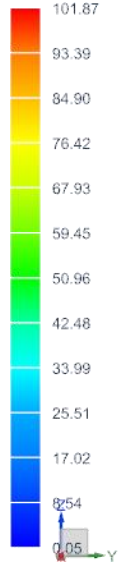
# Constraints

## Constraints matters

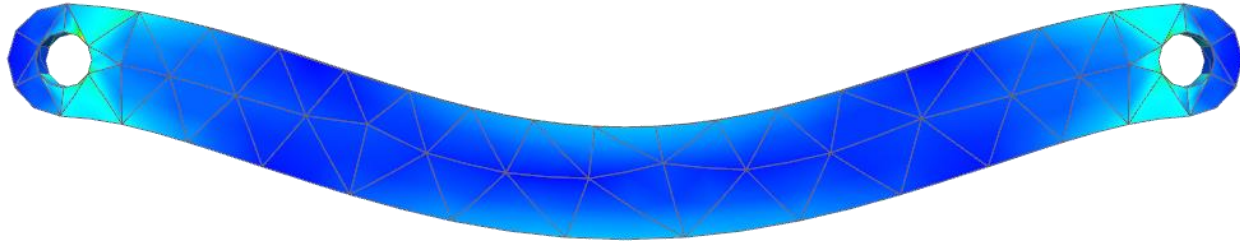
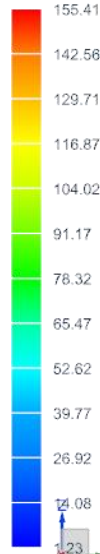
- Fixed or
- Revolute



