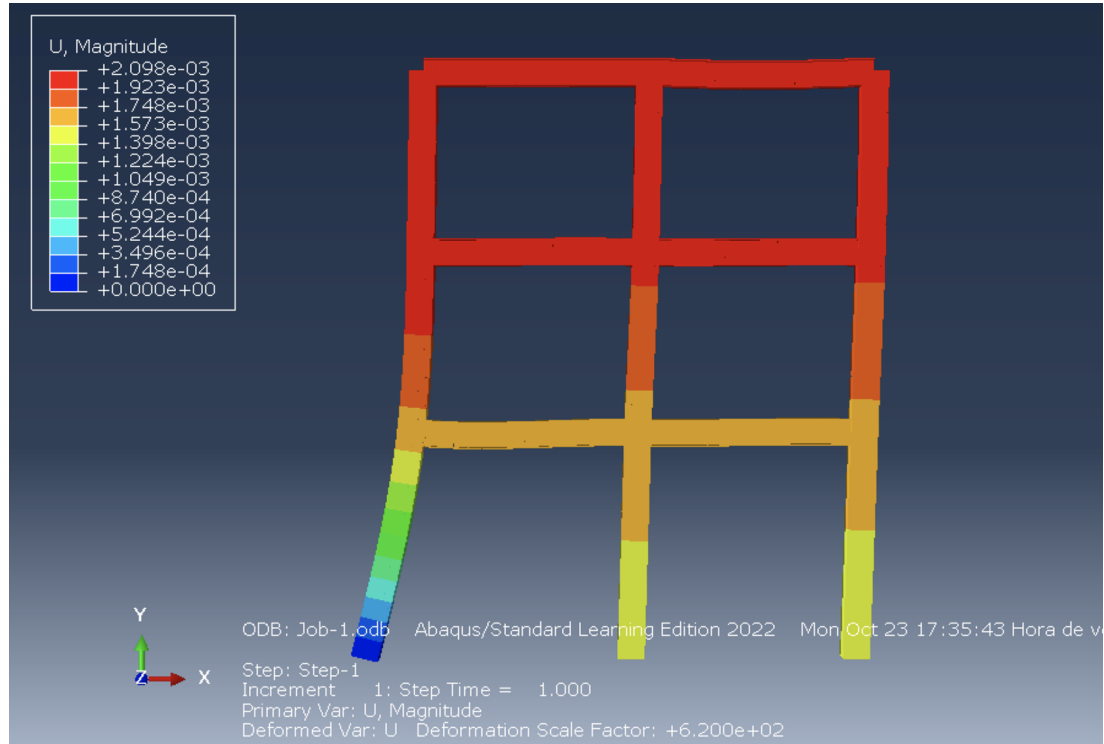


# Finite-element analysis and model calibration

## Session #4, Practice 5.

### Structural elements: beams



GRUPO DE MECÁNICA COMPUTACIONAL, ETSICCP, UPM

October 26th, 2023

## Índice

<b>1. Proposed exercise</b>	<b>2</b>
<b>2. Abaqus Analysis: Model with continuum elements</b>	<b>3</b>
2.1. Part Module	3
2.2. Property module	4
2.3. Assembly module	9
2.4. Step module	9
2.5. Load module	11
2.6. Mesh module	13
2.7. Job module	14
<b>3. Results</b>	<b>15</b>

# 1. Proposed exercise

A 2D portal frame is considered, with the dimensions in meters shown in the attached figure. The material of this portal frame is linear elastic, with mechanical properties of  $E = 2.1 \cdot 10^{11}$  Pa,  $\nu = 0.3$  and  $\rho = 2500$  kg/m<sup>3</sup>. In addition to the self-weight of the portal frame, 4 additional loads will be considered as shown in the figure; two point loads of value 500 N applied at the centers of the second floor beams, another uniformly distributed load of value 1000 N/m on the left side of the first floor and finally, a triangular distributed load of equation  $y = 0.1x$  and maximum value  $-20 \cdot 10^3$  N/m on the roof of the structure. The supports of the portal frame columns on the ground are shown in the same way in the first figure.

The columns have a square cross-section of  $60 \times 60$  cm and the horizontal beams have an I-type cross-section (“I” in Abaqus) with the dimensions shown in the 2.

The model will be made with Timoshenko (B21) linear beam type elements and will be discretized with an approximate element size of 0.3 meters. A 2-D Finite Element model of the structure under the actions of the loads described in the statement will be developed.

