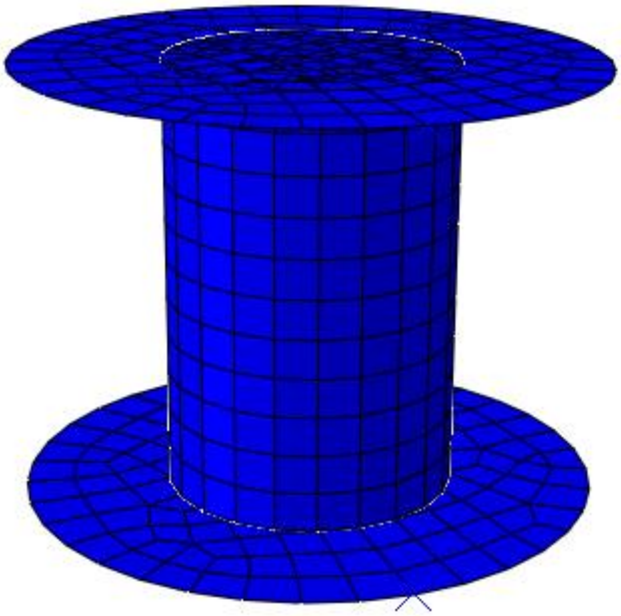
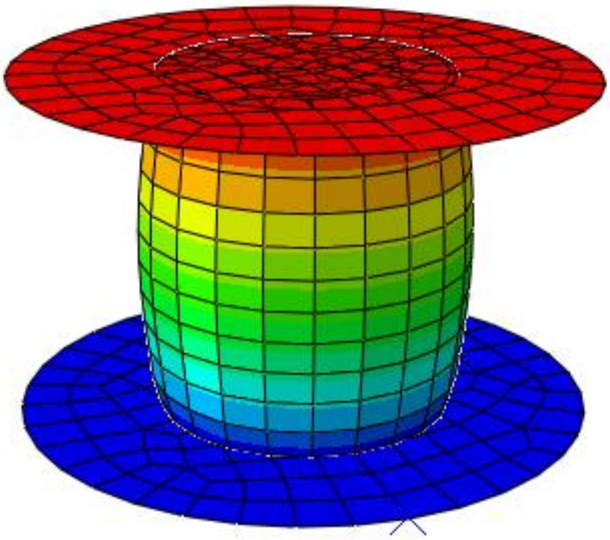
Cold bulk metal forming process with using mechanical type simulations.





1. Introduction

The aim of the task is to prepare a model of the process of the cylindrical sample compression test with dimensions: diameter=100mm and height=150mm. The sample is to be compress with two flat tools with diameter=200mm each. The lower tool is to be locked at all points of freedom, and the upper tool is to be moved from one base of the sample to the other until 20% deformation is achieved.

2. Material data and section creation.

The deformable material is an alloy of steel with specified mechanical properties, including the range of elastic-plastic deformation. Therefore, the material data required for the calculation in the ABAQUS application are material density, Young's modulus, Poisson's coefficient and stress-strain curve. To apply the material data correctly, it must be entered in "Materials" in the appropriate units. The model is created in mm, so the data must be entered into the application in accordance with the principle described in *Tab. 1*.

Tab. 1 Elastic properties of the steel used in the model.

Units SI	Model in *m*	Model in *mm*
Density	$7950\ \frac{kg}{m^3}$	$7.95E-9\frac{tonne}{mm^3}$
Young's module	$210E9\ Pa$	$210000\ MPa$
Poisson's coefficient	0.3	0.3

Insert material data into ABAQUS in the "Materials" window (Fig. 1)

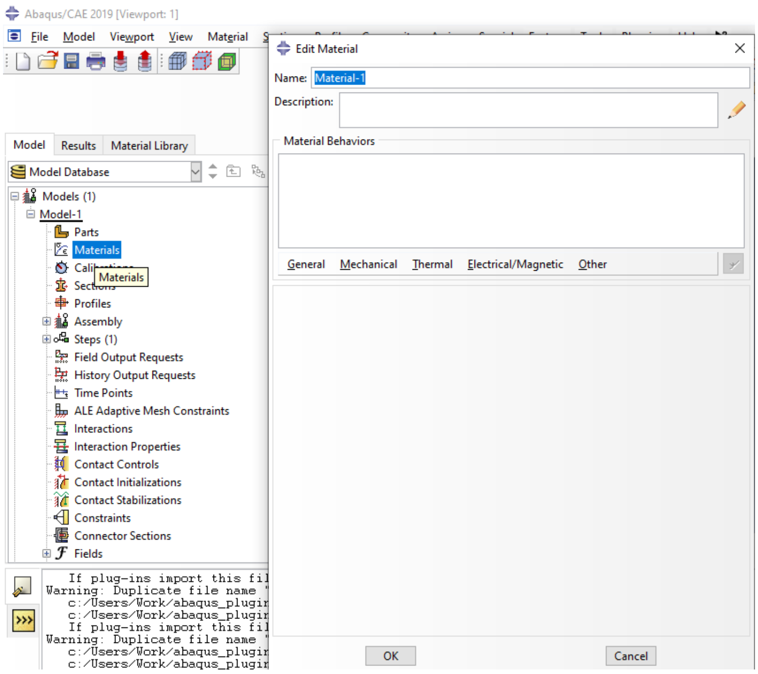


Fig. 1 Material data insertion window.

The material data from the table are entered as **General->Density** and **Mechanical ->Elasticity->Elastic** (Fig. 2).

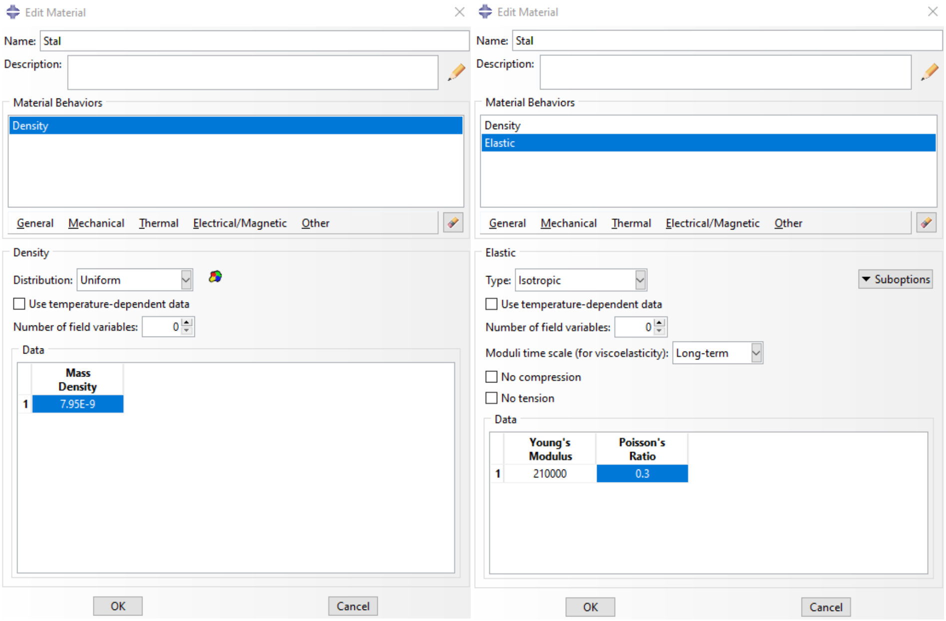


Fig. 2 Density and elastic properties of the material.

The stress-strain curve is taken from the *"Steel_material_datas.xlsx"* file added into the course platform. In the *flow_stress* tab, we take, respectively, column D where the yield stress values are located and column C where the plastic strain values are located. Then insert the data from each column in "Edit Material" in the option **Mechanical->Plasticity->Plastic** (Fig. 3)

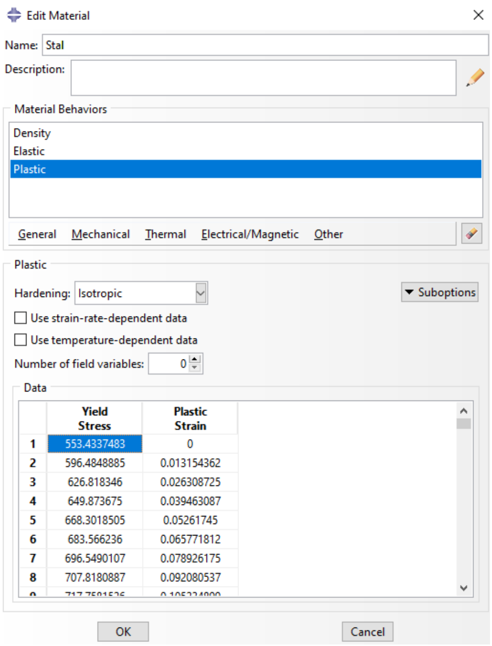


Fig. 3 Plastic properties of the material.

