

[?](#) [Assignments](#) [External tools](#) [Forums](#) [Group choices](#) [Questionnaires](#) [Quizzes](#) [Resources](#)

Syllabus

**Started on** Thursday, 28 September 2023, 3:22 PM

**State** Finished

**Completed on** Thursday, 28 September 2023, 3:31 PM

**Time taken** 8 mins 30 secs

**Grade** 8.77 out of 10.00 (87.69%)

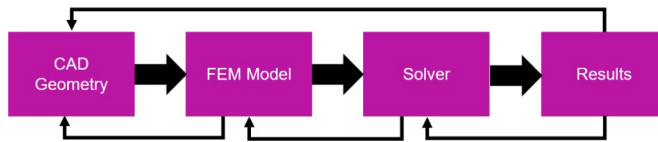
**Feedback** Excellent!



## Information

**FEM Process**

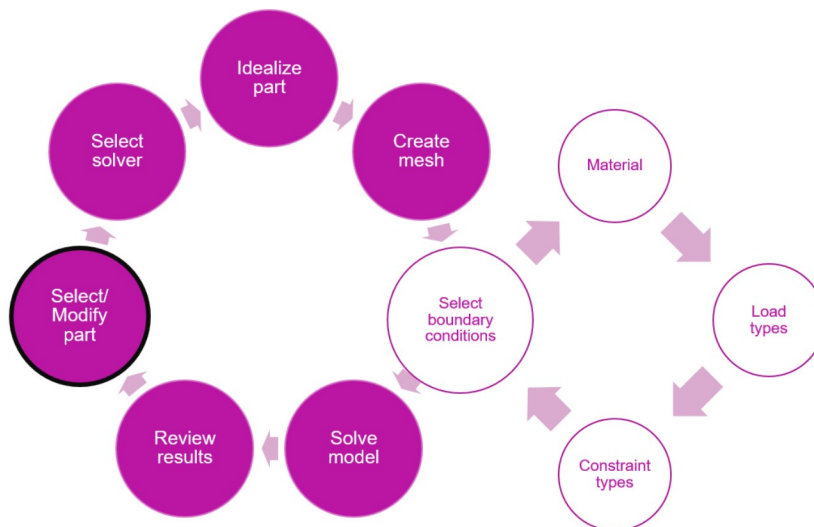
A typical FEM process in this course includes several steps, as seen in the figure below.



With a FEM tool, you can perform

- strength
- vibration
- buckling
- thermal

simulations. For a strength analysis, you need to define a geometry, material, constraints (i.e. how the geometry is attached to the ground) and loads. To attach a geometry to the simulation model, all degrees of freedom need to be fixed. Different FEM programs have somewhat similar processes to create simulation models. A process used in NX Simulate can be seen in the picture below.

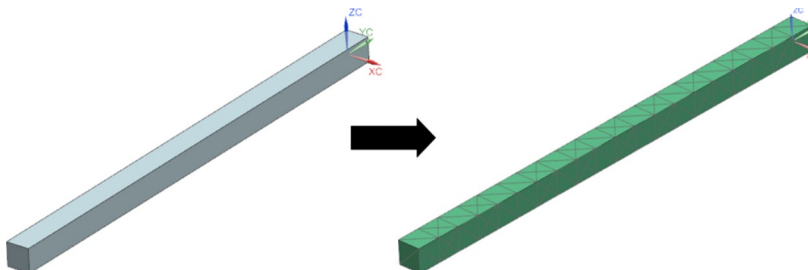
**FEM model creation**

FEM model needs at least four different inputs:

- Geometry (FEM mesh)
- Material (for the mesh)
- Constraints (how the geometry is attached)
- Loads (what forces and where).

