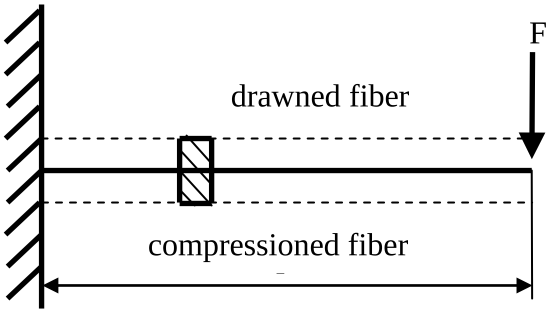
CAD-CAM-CAE example

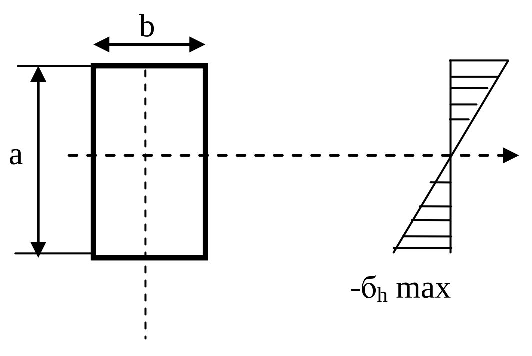
example title:	Fem analysis of a cantilever beam which is fixed at one end
example number:	ÓE-A15
example level:	**basic** - medium - advanced
CAx system:	CATIA V5
Related material part with TÁMOP	CAD, FEM
Job Description:	Build a cantilever beam model which is fixed at one end and make Finite element analysis with CATIA 's Generative Structural Analysis module

1. The task

Determine the deflection of the captured cantilever beam. The cantilever beam's material is steel. See below the main parameters.

F=1000N
a=100mm
b=80mm
l=1000mm
E=210000

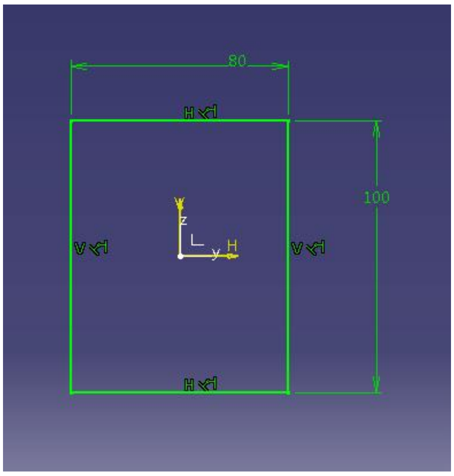




2. The solution steps

2.1. Build the model

First we build the model of cantilever beam. Draw a sketch with this dimension, and use the pad command and enter 1000 mm in the Length field.

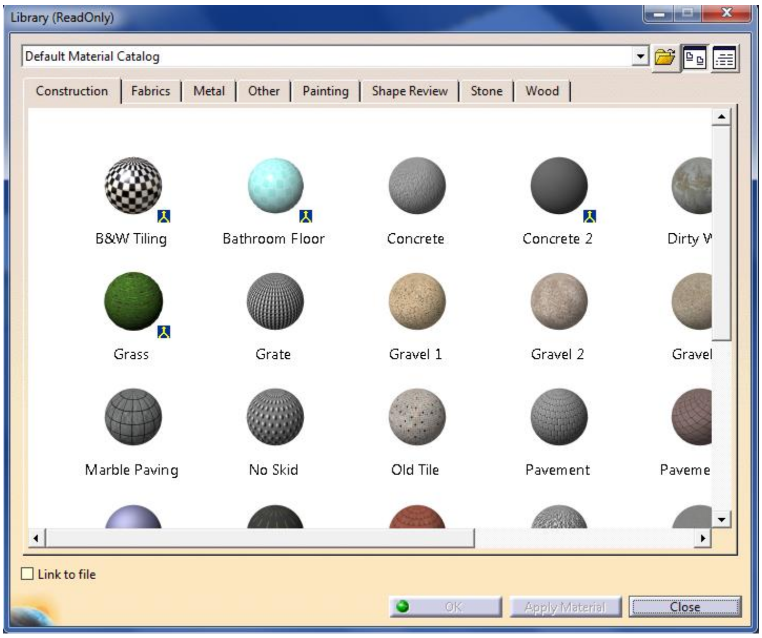


This picture show the sketch.

2.2. Material Properties

In the case no material has been previously assigned to the part, before entering the Generative Structural Analysis workbench, you should proceed as follows:

Select the part in the specification tree. Click Apply Material  icon. The Material library appears. Select a **Metal** material family, then select the desired material from the displayed list, select the **Steel** then click OK. The material is applied.