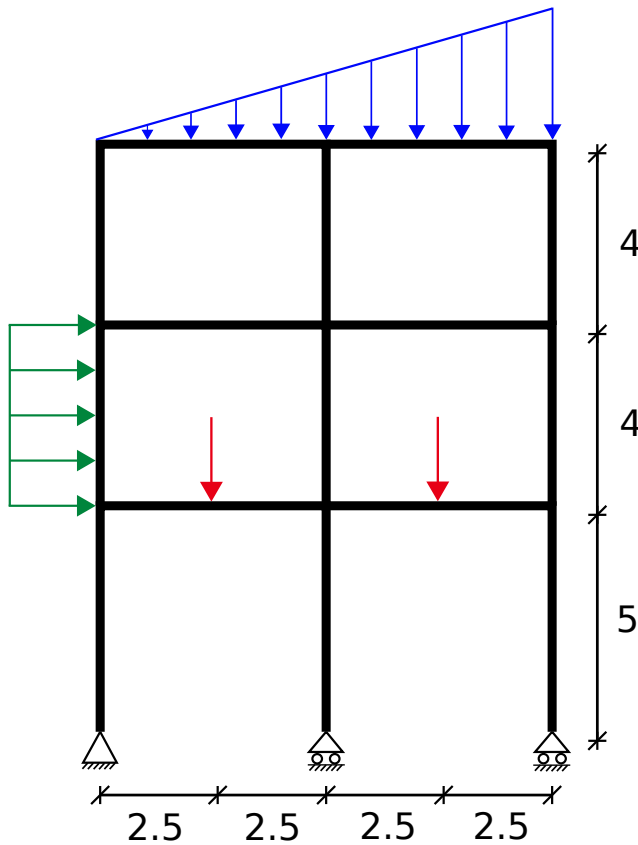


Figura 1: Sketch of the portal frame

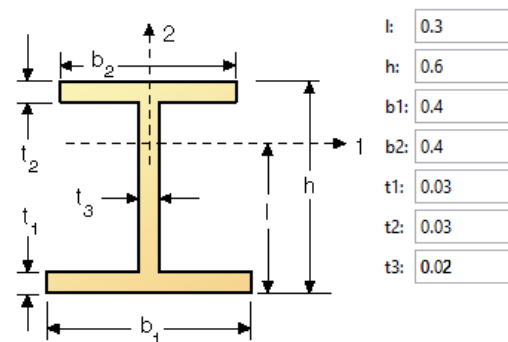


Figura 2: Profile of the I-beam

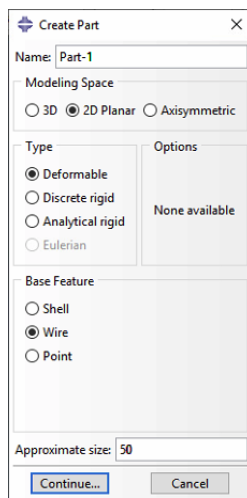
## 2. Abaqus Analysis: Model with continuum elements

### 2.1. Part Module

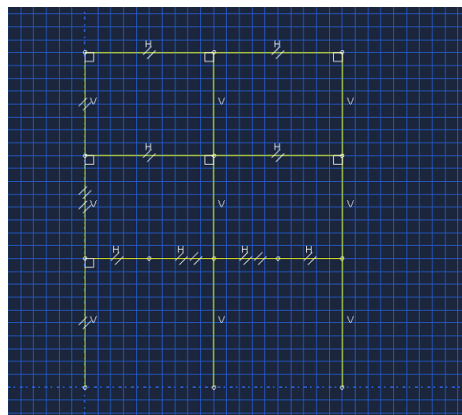
Start by executing *Abaqus CAE* to create a new model. Enter the **part** module, click the icon for creating a new part, which will be defined as 2D, deformable and wire. (Fig. 3(a)).

We can choose any help of the CAD tool of Abaqus. Write the coordinates or re-dimension a draft are the quickest ways. Remember to leave 2 points, in the first floor, to load the portal frame with point loads. The final sketch is seen in Fig. 3(b)).

Once the portal is created it is accepted with the button “Done”. The result obtained is shown in Fig. 4.



(a) Create new part



(b) Sketch of the geometry

Figure 3: Creation of the geometry

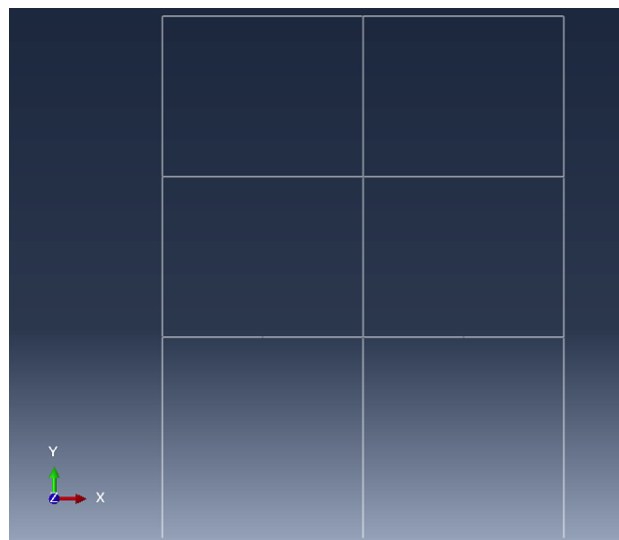


Figure 4: Geometry generated by the sketch of the portal frame

## 2.2. Property module

Select the icon for creating a new material (Fig. 5(a)), choose a linear elastic material and type in the properties (Fig. 5(b)).

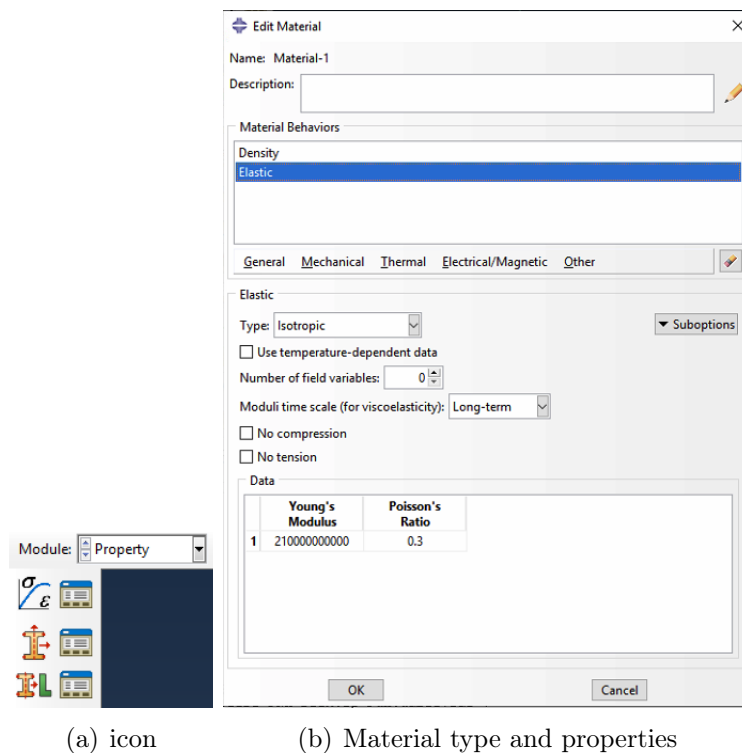


Figura 5: Create new material

Following a “*beam section*” is created (inside the category *Beam*), whose properties must be defined. In this case, since this is a small strain calculation (click on *Before Analysis*), we will not use the assigned material, typing the parameters instead (see Fig. 6.c for the column and 6.d for the beam). It is important to remark that we are typing Young’s modulus, Shear modulus (can be calculated from Young and Poisson’s parameters) and Poisson’s ratio. The density is not needed if we are not calculating a dynamic analysis.