**Geometry**

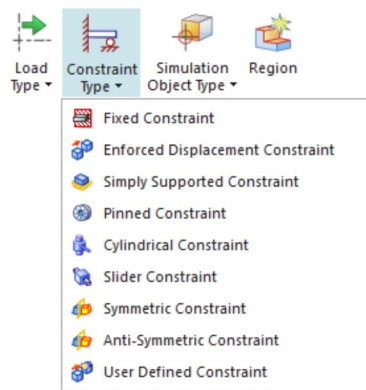
During meshing, a CAD geometry is divided into several elements (mostly tetrahedral or cubic) that are attached to each other using connection nodes (points). An example of a beam can be seen in the picture below. This mesh then works as an input to the simulation model. If a CAD geometry is updated, the mesh needs to be updated as well. A simulation model can have several meshes, but they need to be connected (node-to-node) to ensure that the simulation works.

**Material**

Material defines the behaviour of a mesh (geometry can be seen as a spring and material as a spring constraint). FEM programs offer several different material types in their material libraries.

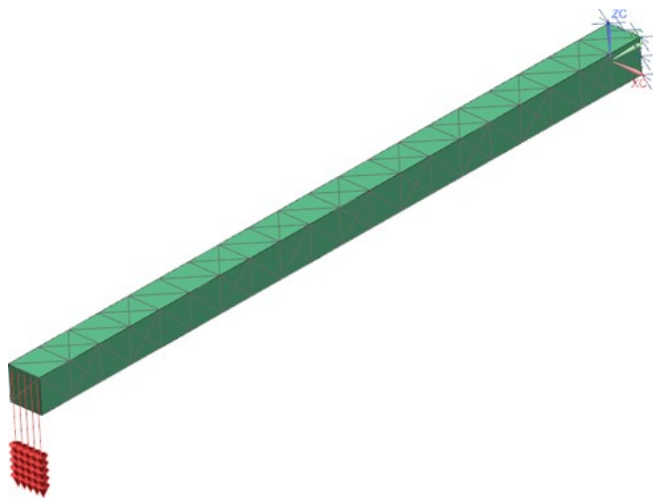
**Constraints**

Constraints are used to fix all degrees of freedom in the simulation model. If there are any DoFs in the model, the simulation can't be performed. Constraints fix the selected entities to the ground (or to each other). For example, if a back surface of a beam is constrained using the Fixed Constraint (6 DoFs fixed), that surface can't move during the simulations at all, not even if a load is applied to the same surface. FEM programs offer several types of constraints.



### Loads

FEM models need to have some inputs that force the geometry to react. Mostly this is done using forces (including gravity) and moments. If there are no forces in the strength calculation model, nothing will happen. The FEM program may remove poorly positioned forces (for example, force to a fixed constrained surface). In the picture below, a beam has a force in the left side (red arrows) and a fixed constraint (blue x-s) in the right size.



### Solutions

In NX, simulation model is a container for meshes, materials, loads and constraints. To perform a simulation, a solution is needed. Solution defines, which kind of analysis is performed to the model. In structural case using default NX Nastran Design solver, three types are provided (linear statics, vibration modes, and linear buckling).



#### Question 1

Mark 1.00 out of 1.00 Correct

**Which of the following simulations can be done using a typical FEM program?**

Select one or more:

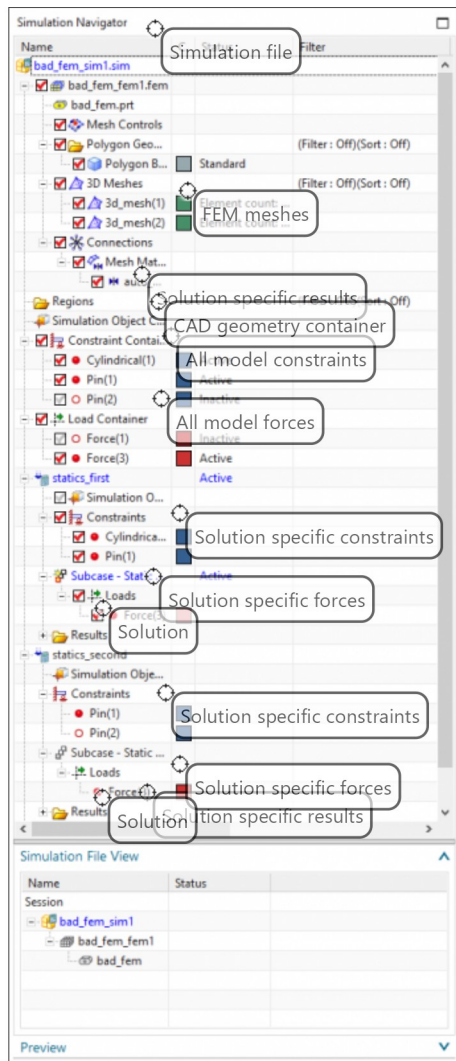
- ☒ a. Strength ✓
- ☒ b. Thermal ✓
- ☒ c. Buckling ✓
- ☐ d. Fluid flow
- ☐ e. Signal processing

