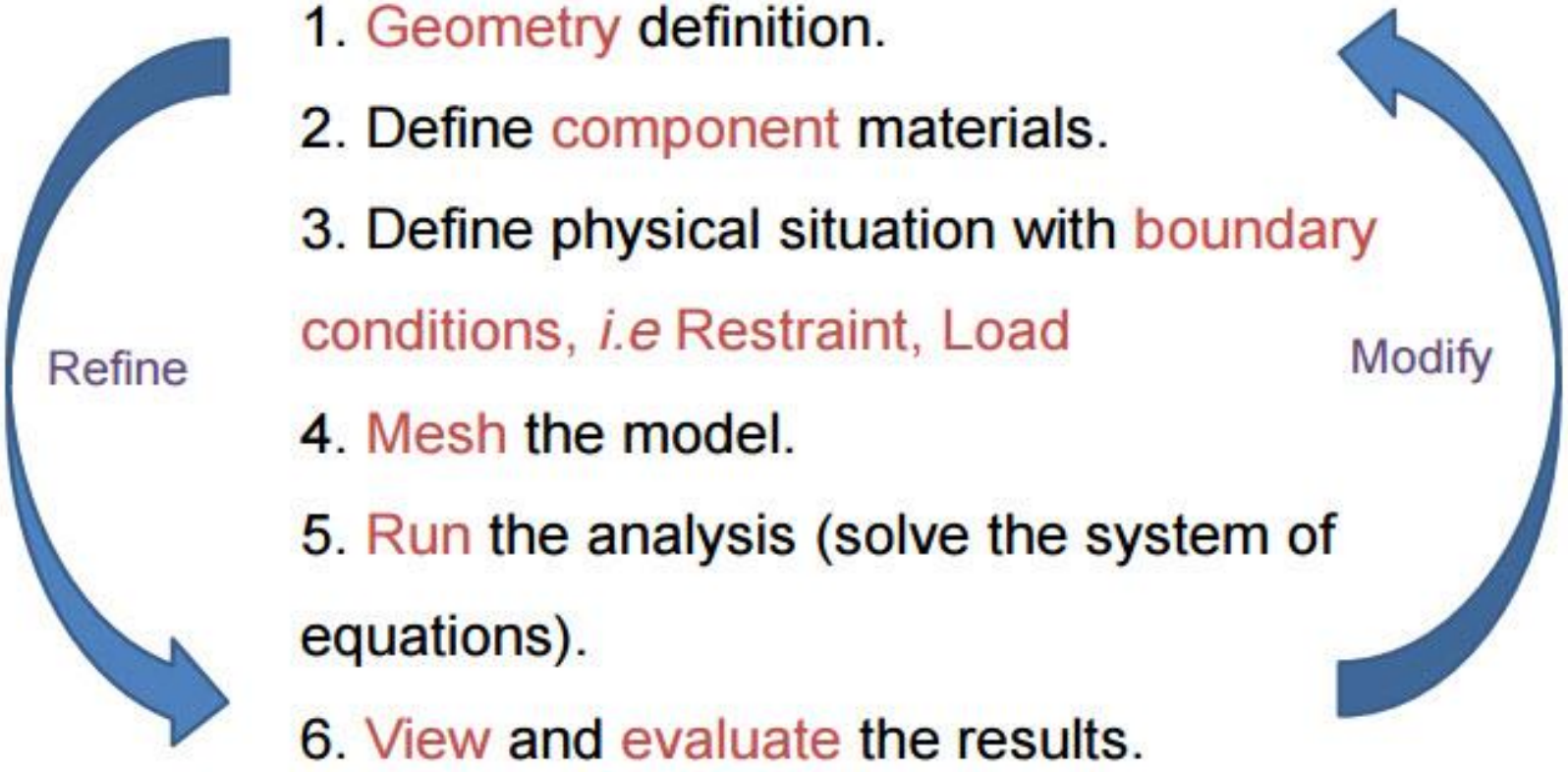
Tutorial 2

Armin Yousefi Kanani

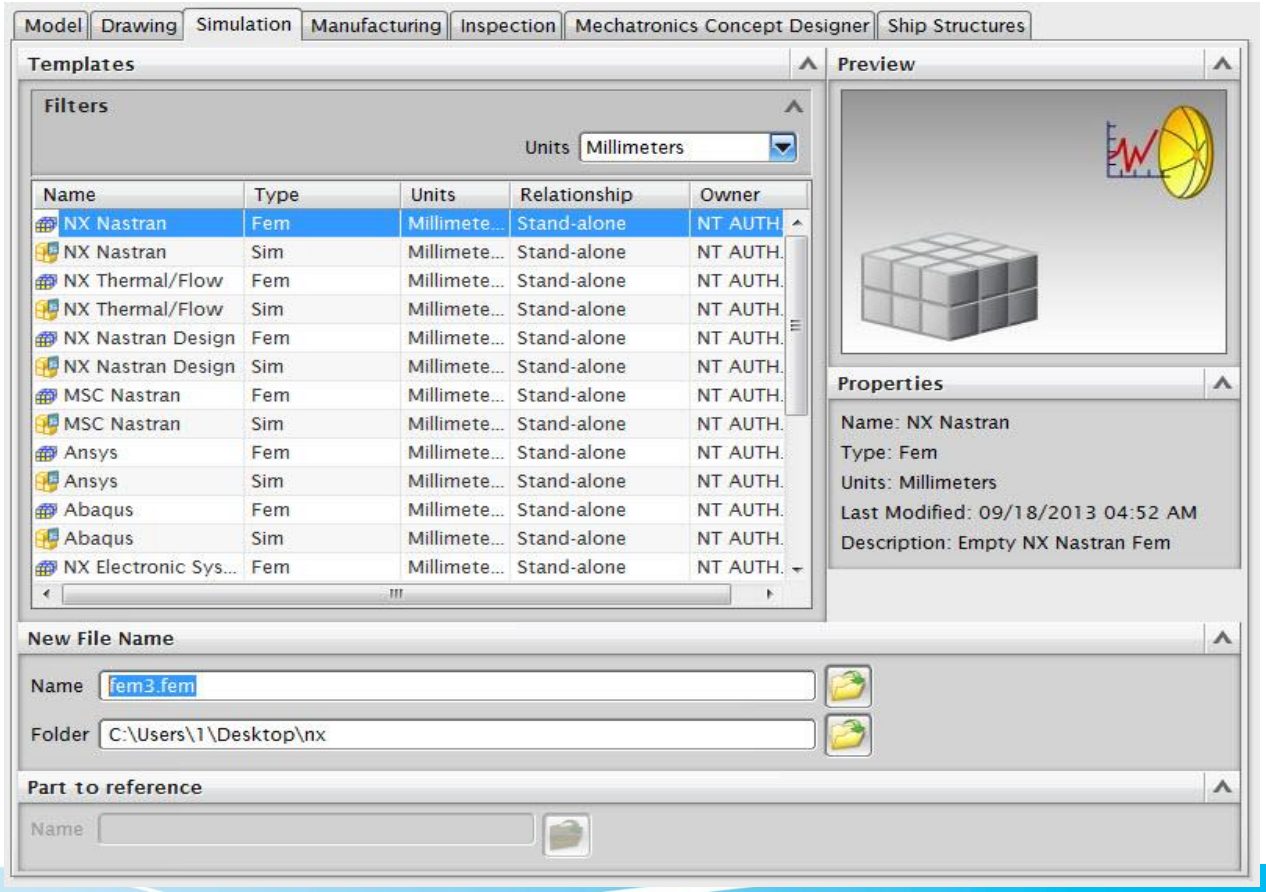
FEA Process in NX/NASTRAN



1- Structure Module

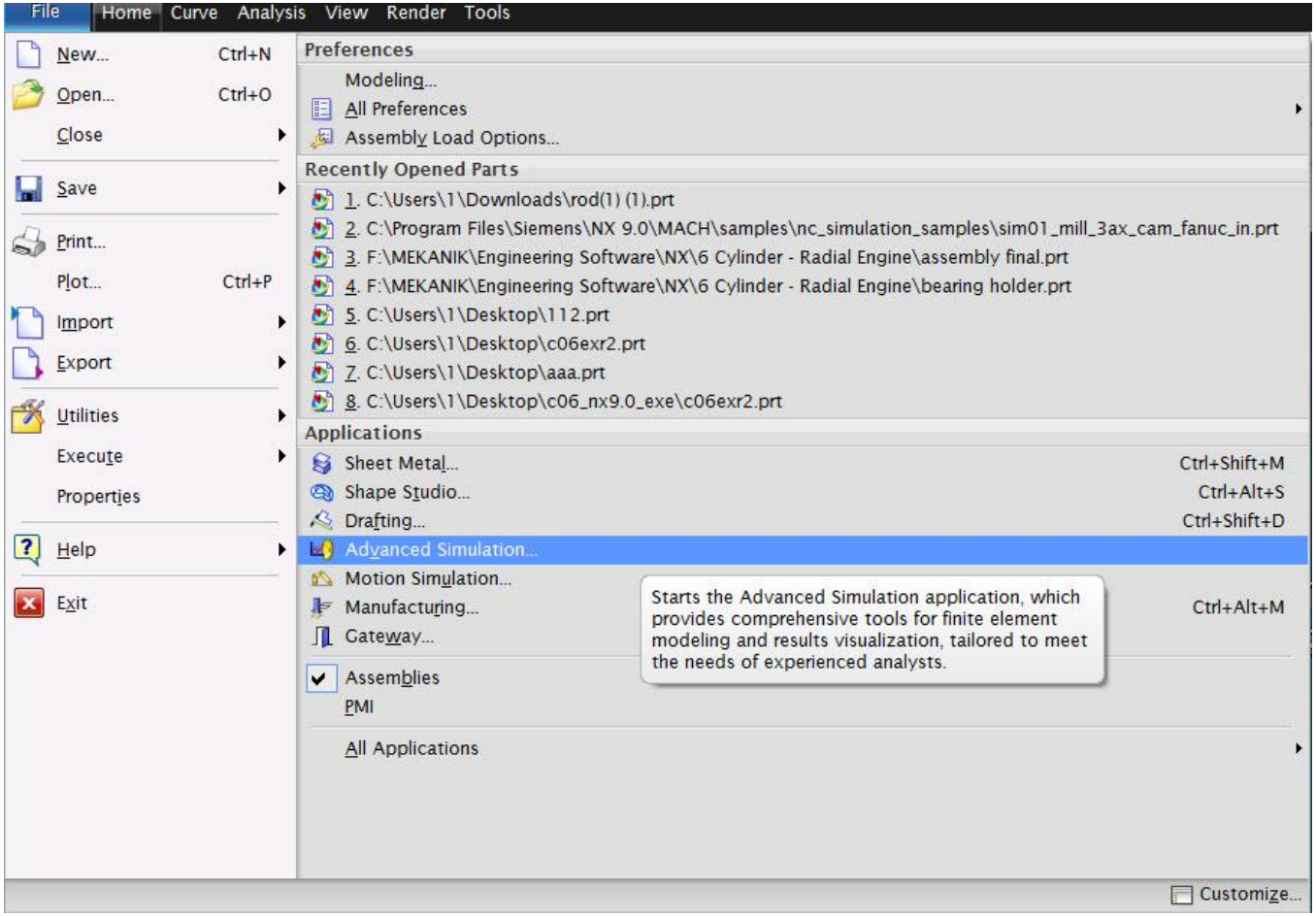
There are two method to open model in Design simulation in NX.

1- Copy and paste the file **Bracket.prt** into a new