

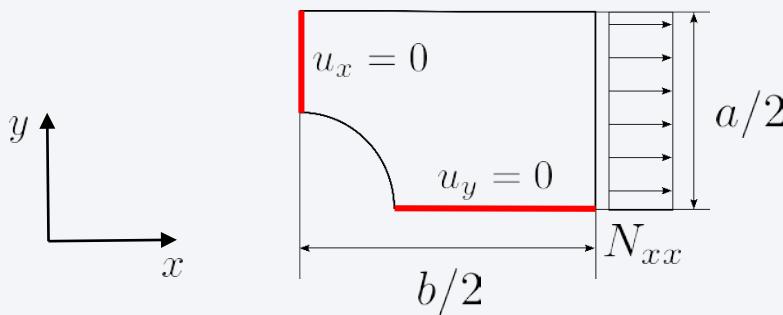
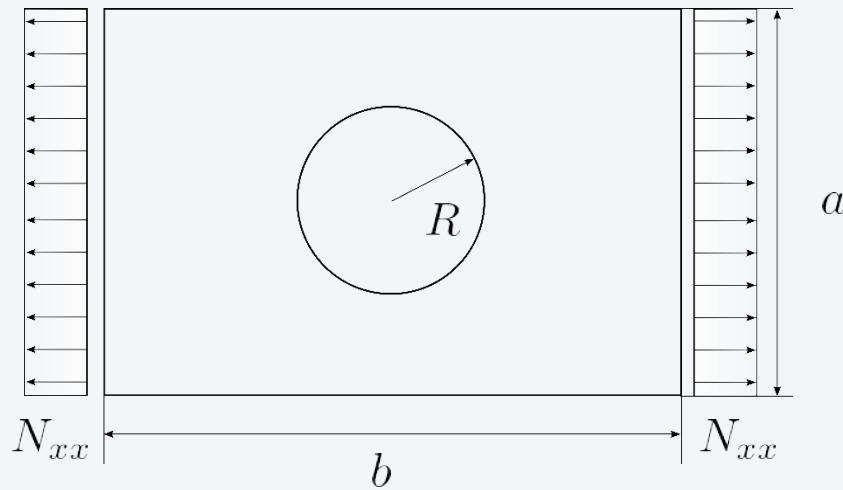
A brief introduction to the use of Abaqus using the CAE

Course of Spacecraft Structures
AY 2017/2018

Riccardo Vescovini
Politecnico di Milano, Department of Aerospace Science and Technology

Example

- Open-hole test: membrane with center hole, loaded in traction (1/4 of the structure due to the double symmetry of the problem)