Your answer is correct.

#### Question 2

Mark 0.77 out of 2.00 Partially correct

**Match items into NX Design Simulation model tree**



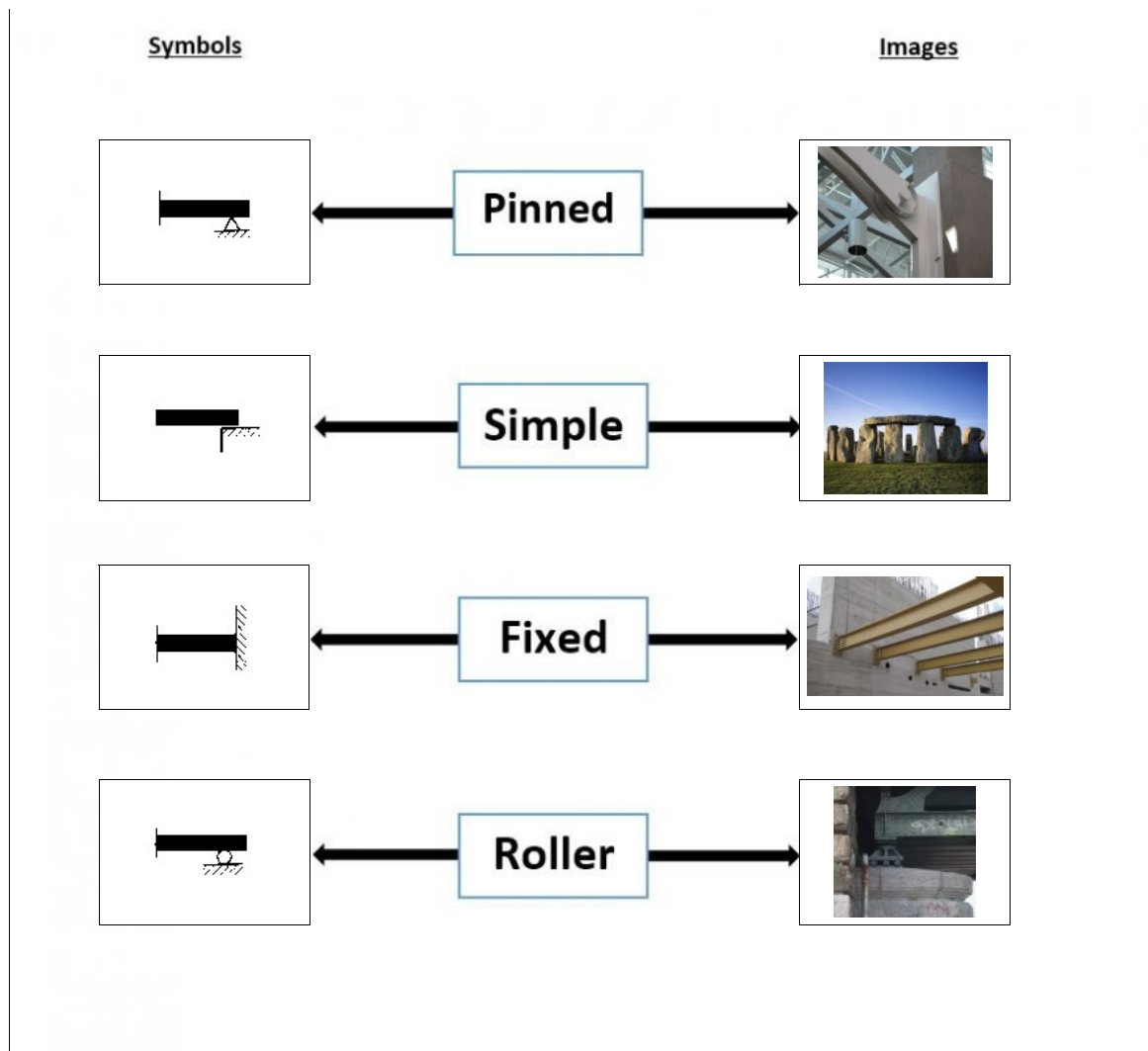
Your answer is partially correct.