folder>> Open this newly copied file. then Click on **NEW** ⟶ **ADVANCED SIMULATIONS** if the part is NOT already opened in the NX window as shown below.



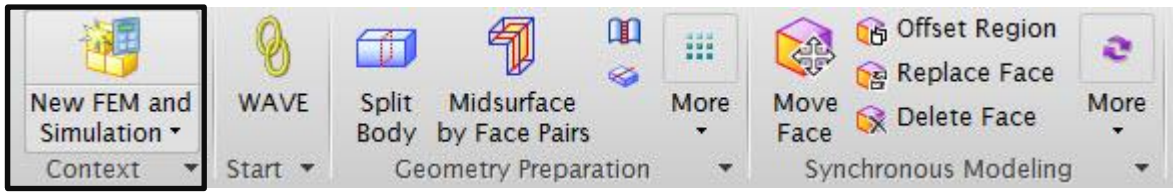
1-1 Second method

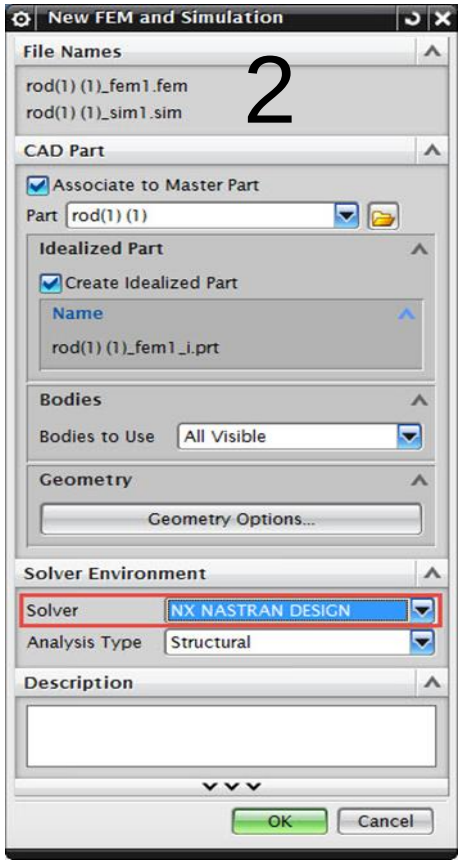
If part is already opened in NX, then click on
NEW ⟶ ADVANCED SIMULATIONS