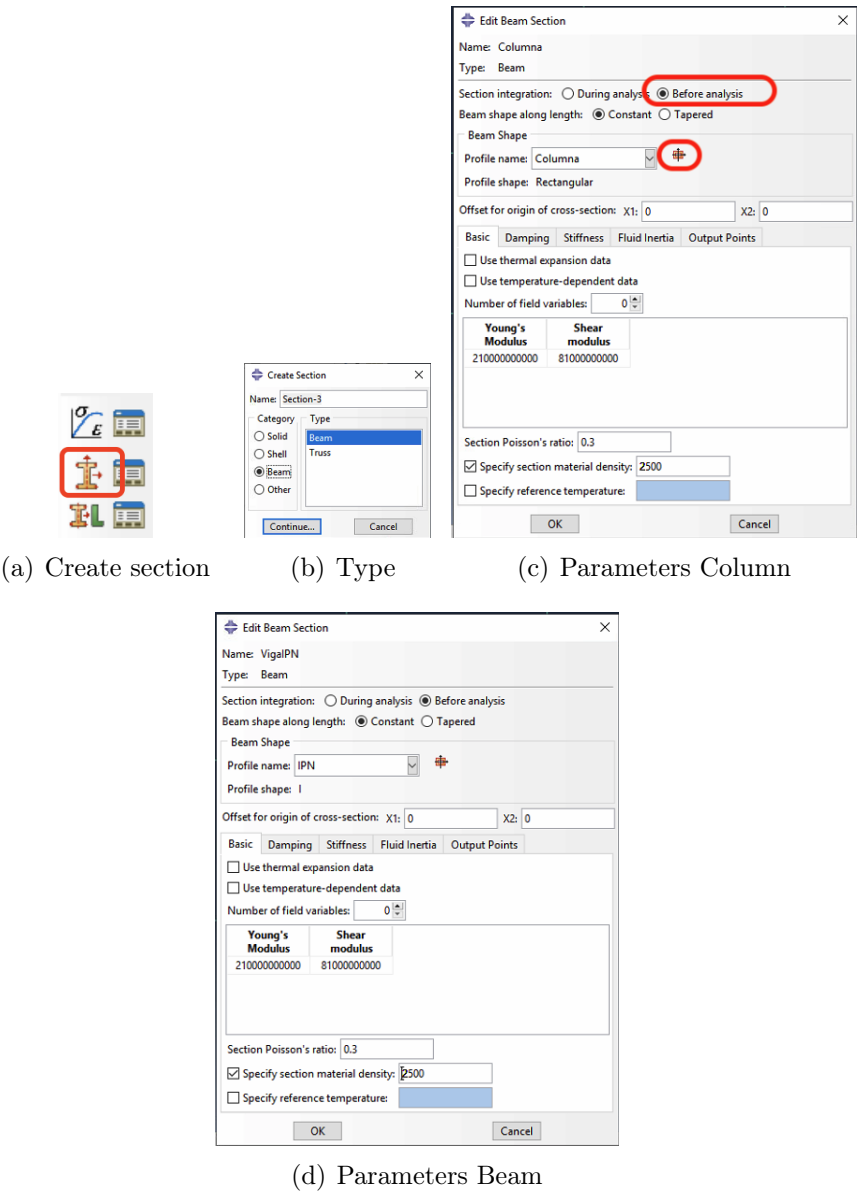


Figure 6: Creation of the section

Finally, we need to define a Profile of our beam. In both column and beam sections, we have to use different profiles. In Figs. 7 and 8 both column and beam profiles are created.

Following, we need to assign these sections to the appropriate wire elements. In Fig. 9.a we can see which are the elements where column section is assigned, as well as in Fig. 9.b.