Input data

$$a = 100 \text{ mm}$$

$$b = 150 \text{ mm}$$

$$R = 25 \text{ mm}$$

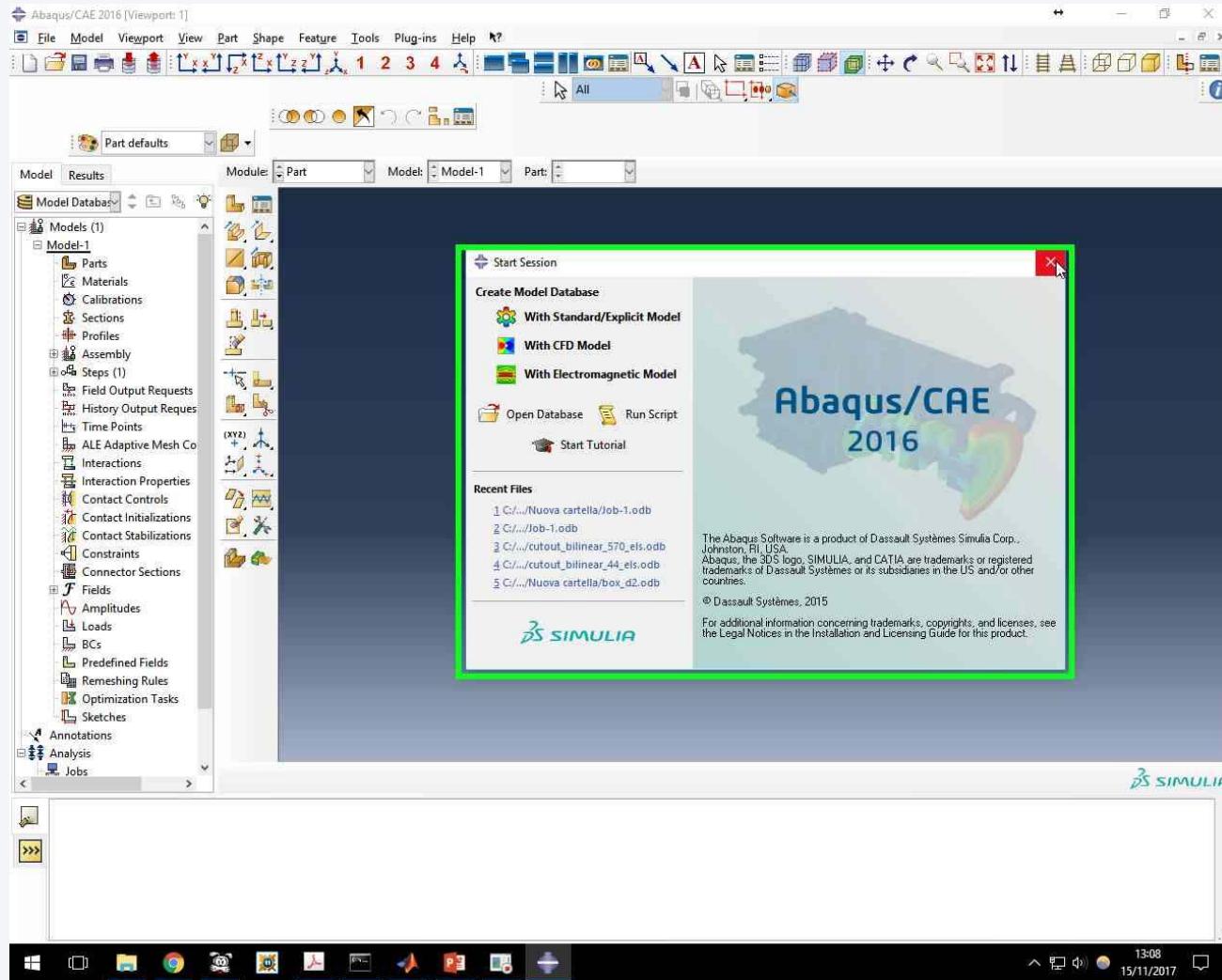
$$t = 1 \text{ mm}$$

$$E = 72 \text{ GPa}$$

$$\nu = 0.3$$

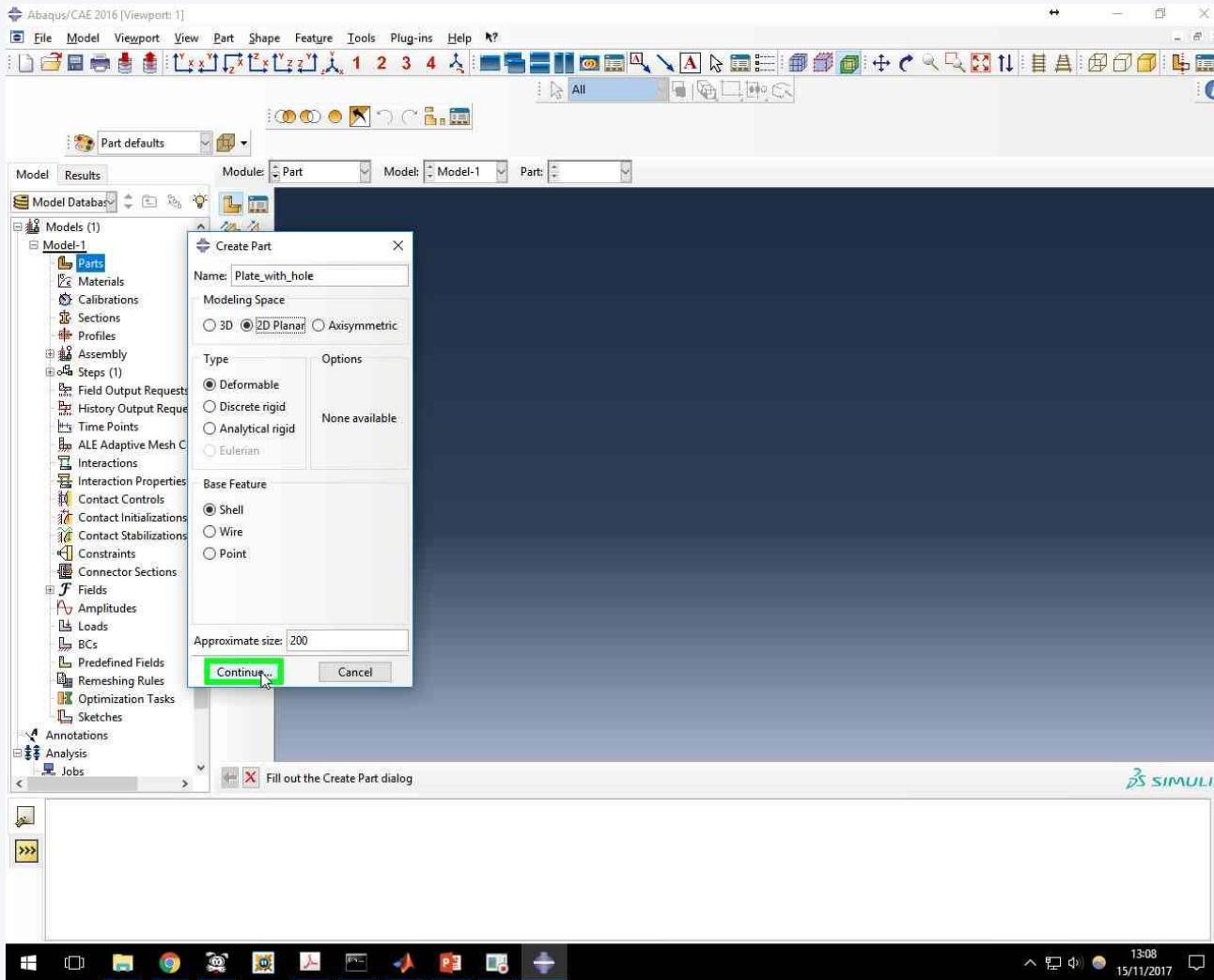
$$N_{xx} = 20 \text{ N/mm}$$

Start session



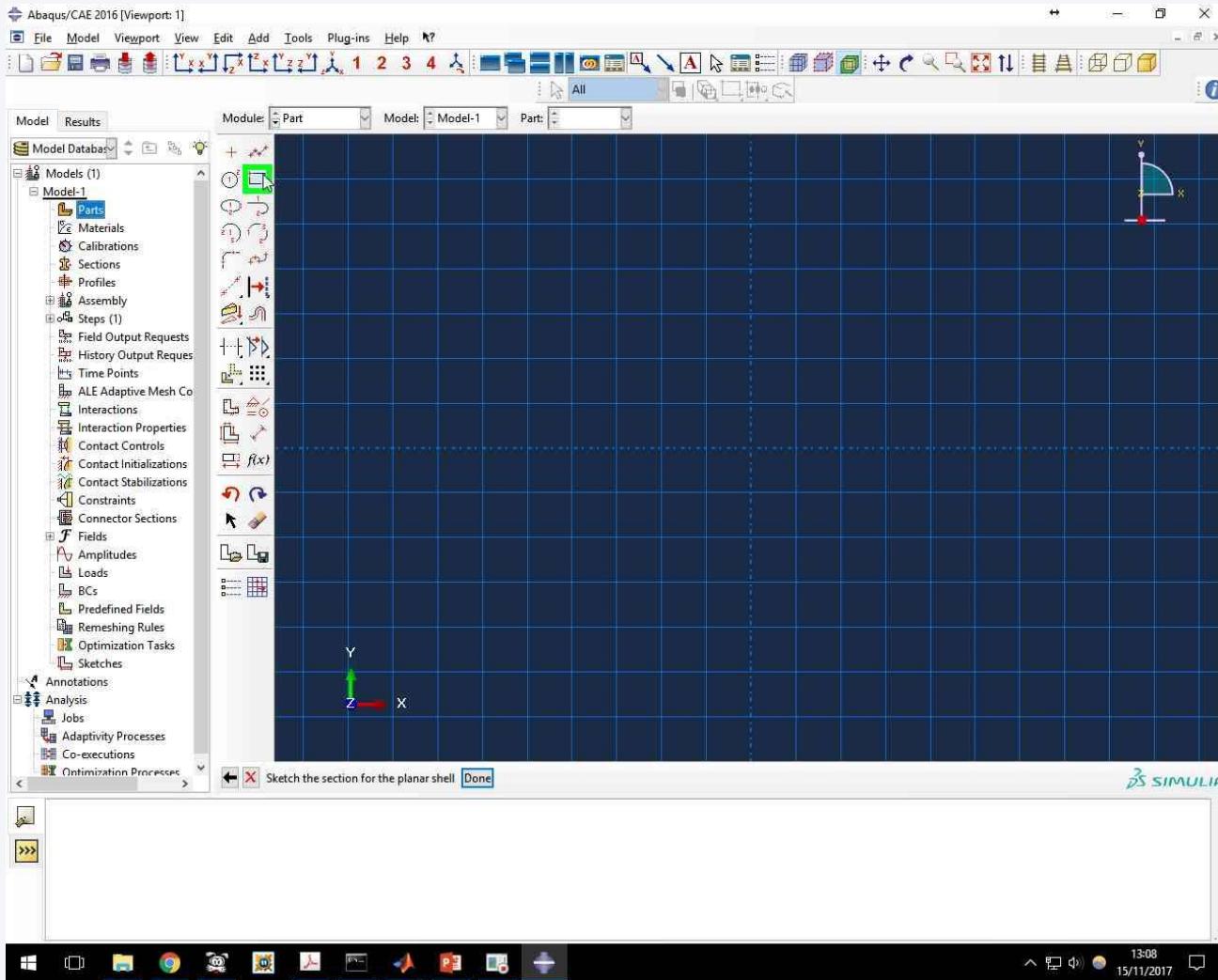
- + Create a folder
- + Open the command prompt and move to the folder (the syntax is «cd c:\my_address\....\My_folder»)
- + Launch the Abaqus command interface with the command «abaqus cae»

Geometry



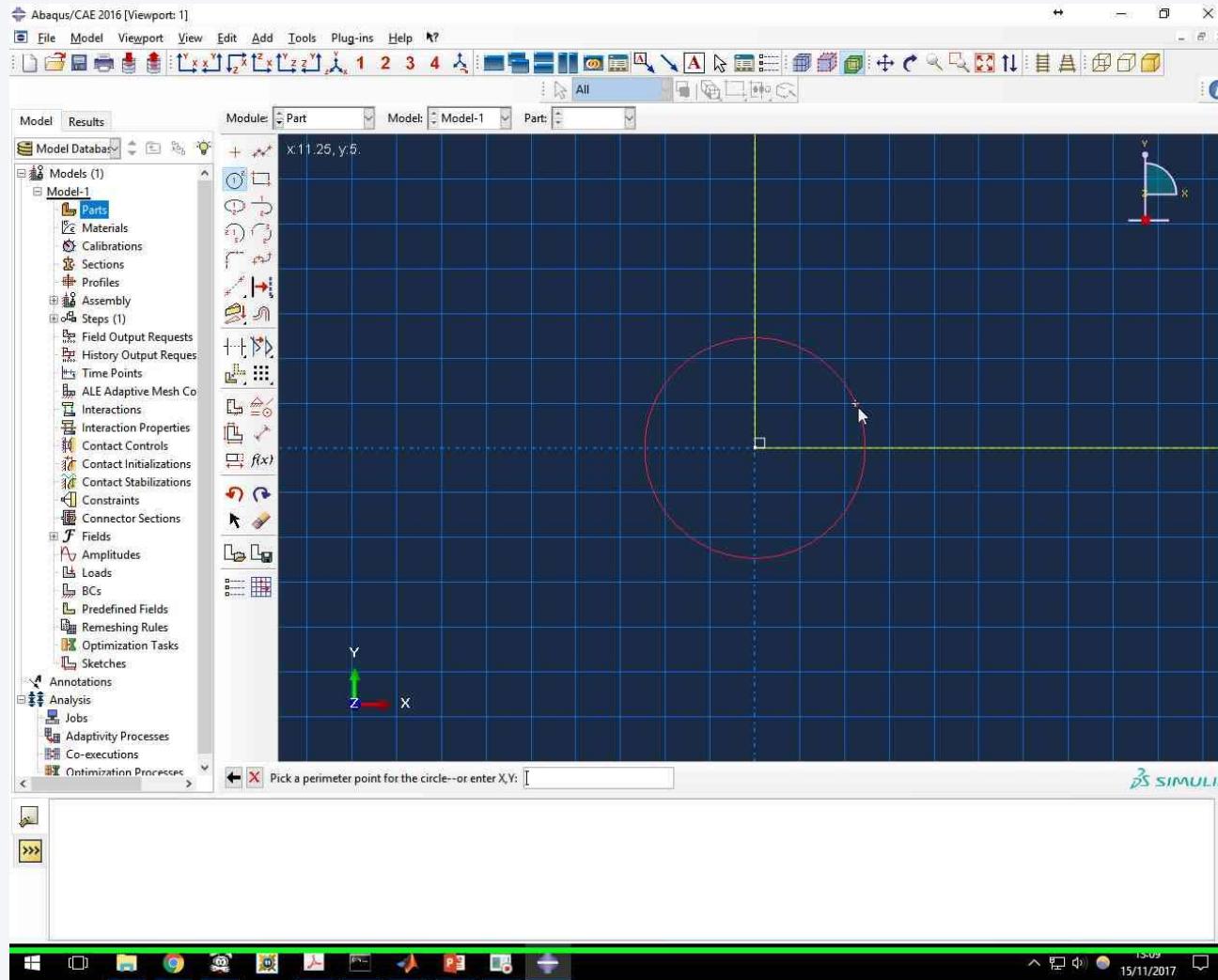
The first step consists in generating the geometry. Note that the geometry has the only role of supporting the generation of the mesh. The finite element model does not involve the geometric quantities defined at this stage.

Geometry



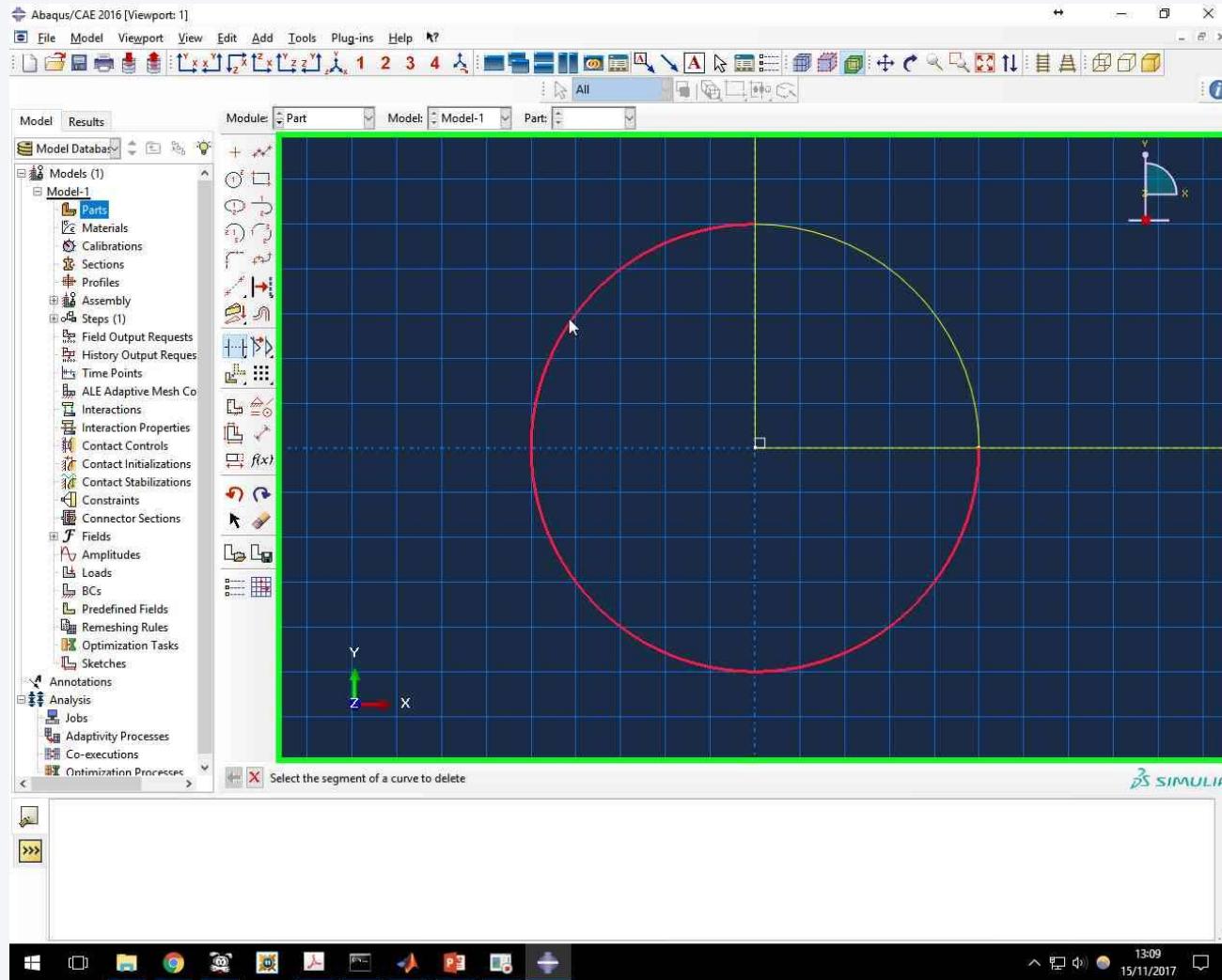
Sketch the rectangle. The corner positions can be defined by inserting the coordinates or by clicking in the corresponding positions.