# Mesh Size



**Nodal stresses  
(MPa)**



# Example Stand-alone FEM Programs

 COMSOL

 ABAQUS

 FEMAP

ANSYS<sup>®</sup>

# Courses about FEM

## **MEC-E1050 Finite Element Method in Solids**

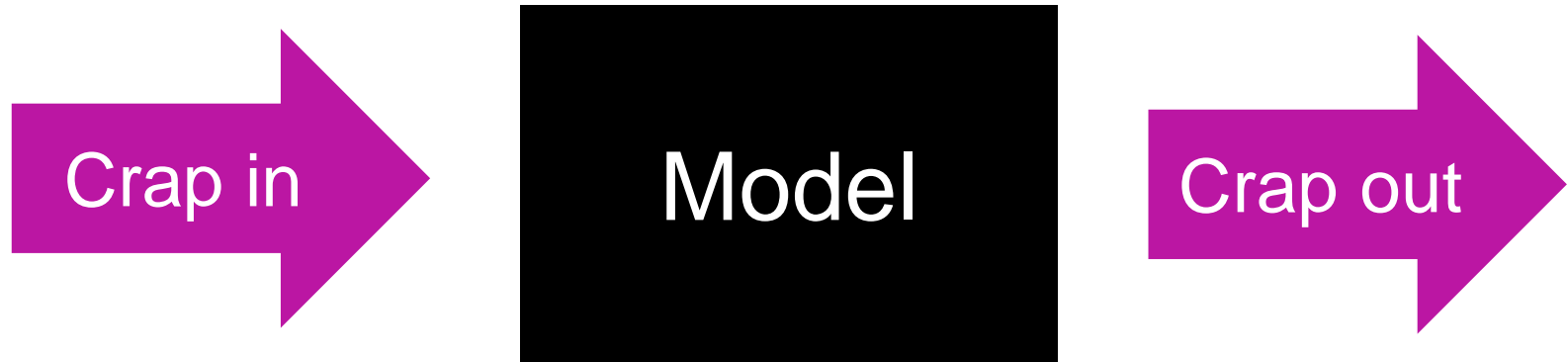
- 5 cr, starts on 2<sup>nd</sup> period

## **MEC-E8001 Finite Element Analysis L**

- 5 cr, starts on 3<sup>rd</sup> period



# User's Responsibility



# FEM Process in NX

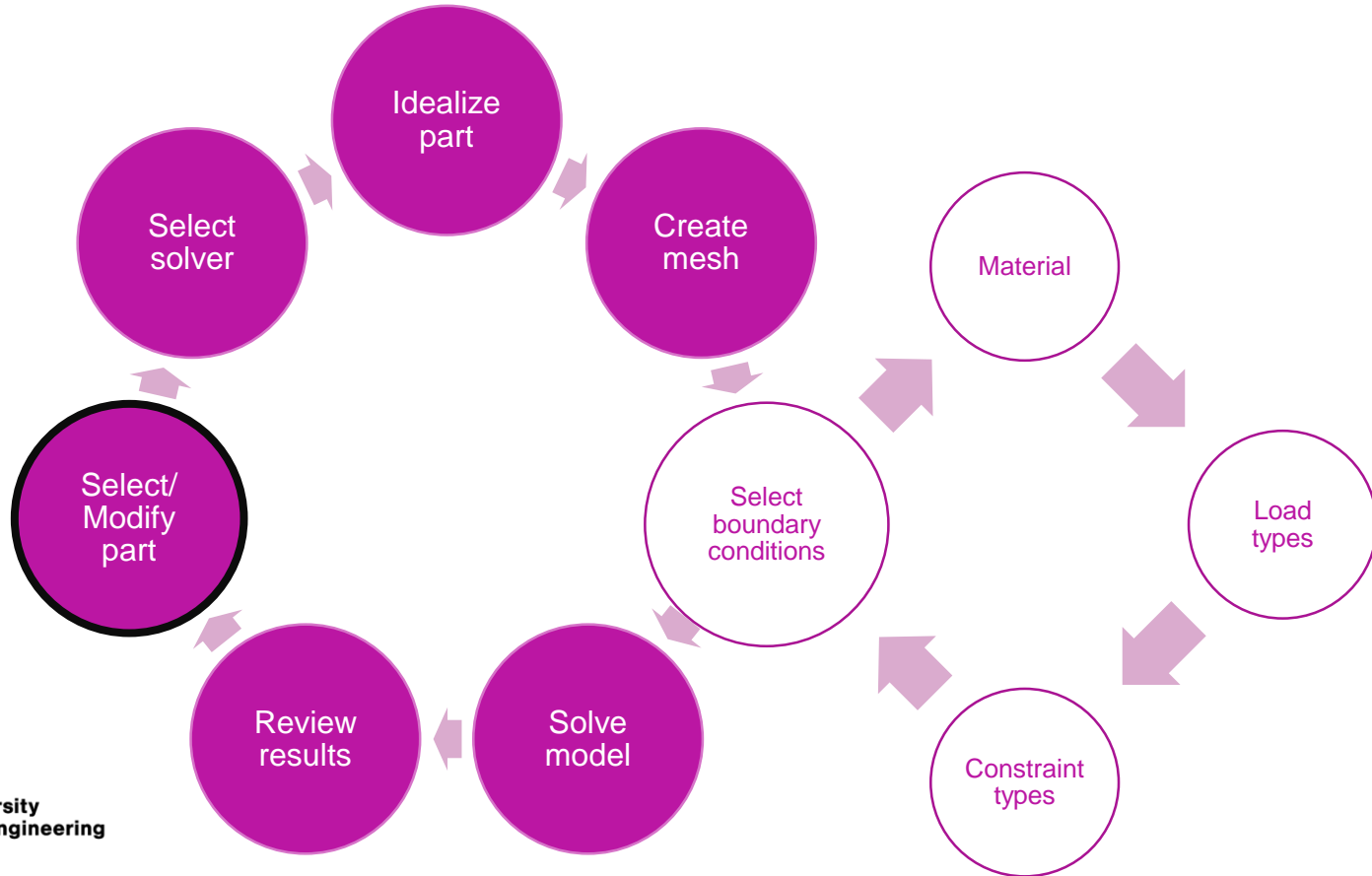
# **NX Simulate**

**Integrated into Siemens NX**

**Uses NASTRAN solver**

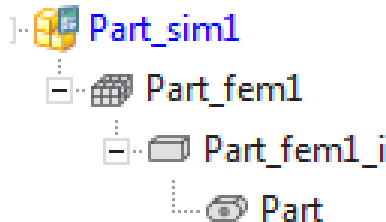
**Export to other solvers also possible**

# FEA Process



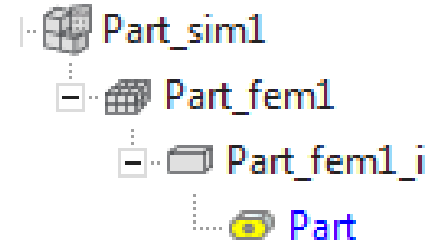
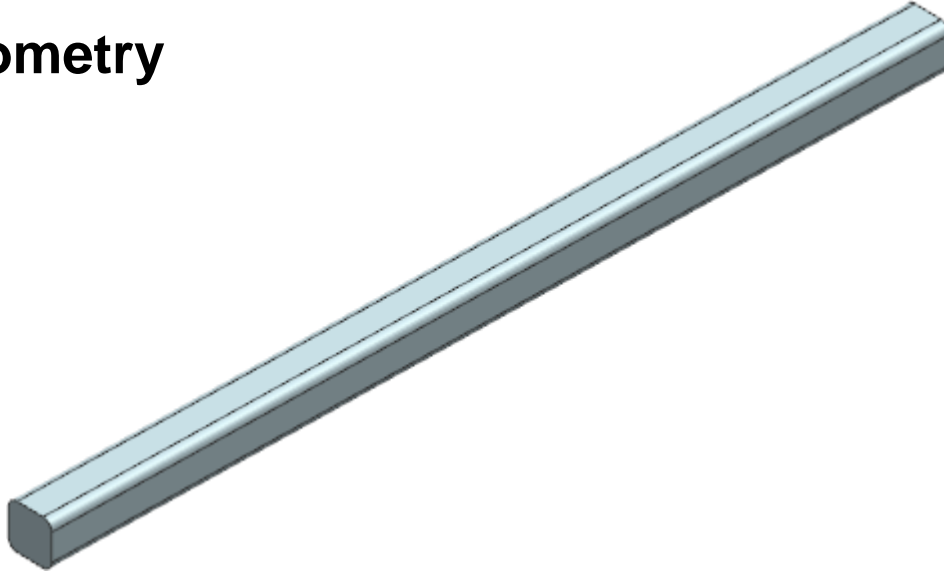
# Different Parts in NX