The material data must then be assigned to the appropriate section defined in this case as homogeneous (Fig. 4).

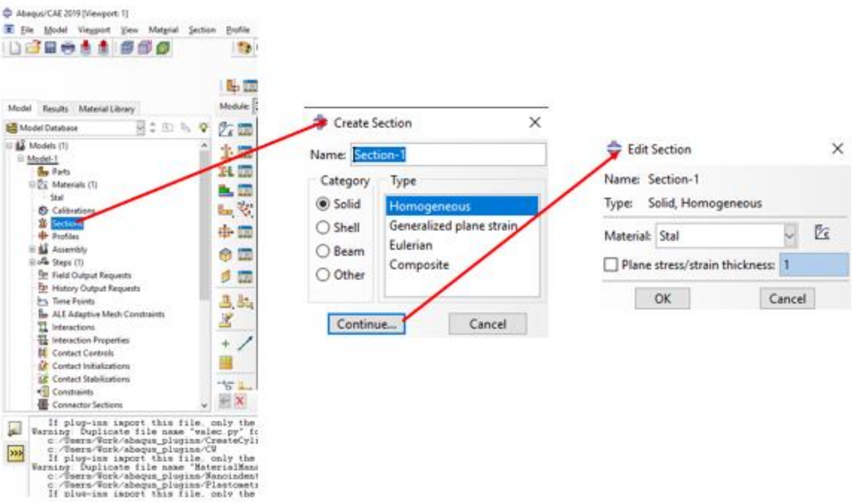


Fig. 4 Creating sections and assigning material data.

3. Technical drawing

   We make two technical drawings representing the sample and forming tools. The assembly to be done is shown, which consists of three parts (probe, lower tool and upper tool) in Fig. 5

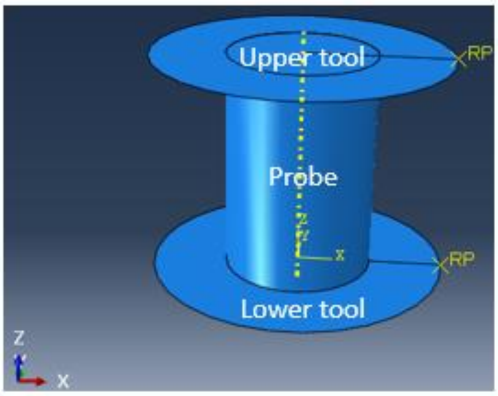


Fig. 5 Model assembly.

The drawing of the forming tool is done once, as it will be double-loaded in the fold for the top and bottom tool respectively. The sample and tool dimensions are presented below.

- Sample dimensions (*Fig. 6*): R = 50mm and H = 150 mm (**3D->Deformable->Solid ->Extrusion**)

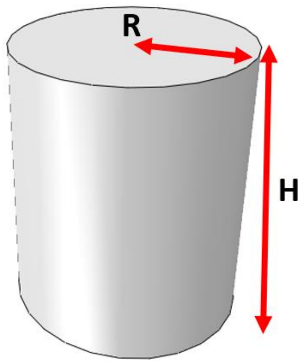


Fig. 6 Sample model.

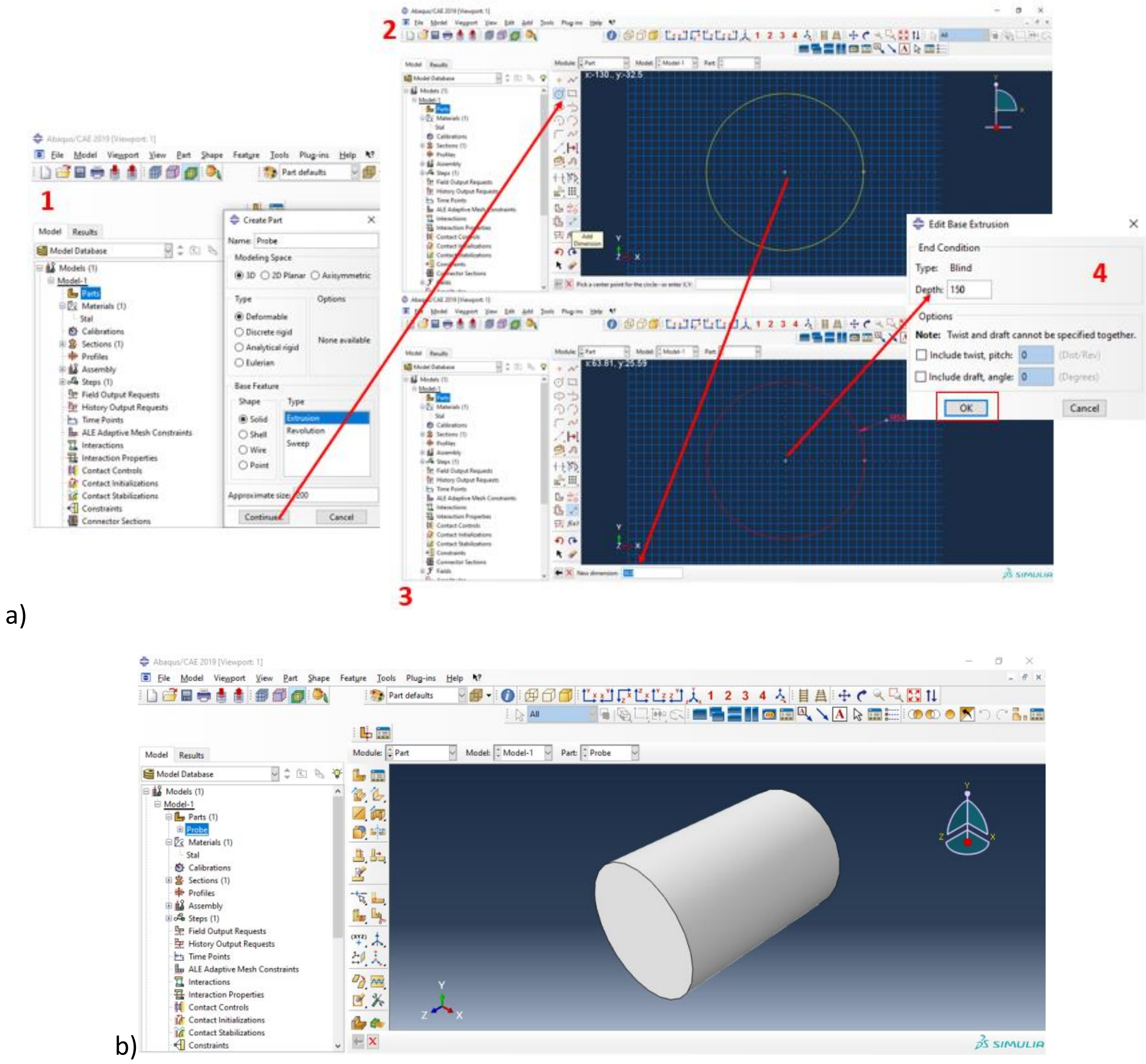


Fig. 7 Creating a technical drawing of the sample in the ABAQUS environment.