Create Simulation

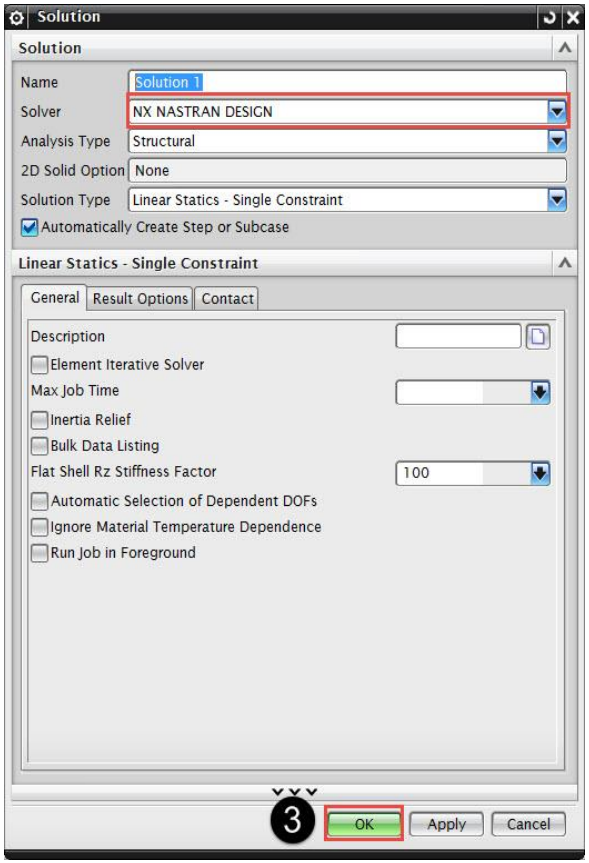
1



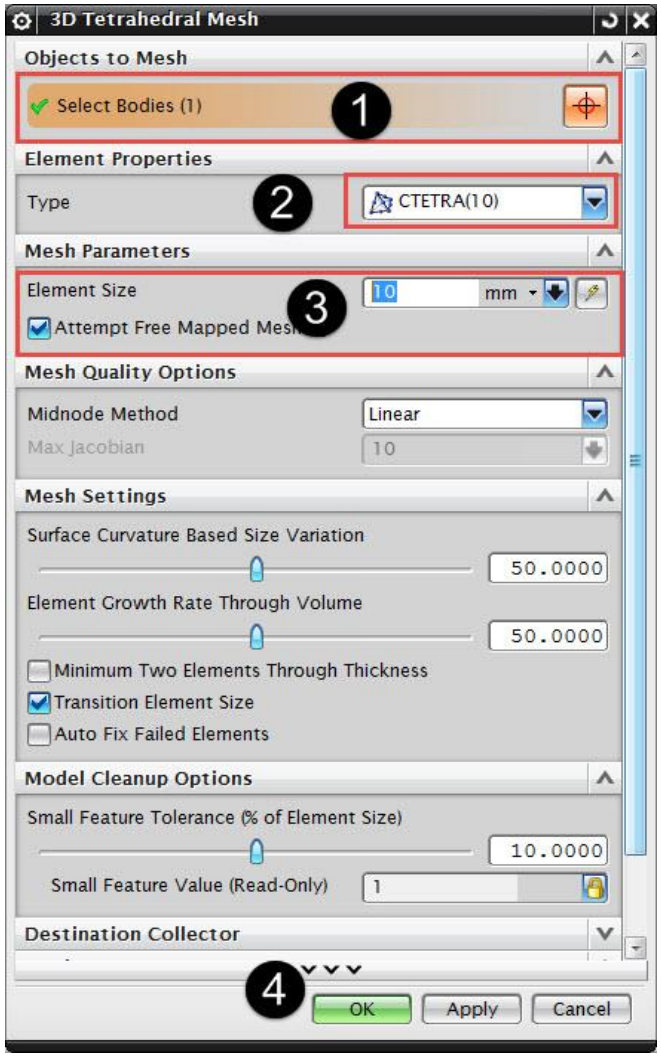


In first step you need to define the type of simulation in Ansys.

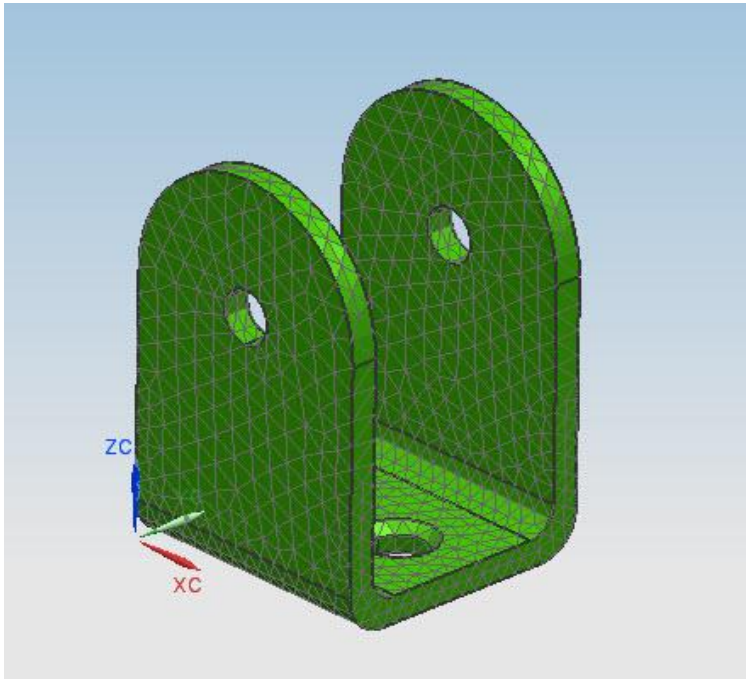
In this tutorial we are going to use **NX NASTARAN DESING** as solver and then click on **okay**.