You have correctly selected 5.

### Question 3

Mark 1.00 out of 1.00 Correct

Drag the equivalent constraint symbols to the left of their respective labels and the images to the right to the labels

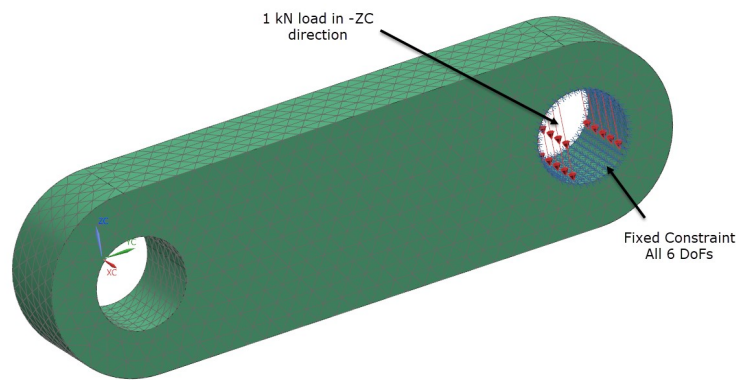


Your answer is correct.

#### Question 4

Mark 1.00 out of 1.00 Correct

Does the following FEM simulation give meaningful results?



Select one:

- ☐ a. Yes, only stresses can be calculated
- ☒ b. No, load and constraint are attached to the same surface ✓
- ☐ c. Yes, stresses and displacements can be calculated
- ☐ d. Yes, only displacements can be calculated

Your answer is correct.

#### Question 5

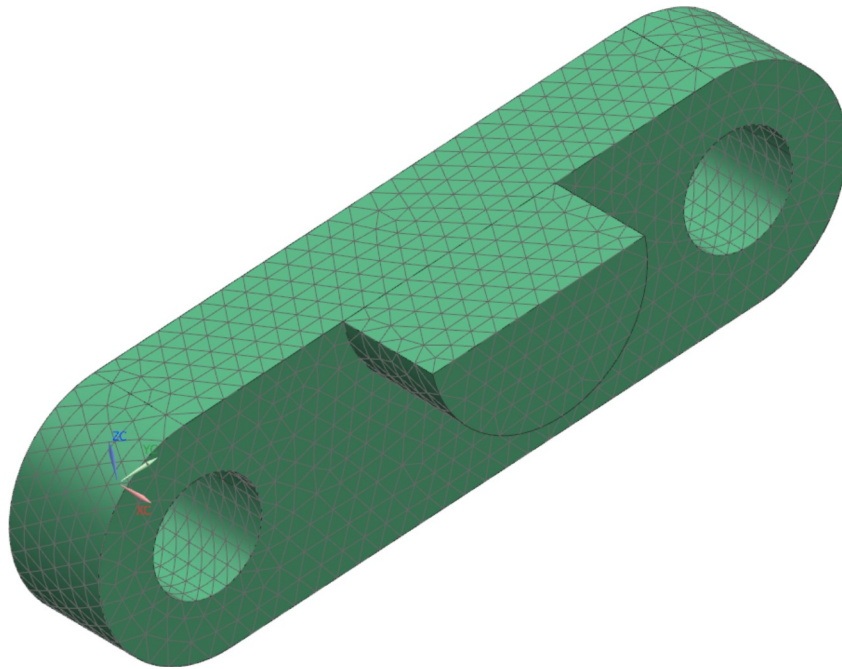
Mark 1.00 out of 1.00 Correct

Which of the meshes below is OK for simulation?