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

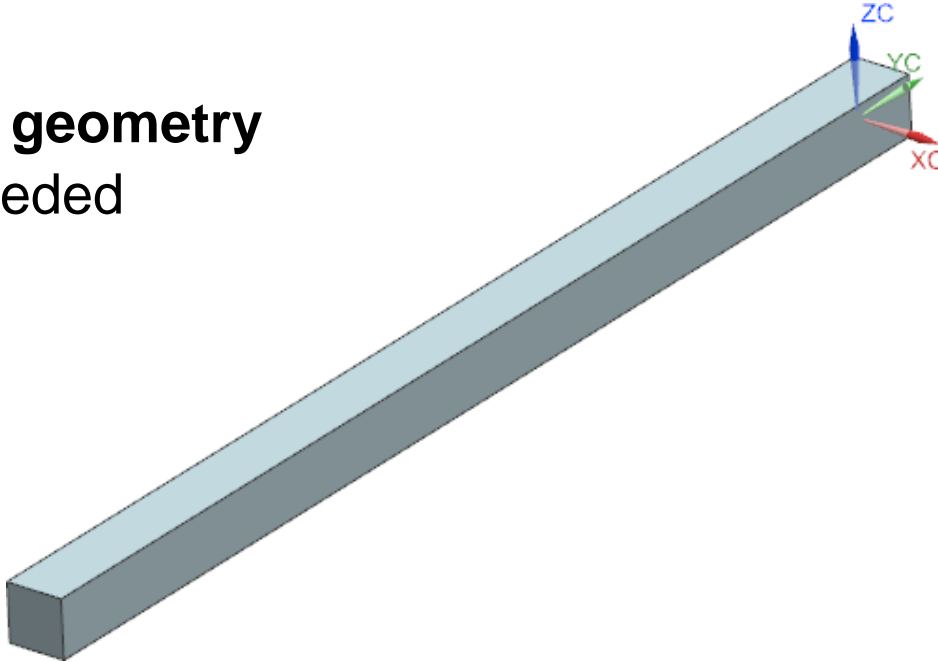
CAD geometry



# Displacement of a Beam

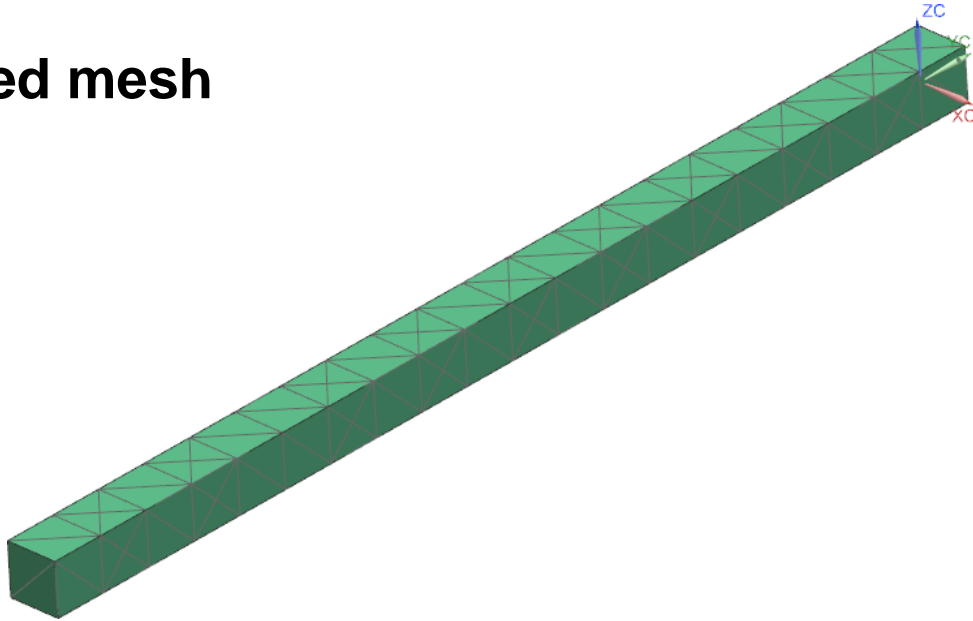
## Idealized geometry

- If needed



# Displacement of a Beam

Generated mesh

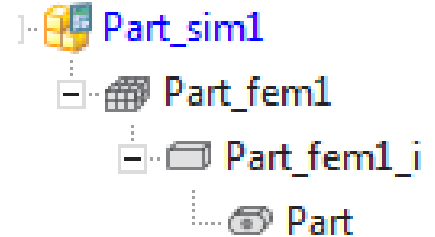
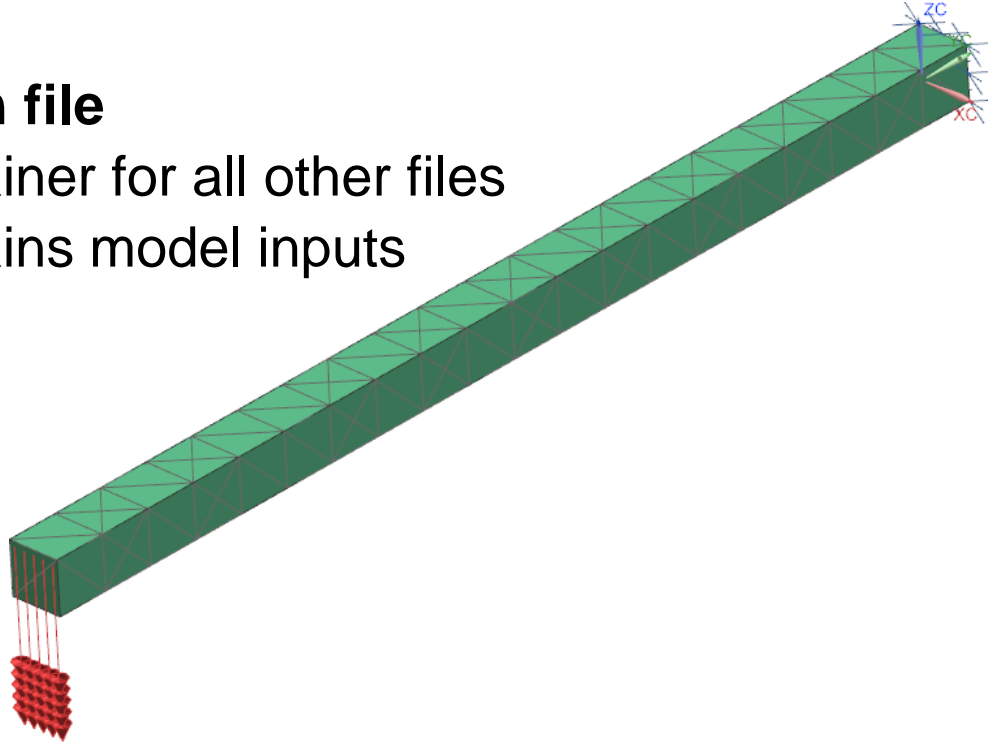