Geometry



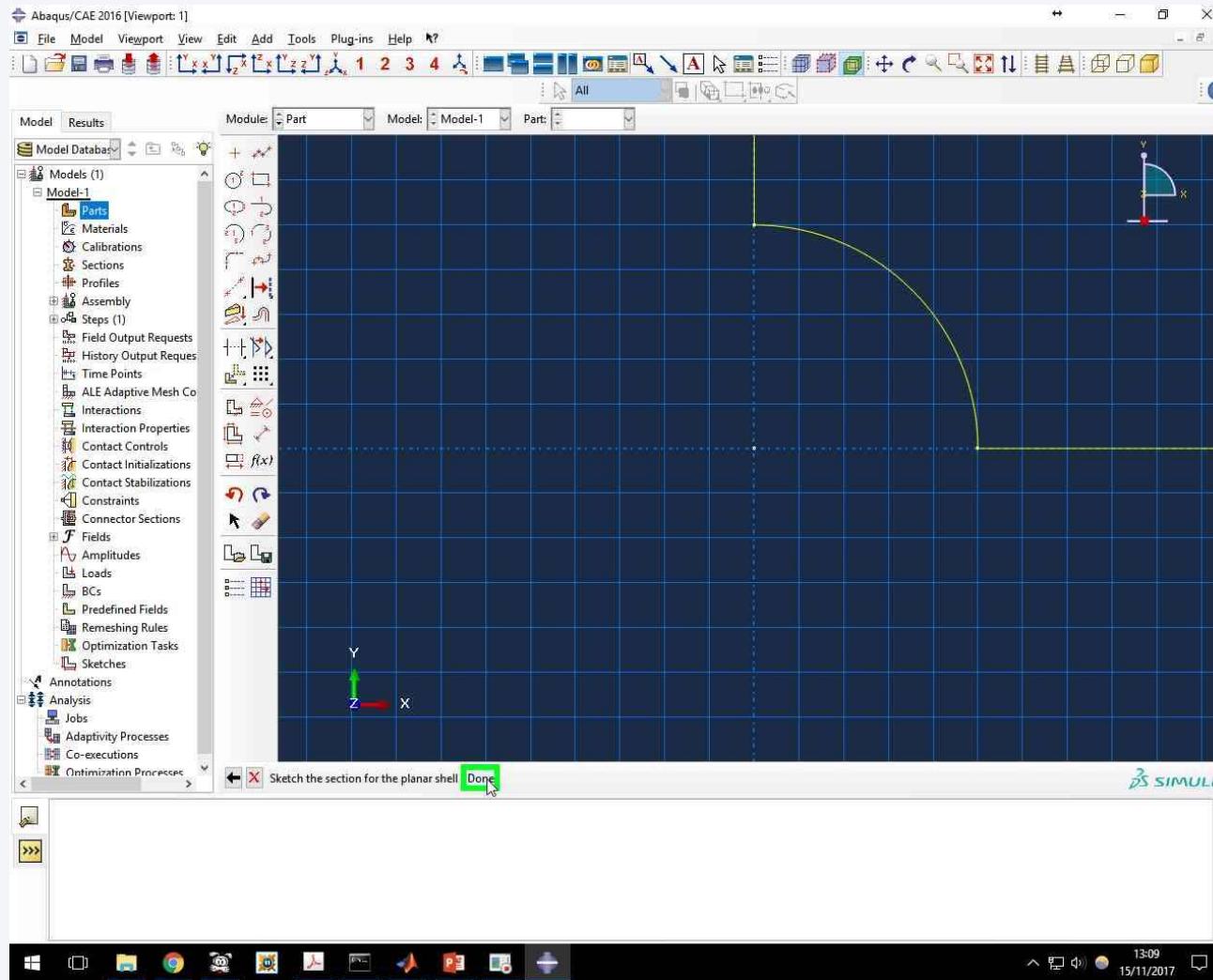
Sketch now the circle to define the hole.

Geometry



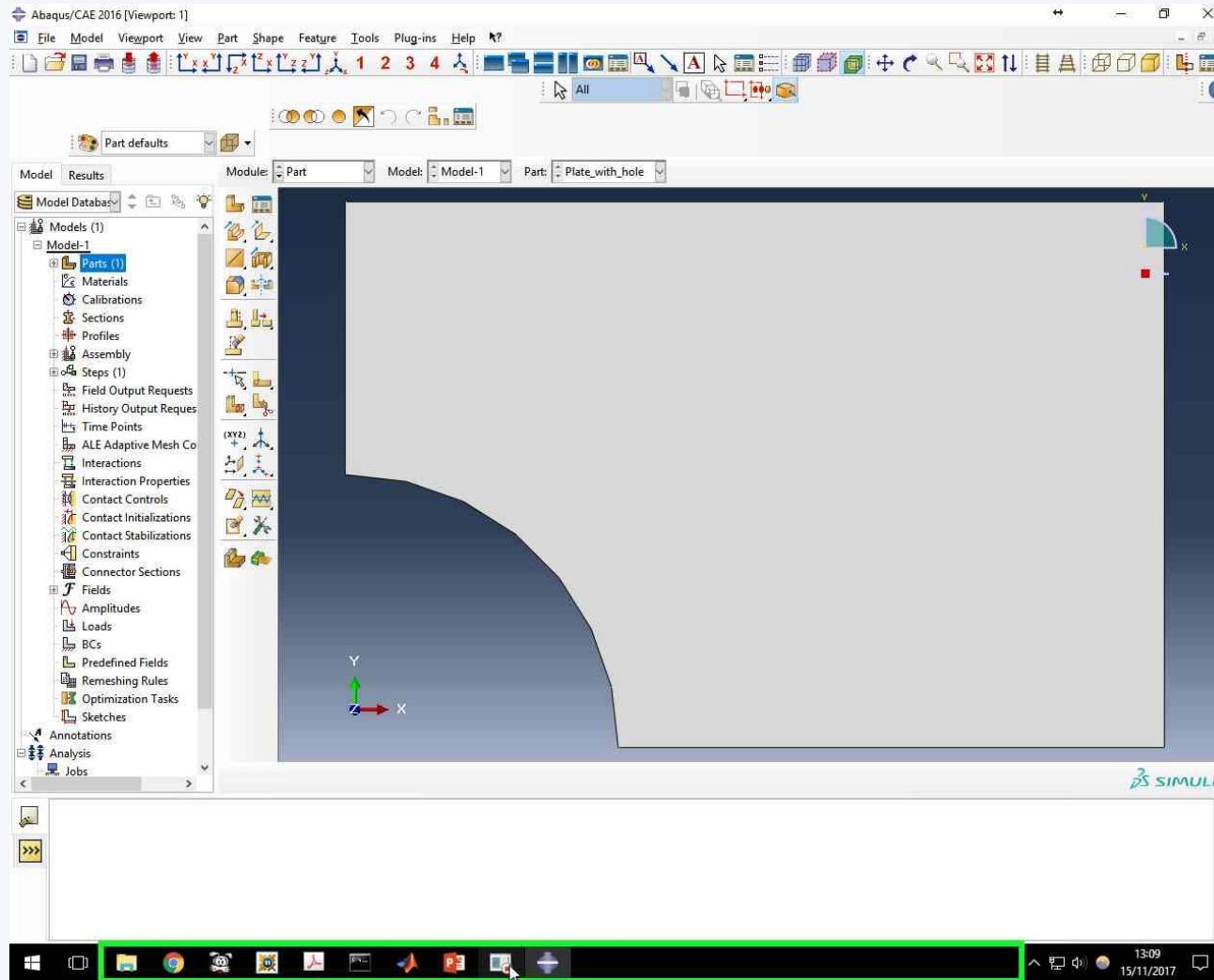
Use the auto-trim command to delete the outer portions of the circle.

Geometry



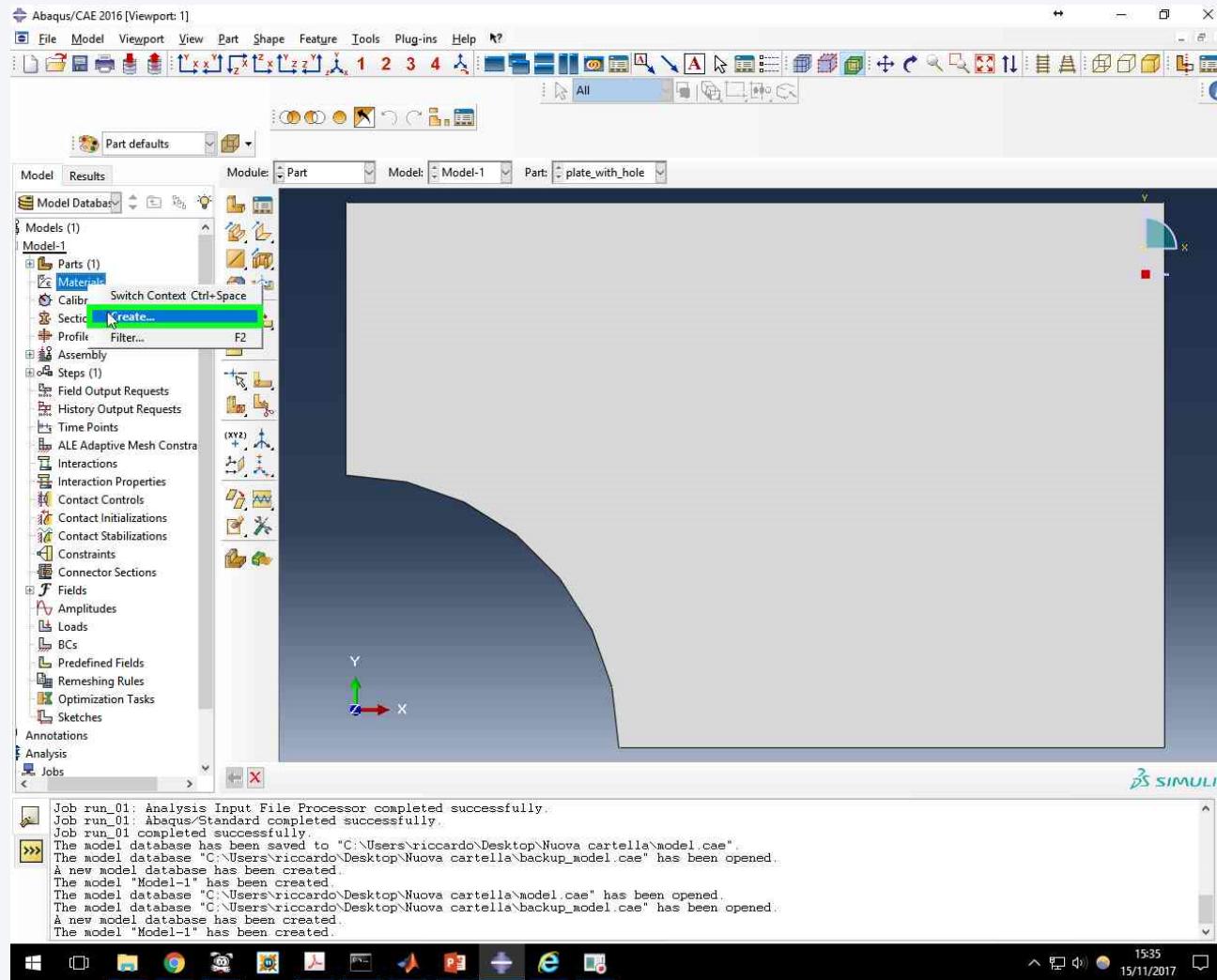
Confirm by clicking on Done.

