LGCIV2041: Numerical Analysis of Civil Engineering Structures – Assignment

Handout: 19.02.2024 **Due:** Sunday 31.03.2024

Professors: João Pacheco de Almeida, Hadrien Rattez

Teaching Assistant: Alexandre Sac-Morane

This is a document to explain the Abaqus file given to solve Pb2 and how to use it.

Abaqus file is distributed as the implementation of the Ramberg-Osgood spring is a specific manipulation in Abaqus. If you have any questions, remarks, wills for script modifications, please mail me at **alexandre.sac-morane@uclouvain.be**.

The files are named **pb2_abaqus_file_deformable.inp** and **pb2_abaqus_file_rigid.inp**.
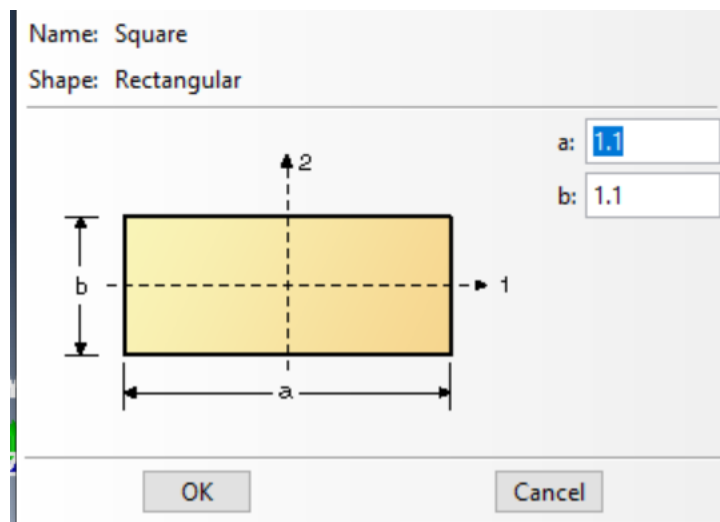
About the Part Module

A wire is build in a 2D space.
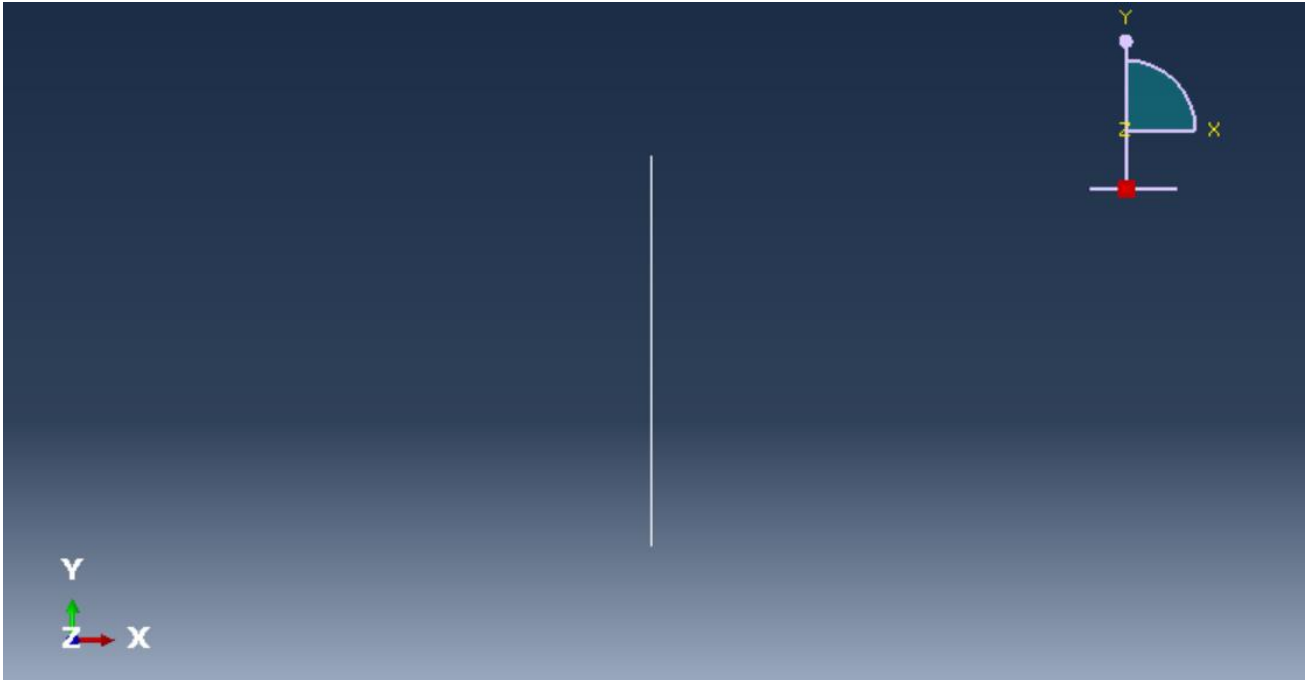
About the Property Module