- Tool dimensions (Fig. *7*): R = 100 mm (**3D->Discrete Rigid->Shell->Revolution**)

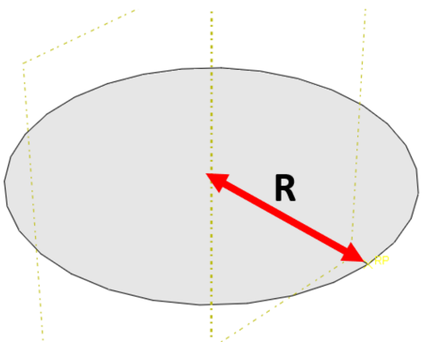
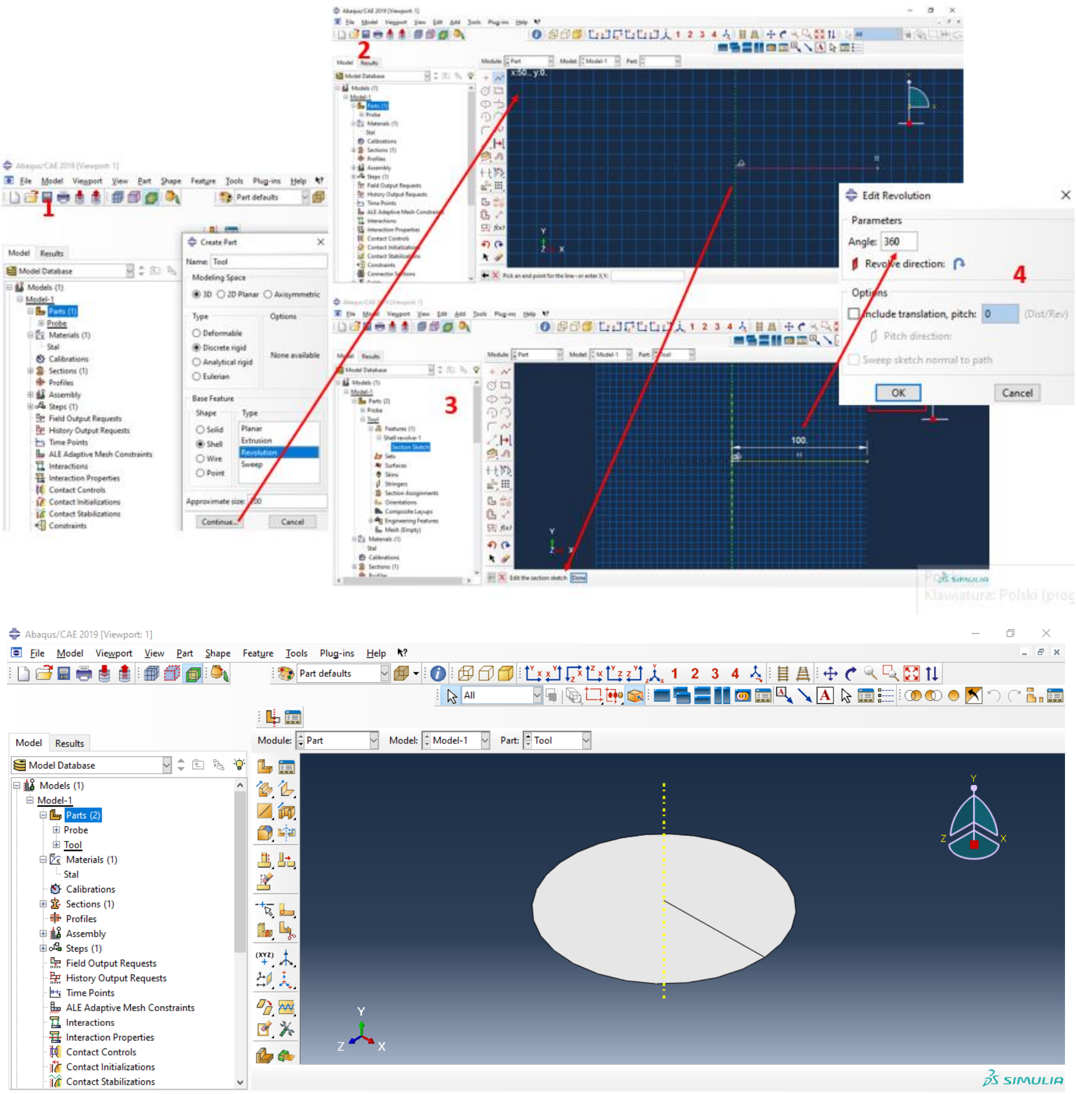


Fig. 8 Upper and lower tool model.

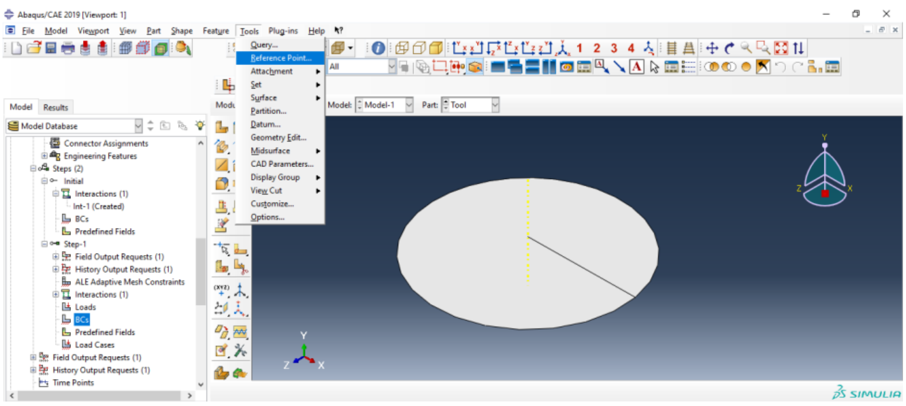


a)

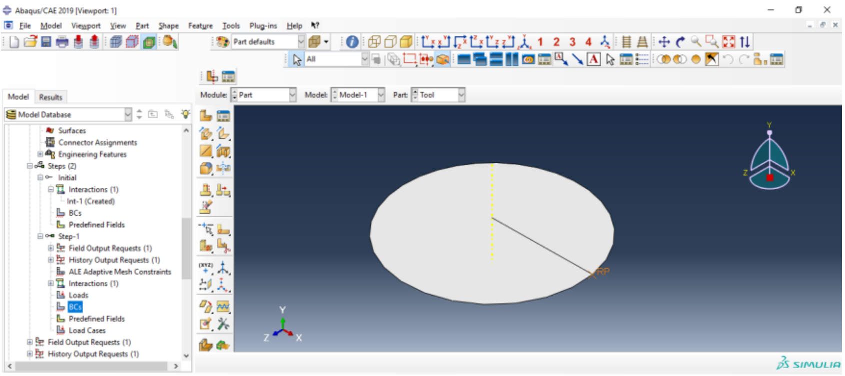
b)

Fig. 9 Create a tool drawing in the ABAQUS environment.

In order to be able to create any boundary conditions on tools that are of the "Rigid" type, a special reference point [RP] must be inserted in the "Tool" section. Use the option from the top bar to do this **„Tools"-> Reference Point** (Fig. 10)**.** The location of such a point is arbitrary.



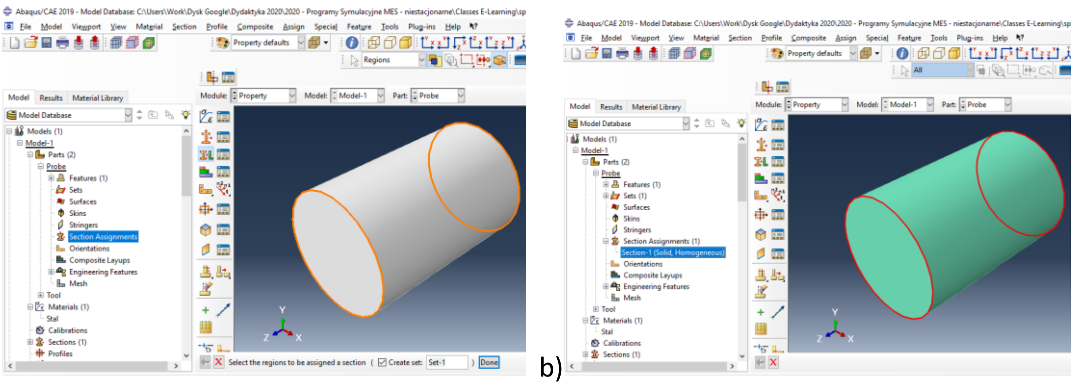
a)



b)

Fig. 10 Inserting a reference point.