Geometry



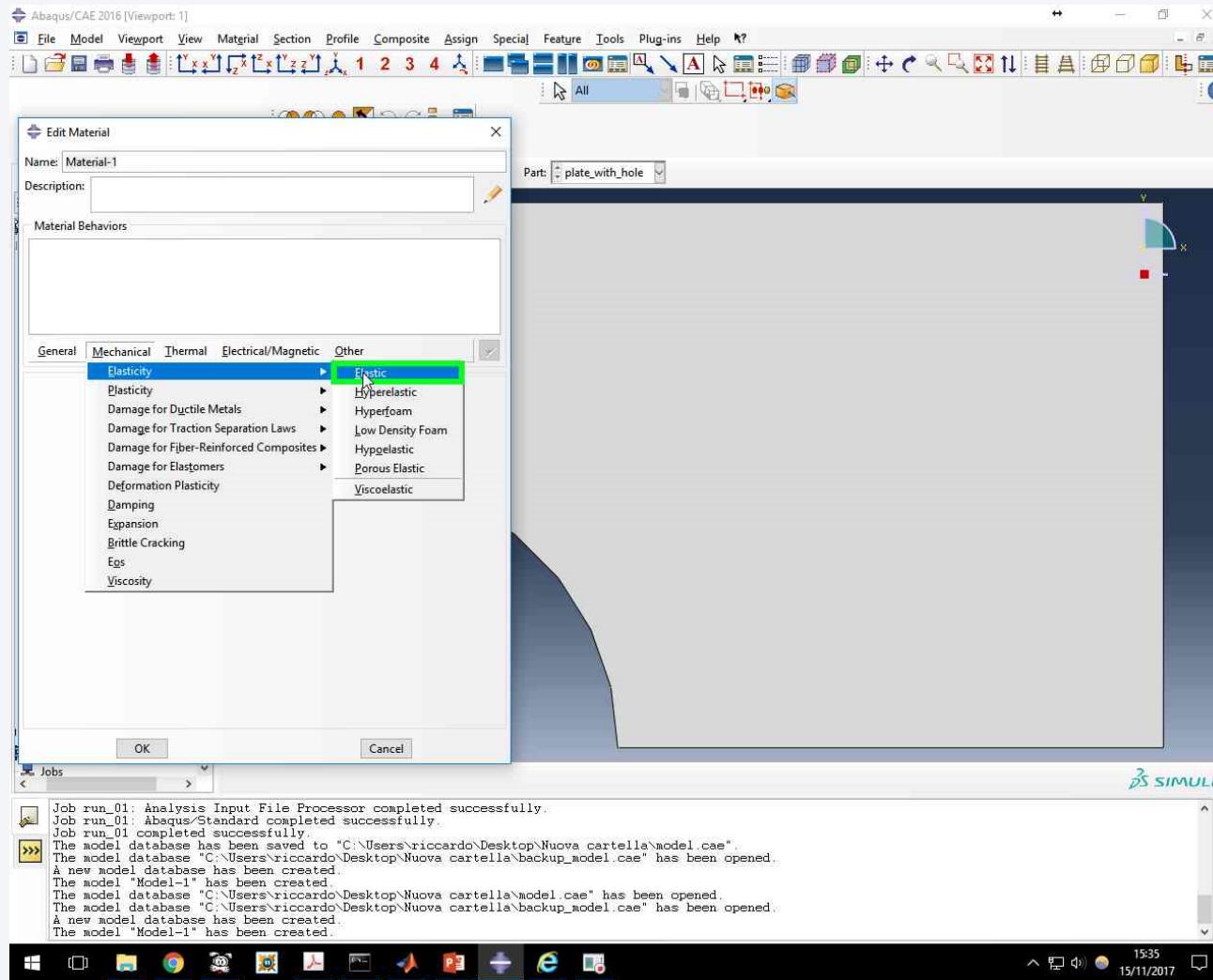
The Part should now appear as illustrated.

Material definition



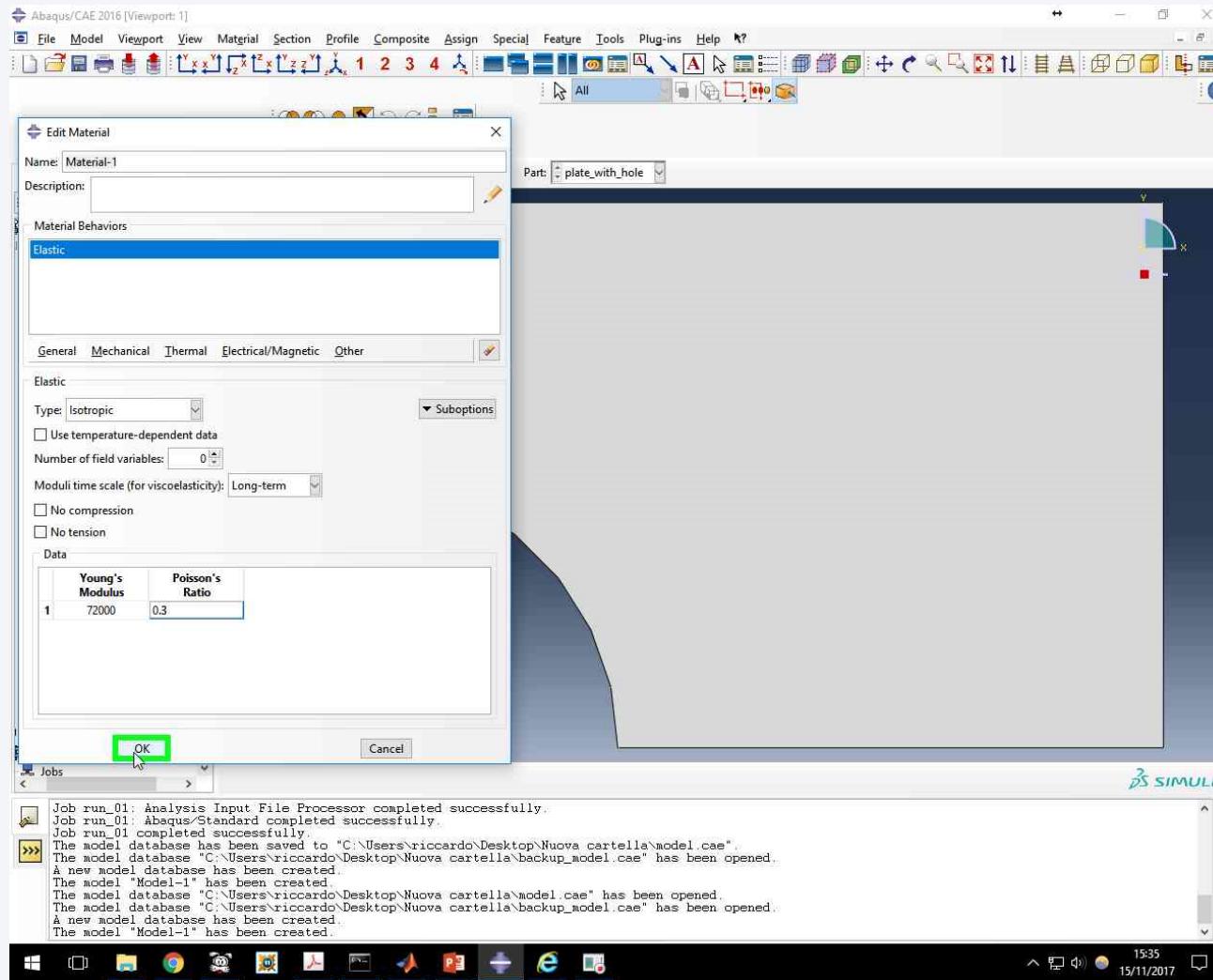
Create the material definition.

Material definition



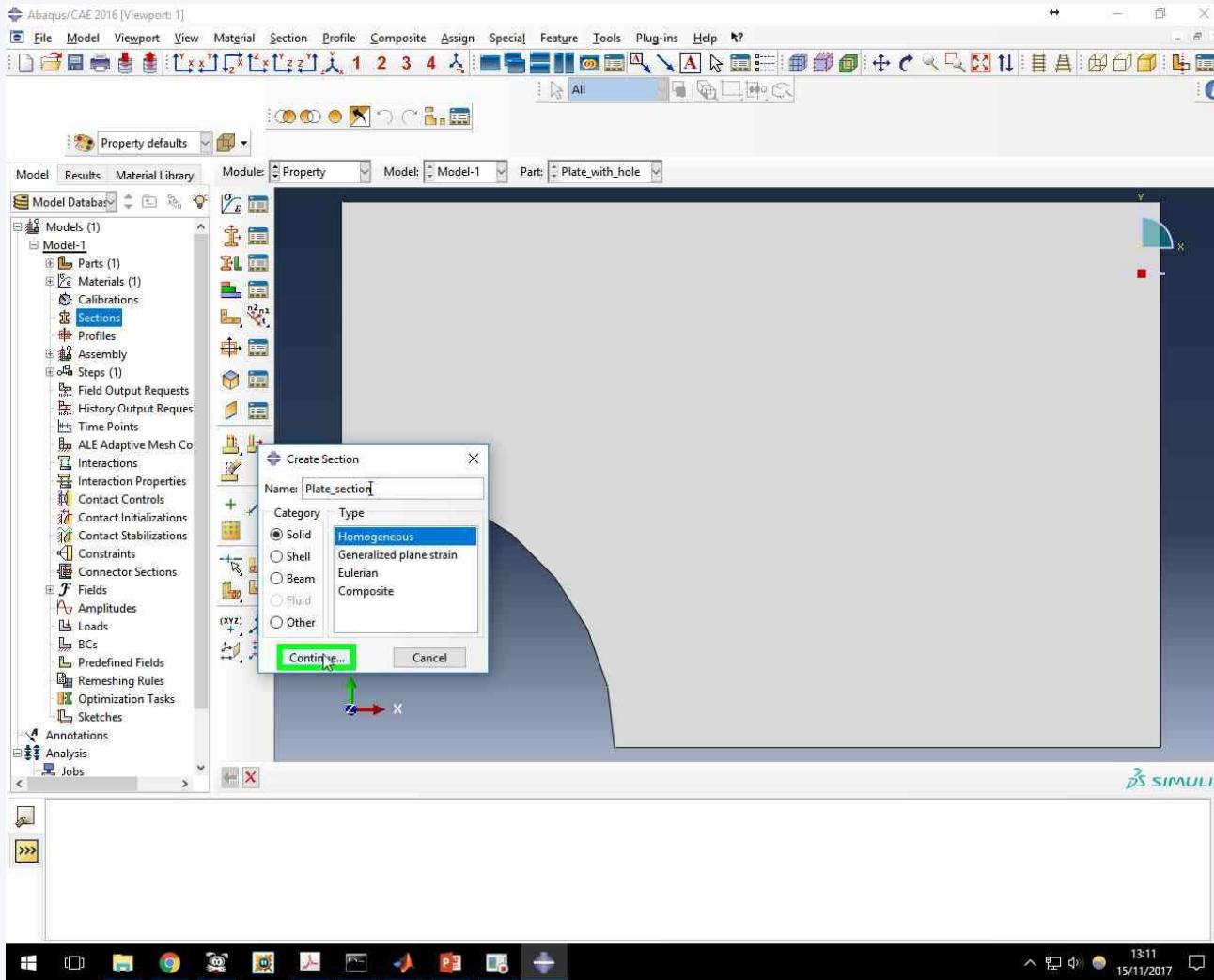
In this case a linear elastic material is considered.