# Displacement of a Beam

## 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

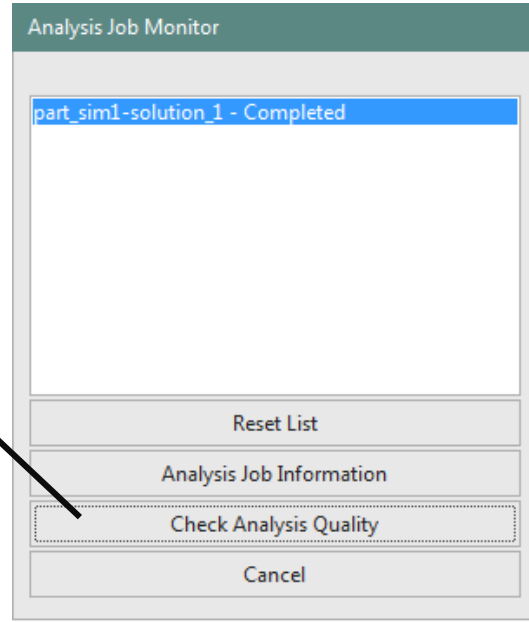
Strain Energy Error [ 3.937797% ]

Steady Stress (Relative) Error [ 4.002476% ]

Steady Stress (Absolute) Error [ 4.135979N/mm<sup>2</sup> (MPa) ]



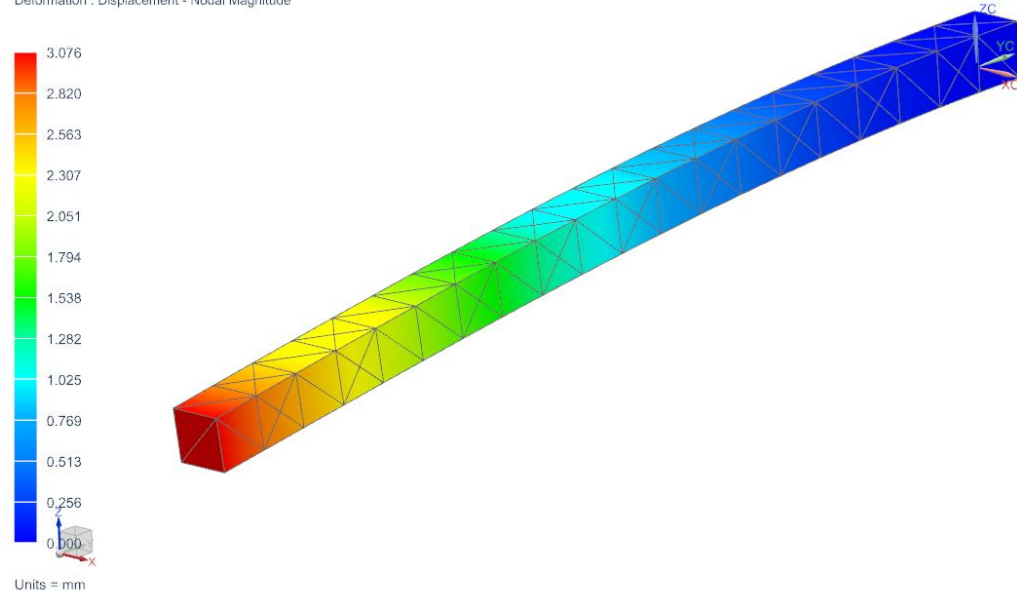
Aalto University  
School of Engineering



# Results

## Displacement

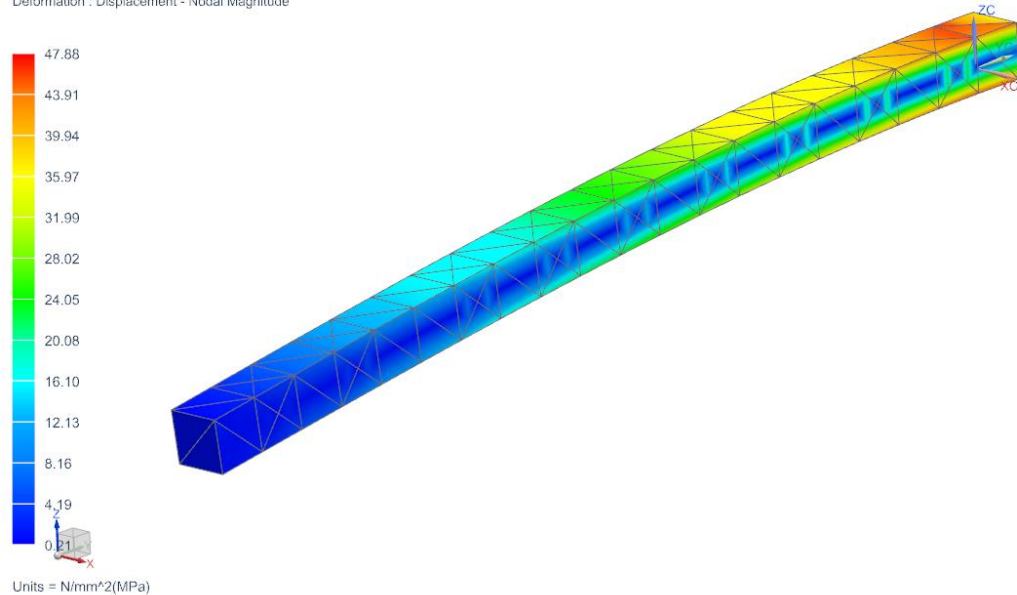
Part\_sim1 : Solution 1 Result  
Subcase - Static Loads 1, Static Step 1  
Displacement - Nodal, Magnitude  
Min : 0.000, Max : 3.076, Units = mm  
Deformation : Displacement - Nodal Magnitude



# Results

## Stress

Part\_sim1 : Solution 1 Result  
Subcase - Static Loads 1, Static Step 1  
Stress - Element-Nodal, Unaveraged, Von-Mises  
Min : 0.21, Max : 47.88, Units = N/mm<sup>2</sup>(MPa)  
Deformation : Displacement - Nodal Magnitude