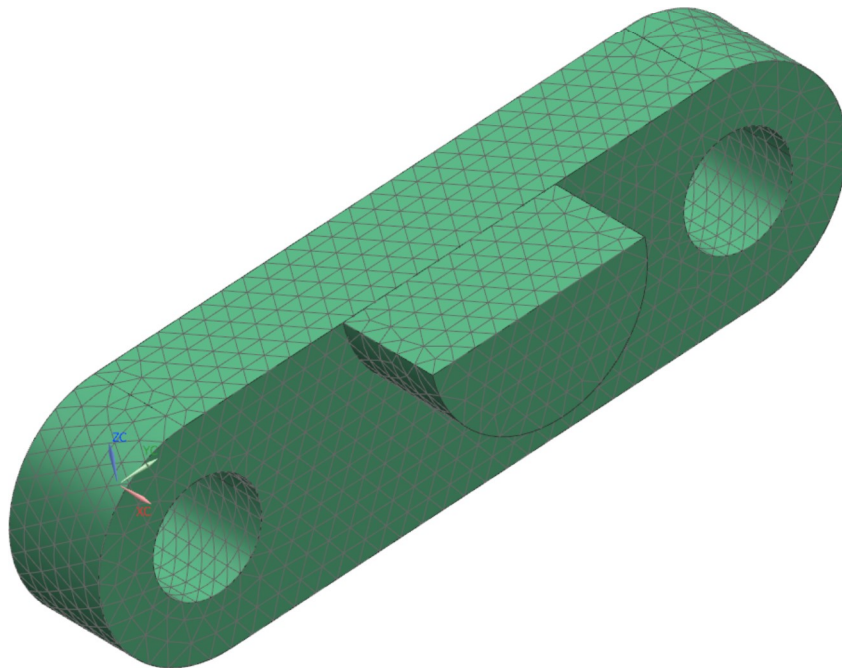


Select one:

☒ a.



☐ b.

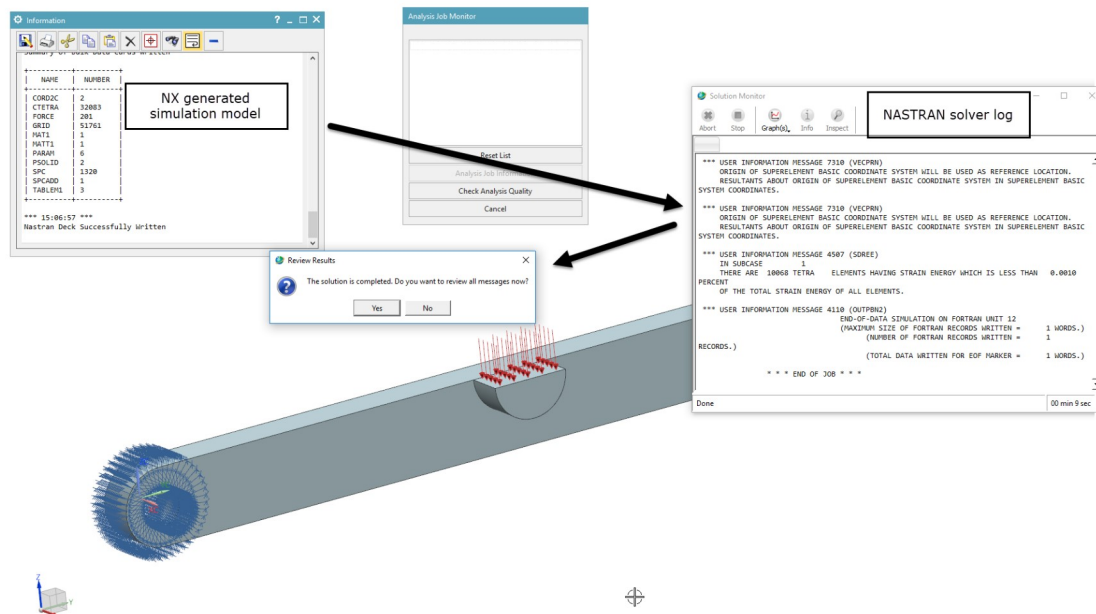


Your answer is correct.