Then assign the section created in point 3 to the sample in **"Section Assignments"** by following the commands given in the bottom bar (*Fig. 11*).



a) b)

Fig. 11 Assigning sections to the sample

4. Define the time step.
   The time step defines which type of simulation is to be performed. In this task we use the solver Implicit, so using the ABAQUS environment nomenclature we use the solver types Standard. Double click **Steps->Static**, **General (Step-1)** (Fig. 12)

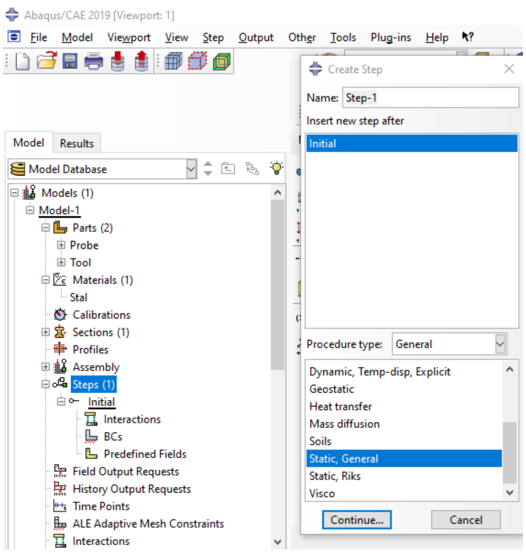


Fig. 12 Create a time step in the ABAQUS environment.

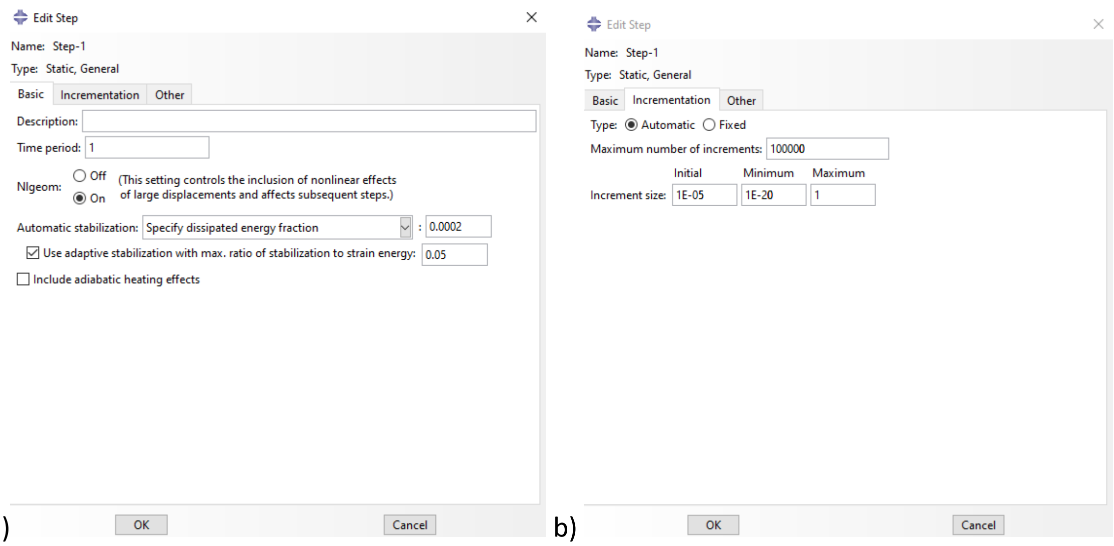


Fig. 13 Internal setting of the time step.

5. Preparation of the assembly.

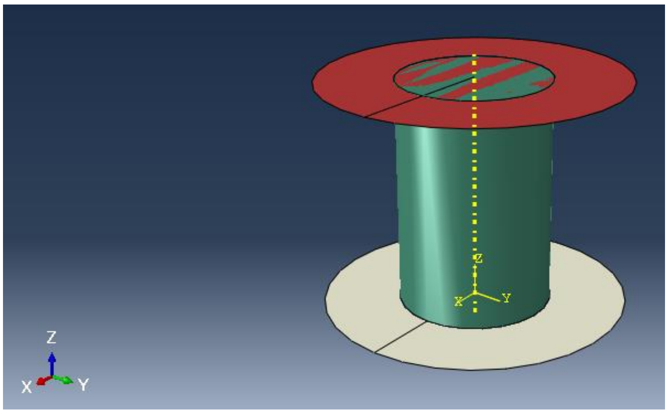


Fig. 14 Assembly.

Now, with the help of previously prepared parts, you need to prepare the model assembly. The sample in the assembly is to be placed in the middle between the two tools. To do this, you will need to use the move and rotate options to create the required assembly part setting. So double click **"Assemble->Instances"**. Load both **"Probe"** and **"Tool"** models using the **"Shift"** button. (Fig. 15).

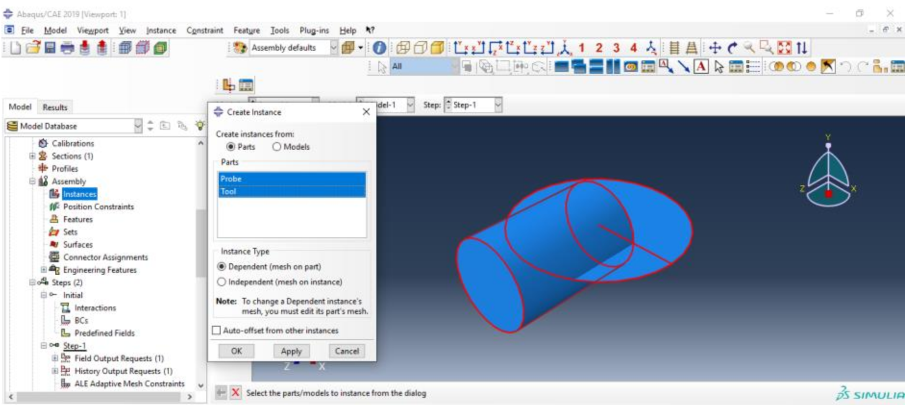


Fig. 15 Loading parts for assembly

Now turn the "Tool-1" so that the possible contact surface is consistent with the lower surface of the sample.