Material definition



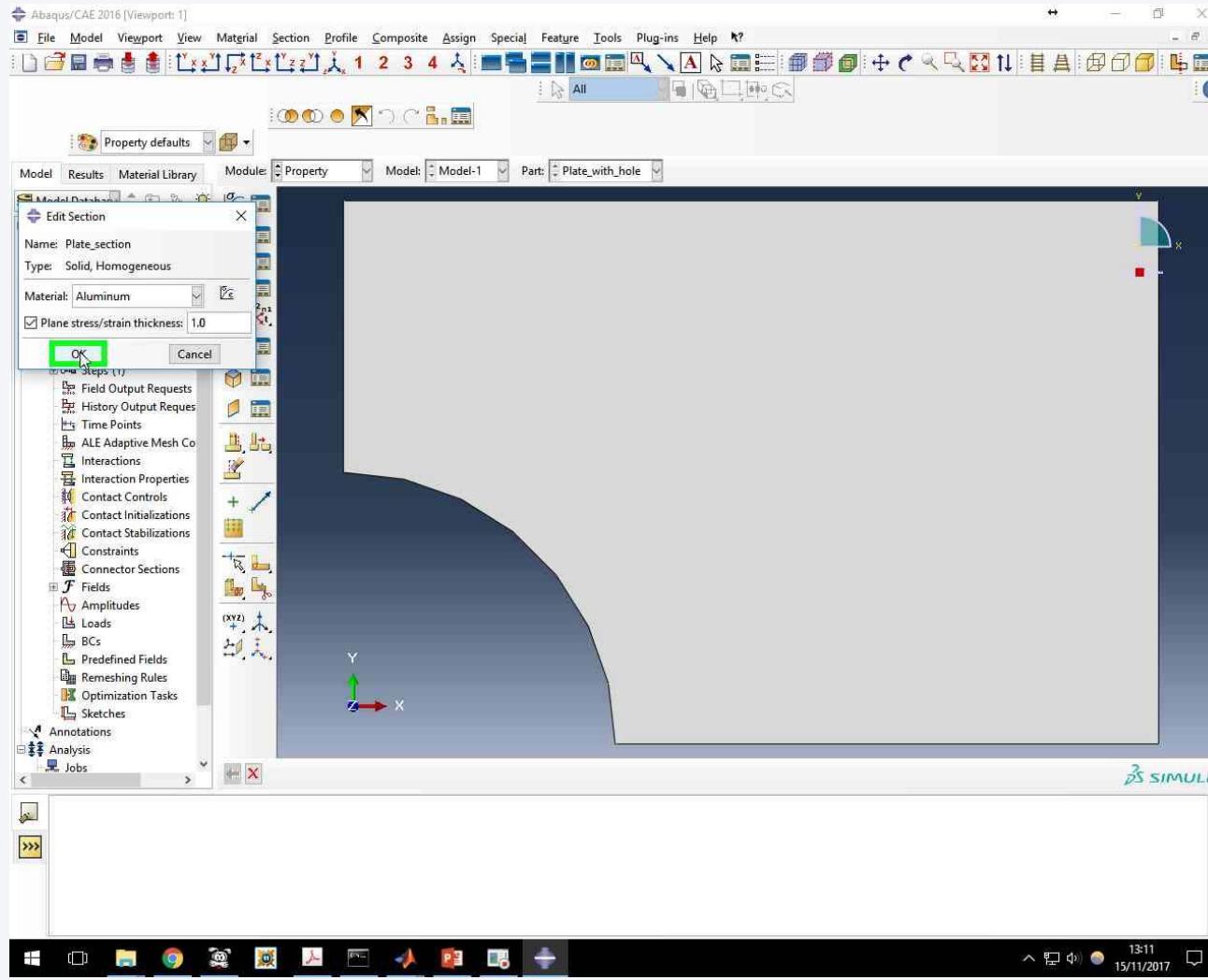
Define the Young's modulus and Poissons's ratios.

Section properties



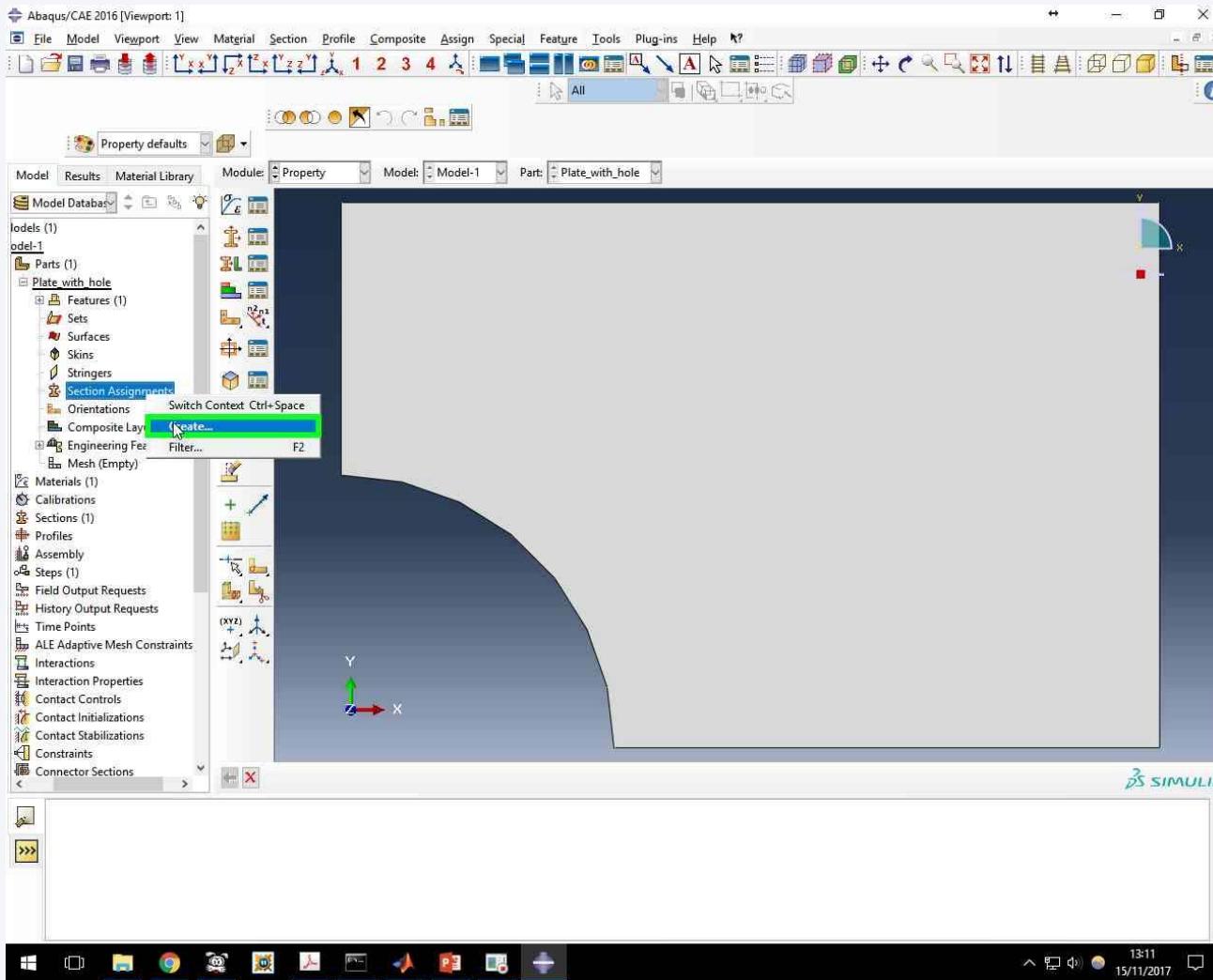
Create now the Section properties. The section, in this case, is used to specify the thickness of the membrane (recall that the membrane model is given by a 2D surface, thus the thickness should be defined to calculate the membrane stiffness).

Section properties



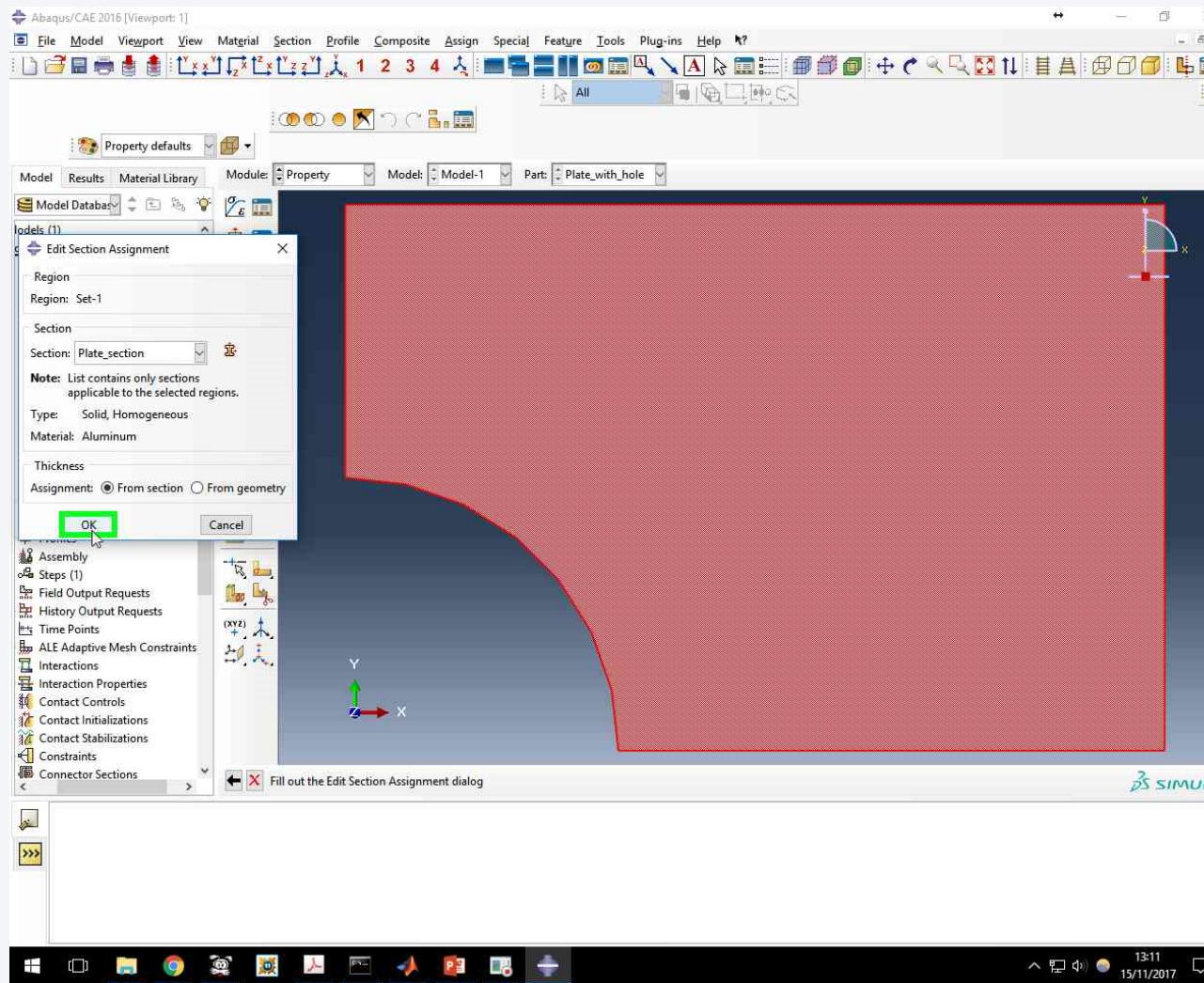
Select the material and confirm.