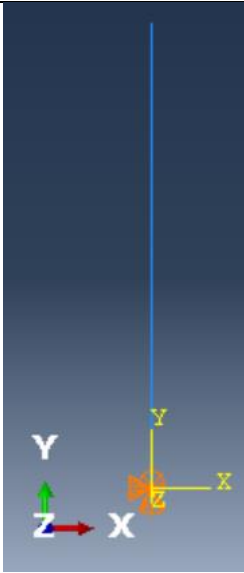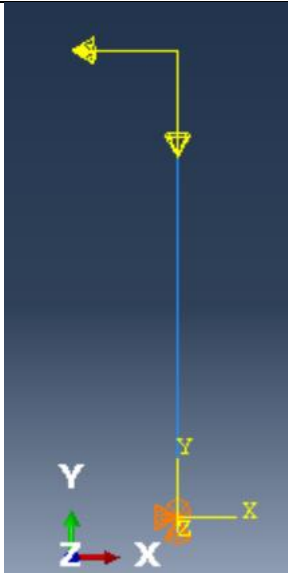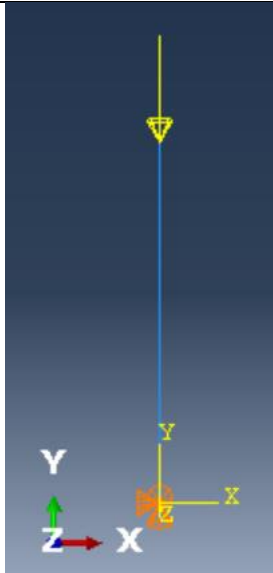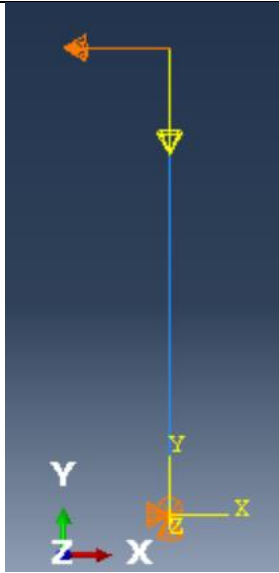A homogeneous material is designed with a Young's modulus and a Poisson's ratio (=0.3).

pb2_abaqus_file_deformable.inp	pb2_abaqus_file_rigid.inp
Y = 3e8 Pa (to allow deformation of the body)	Y = 1e12 Pa (to verify the rigid body assumption)

A square section of size 1.1 m is created and assigned to the part.

About the <u>Step Module</u>

4 steps are created:

Initial	Linear	Reinitial	Unlinear
The node at the bottom is pinned (U1=U2=U3=0) and a Ramberg-Osgood spring is created at the bottom.	The forces W (=2500 kN) and Flat (=900 kN) are applied at the top node. Linear geometry is assumed.	The force Flat is removed. Only the force W is applied at the top node. Linear geometry is assumed.	The force W is applied at the top node. A displacement control ($\Delta_{lat} = -1.2$ m) is applied at the top node. Nonlinear geometry is assumed.

About the <u>Mesh Module</u>

20 nodes are used to mesh the beam with shear-flexible elements.
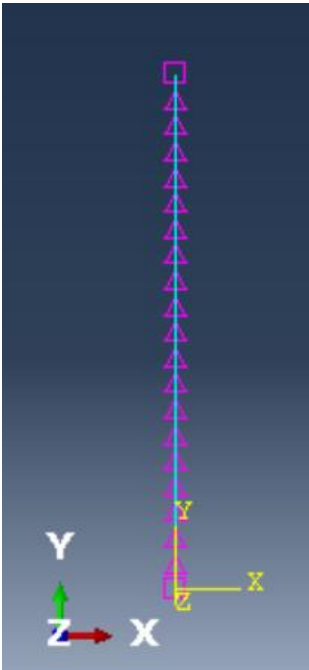
About the <u>Visualization Module</u>

