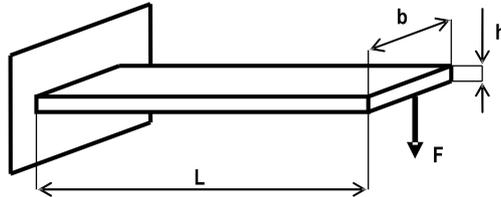


# A cantilever beam

Exercise no. 1 of the Finite Element Method I. course

## Aim of the exercise

The exercise is a practical introduction to the program ABAQUS illustrated on a simple problem – a cantilever beam loaded by a single force. A detailed step by step procedure is provided in the text below.



$L = 100$  mm,  $h = 5$  mm,  $b = 3h$ ,  $F = 1000$  N,  $E = 210000$  MPa,  $\nu = 0.3$

Determine the displacement of the free end of the beam. Consider 3D, 2D and 1D types of geometry of the beam and static loading.

---

## Solution

After starting the program ABAQUS, the start menu appears on the screen from which you select *Create model database - With Standard/Explicit Model* - Fig. 1. There is also the option to run the ABAQUS tutorials.

### 3D beam

#### The Part module

Beam geometry can be created in the module *Part* either by clicking on *Part* → *Create*, or by clicking on the *Create Part* icon. The part is called "beam\_3D" and will be modeled as **3D deformable** body. Beam geometry will be created by selecting **Solid** and **Extrusion**. Finally, you must choose a suitable approximate size of the part (**Approximate size**, Fig. 2). By pressing the *Continue* button you will be redirected to the *Sketch* environment, where you can create a profile of the beam and its 3D representation. Profile of the beam is modeled as a rectangle that is specified by two opposite corners. The coordinates of these corners are [0,0] and [15,5]. Location of the opposite corners can be supplied either by coordinates or by using the mouse click in the appropriate position on canvas Fig. 3. Modeling of the profile is ended by pressing the *Cancel procedure* button Fig. 4 and *Done* Fig. 5. Beam geometry is finished by extruding the profile to 100 mm length Fig. 6, Fig. 7.

#### The Property module

Material properties of the beam are specified in the module *Property*. The material may be called "steel" and will be modeled as a homogeneous and **isotropic elastic** material Fig. 8. This material is defined by the Young's modulus of elasticity  $E = 210000$  MPa (can be entered in the format 2.1e5) and Poisson's ratio  $\nu = 0.3$  Fig. 9. *Note: Be careful with units - if dimensions of the part are specified in [mm], then the Young's modulus must be given in [MPa] and the resulting stress fields are in [MPa]. If dimensions are entered in [m], then the Young's modulus must be entered in [Pa] and results of stress are also in [Pa]!*

Material properties are assigned to parts through sections, which are defined by the *Create Section* function. For our purpose we define a **homogeneous solid** section Fig. 10, and the previously specified properties of the steel are assigned to it Fig. 11. The created section is assigned to the geometric model of the beam. Click on *Assign section*, select the beam in the viewport and confirm by *Done* Fig. 12 and Fig. 13. If assigned properly, geometry of the part changes its color to green Fig. 14.

#### The Assembly module

Although the model consists of only a single part, you need to create **instance** of this component in the **Assembly** module by using *Instance Part*. You may select between **Dependent and Independent Instance Type** Fig. 15.

This setting will have effect on the mesh creation process in the Mesh module. Mesh of the dependent instances is a copy of mesh, which was created on part. Conversely, mesh of the independent instances may be unique for every single instance derived from the same part.

### The Step module

Time partitioning and physics of the analysis is set in the module Step by clicking on the *Create step* and by defining the analysis steps. ABAQUS creates the **Initial step** by default and it cannot be changed. We assume the **static** Fig. 16 (we are not interested in the inertia forces) and **nonlinear** analysis (effect of the large deformations will be considered – *Nlgeom: On* Fig.17). We need to define a single analysis step since the beam is loaded only by a single force from zero to max. value. The size of the initial increment (increment - a portion of step) may be 0.05, the minimum size is 1e-5, and the maximum size of the increment is 0.2 Fig. 18.

### The Load module

Loads are set in the module Load. At first we need to divide geometry of the beam on two cells along the longitudinal axis. Cutting plane may be created by using *Partition cell* and by selecting three points (*3Points* button) Fig. 19, that we select on the edges of the beam and confirm by *Create Partition* Fig. 20. When accomplished, terminate this function by the *Done* button Fig. 21. The beam will be loaded (set by using the *Create Load* function) by a single force (*Concentrated Force* option) Fig. 22, which is applied on the edge of the free end of the beam Fig. 23. The magnitude of this force is  $F = 1000$  N and acts in the negative Y direction (the coordinate axes are marked with numbers in ABAQUS - XYZ  $\rightarrow$  123) Fig. 24 and Fig. 25.

*Note: The external forces and other kinds of loads can only be entered in the user defined steps (i.e. Step-1 which we have defined previously). In the Initial step you can enter only kinematic boundary conditions and interactions.*

The kinematic boundary conditions are set also in the Load module. The beam is constrained (by selecting *Create Boundary Condition*) Fig. 26 on the opposite side from the force application point (you need to hold the Shift key while selecting both surfaces, where the boundary condition will be applied Fig. 27). Zero displacement condition in all three directions must be prescribed Fig. 28.