Section properties



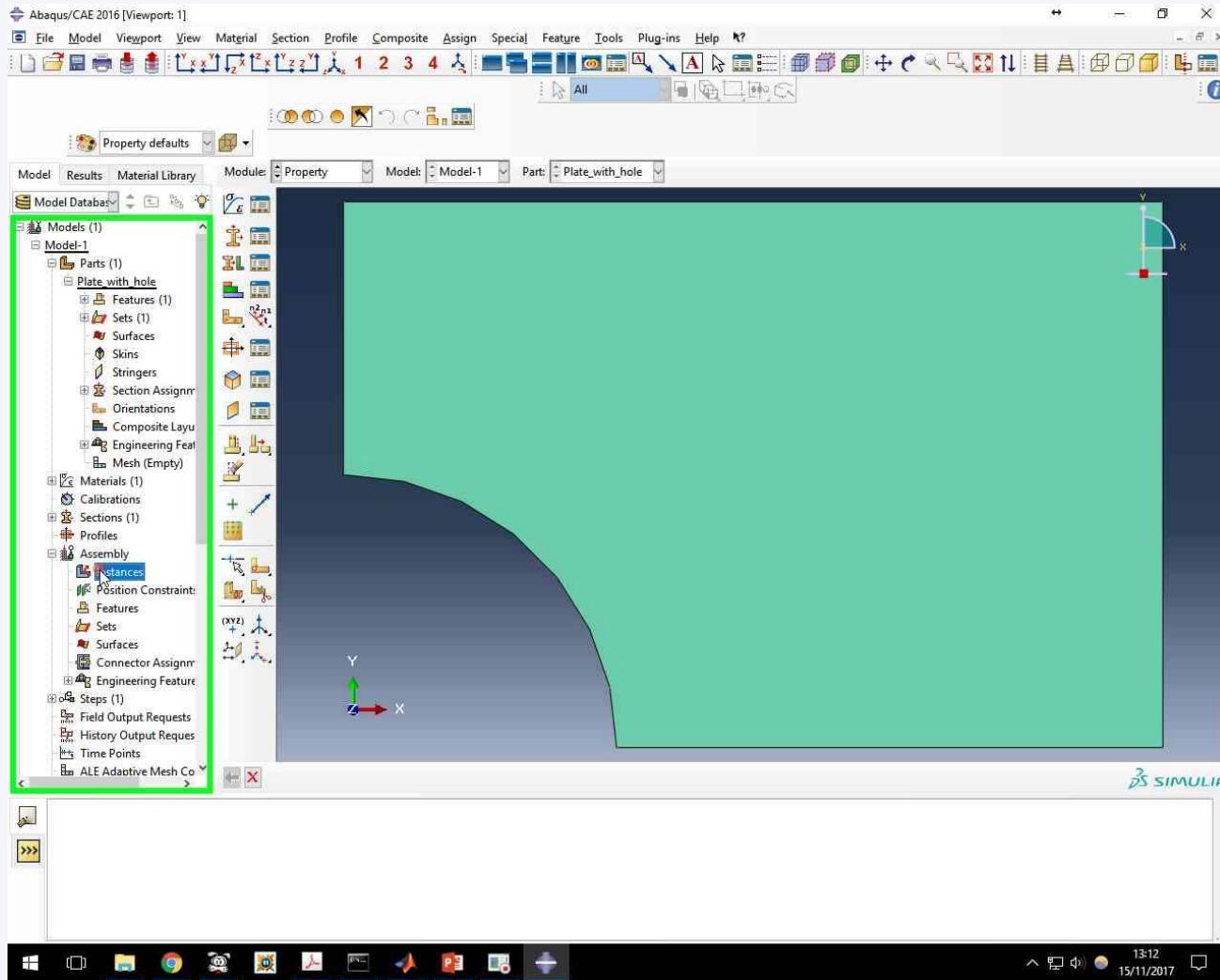
The section previously defined is now assigned to the part. In other words, it is specified that the geometric entity previously sketched is made of Aluminum and has thickness equal to 1.0.

Section properties



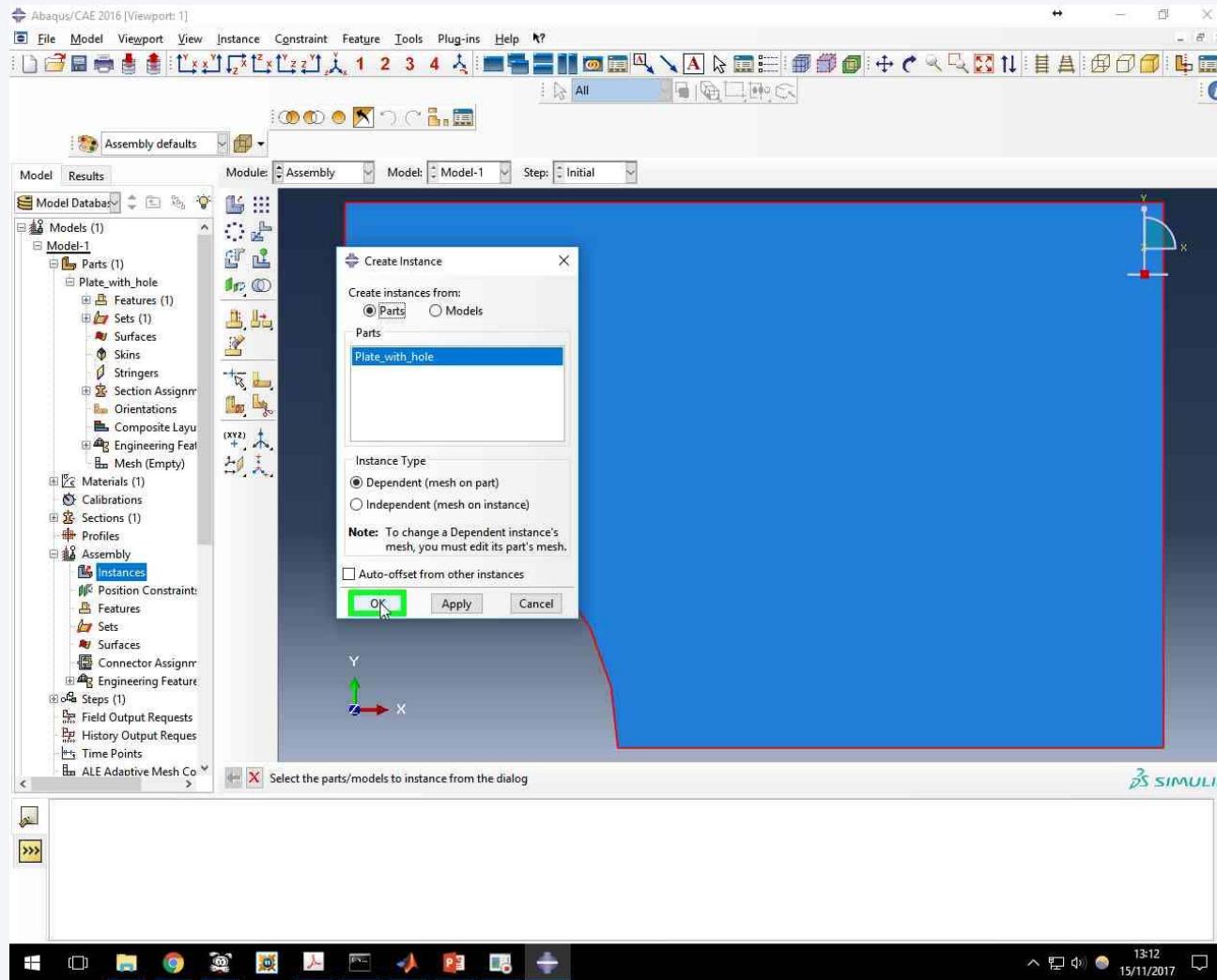
Confirm.

Creating an instance

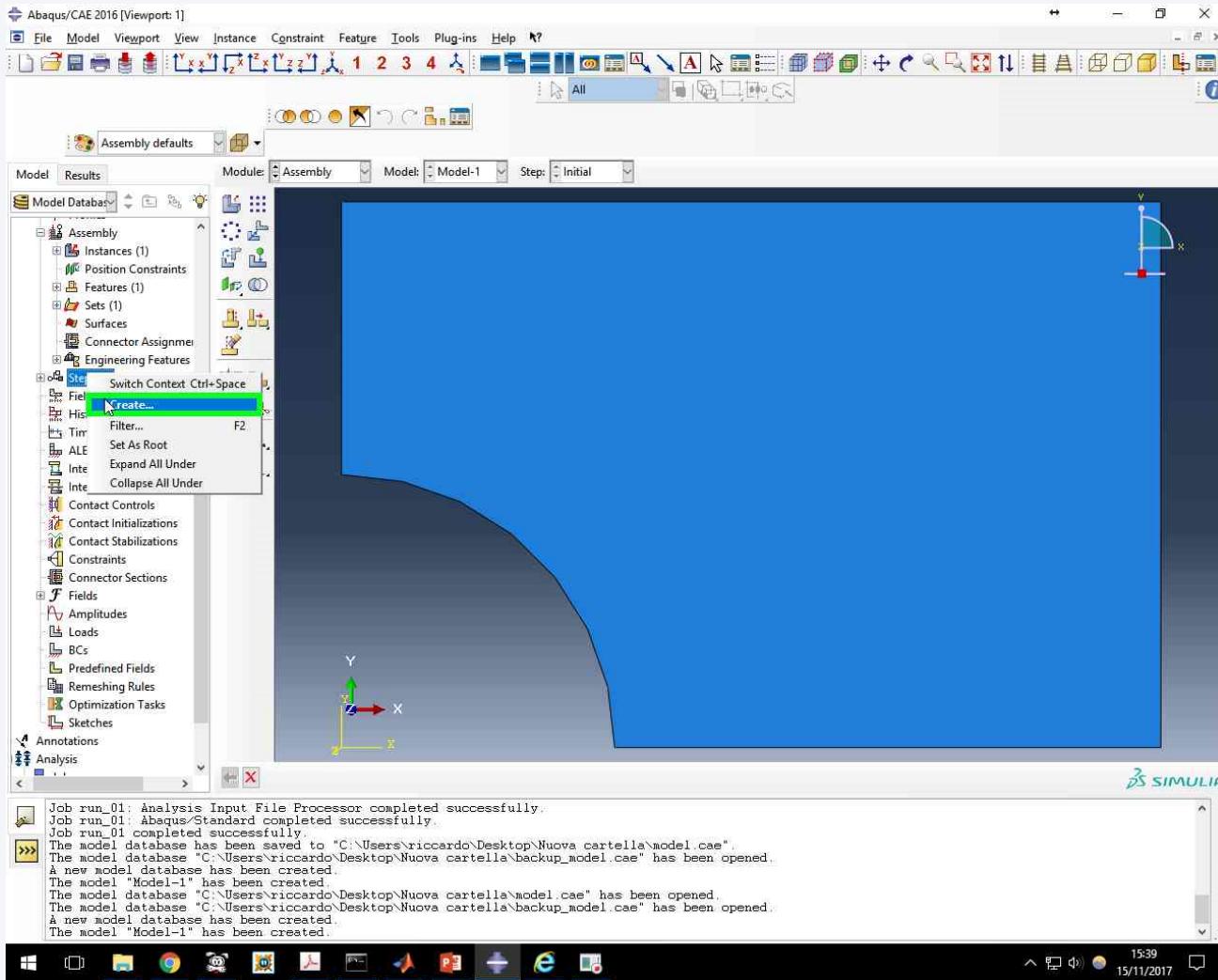


To proceed with the generation of the model it is necessary to create an instance. Right-click on Instance.