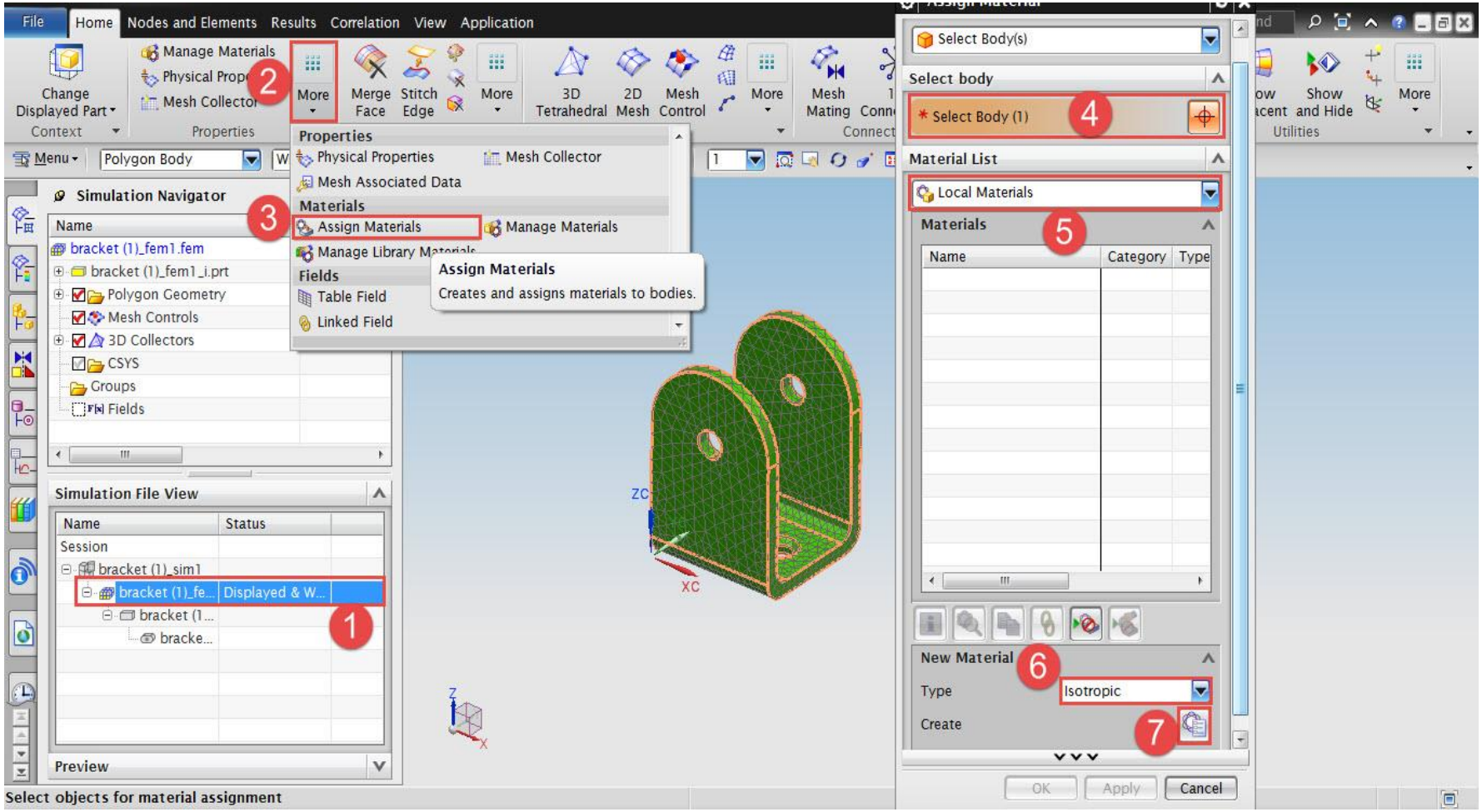
Mesh:



In this step mesh will apply to geometry.
Mesh definition>>>refer to >>previous tutorial.
First select body then give element size 10 mm.
For more exercise change element type and compare result with each other.



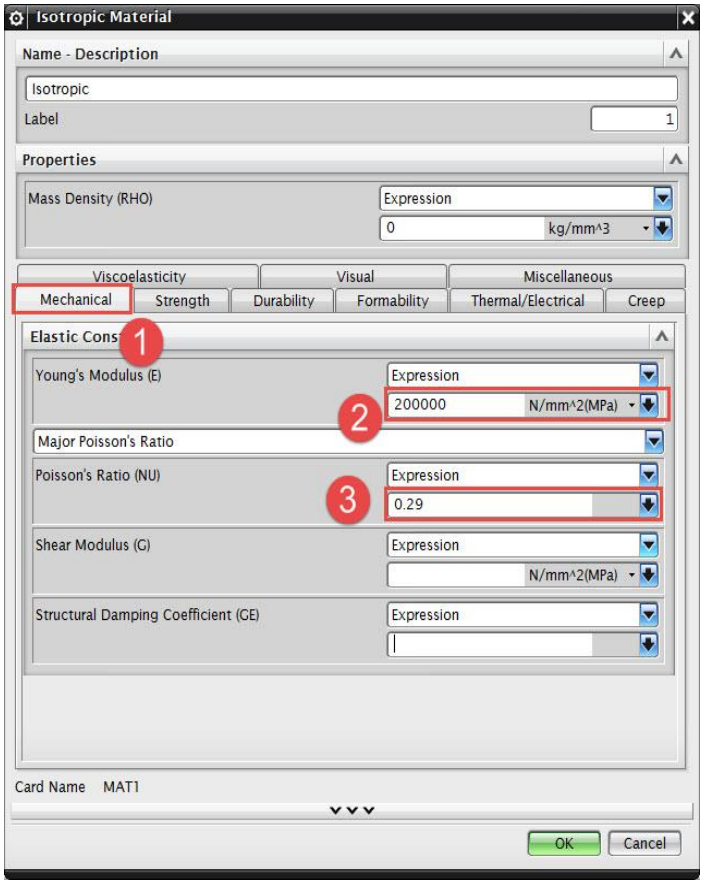
Material:

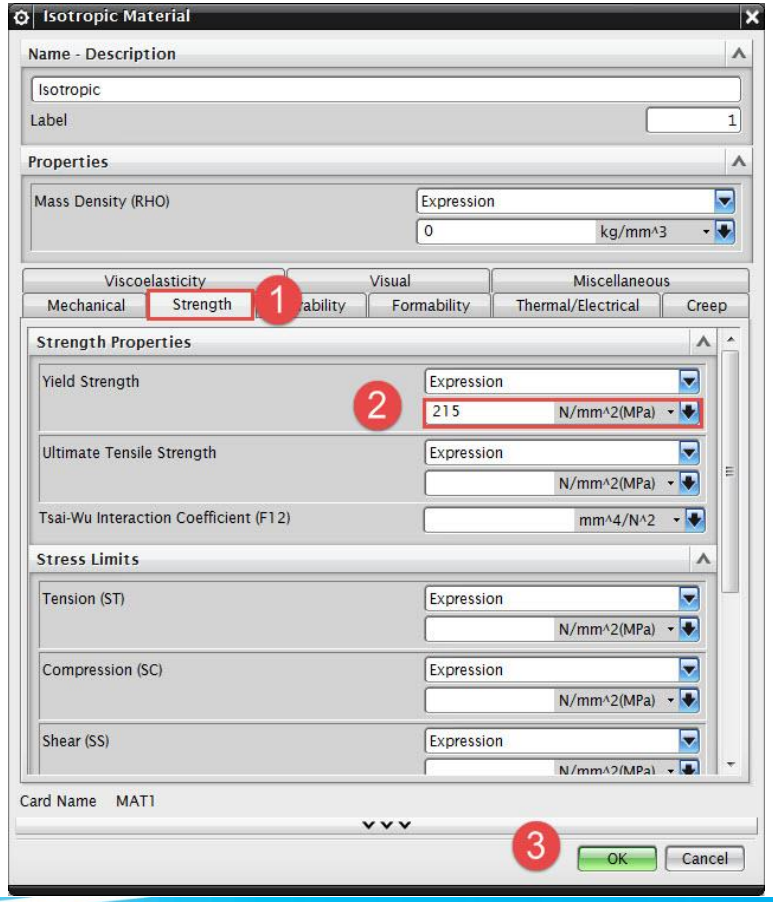

File
Home
Nodes and Elements
Results
Correlation
View
Application
Manage Materials
Physical Properties
Mesh Collector
Properties
More
Merge Face
Stitch Edge
3D Tetrahedral
2D Mesh
Mesh Control
Mesh Mating
Change Displayed Part
Context
Show and Hide
Utilities
Menu
Polygon Body
Simulation Navigator
Name
bracket (1)_fem1.fem
bracket (1)_fem1_i.prt
Polygon Geometry
Mesh Controls
3D Collectors
CSYS
Groups
Fields
Properties
Physical Properties
Mesh Collector
Mesh Associated Data
Materials
Assign Materials
Manage Materials
Manage Library Materials
Fields
Table Field
Linked Field
Assign Materials
Creates and assigns materials to bodies.
Simulation File View
Name
Status
Session
bracket (1)_sim1
bracket (1)_fe...
Displayed & W...
bracket (1...
bracke...
Preview
ZC
XC
Select Body(s)
Select body
Select Body (1)
Material List
Local Materials
Materials
Name
Category
Type
New Material
Type
Isotropic
Create
OK
Apply
Cancel
1
2
3
4
5
6
7
Select objects for material assignment

Material

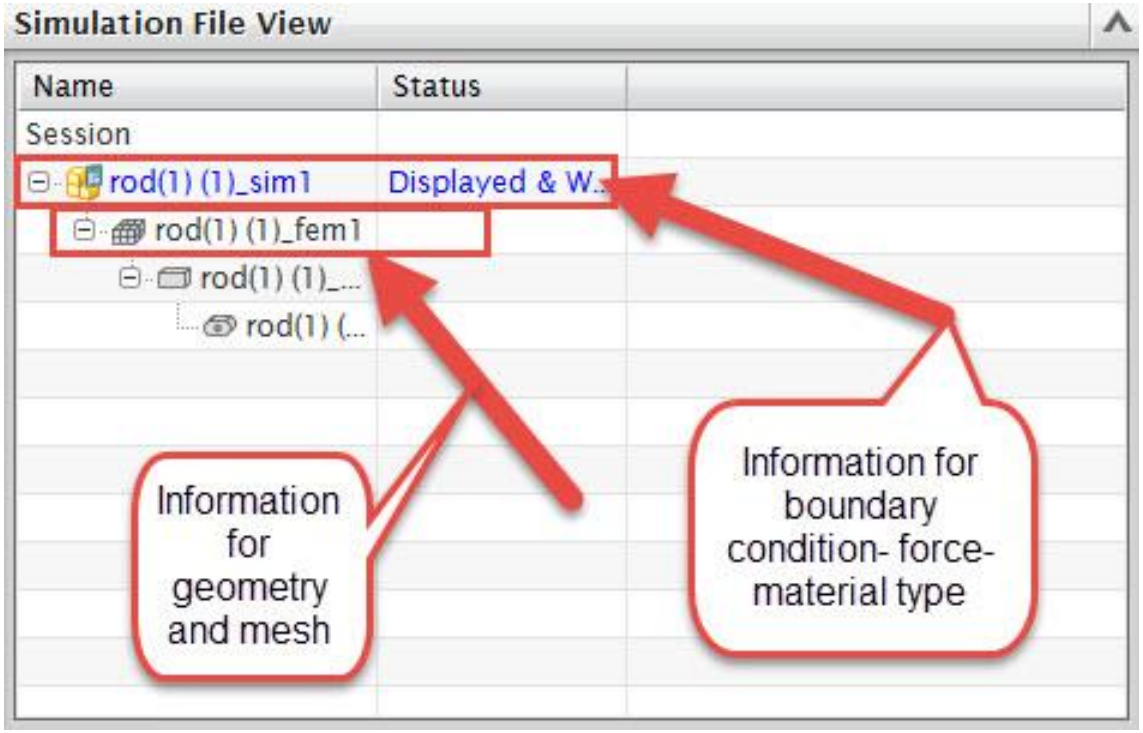
AISI Type 304 Stainless Steel

Physical Properties	Metric	English
Tensile Strength, Yield	215 MPa	31200 psi
Elongation at Break	70 %	70 %
Modulus of Elasticity	193 - 200 GPa	28000 - 29000 ksi
Poisson's Ratio	0.29	0.29