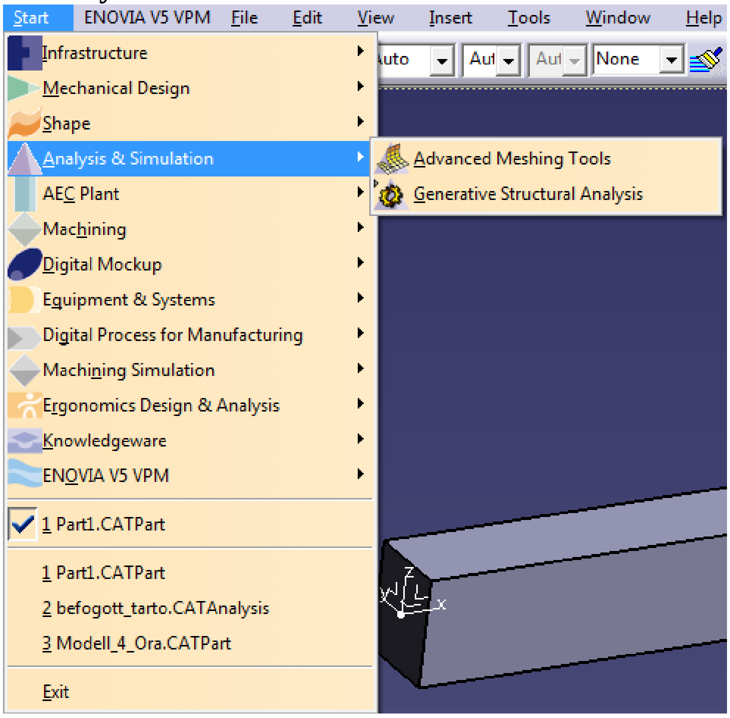


You can visualize the material properties and its analysis characteristics by selecting the material in the specification tree and using Edit > Properties > Analysis.
If you have special material enter the properties here.

3. Finite Element Analysis

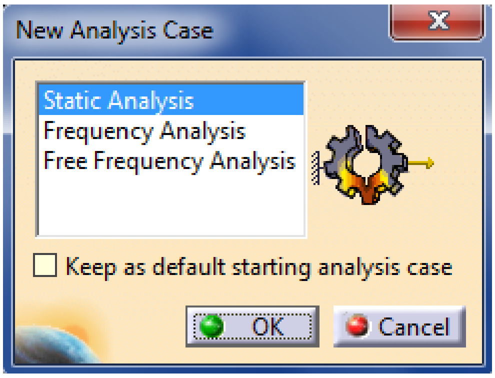
3.1. Enter Generative Structural Analysis Workbench.

Select **Start** > **Analysis & Simulation** > **Generative Structural Analysis.**

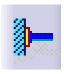


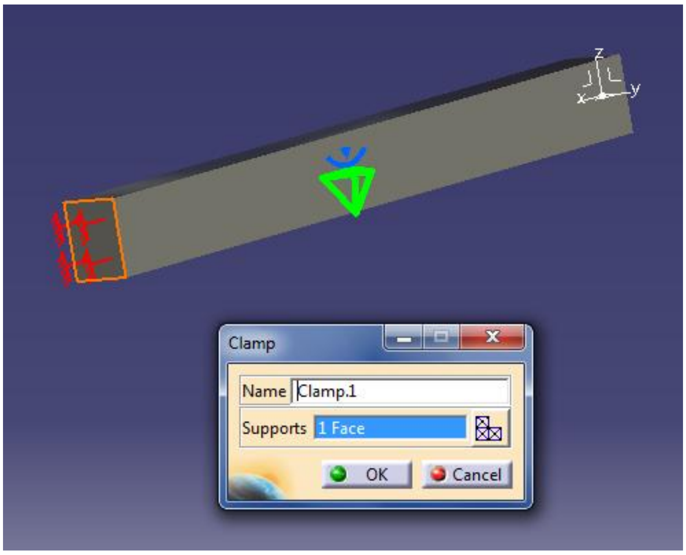
The New Analysis Case dialog box appears with Static Analysis as default option. Static Analysis means that you require a linear static computation by referencing restraints and loads. In this particular case, also keep Static Analysis type selected. Click OK. The CATAnalysis document now opens. It is named Analysis1. You will now perform different operations in this document.

A link exist is between the CATPart and the CATAnalysis document.