- Choose Rotate->Instance

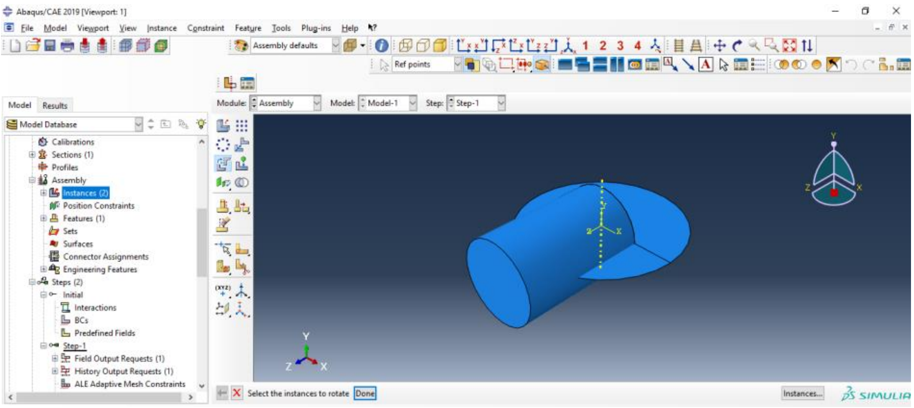


Fig. 16 Select Rotate->Instance

- Select the bottom tool

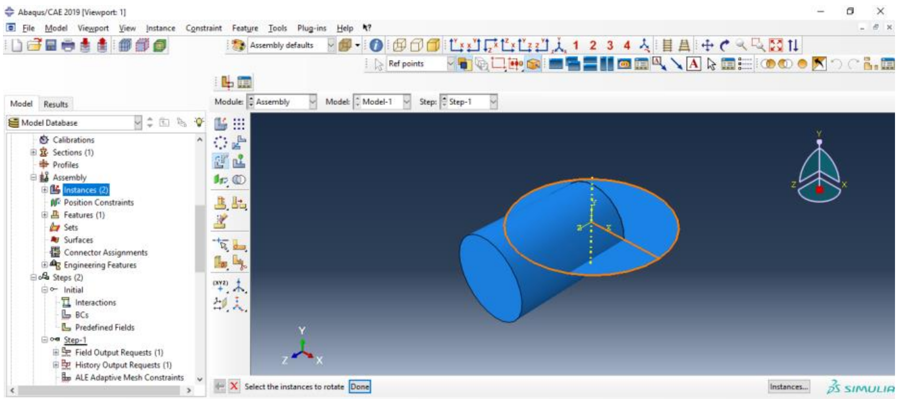
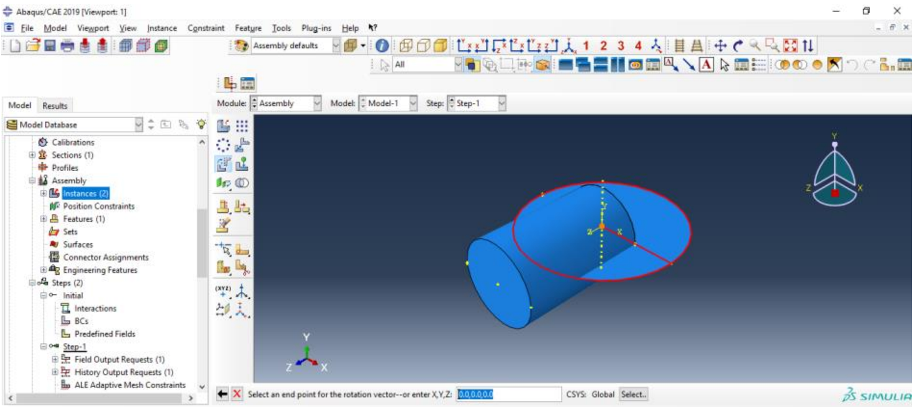
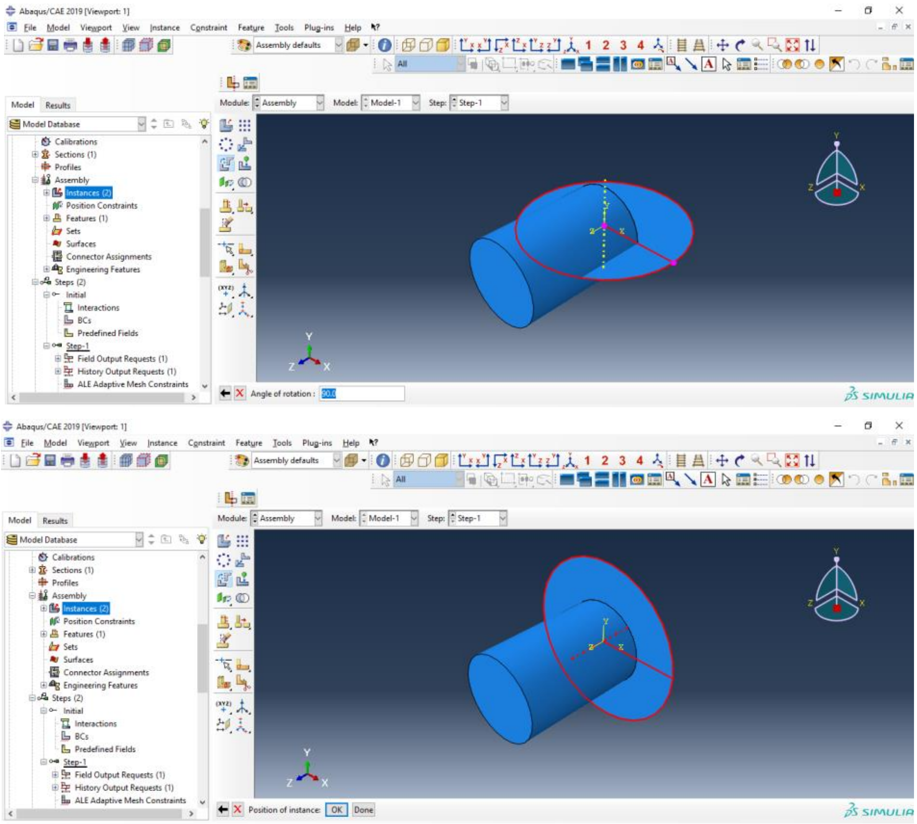


Fig. 17 Select the bottom tool

- Select the axis of rotation against which you rotate the tool model by 90 degrees.

a)





b)

c)

Fig. 18 Select the axis of rotation against which you rotate the tool model by 90 degrees.

Now we load the second tool "Tool-2", move it to the upper surface of the sample and then rotate it in the same way as it was done with the lower tool (Fig. 19).

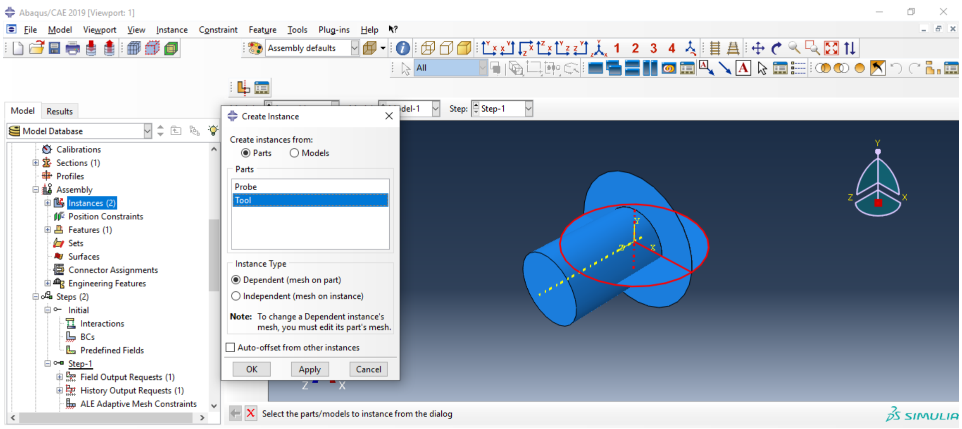
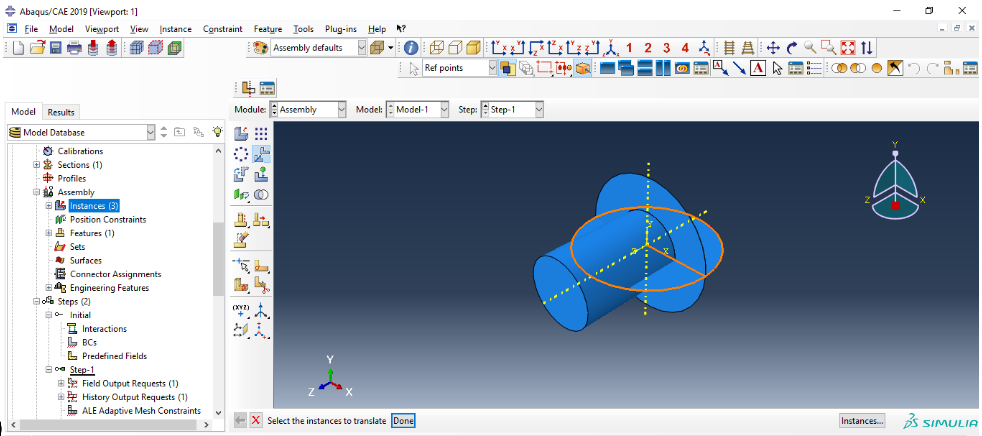
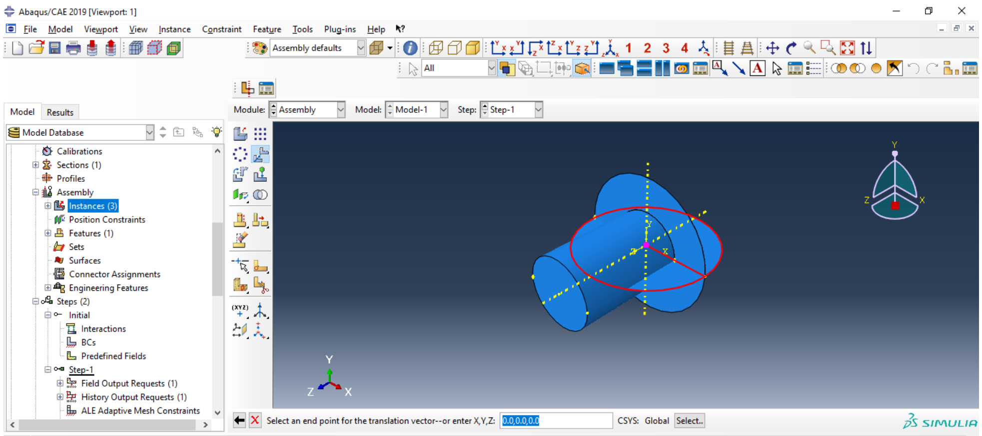
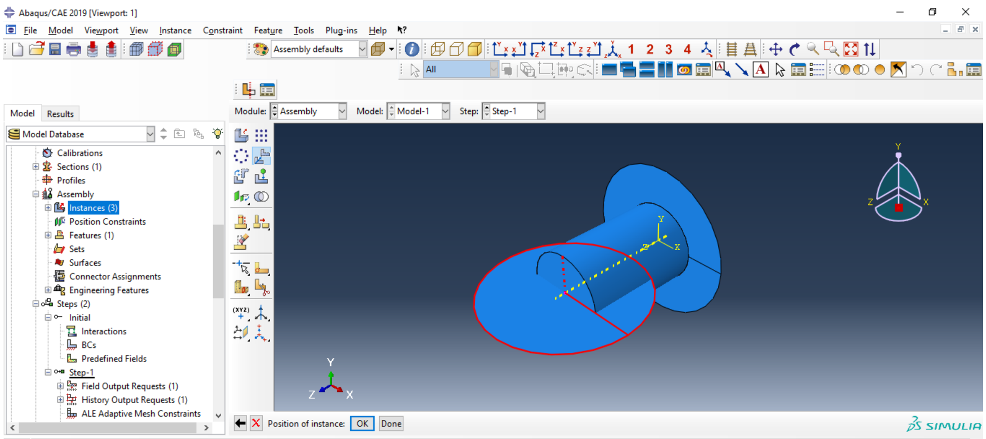
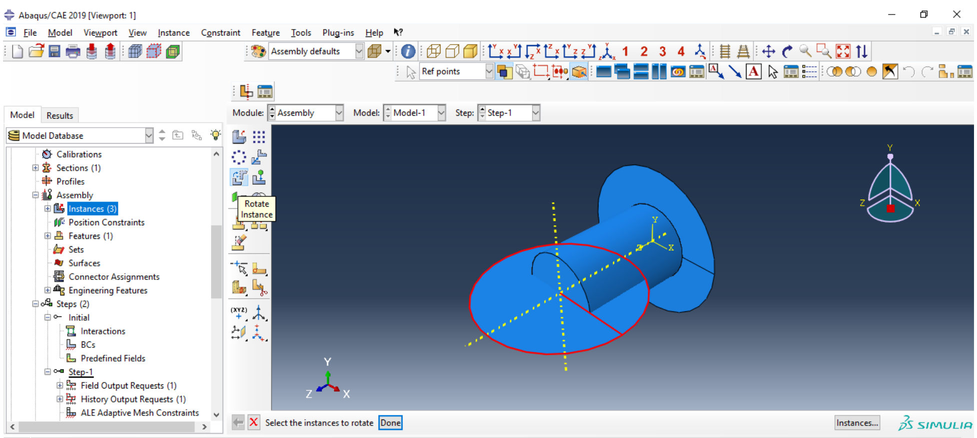