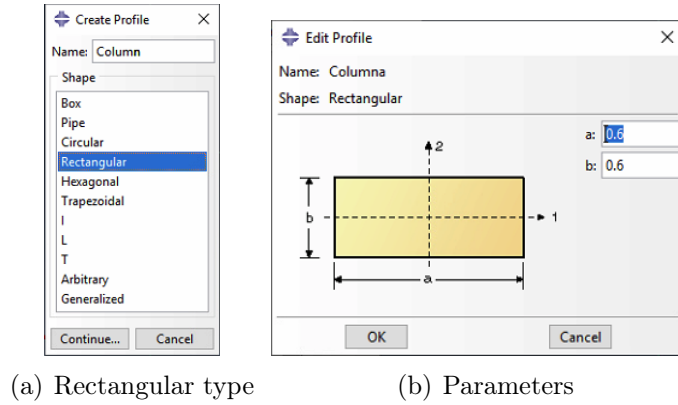


Figure 7: Creation of the column profile

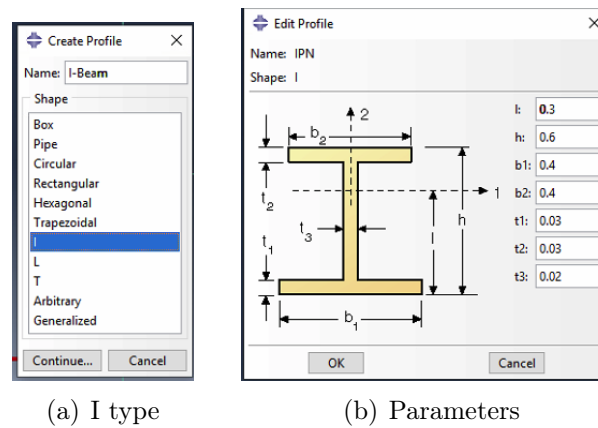


Figure 8: Creation of the beam profile

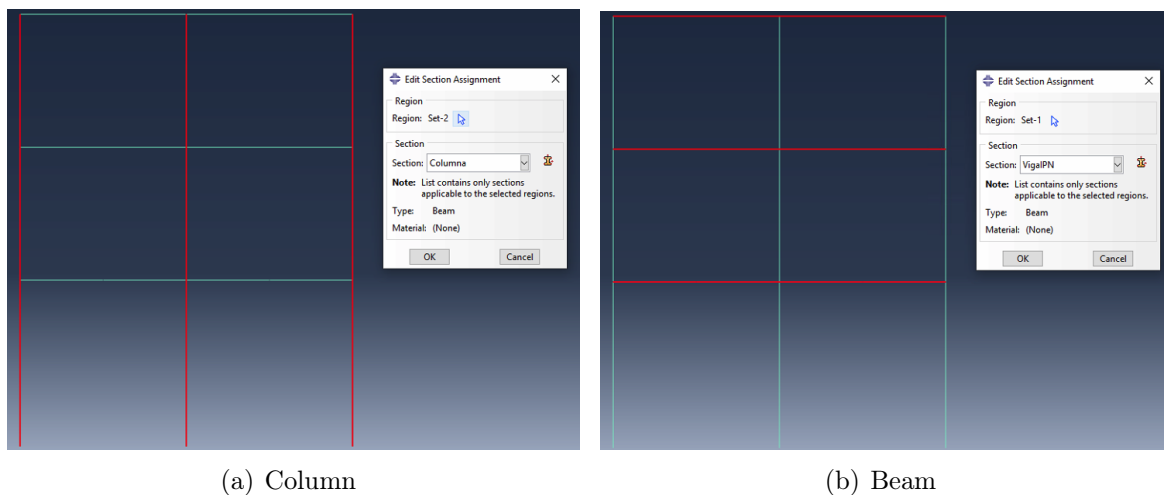


Figure 9: Assignments of both section types

In the definition of all the components of the beam finite element, it is important to define the orientation of the line elements, since the profiles must be oriented properly. Abaqus uses 3 axes to define the profile,  $t$  for the tangent and 1 and 2 for the perpendicular. We can see them in Fig. 10.

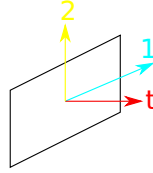
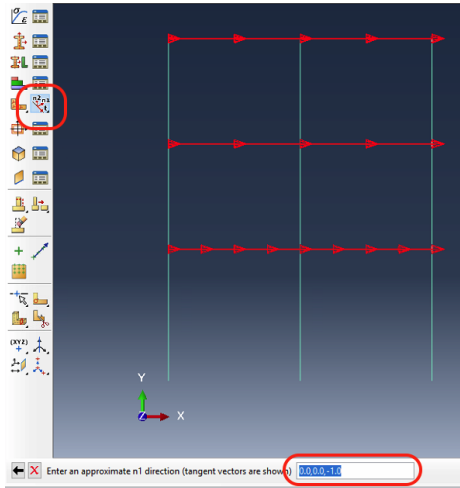


Figura 10: Direction of the axes of the beam

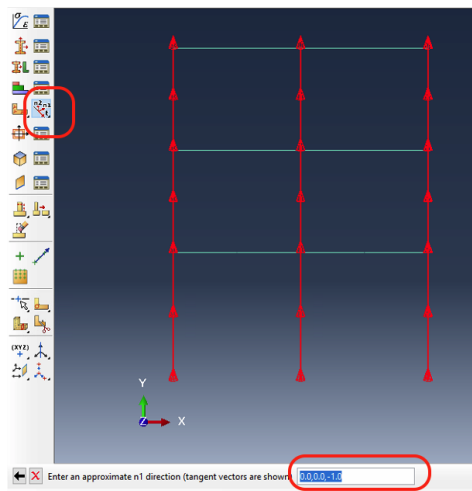
Once we click on *Assign Beam Orientation*, first we have to select the beams (or a set created before with only beams). According to Fig. 10, the vector of 1 that Abaqus requires is  $(0,0,-1)$ , see Fig. 11.a. The results are seen in Fig. 11.b. For the column the procedure is similar, seeing the process and the results in Figs. 11.c and 11.d. respectively.



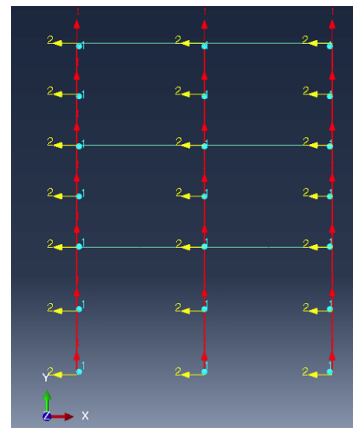
(a) Beam assignemnt



(b) Beam result



(c) Column assignemnt



(d) Column result

Figura 11: Assignments of orientations

Finally, to check if the orientation was correct, select *View/Part Display Options* and click on *Render Beam Profiles*. The results are seen in Fig. 12.