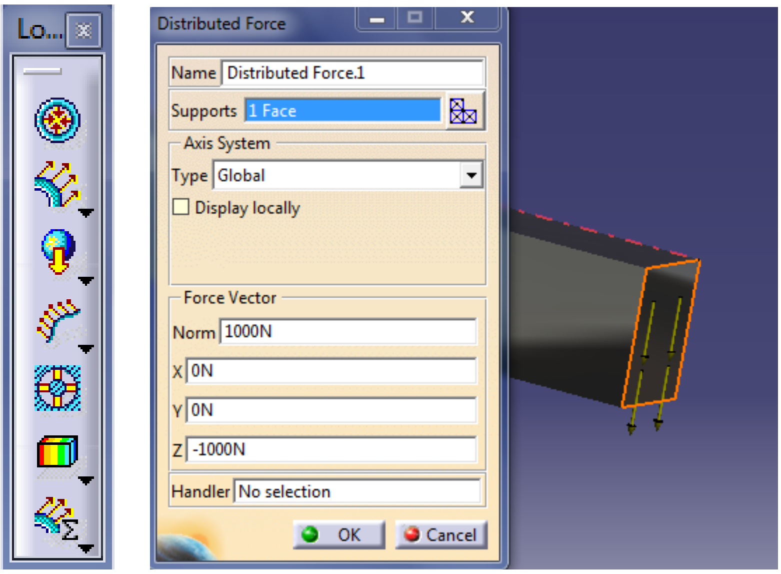
3.2. Restraint

Click **Clamp**  in the Restraints toolbar.
Clamps are restraints applied to surface or curve geometries, for which all points are to be blocked in the subsequent analysis. The Clamp dialog box appears. Select the face at the end of body.



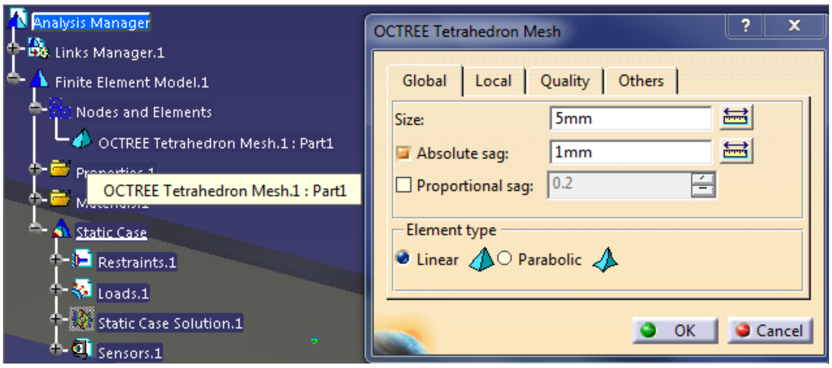
3.3. Loads

Click Distributed Force  in the Loads toolbar. The Distributed Force dialog box appears. Select the supports on which you want to apply a distributed force. Select the face at the end of body opposite clamp side. Enter the -1000 N value in the Z field of Force Vector section.

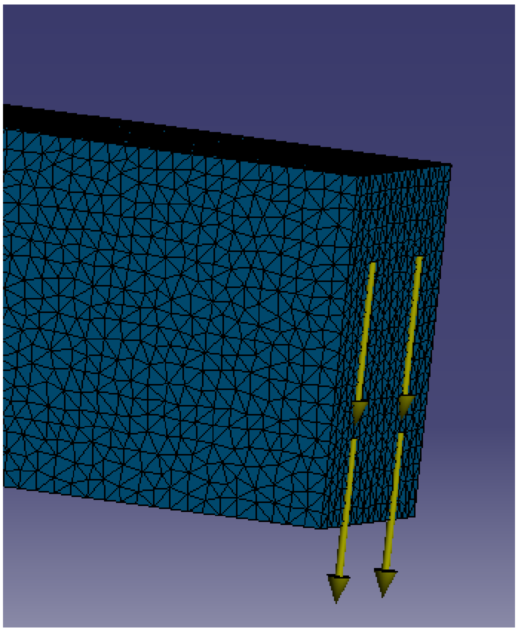


3.4. Modify of the Mesh size

Select the mesh part in the specification tree which you want to modify. Click twice on the "OCTREE Tetrahedron Mesh.1 : Part1" in the specification tree on the right side of screen. The OCTREE Tetrahedron Mesh dialog box appears.



Change global parameters in the Global tab. Enter 5mm in the Size field. Enter 1mm in the Absolute sage field. Click OK in the OCTREE Tetrahedron Mesh dialog box.