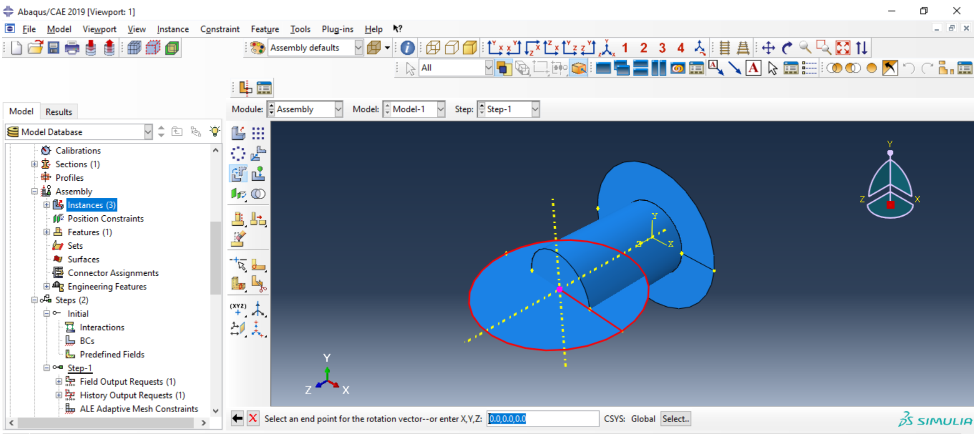
Fig. 19 Loading the second tool with the Instances option.


Abaqus/CAE 2019 [Viewport: 1]
Module: Assembly
Model: Model-1
Step: Step-1
Select the instances to translate
Done
SIMULIA
a)

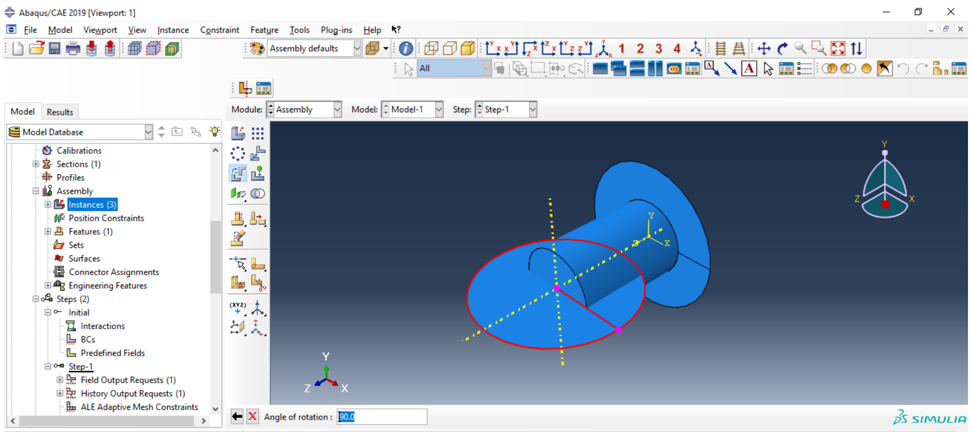

Abaqus/CAE 2019 [Viewport: 1]
Module: Assembly
Model: Model-1
Step: Step-1
Select an end point for the translation vector--or enter X,Y,Z:
CSYS: Global
SIMULIA
b)


Abaqus/CAE 2019 [Viewport: 1]
Module: Assembly
Model: Model-1
Step: Step-1
Position of instance:
OK
Done
SIMULIA
c)

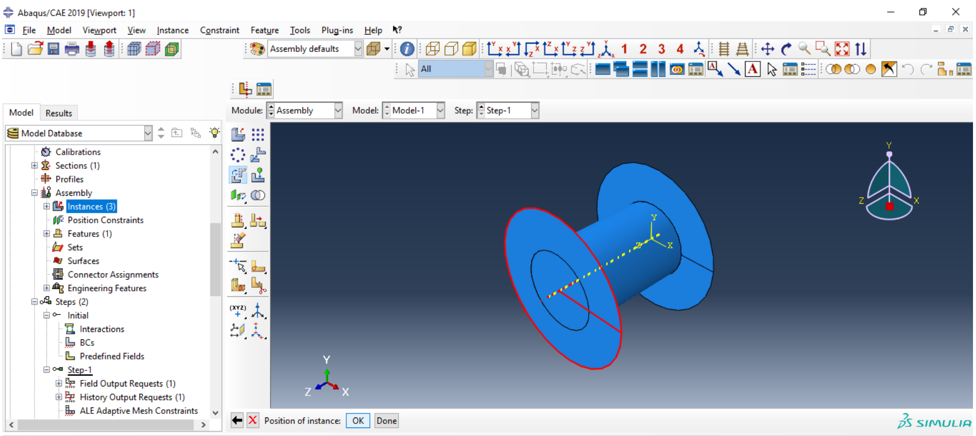

Abaqus/CAE 2019 [Viewport: 1]
Module: Assembly
Model: Model-1
Step: Step-1
Rotate Instance
Select the instances to rotate
Done
SIMULIA
d)



e)



f)



g)

Fig. 20 Loading the second tool with the Instances option.