Creating an instance

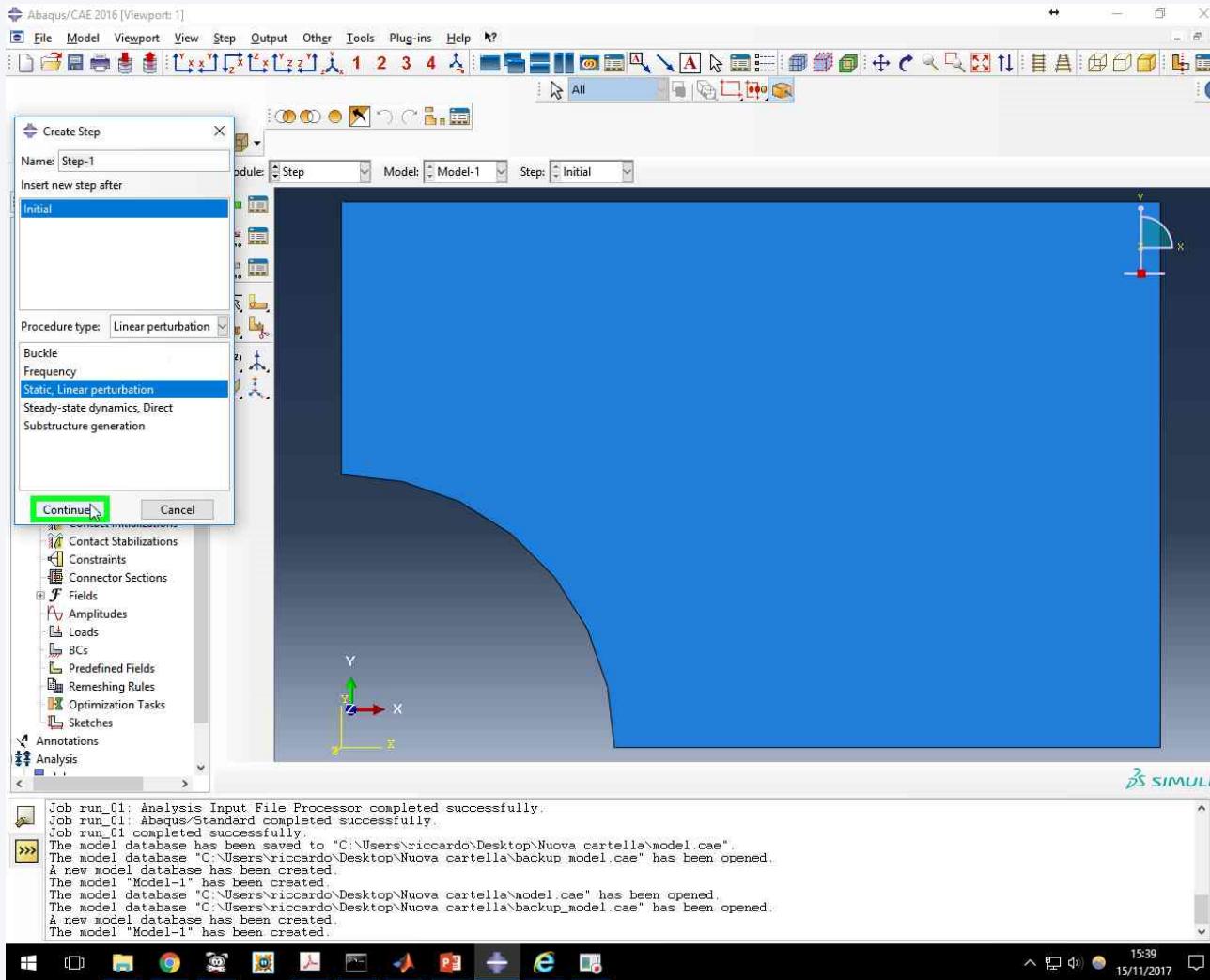


Step definition



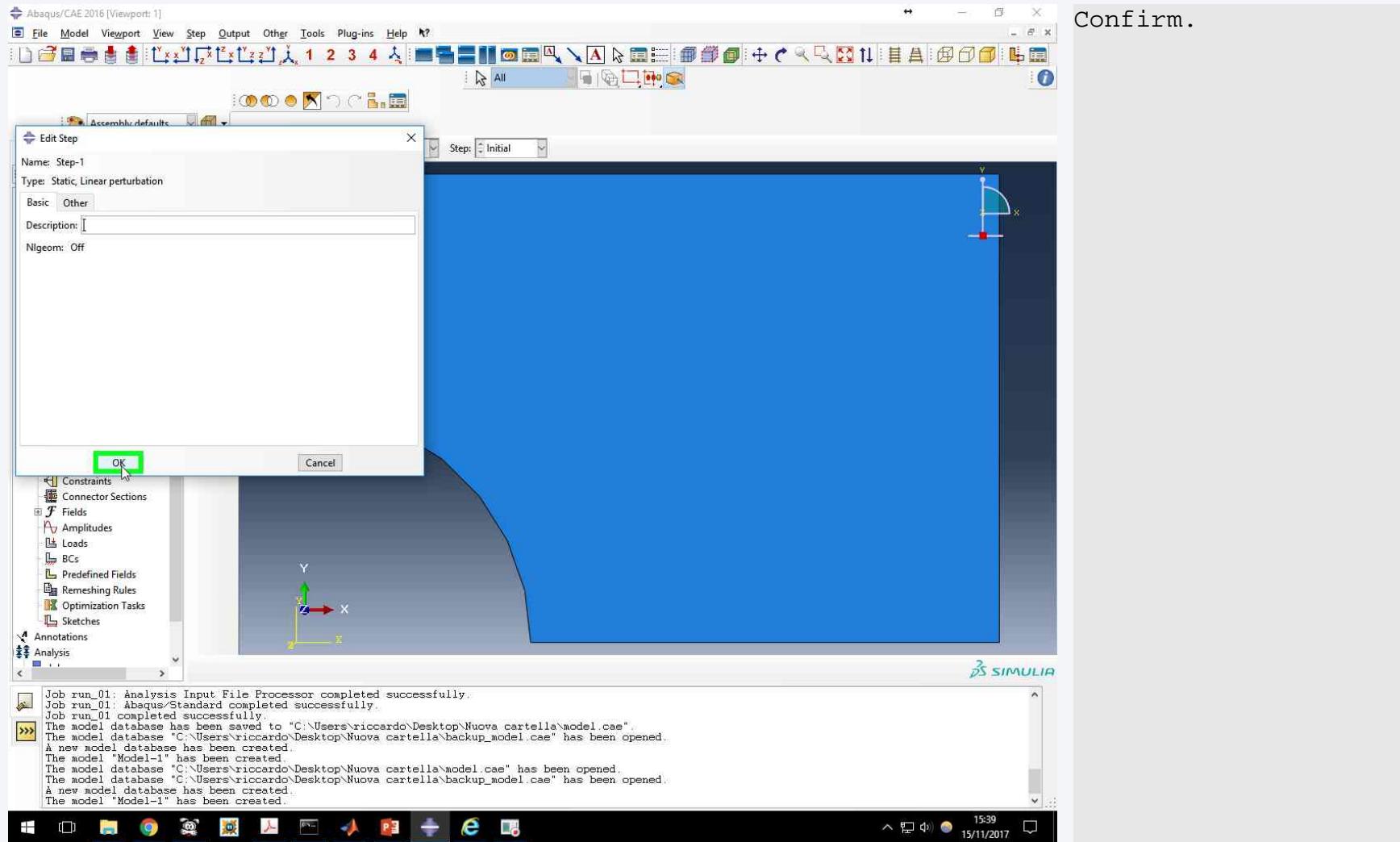
The Step defines the analysis procedure (in this case linear static). In principle, one single analysis can be composed of several steps. For instance, the initial part of the analysis could be a thermal step, and a subsequent step can be the introduction of a mechanical load. In this case one single step is considered.

Step definition

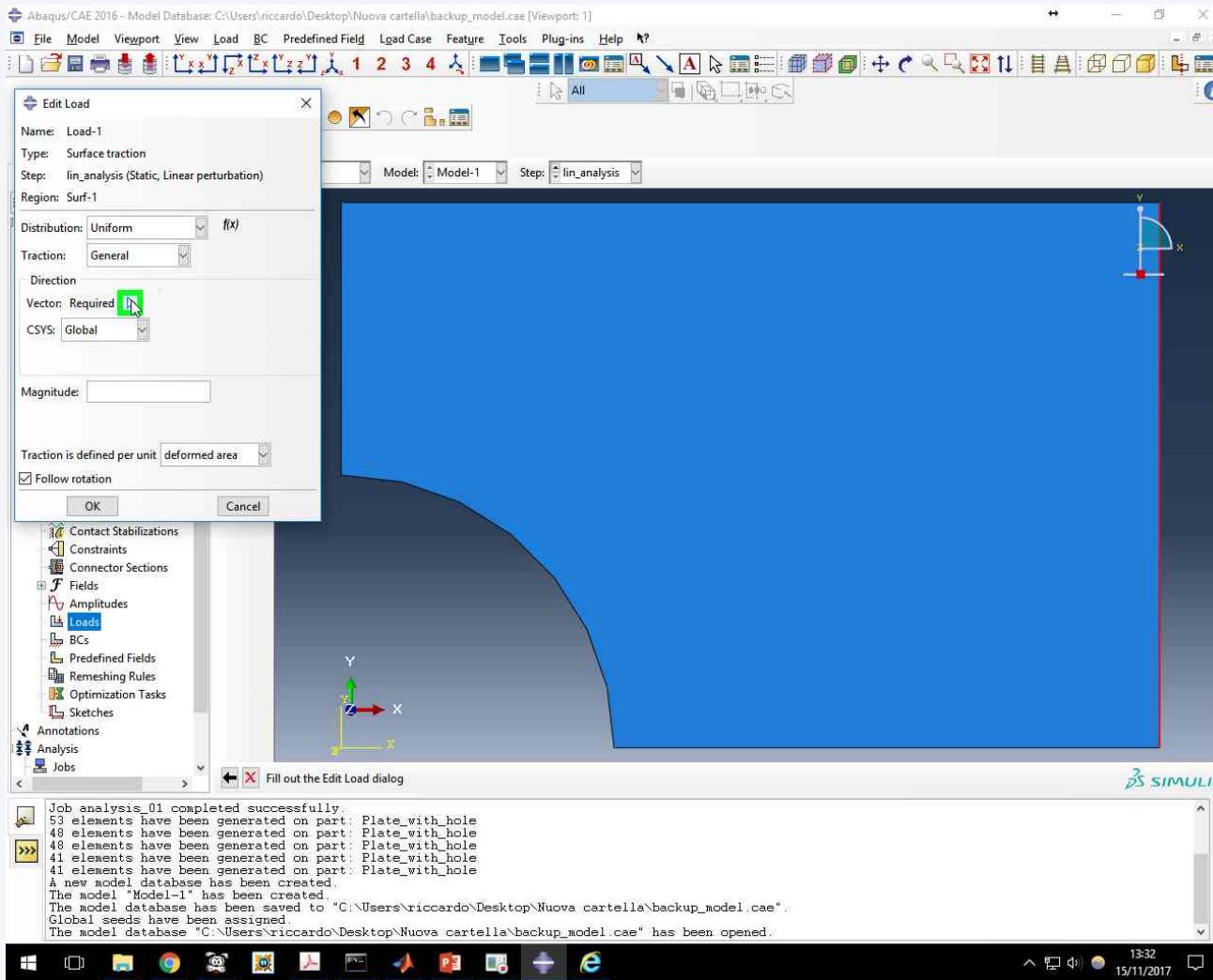


The analysis type is selected among the linear perturbation ones.