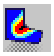


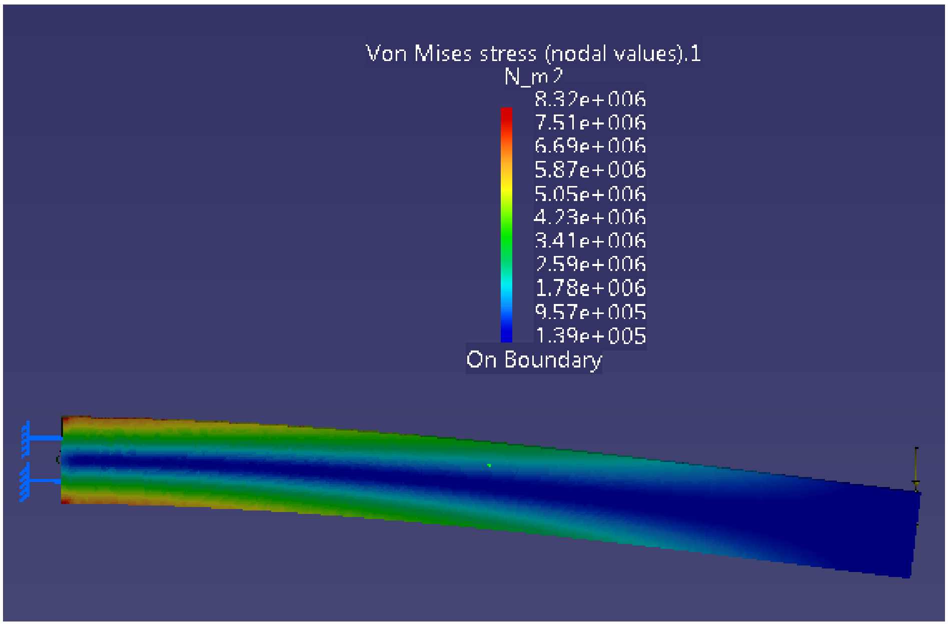
3.5. Calculation

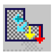
Click **Compute**  icon. Computing a mesh will enable the analysis of any object of Restraints, Loads and Masses type, without requiring the computation of a solution. The Compute dialog box appears. Select **All** from the list then click OK. Both the Computation Status dialog box and the Computing... progress bar appear. The progress bar provides a series of status messages (Meshing, Factorization, Solution) that inform you of the degree of advancement of the computation process.

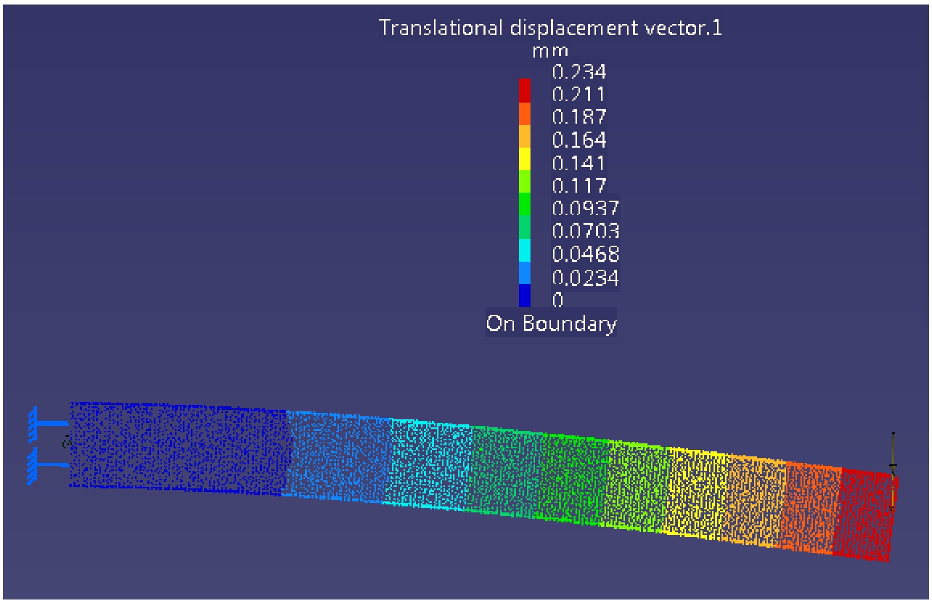
3.6 Results Visualization

Click **Deformation**  in the **Image toolbar**. The image is also visualized.

Click **Von Mises Stress**  in the Image toolbar. The Von Mises stress image is displayed, and a Von Mises Stress (nodal value).1 image appears in the specification tree under the active static case solution. The Von Mises stress distribution on the part is visualized in Iso-value mode, along with a colour palette. You can visualize the Von Mises stress image in different ways by modifying the custom view modes. To do this, select View > Render Style > Customize View. When the mouse cursor is passing over finite elements of the mesh, the values of the Von Mises stress are visualized at each of their nodes.



Click Displacement  in the Image toolbar. The translational displacement vector distribution on the part is visualized in arrow symbol mode, along with a colour palette.



A Translational displacement vector.1 image also appears in the specification tree under Static Case Solution.1. When the mouse cursor is passing over vector arrow symbols, their components with respect to the global reference frame are visualized.