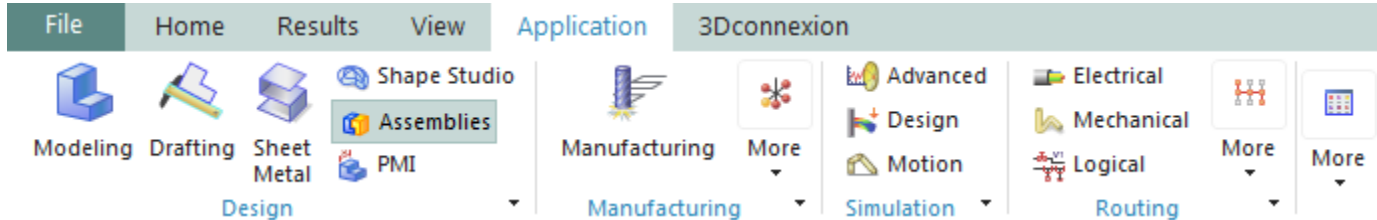
# Different modes

## To update CAD geometry

- When part file is active, Application → Modeling

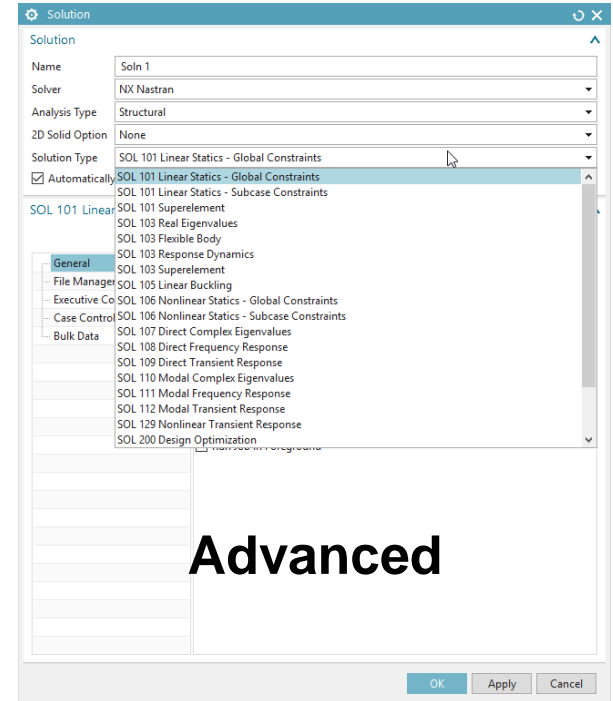
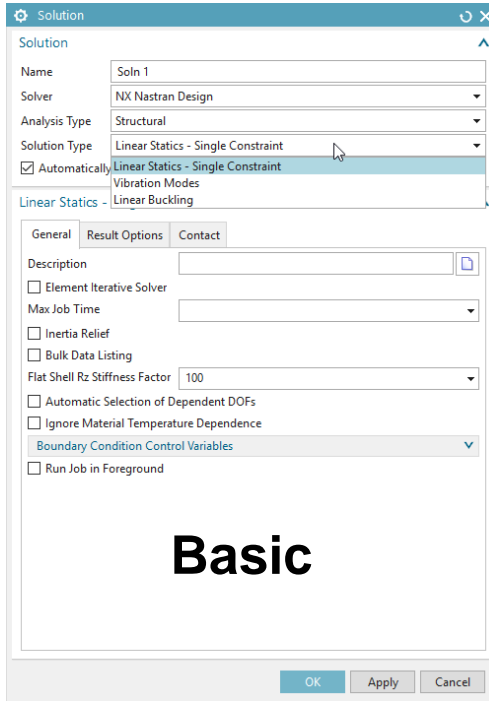
## To access FEM model