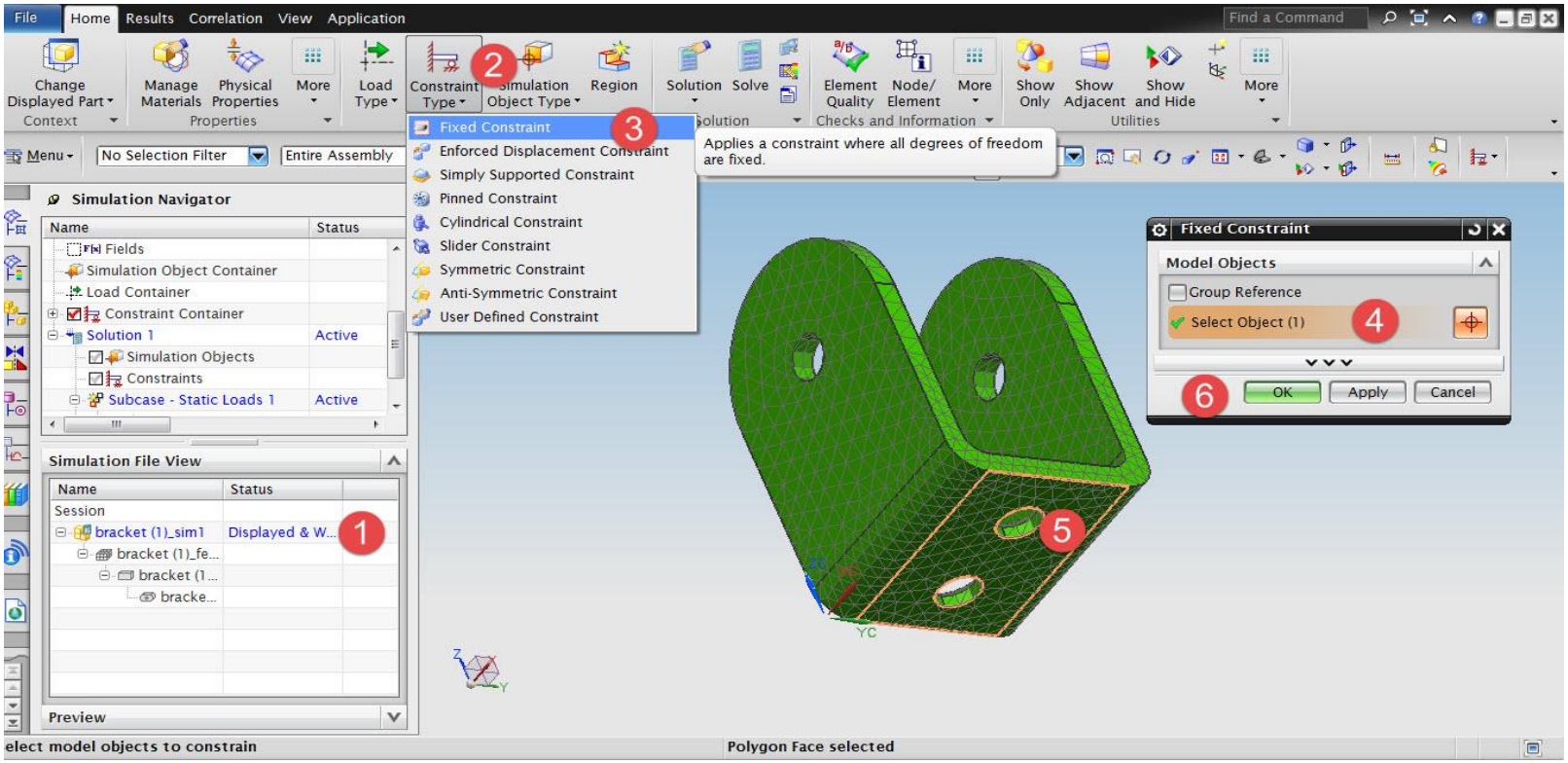




How to switch from fem to sim

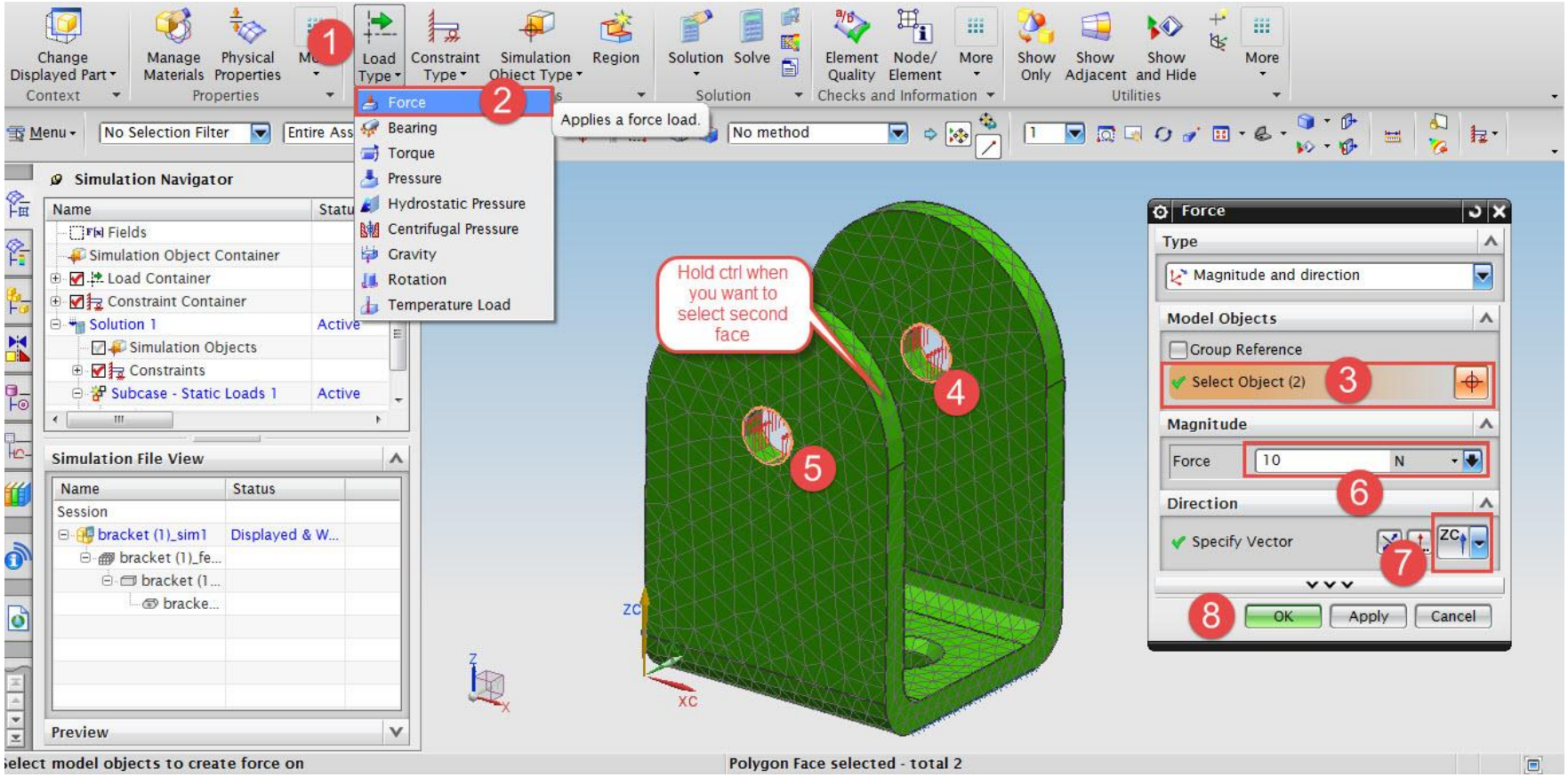


Boundary Condition:



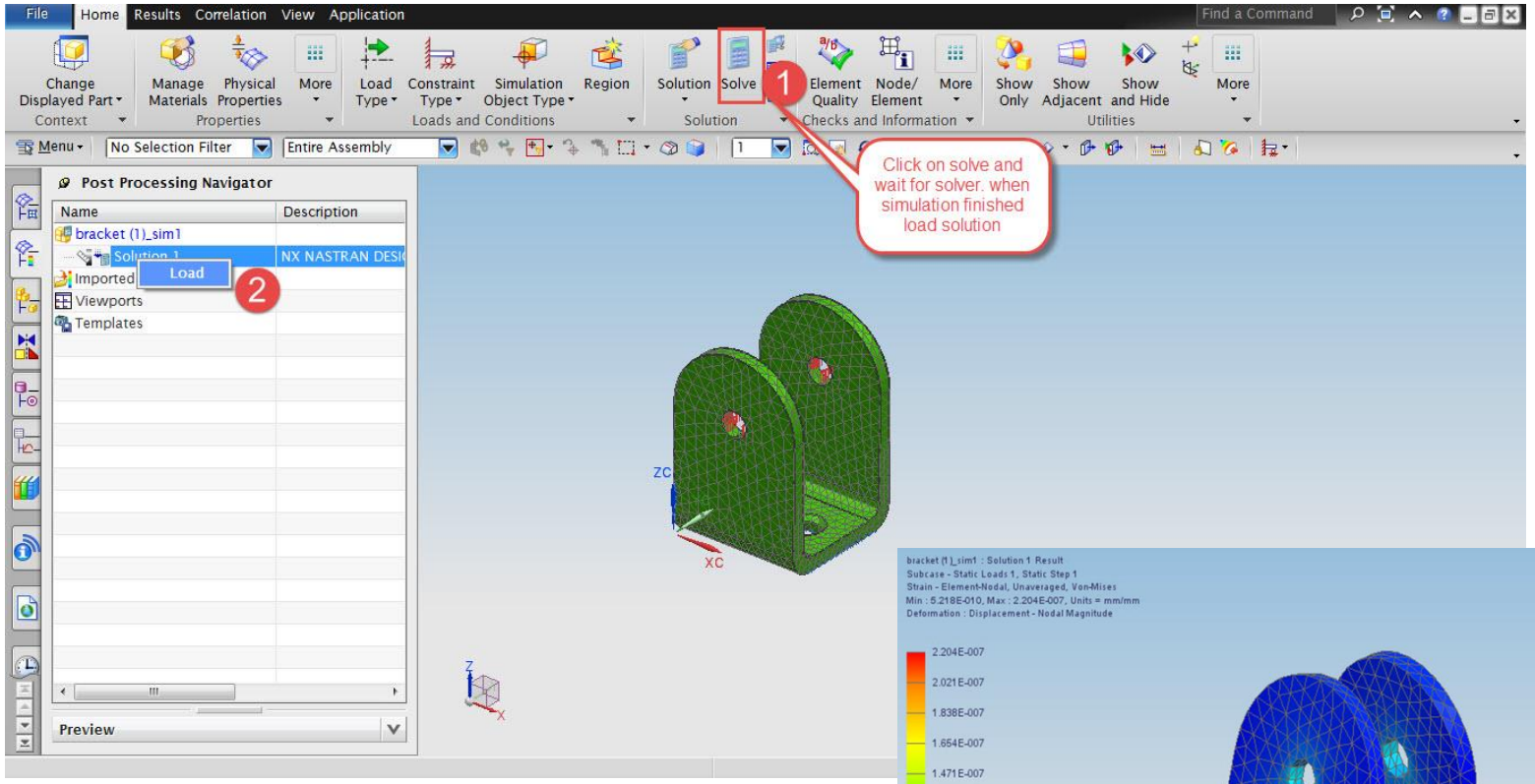
Boundary Conditions: Boundary conditions are surfaces that are fixed to arrest the degrees of freedom. Some surfaces can be rotationally fixed and some can be constrained from translational movement.

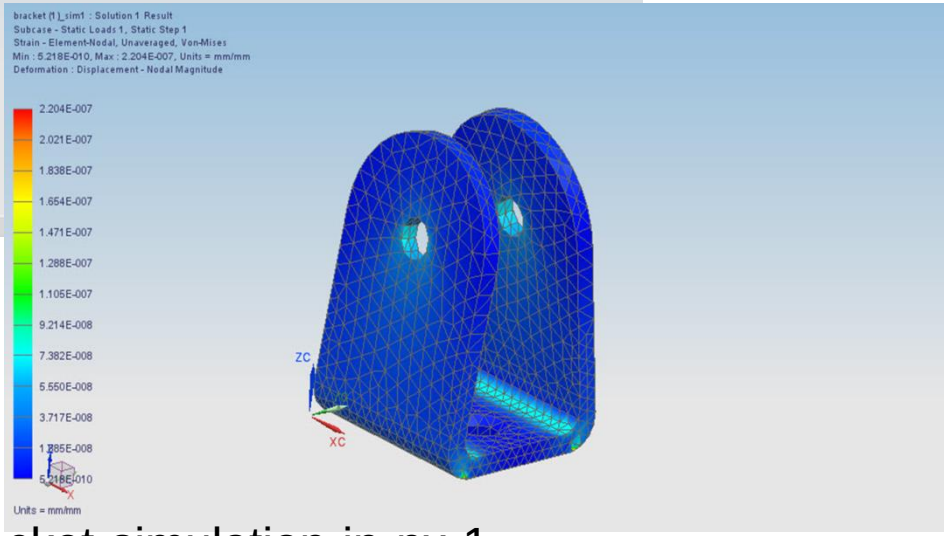
Forces:



Force: This option allows you to exert different types of forces and pressures to act on the solid along with the directions and magnitudes.

Solve and Results



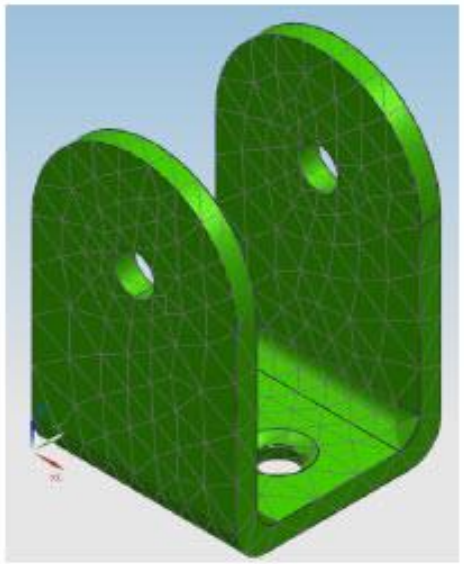


Final model:

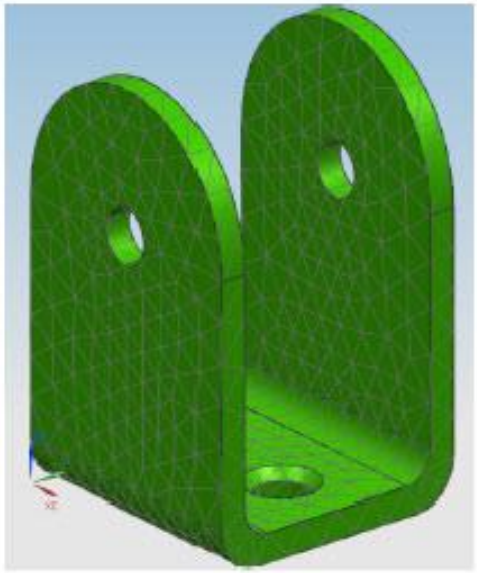
https://grabcad.com/library/tutorial-32-bracket-simulation-in-nx-1