6. Selection of boundary conditions.

   In the example, the lower tool is to be locked in all degrees of freedom. Therefore, the **"Encaster"** option is set to the lower tool reference point in **Step-1->BC-1->Mechanical ->Symmetry/Antisymmetry/Encaster.**

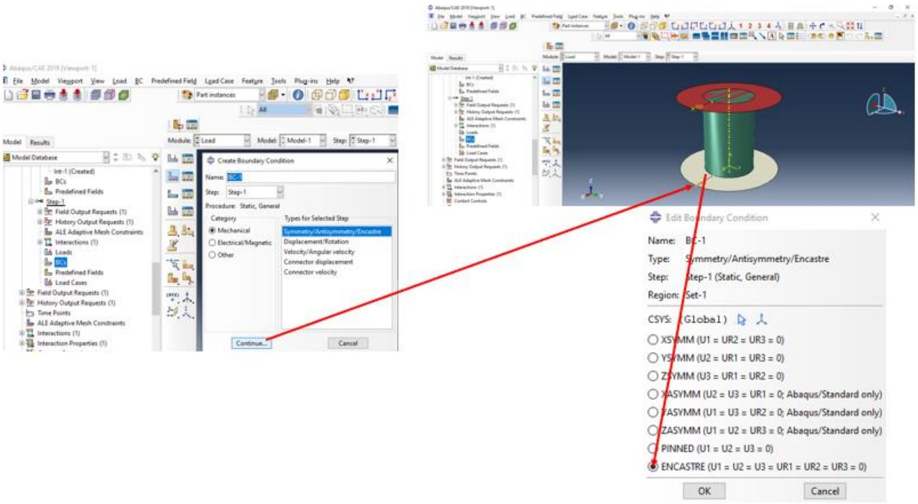


Fig. 21 The first boundary condition is placed on the lower tool.

The upper tool moves by 30mm, deforming the sample by 20% from its height to Fig. 22.

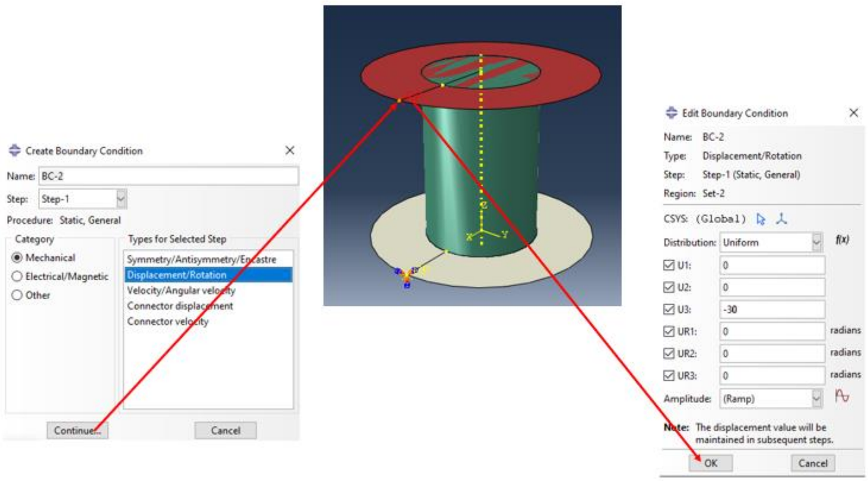


Fig. 22 The second boundary condition is attached to the upper tool.

7. Define interactions.
   In the "Initial" step, select **"General contact (Standard)".** This option allows to create an automatic contact in ABAQUS application. The method of establishing a "**General**" type of contact is presented on Fig. 23.
   After selecting the option **"General contact",** select the way of contact between the tools and the sample. Here you must select the coefficient of friction and the value of the coefficient of friction through **Contact->Mechanical->Tangential Behaviour->Friction Formulation (Penalty)->Friction Coeff (0.3).**

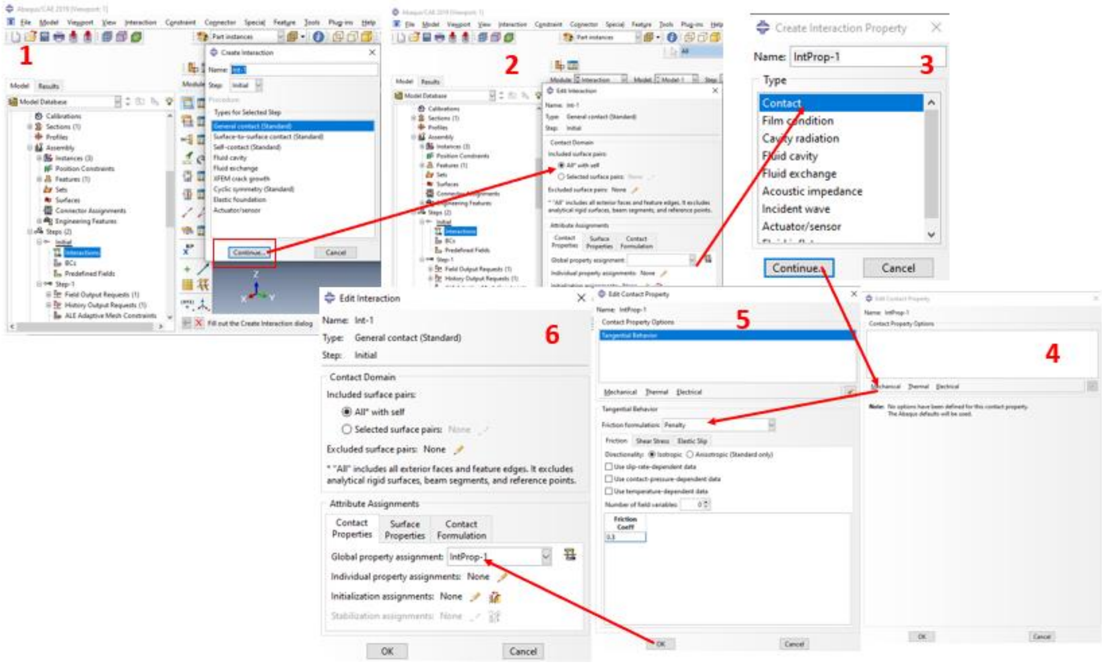


Fig. 23 Creating General contact.

8. FEM discretization.
   The discretization is done on both parts in parts in the module (Parts) and not in the module (Assembly). Please note that the student version does not allow for discrediting more than 1000 nodes per model.

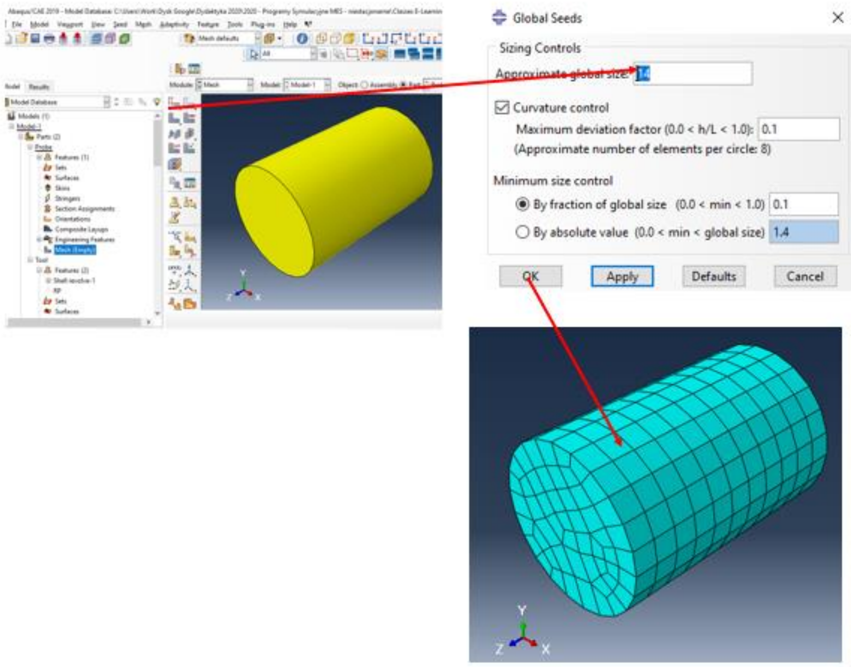


Fig. 24 Sample FEM discretization.