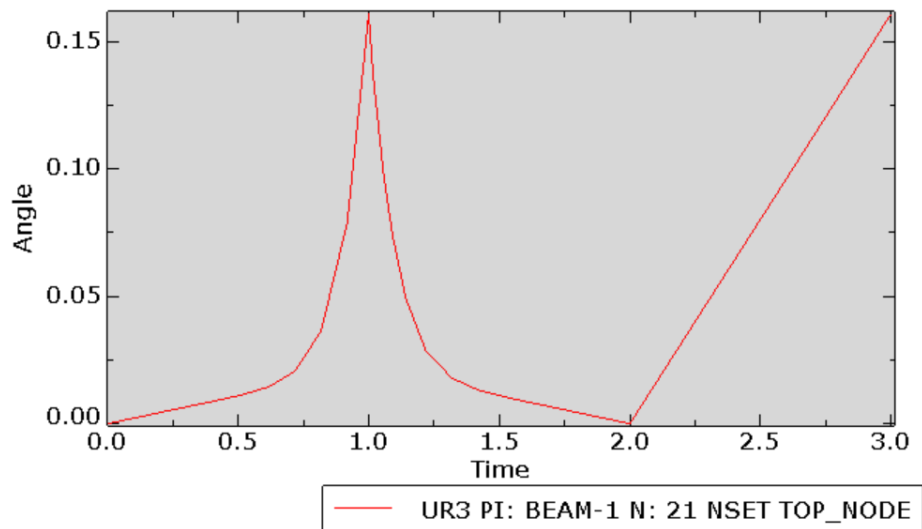Different field outputs are available:

- RF, RM3, S, SF, SM, U, UR

Steps are solved with multiple iterations. To navigate between iterations and steps, please use the following tools.

⏮ ◀ ▶ ⏭

Different history outputs are available:

- S, SF, SM, U, UR at the top and at the bottom

In the following, an example is given with the history of the variable UR3 at the top.


─── UR3 PI: BEAM-1 N: 21 NSET TOP_NODE

- From t=0s to t=1s, the element is going to step Initial to Linear.
  - ○ The force W and Flat are applied with a linear interpolation ($P(t) = P \times t$, with P the components given and t the time).
- From t=1s to t=2s, the element is going to step Linear to Reinitial.
  - ○ The force W is applied with the final value.
  - ○ Flat is unapplied with a linear interpolation.
- From t=2s to t=3s, the element is going to step Reinitial to Unlinear.
  - ○ The force W is applied with the final value.
  - ○ Δ_{lat} is applied with a linear interpolation.
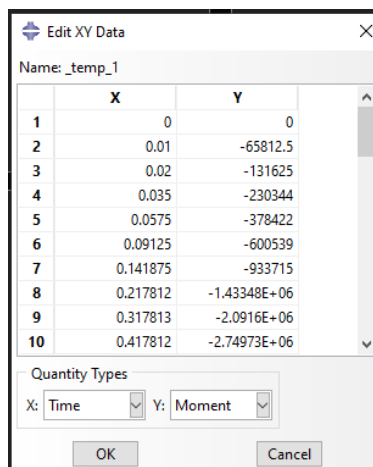
<u>If you want to export data into an excel file.</u>