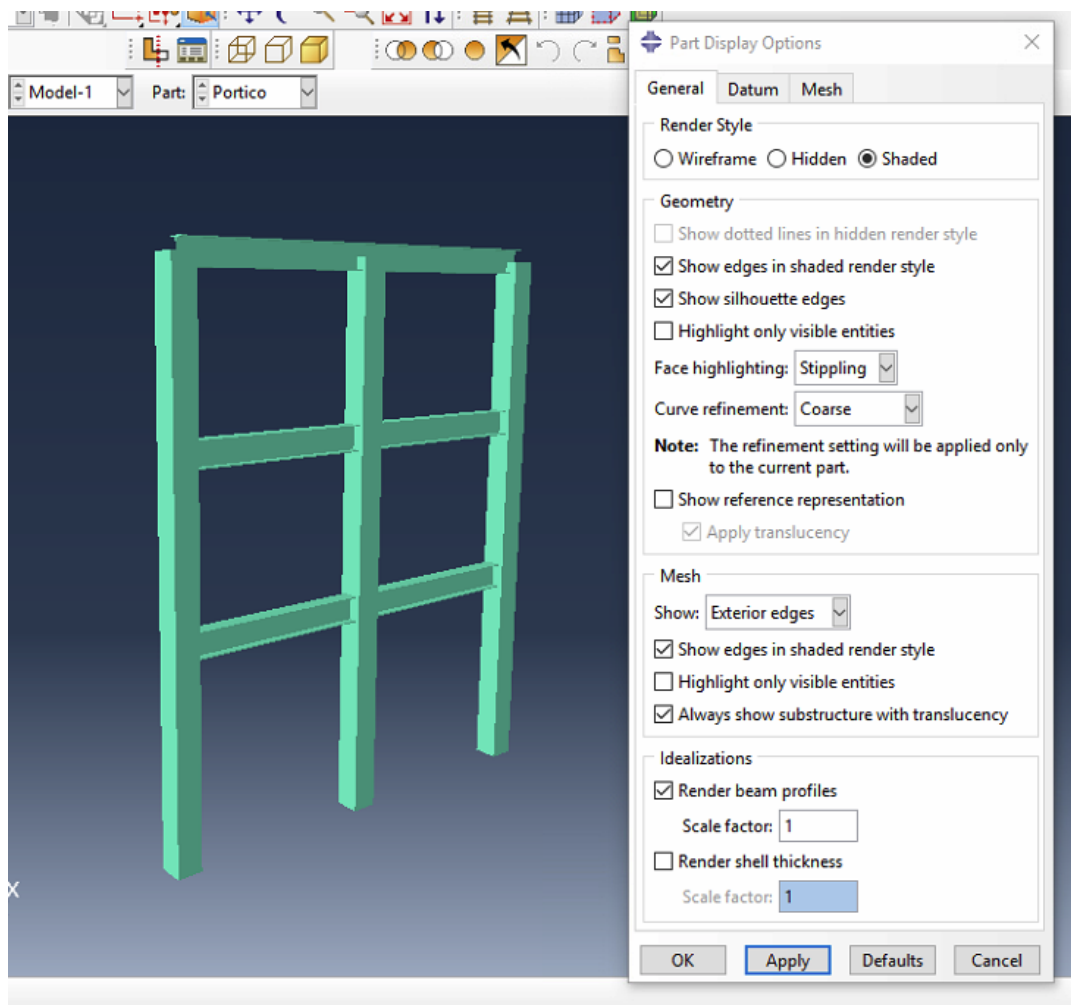


Figura 12: Render beam profiles



## 2.3. Assembly module

In this module one has only to create an “instance” from the part, accepting the default options:

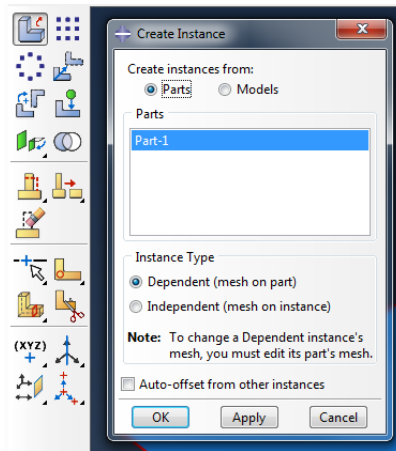


Figura 13: Assembly: create an instance of the part

## 2.4. Step module

Now create a “step” with the procedure type “Static, general”. Take the default options (Fig. 14).

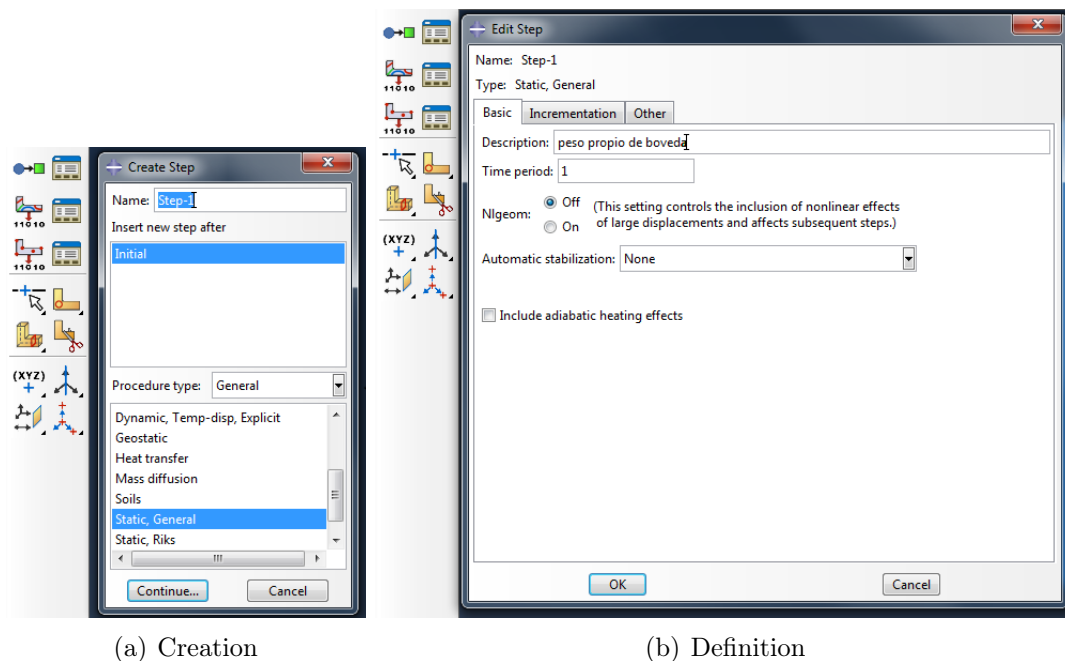


Figura 14: Create and define the “step”

In the Field Output we can check which are the variables that are included. To do so, click on the manager of field outputs, and edit, and include verify the fields to show (Fig. 15).

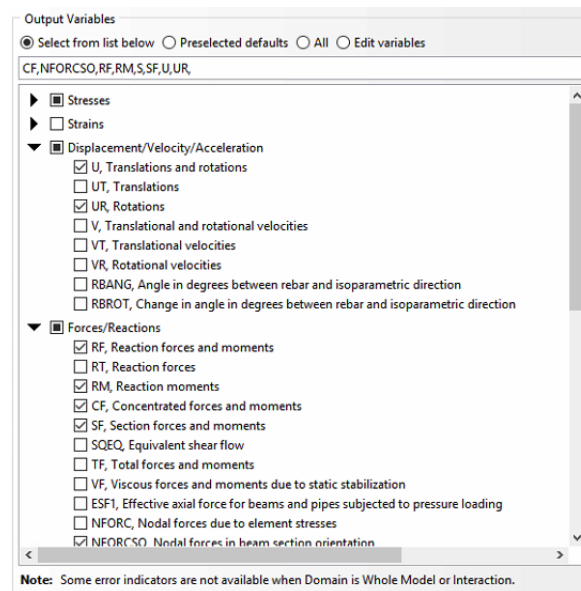


Figure 15: Variables in “Field output”

## 2.5. Load module

In the **Load** module both the applied loads as well as the boundary conditions will be defined. First, the three loads are defined, corresponding with “*Concentrated force*” and “*Line load*”. The concentrated loads are defined in Fig. 16. Secondly, a lateral load is defined in Fig. 17. Finally, the non-constant line load definition is depicted in Fig. 18. It is important to remark that we have to create an *Analytical Field* with an equation  $0.1 \cdot X$  and assign to distribution.