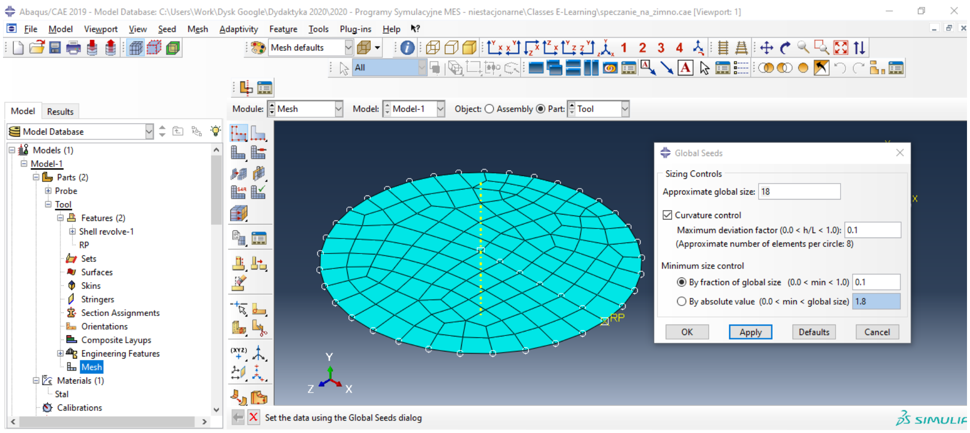


Fig. 25 Tool FEM discretization.

9. Create a task and perform calculations.
   The last step is to prepare the input file (Job-1.inp) for Solver Standard. Double click on "Job" to enter the calculation settings.

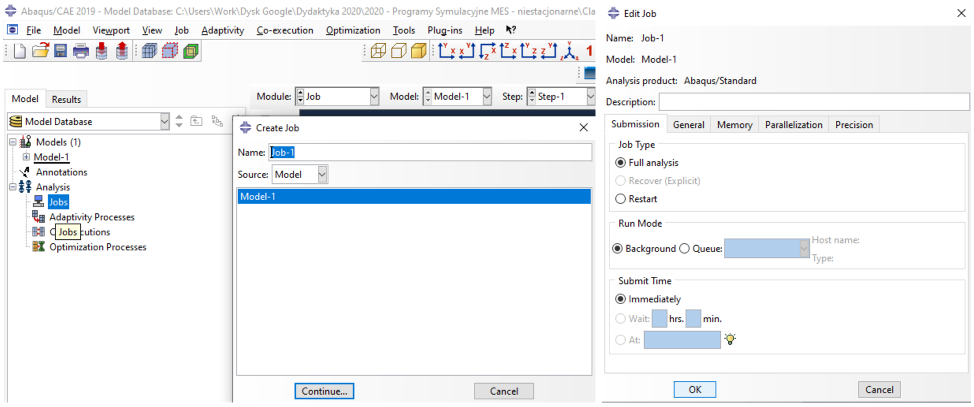


Fig. 26 Order calculations from the model.

Now you have to right-click and run the "Submit" option, which sends calculations to the solver that is installed locally.

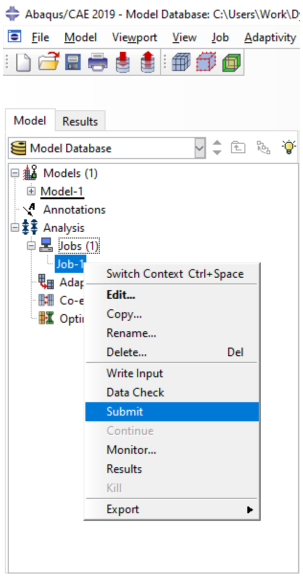


Fig. 27 Starting calculations with the Submit option.

10. Visualisation of results.
    If you click the right mouse button on the "Job-1" and enable the **Monitor** option, you can see the numerical values or the message **"Complete"** is written, it means that ABAQUS counted the simulation correctly. Errors or warnings may appear then **"Error"** message will appear then you need to see in the logs what the problem is and correct it according to the above instruction. If everything calculated correctly you can go to the results analysis by right clicking on "Job-1" and **"Results".** Clicking **"Plot Contours on Defromed Shape"** allows you to view the distributions of different calculated values such as stresses, deformations and displacements (Fig. 28).

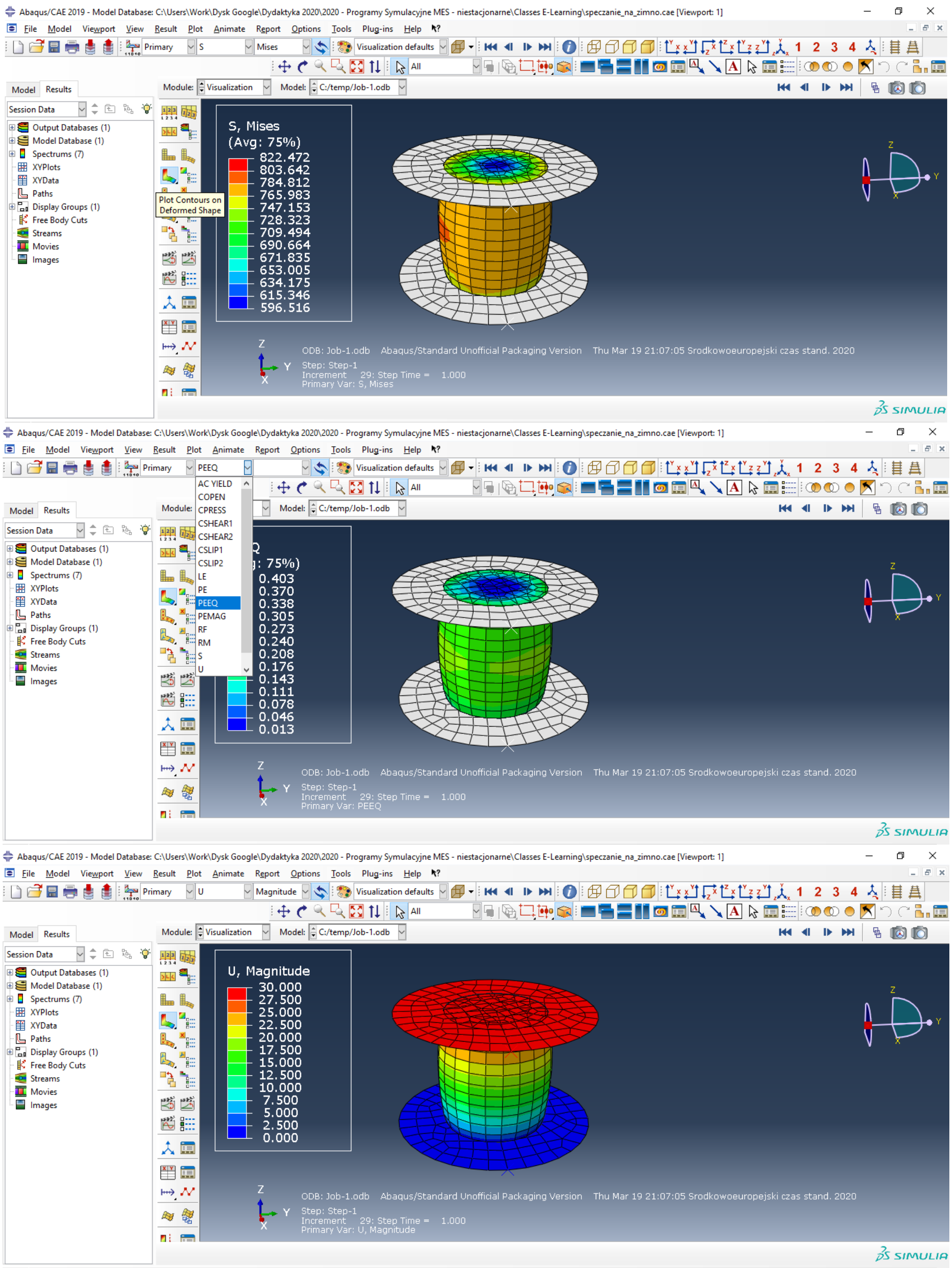


Fig. 28 Results of calculations.