In the Visualization Module, select the history output you want to export.
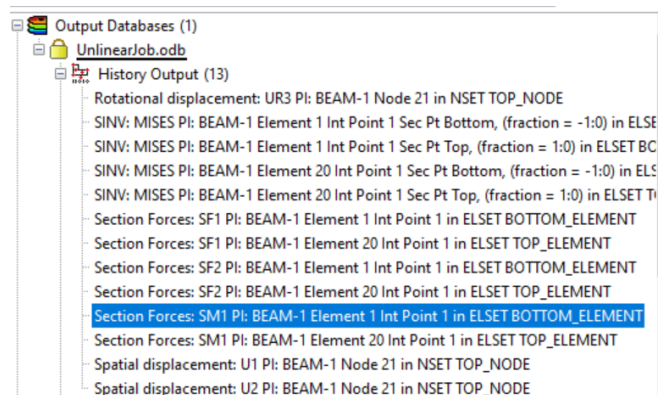
Then open the XY Data Manager .
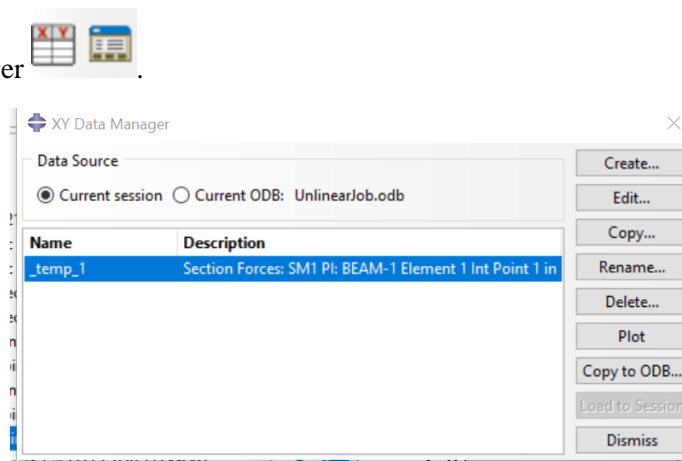
Double click on the data XY Data.

Select, copy and paste the data you want to export.

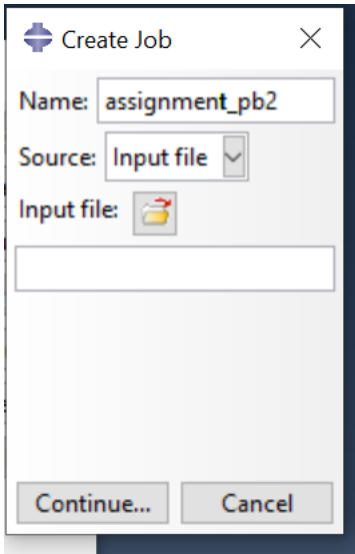## How to run the simulation and obtain results.

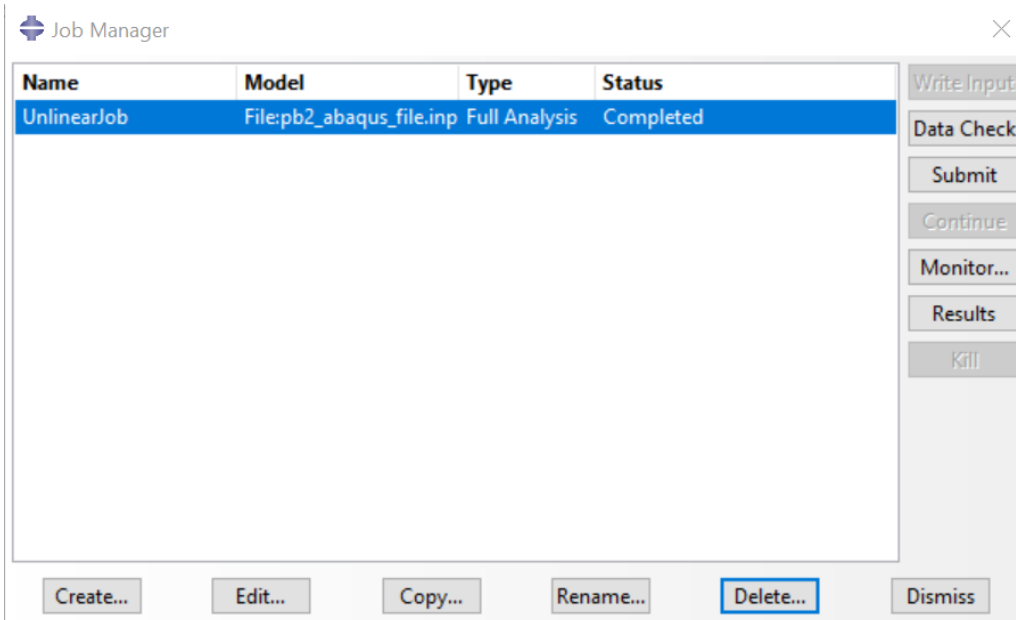Download the file.

Open Abaqus.

Go to the Job Module.

Create a Job.

In Source, select **Input file.**

Select the file.

In the Job Manager, select your job and click on Submit.

Once the simulation is done, click on Results.

You are good at analyzing the results.

If you have any questions, remarks, wills for script modifications, please mail me at **alexandre.sac-morane@uclouvain.be**.