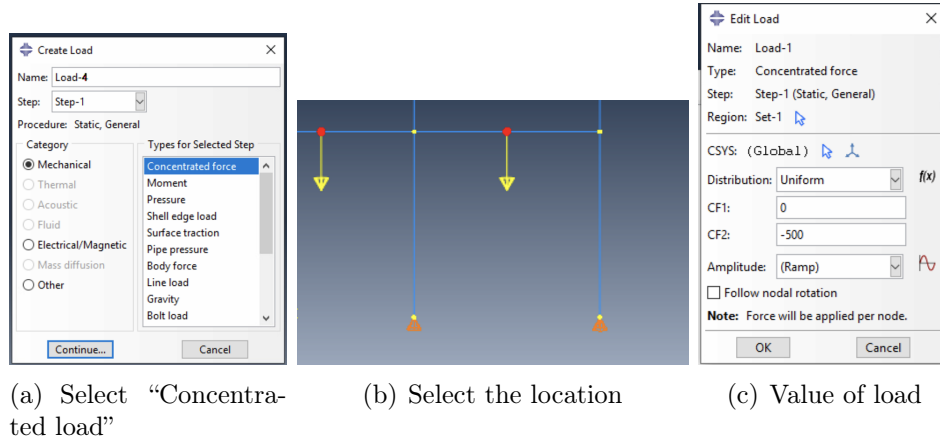


Figure 16: Definition of concentrated load

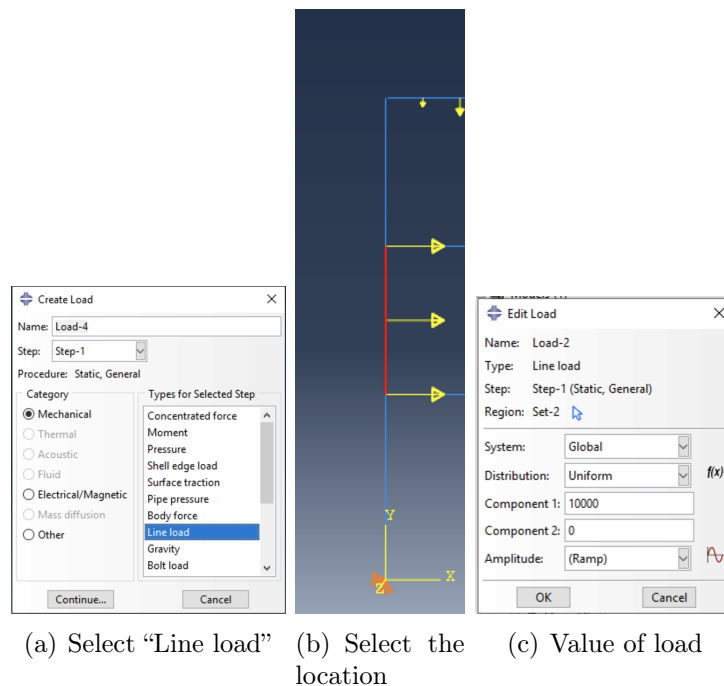


Figure 17: Definition of constant line load

Next the boundary conditions are defined. First, open the “*manager*” for boundary conditions and create the appropriate displacement condition on the left support, whose  $X$  and  $Y$  displacements are neglected (Fig. 19.a). Following, neglect the vertical displacement on the other 2 supports (Fig. 19.b).