#### Information

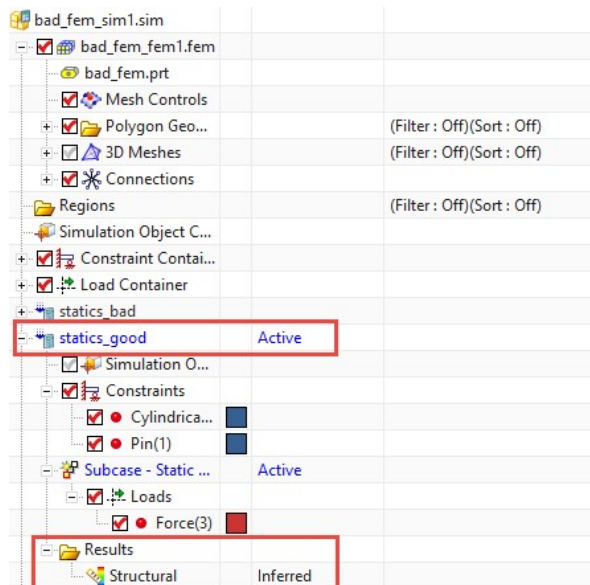
##### FEM Solving Prosess in NX

NX comes with NASTAN FEM solver. User defines simulation model in NX (mesh, loads, constraints, etc.) and a Solution. Then NX writes a simulation deck for NASTRAN (first pop-up window during simulation). NASTRAN reads this deck and solves the case. This means, that if the model has some errors/mistakes, you should check both windows' logs to find out what happened.



## Results

If the simulation was a success, you should have a Results folder under your Solution. Double-clicking it allows you to get access to Results post processing.



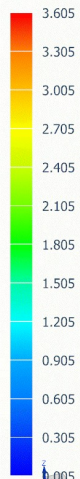
## Question 6

Mark 1.00 out of 1.00 Correct

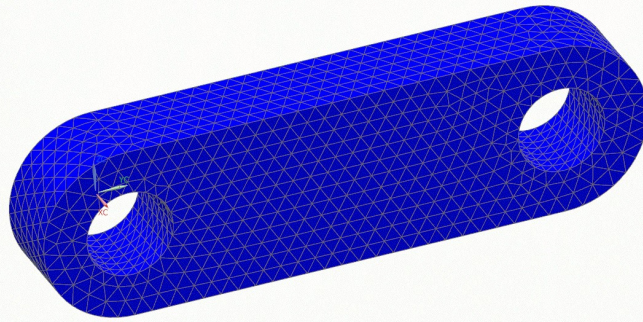
What can be seen from the simulation result below?



bad\_fem\_sim1 : statics\_good Result  
Subcase - Static Loads 1, Static Step 1  
Stress - Element-Nodal, Unaveraged, Von-Mises  
Min : 0.005, Max : 3.605, Units = N/mm<sup>2</sup>(MPa)  
Deformation : Displacement - Nodal Magnitude  
Animation Frame 1 of 8



Units = N/mm<sup>2</sup>(MPa)



Select one or more:

- ☒ a. The simulation was success ✓
- ☒ b. Maximum stress of the model ✓
- ☐ c. Maximum displacement of the model
- ☒ d. How the model moves under the loads ✓
- ☐ e. Different vibration models

Your answer is correct.