*Note: Although encastre constrains all 6 degrees of freedom (three displacements and three rotations) in 3D case, remove just the three displacements - the reason is that the mesh will be formed by 8-node spatial continuum elements C3D8 (C - continuum (solid) element, 3D - space, 8 - 8 nodes), which have only three degrees of freedom - displacements). The applied boundary conditions are displayed as shown in Fig. 29.*

### The Mesh module

To mesh the model you need to set the element size at first. We choose *Seed Part* instance, set the global size of elements to 1 mm Fig. 30 and confirm by *Done* Fig. 31. Then select the cells you want to mesh, click on *Assign Mesh Controls* Fig. 32 and select the method of meshing (in our case it will be **structured hexagonal grid**) Fig. 33. The next step is to determine the type of the elements - linear, 8-node continuum elements C3D8 (select geometry you want to mesh and choose *Assign Element Type*) Fig. 34. The last step is to mesh the model by *Mesh Part Instance* Fig. 35 and Fig. 36.

### The Job module

Now the model of the beam is ready to start the calculation. Parameters of the calculation are specified in the module Job, select *Create Job* and name the job as "beam\_3D" Fig. 37. From the available settings (*Edit Job* window) select *Full Analysis* Fig. 38. When everything is set, we can click on the *Job Manager* and start the analysis by *Submit* Fig. 40. Progress of the analysis may be monitored in the log Fig. 41. When the analysis is complete, the results may be viewed by clicking on *Results* in the *Job Manager* window Obr. 42. ABAQUS creates a set of files in the current work directory (check *File*  $\rightarrow$  *Set Work Directory...* in the main menu to see where it is) during the analysis run. Names of the files are derived from the name of the job. The most important is *jobName.odb*, which stores results of the analysis.

### The Visualization module

How to determine the magnitude of deflection at the free end of the beam is shown in Fig 43.

To be able to continue with creating the 2D beam in the same model database, rename the existing model to "beam\_3D" Fig. 44 and Fig. 45. Then we create a new 2D model as is shown in Obr. 46 and Fig.47.

## 2D beam

*Note: In the following text the procedure of the model creation will not be described in such detail as in the case of the 3D beam. Only the settings that are specific to the 2D beam are mentioned.*

### The Part module

The beam will be modeled as a 2D planar and deformable body. Beam geometry will be created by a surface (shell) Fig.1. In the *Sketch* environment we will create a rectangle with two opposite corners of the coordinates [0,0] and [100,5] Fig. 2. Resulting geometry is in Fig. 3.

### The Property module

First, we define the material - steel (prescribe the same behavior and parameter values as in the previous case). Then we need to create a homogeneous solid section Fig. 4 and assign the material properties of the steel to it. The thickness of the beam is 15 mm Fig. 5. Finally, assign the created section to geometry of the model.

### The Assembly module

Create the assembly the same way as in the 3D beam case.

### The Step module

Use the same settings as previously - define a single static step, assume large displacements.

### The Load module

Define the concentrated force on the free end of the beam. No modifications of geometry are needed. Apply the boundary conditions on the edge, which is located on the opposite side to the application point of the force. Displacements in direction 1 and 2 must be constrained Fig. 6.

### The Mesh module

At first, we set the global element size to 1 mm (use the *Seed part* method). Then set the mesh controls - quadrilateral elements, structured mesh. The next step is to determine the type of the elements assuming the plane stress conditions - continuum planar linear elements CPS4 (C - continuum (solid) element, PS - plane stress, 4 - 4 nodes) Fig. 7.

## 1D beam

### The Part module

The 1D beam will be modeled as a 3D deformable body. Geometry will be created by a single line (wire). As with 3D beam, when all settings in the *Create Part* dialogue are done, click the *Continue* button to get to the *Sketch* environment. Create the line by two points [0,0] and [100,0].

### The Property module

First, we define the material - steel. Then we create a new beam profile (*Create Profile*, select a rectangular profile and call it "beam") Fig. 4. Here, it is very important to specify the profile of the beam in the correct orientation Fig. 5! The orientation of the beam is defined by its tangent and direction of axis 1 unit vector. Select the beam, choose *Assign Beam Orientation* and determine the orientation of the tangent Fig. 6 and n1 vector Fig. 7. The result of the orientation assignment can be seen in Fig. 8. Then create a section representing material and cross-sectional properties of the beam. Choose the *Category Beam* and *Beam Type* Fig. 9. Specify the beam profile, material, and Poisson's ratio Fig. 10. Subsequently we assign this section to the geometrical model by using the *Assign Section* function.

### The Assembly module

Create the assembly the same way as in the previous cases.

### The Step module

Use the same settings as previously - define a single static, nonlinear step.

### The Load module

Define to concentrated force at the free end of the beam.

Select a node on the very end of the beam to define the kinematic boundary conditions. We need to constrain all six degrees of freedom because 3D beam elements have all of them active [Fig. 11](#).

### **The Mesh module**

First we set the global size of elements to 1 mm. The next step is to choose the type of the elements - linear, 2-node beam elements B31 using the Assign Element Type [Fig. 12](#).

### **The Job module**

Define a new job and submit it for analysis.