- Application → Design



# Different Solvers

Basic solver should be fine for this course



# Geometry Optimization

## Goal

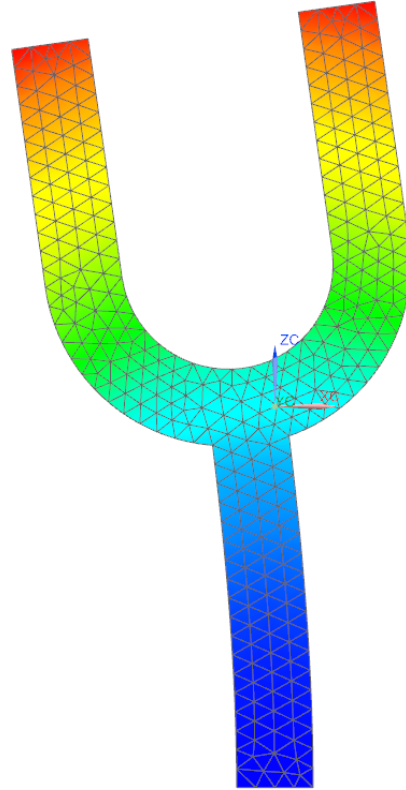
- First modal mode to 440 Hz

## Vibration analyze result

- First mode 279,3 Hz

## Starting geometry

- Cross-section 20



# Geometry Optimization

Requires ready made simulation as input

Create Geometry Optimization Solution

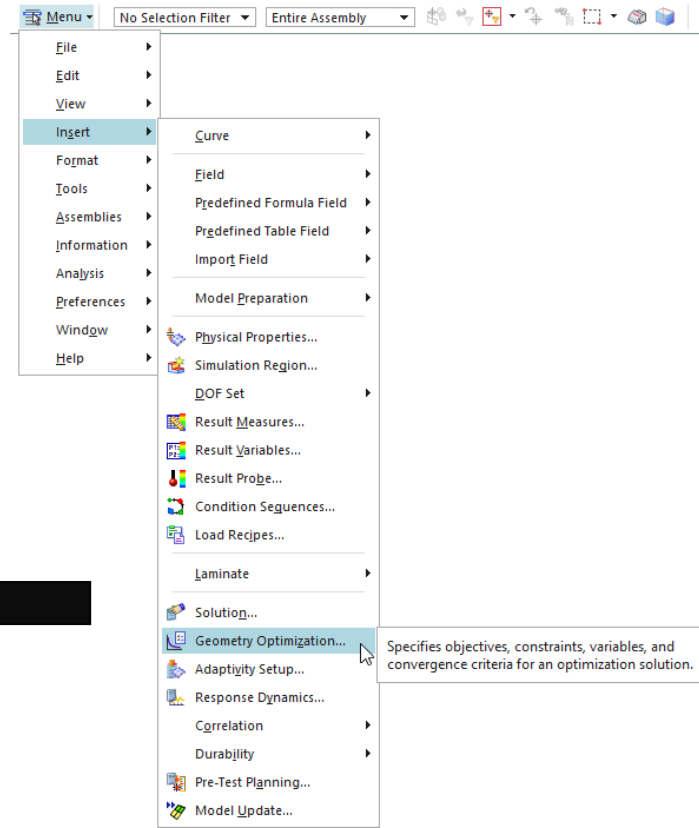
Name  
DemoOptimization

Solution List

Type	Name
Vibration Modes	Modal Analysis

Process Type  
Optimizer

OK Cancel

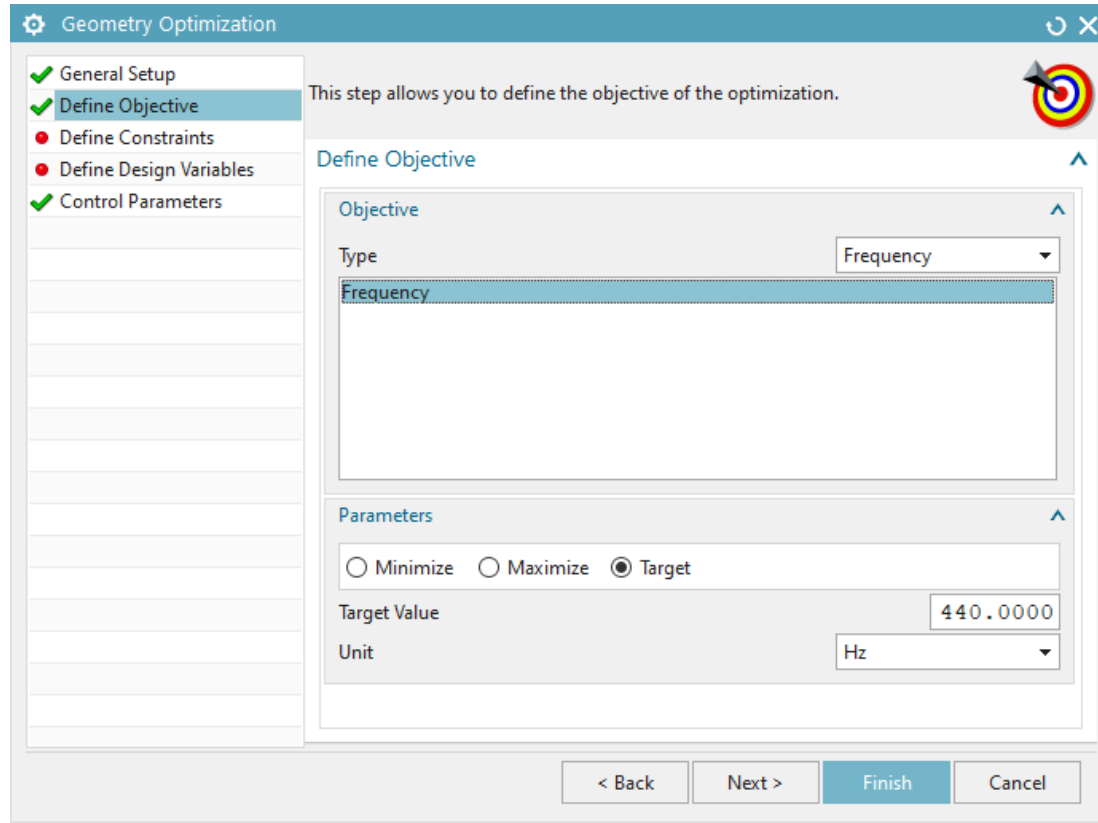




# Geometry Optimization

## Objective

- First mode frequency to 440 Hz



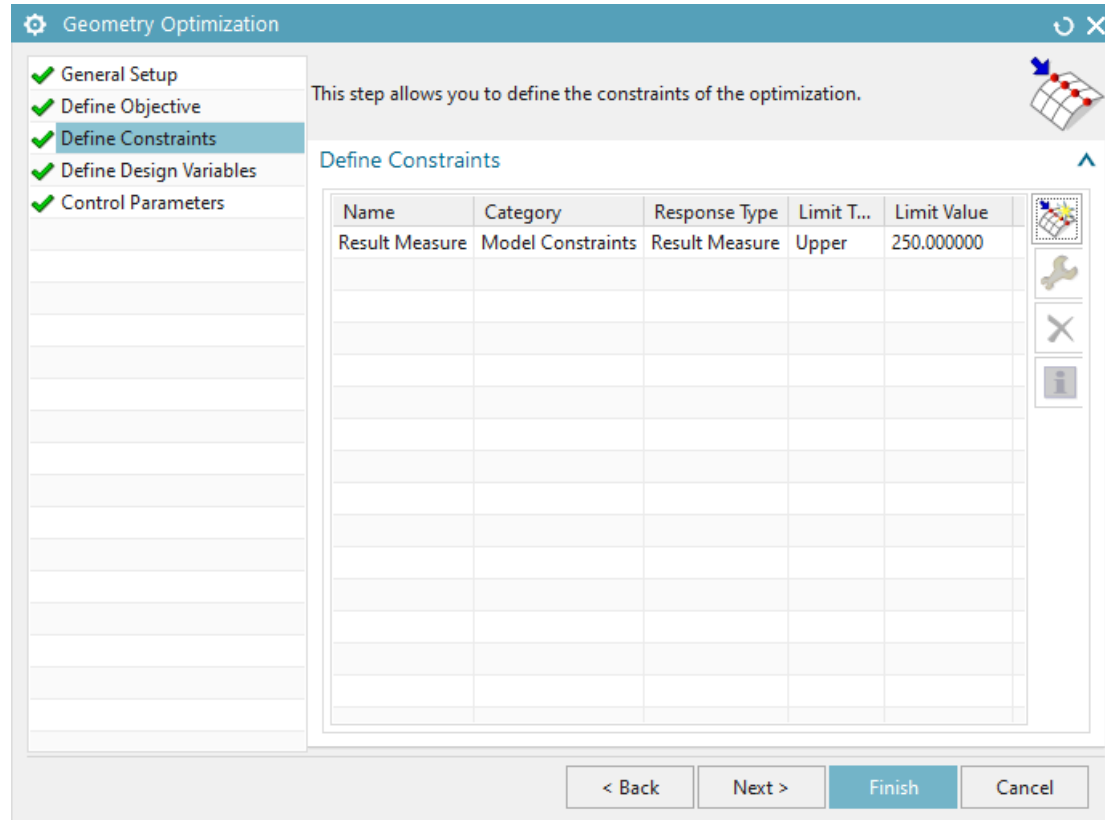
The screenshot shows the 'Geometry Optimization' window. On the left is a sidebar with a list of steps: 'General Setup' (checked), 'Define Objective' (checked and highlighted), 'Define Constraints' (unchecked), 'Define Design Variables' (unchecked), and 'Control Parameters' (checked). The main area is titled 'Define Objective' and contains the following fields:

- Objective:** A dropdown menu set to 'Frequency'.
- Type:** A dropdown menu set to 'Frequency'.
- Parameters:** A section with three radio buttons: 'Minimize' (unchecked), 'Maximize' (unchecked), and 'Target' (checked).
- Target Value:** A text input field containing '440.0000'.
- Unit:** A dropdown menu set to 'Hz'.

At the bottom of the window are four buttons: '< Back', 'Next >', 'Finish' (highlighted in blue), and 'Cancel'.

# Constrains

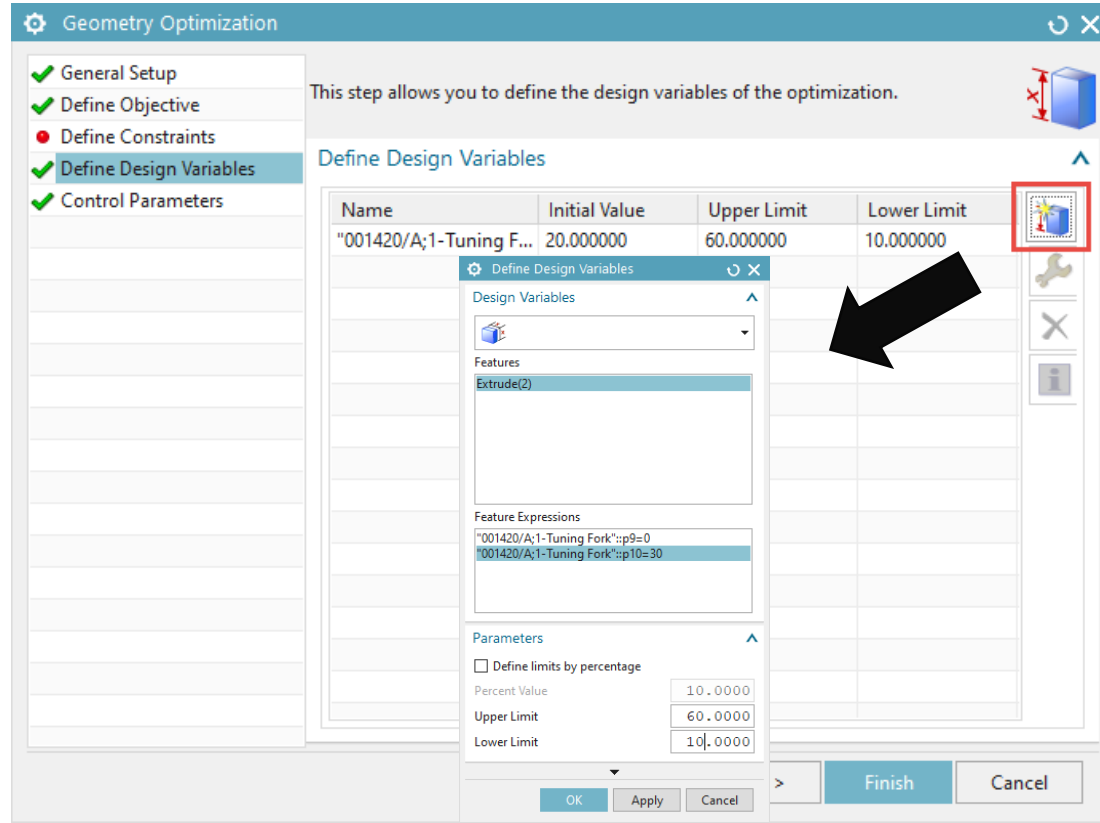
- A?** Aalto University  
School of Engineering