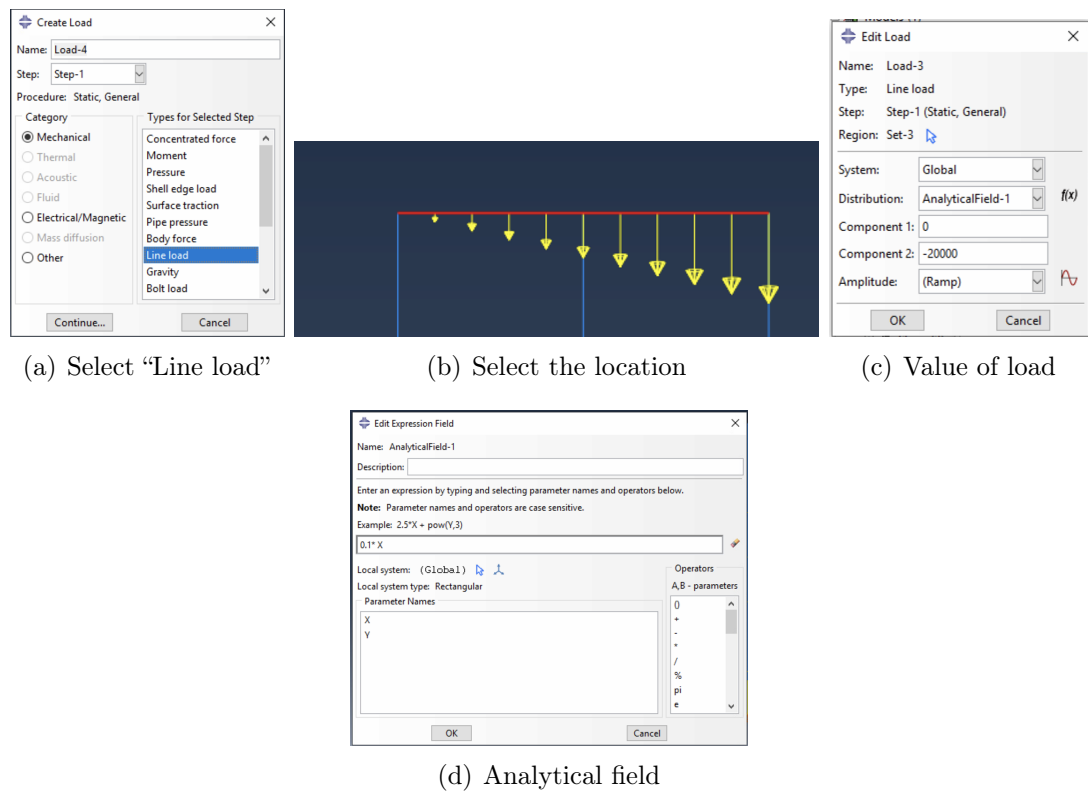


Figure 18: Definition of non-constant line load

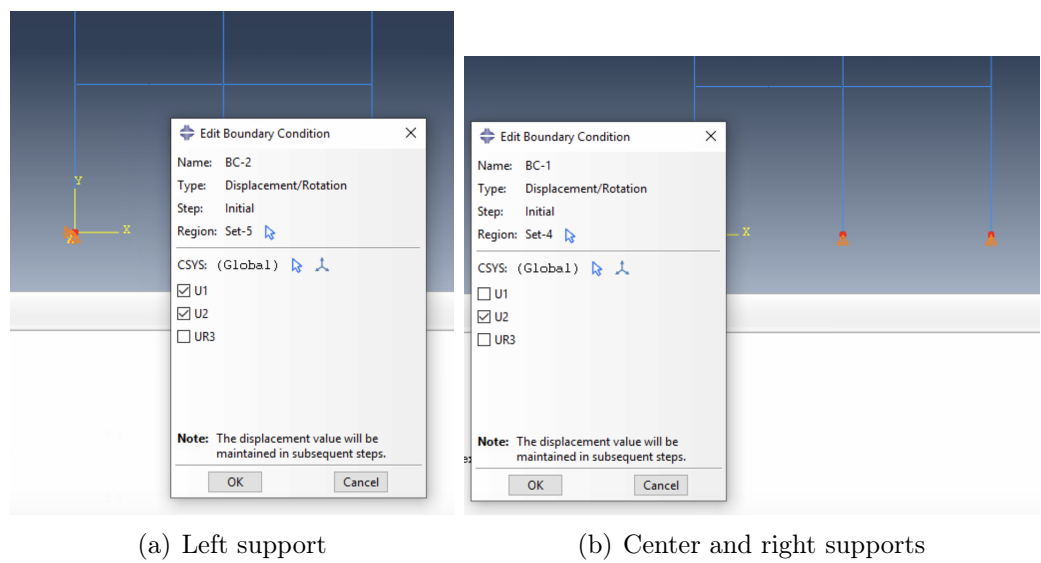


Figure 19: Boundary conditions

See Fig. 20 in order to see all the boundary conditions and loads of the problem.

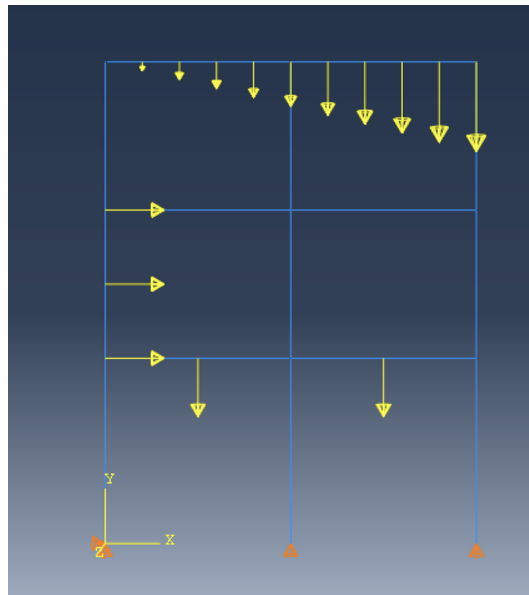


Figura 20: Sketch of BC and loads.

## 2.6. Mesh module

Within the **Mesh** module, start by selecting the part by expanding the tree of the model at the left side, and activating with the right button the icon “*Mesh*”. There are no “Mesh/controls” for wire elements. For the meshing, just select the seeds that the system suggests as default options, figure 21. For the element type we are going to evaluate the model with the Timoshenko element, *B21*, Fig. 22,

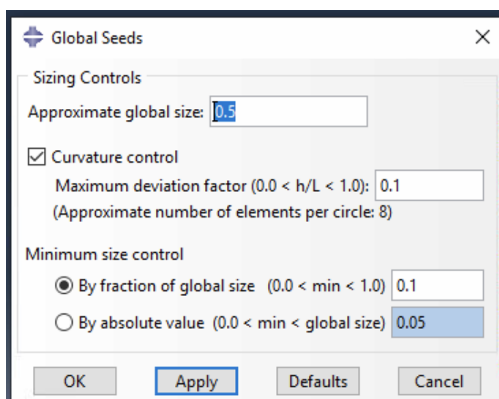


Figura 21: Seeds for the model.

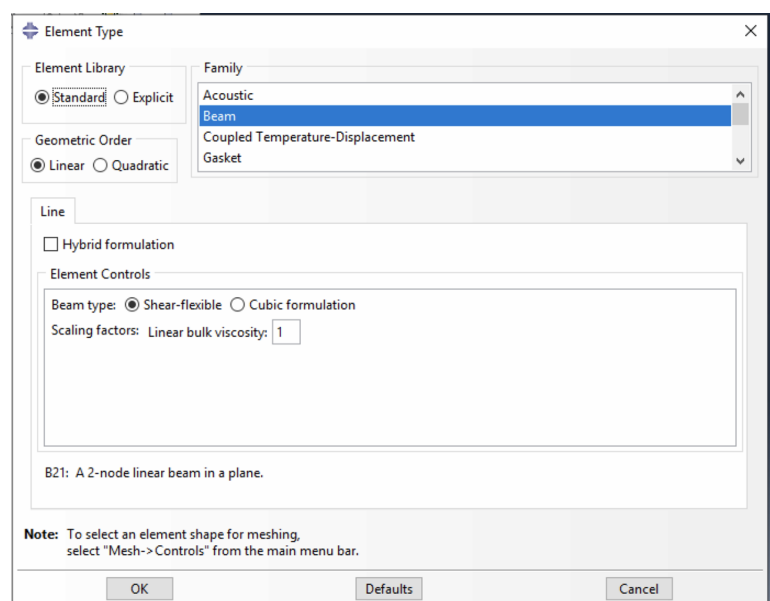


Figura 22: Selection of element type (Timoshenko Beam, B21) within “Mesh / element type”.

## 2.7. Job module

Once the model is completed, create a “Job” with the default options (Fig. 23).