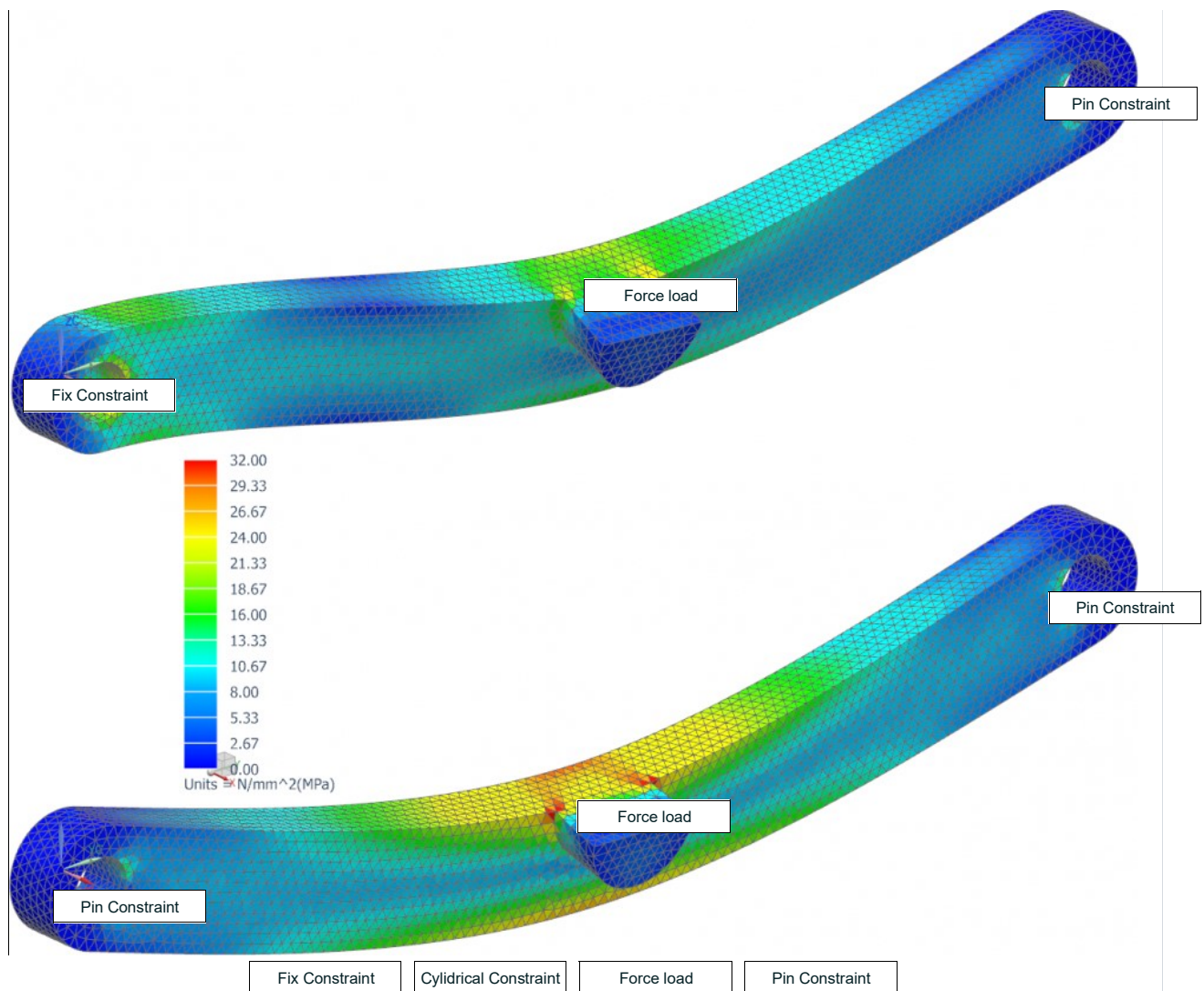
#### Question 7

Mark 2.00 out of 2.00 Correct

#### Drag the correct constraints and loads into the results pictures

Please notice, that results are presented using the same scale.





Your answer is correct.



#### Information

#### Different FEM related parts in NX

Creating a FEM simulation generates several files.

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

#### Visibility in Teamcenter

Teamcenter stores simulation files as their own objects under the same ID as the Part itself (Part can be used in multiple simulations).