NX/PLM- Mesh Refinement

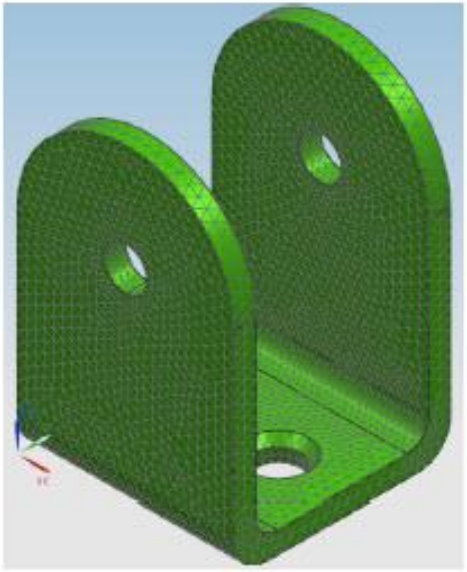
F = 10N



Element size = 20mm



Element size = 15mm



Element size = 5mm

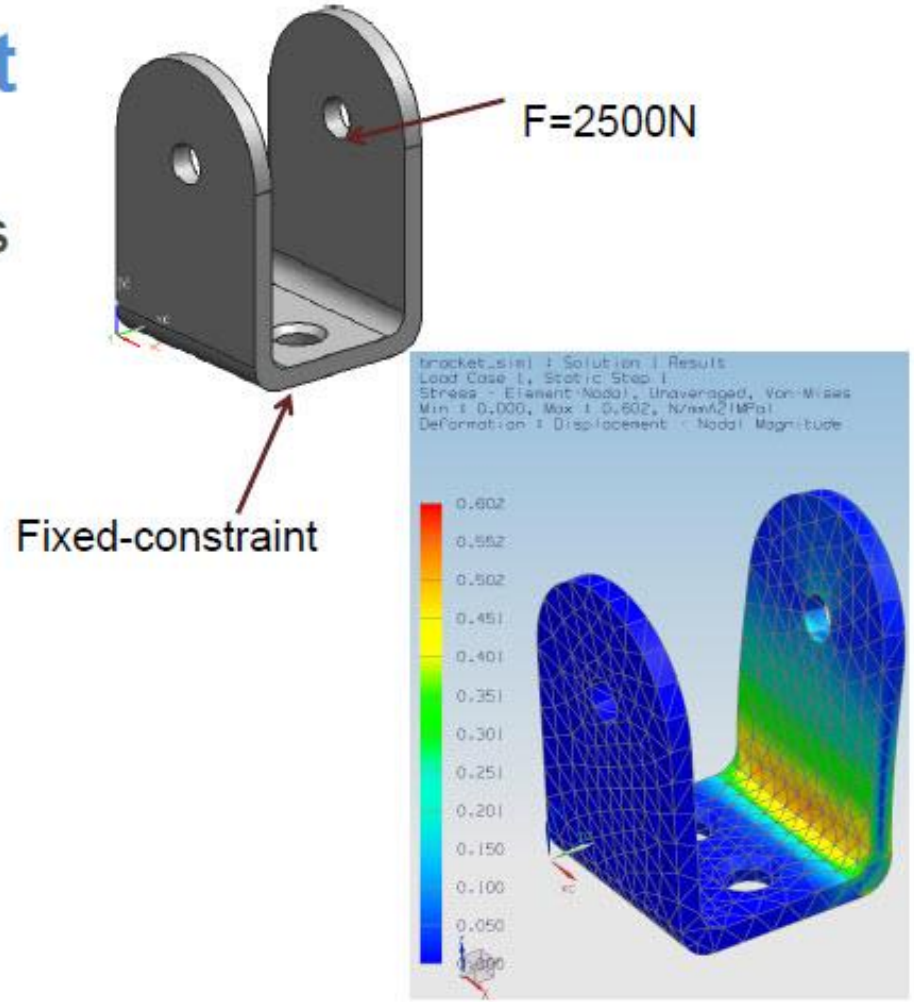
Repeat simulation for three different mesh size

Finish

Example 1:

NX/PLM-Bracket

- Material Properties
- Steel
- Yield strength = 179MPa
- Restraint
- Boundary Conditions
- Mesh
- Load (F = 2500N)
- Analysis



Results- von-Mises

Yield stress = 179MPa

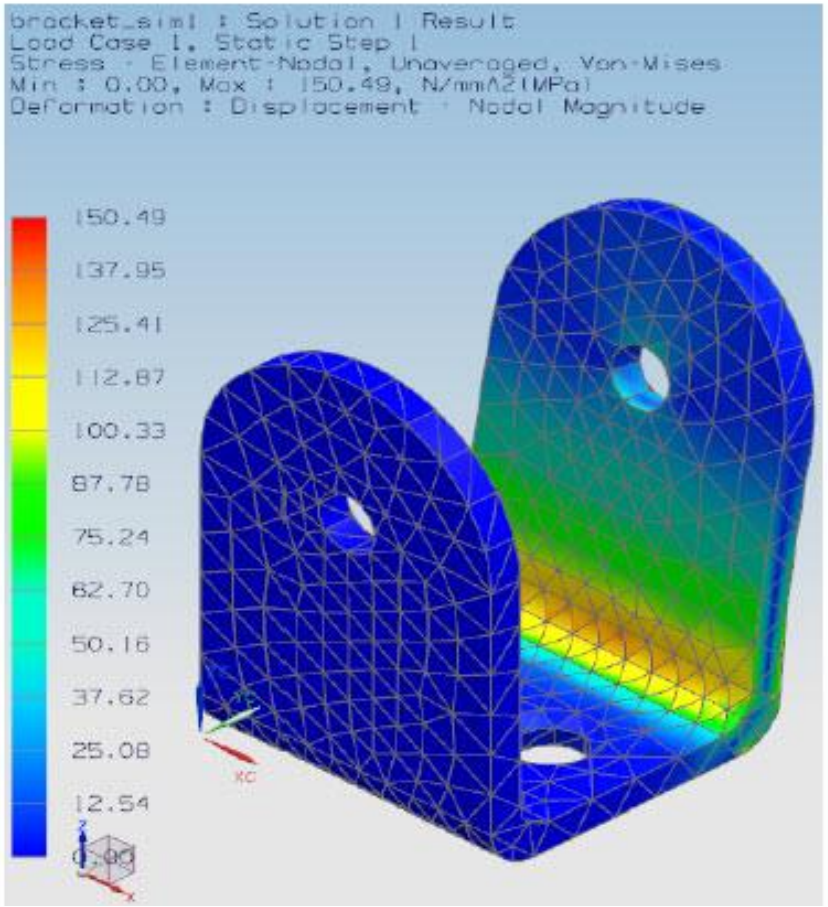
FoS = 1.18

Improvement of FoS to 2?

- Material selection
- Geometry optimisation

DO NOT FORGET:

Weight should be constant



Example 2

1-Download Bar from studyspace
2- open file in Nx
3- Mesh size 10 mm
4- Material same as pervious question make local material E=200 Gpa v=0.3 Yield strength: 215 Mpa
5- Boundary condition and force are shown in pic
7- solve