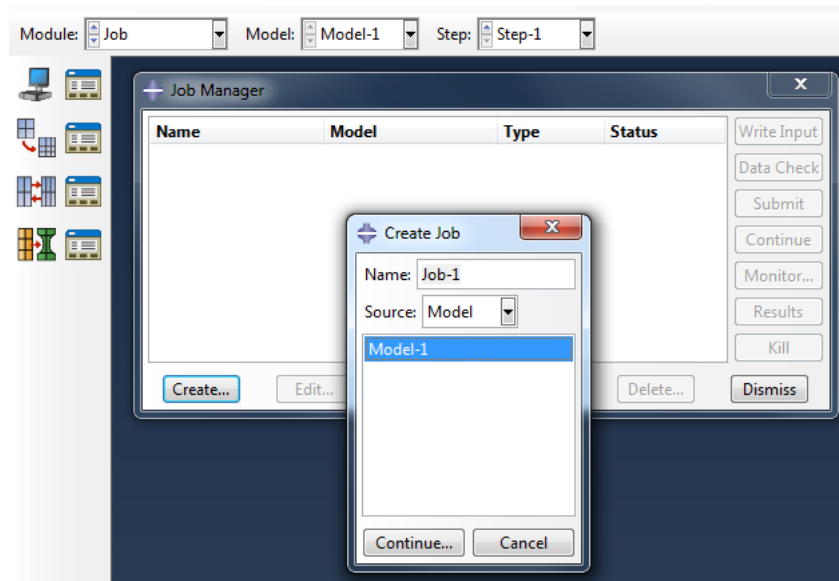


Figura 23: Creation of the “Job”

It may now be submitted for calculation through the “Submit” button. The *Status* should change from “Submitted” → “Running” → “Completed” (Fig. 24) If no error message is shown

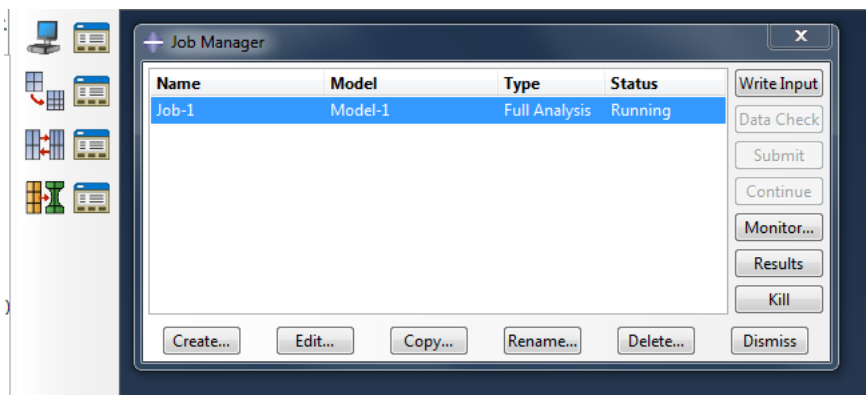


Figura 24: Submission of the “Job”

the job is finished and one can enter now the visualization module to view the results.

### 3. Results

Following, displacements and rotations are depicted for the case of study in Figs. 25 and 26 respectively.

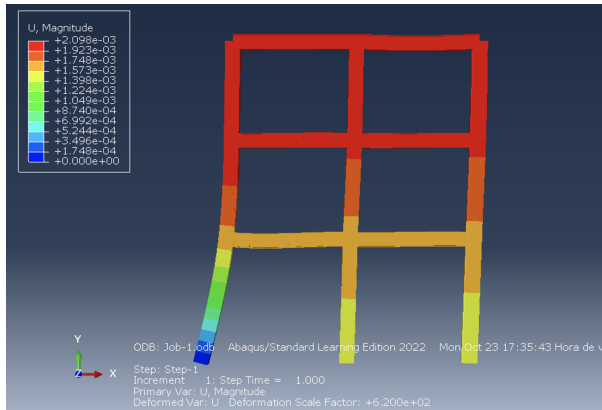


Figura 25: Displacements.

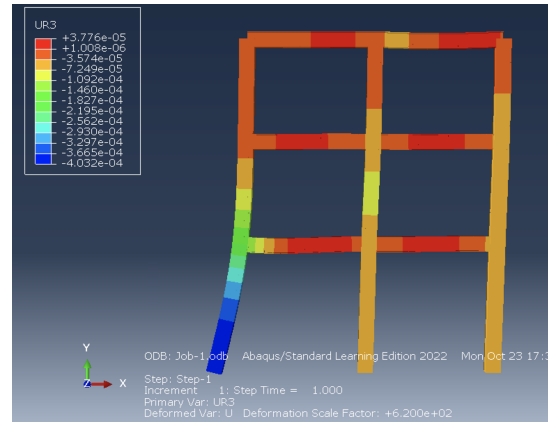


Figura 26: Rotations.

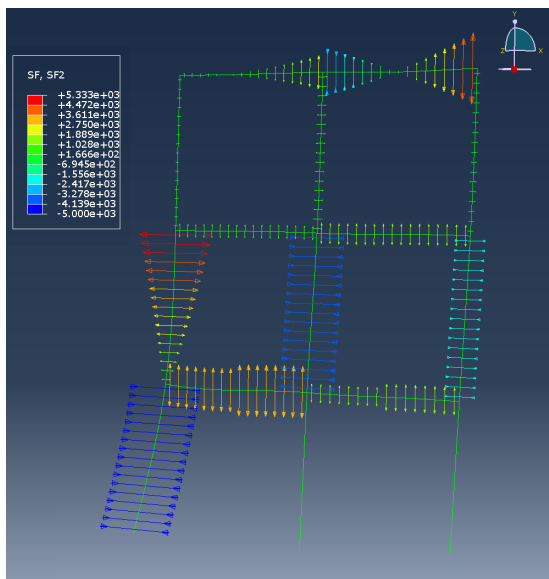


Figura 27: Shear.

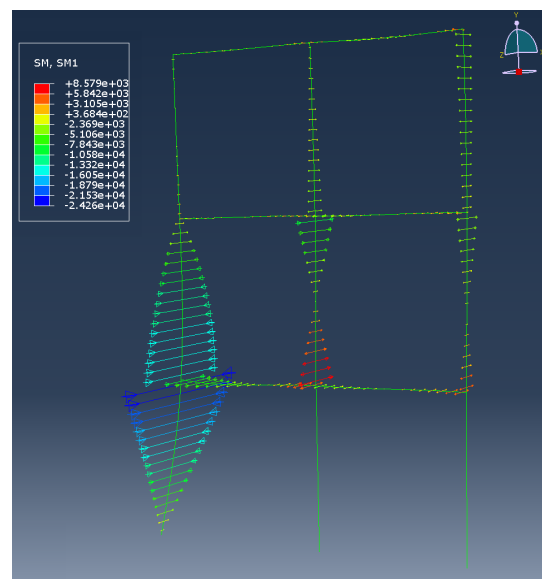


Figura 28: Bending moment.