- ▼ 001421.sim1
  - 001421.sim1
  - ▼ 001421.sim1/A;1
    - 001421.sim1/A
    - > 001421.sim1-A
    - View
  - ▼ 001421.fem1/A;1
    - 001421.fem1/A
    - > 001421.fem1-A
    - > 001421/A;2-FEM Beam Example
    - > 001421.i1/A;1
    - > View

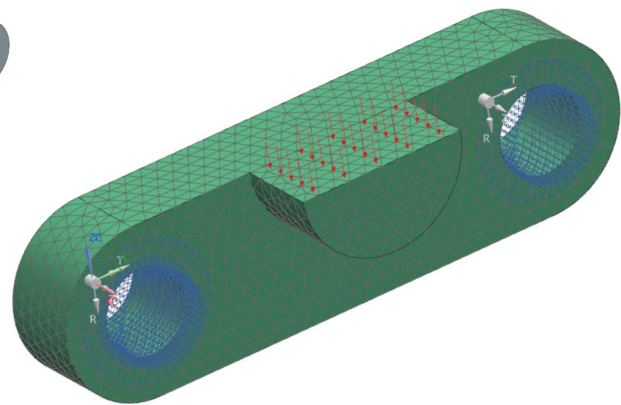
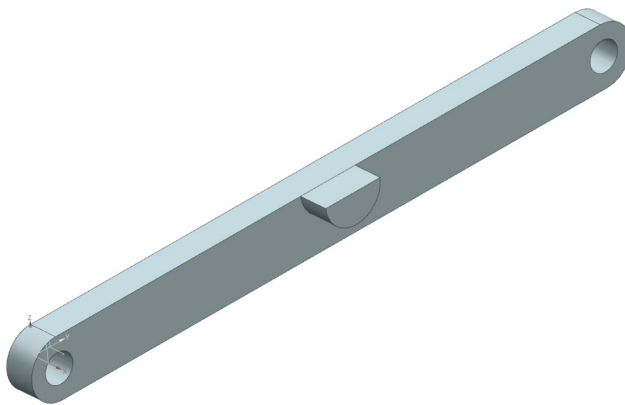
When opening a simulation model from Teamcenter, it does not load the part (it does not need it, because FEM mesh works as an input). To load a part to enable changes to it, the Load tool has to be used.

| Simulation Navigator   |    |                  |                                |
|------------------------|----|------------------|--------------------------------|
| Object                 | C. | Status           | Filter                         |
| 001421.sim1/A;1        |    | Displayed & W... |                                |
| 001421.fem1/A;1        |    | Not Loaded       |                                |
| 001421.i1/A;1          |    |                  |                                |
| Mesh Controls          |    |                  | Refresh Teamcenter Information |
| Polygon Geo...         |    |                  | Load                           |
| 3D Collectors          |    |                  |                                |
| Fields                 |    |                  | (Filter : Off)(Sort : Off)     |
| CSYS                   |    |                  | (Filter : Off)(Sort : Off)     |
| Groups                 |    |                  | (Filter : Off)(Sort : Off)     |
| Fields                 |    |                  | (Filter : Off)(Sort : Off)     |
| Modeling Objects (F... |    |                  | (Filter : On)(Sort : Off)      |
| Regions                |    |                  | (Filter : Off)(Sort : Off)     |
| Simulation Object C... |    |                  | (Filter : Off)(Sort : Off)     |
| Constraint Contai...   |    |                  | (Filter : Off)(Sort : Off)     |
| Load Container         |    |                  | (Filter : Off)(Sort : Off)     |
| Solver Sets            |    |                  | (Filter : Off)(Sort : Off)     |
| Soln 1                 |    | Active           |                                |
| Simulation O...        |    |                  | (Filter : Off)(Sort : Off)     |
| Constraints            |    |                  | (Filter : Off)(Sort : Off)     |
| Results                |    |                  |                                |

## Question 8

Mark 1.00 out of 1.00 Correct

When CAD geometry (\*.par) looks like in picture in left and simulation model (\*.sim) like picture in right, what has to be done?



Select one:

- ☒ a. Activate \*.fem file and update mesh ✓
- ☐ b. Update simulation (\*.sim) file
- ☐ c. Delete simulation (\*.sim) and create a new one

Your answer is correct.

Previous activity

◀ Status Survey Week 5

Next activity

Status Slides Week 4 25.9. ▶