# Geometry Optimization

Dimensions that optimization tool can change

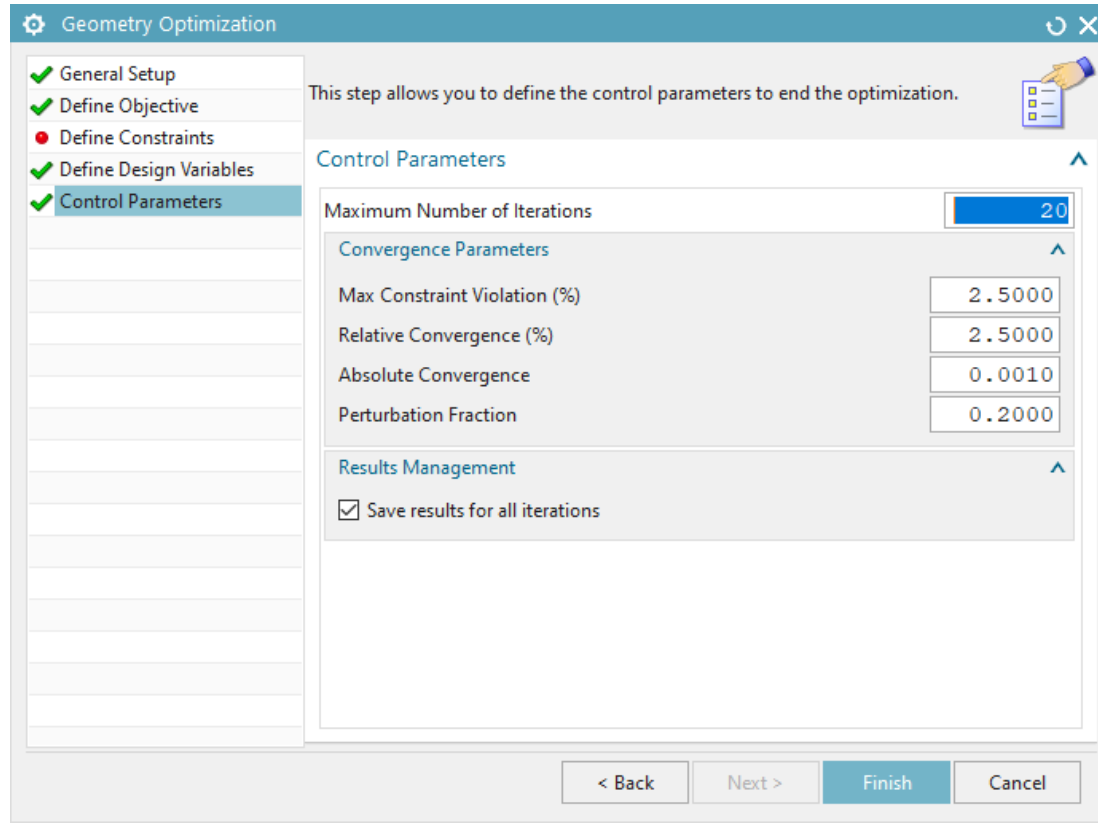
- Within limits



# Geometry Optimization

## Control parameters for solver

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



The screenshot shows the 'Geometry Optimization' window. On the left is a sidebar with a list of steps: 'General Setup' (checked), 'Define Objective' (checked), 'Define Constraints' (not checked), 'Define Design Variables' (checked), and 'Control Parameters' (checked and highlighted). The main area has a title bar 'Geometry Optimization' and a close button. Below the title bar is a message: 'This step allows you to define the control parameters to end the optimization.' To the right of this message is a small icon of a hand pointing to a list. The main content area is titled 'Control Parameters' and contains several sections: 'Maximum Number of Iterations' with a value of 20, 'Convergence Parameters' with sub-sections for 'Max Constraint Violation (%)' (2.5000), 'Relative Convergence (%)' (2.5000), 'Absolute Convergence' (0.0010), and 'Perturbation Fraction' (0.2000). Below these is a 'Results Management' section with a checkbox 'Save results for all iterations' which is checked. At the bottom of the window are four buttons: '< Back', 'Next >', 'Finish', and 'Cancel'.

Geometry Optimization

This step allows you to define the control parameters to end the optimization.

Control Parameters

Maximum Number of Iterations: 20

Convergence Parameters

Max Constraint Violation (%): 2.5000

Relative Convergence (%): 2.5000

Absolute Convergence: 0.0010

Perturbation Fraction: 0.2000

Results Management

☒ Save results for all iterations

< Back Next > Finish Cancel

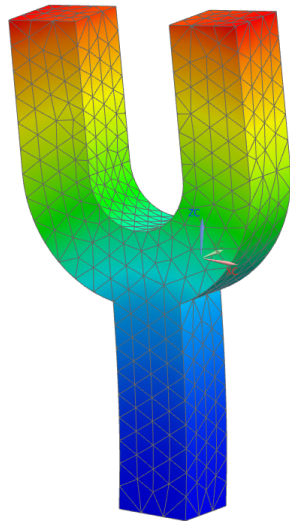
# Geometry Optimization

## Results (notice the 2.5% convergence margin)

Post Processing Navigator		
Name	Description	Status
001420.sim1/A/1		
Modal Analysis	NX Nastran Design, NX Nastran De...	Inferred
Copy of Modal Analysis	NX Nastran Design, NX Nastran De...	Inferred
Structural		Inferred
Modal Optimization	Optimization Setup on Modal Anal...	Inferred
Setup 1	Optimization Setup on Copy of M...	Inferred
Mode 1, 279.847 Hz		
Mode 1, 419.275 Hz		
Mode 1, 439.79 Hz		
Mode 1, 435.015 Hz		
Displacement - Nodal		
Rotation - Nodal		
Stress - Elemental		
Stress - Element-No...		
Strain - Elemental		
Strain - Element-No...		
Strain Energy - Elem...		
Strain Energy Density...		
Reaction Force - No...		
Reaction Moment - ...		
Imported Results		
Fringe Plots		
Post View 2	(MASTER) Displacement - Nodal D...	
Templates		

001420.sim1\_A : Setup 1 Result  
COPY OF MODAL ANALYSIS, Mode 1, 435.015 Hz  
Displacement - Nodal, Magnitude  
Min : 0.000, Max : 1.126, Units = mm  
Deformation : Displacement - Nodal Magnitude

1.126  
1.032  
0.938  
0.844  
0.750  
0.657  
0.563  
0.469  
0.375  
0.281  
0.188  
0.094  
0.000  
Units = mm



### 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

"001420/A;1-Tuning Fork"::p10=20

0	1	2	3
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

# 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