Step definition



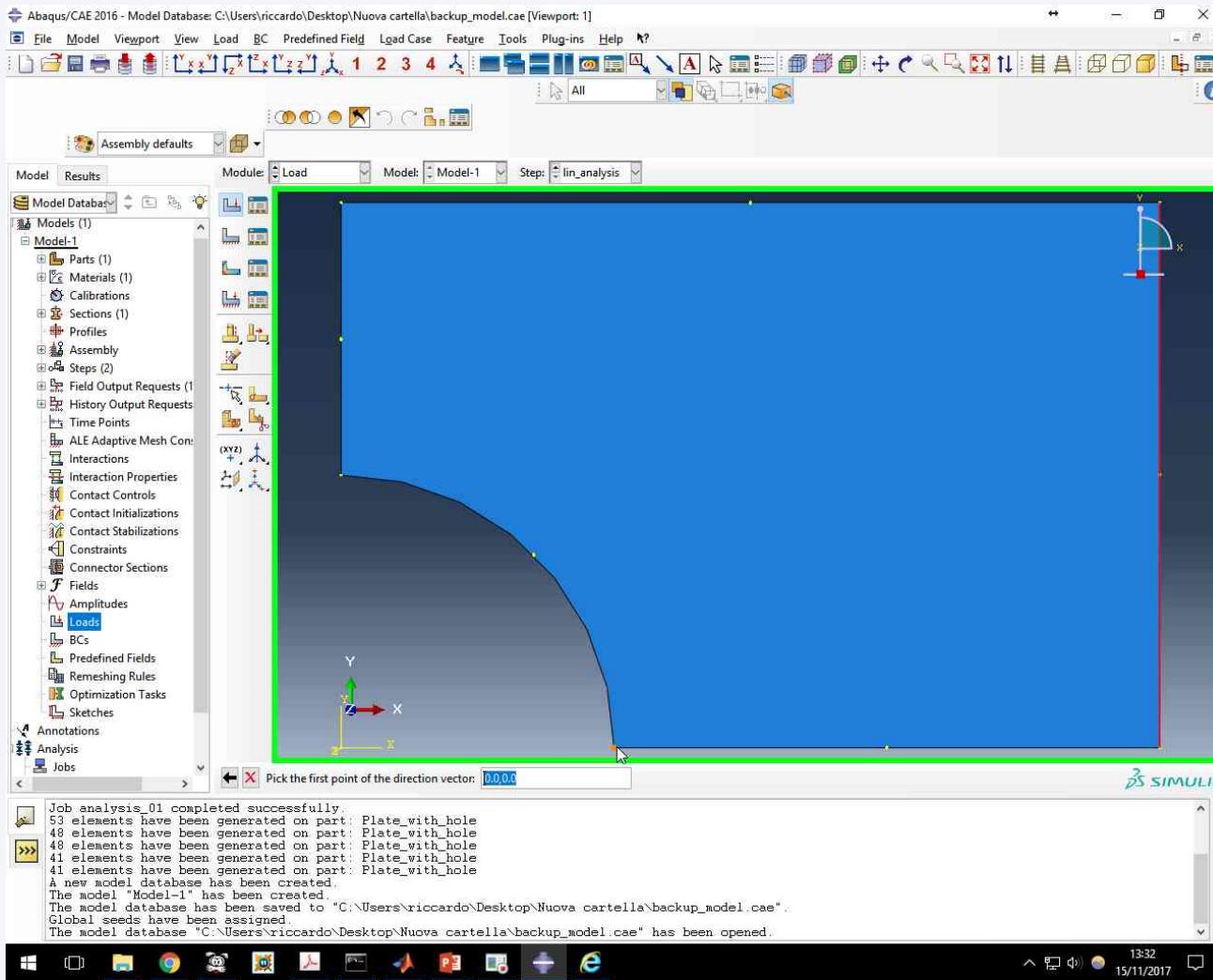
Application of loads



Right click on loads and specify the loading condition. In this case the load is introduced as a distributed force, denoted as "Surface traction".

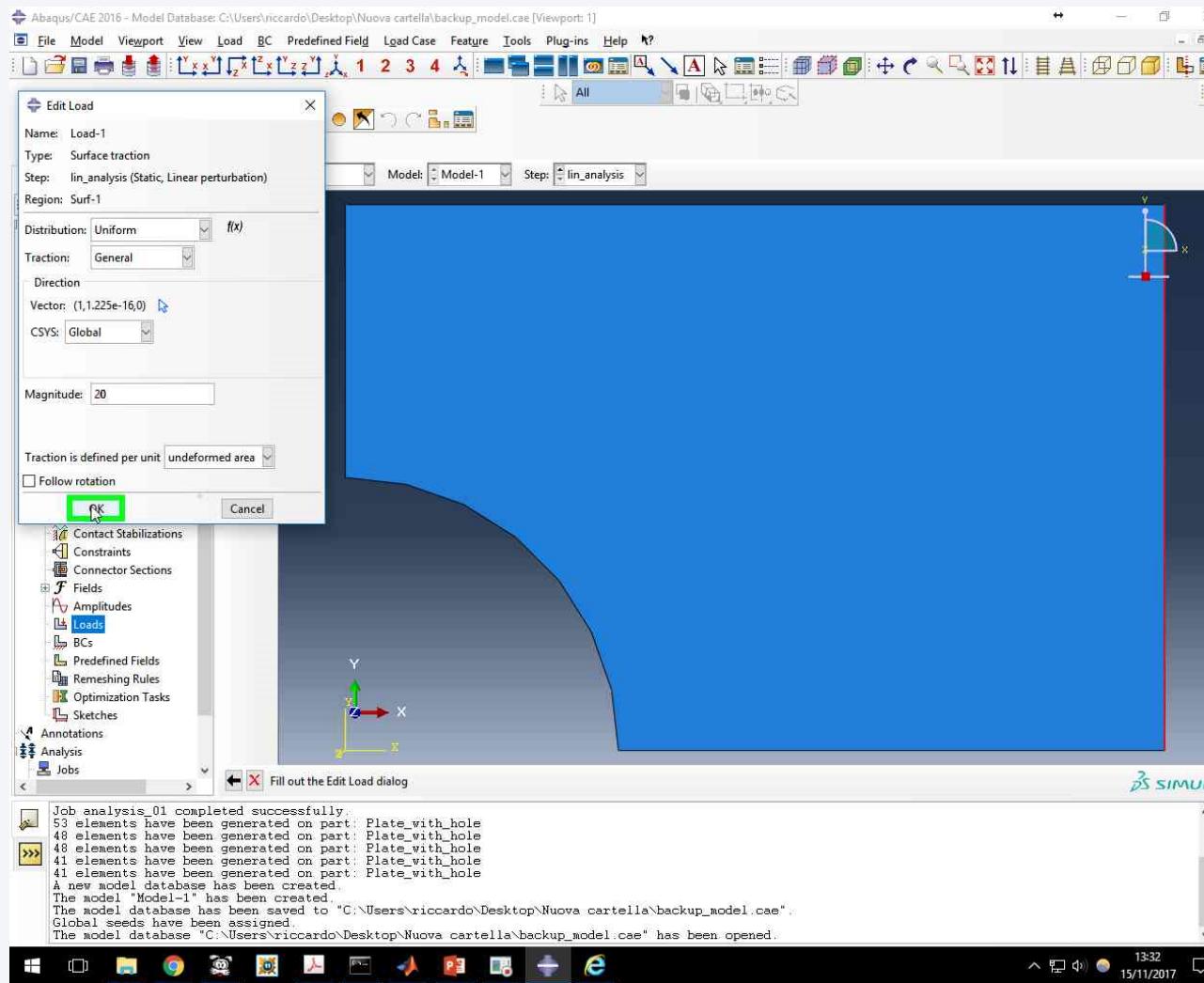
Important remark: the (many) features to introduce the distributed loads are a user-friendly way for quickly specifying the loading conditions. Indeed the user is not required to evaluate by hand the consistent nodal forces. The conversion is done by software, but the final result is always in the form of nodal forces.

Application of loads



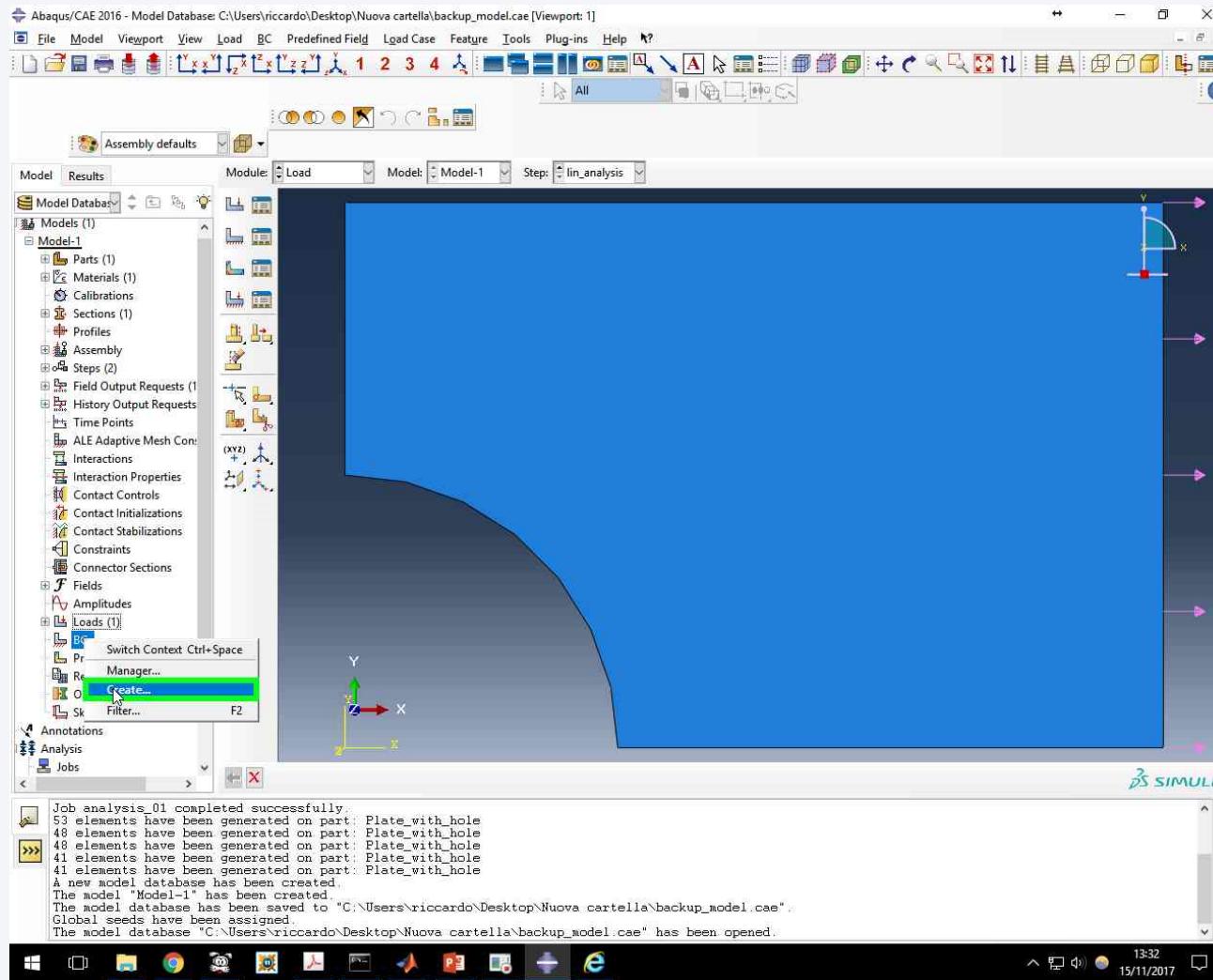
In this case, a vector describing the direction of the distributed force has to be specified. This is done by specifying the coordinates of the vector or by clicking over two points to define the vector.

Application of loads



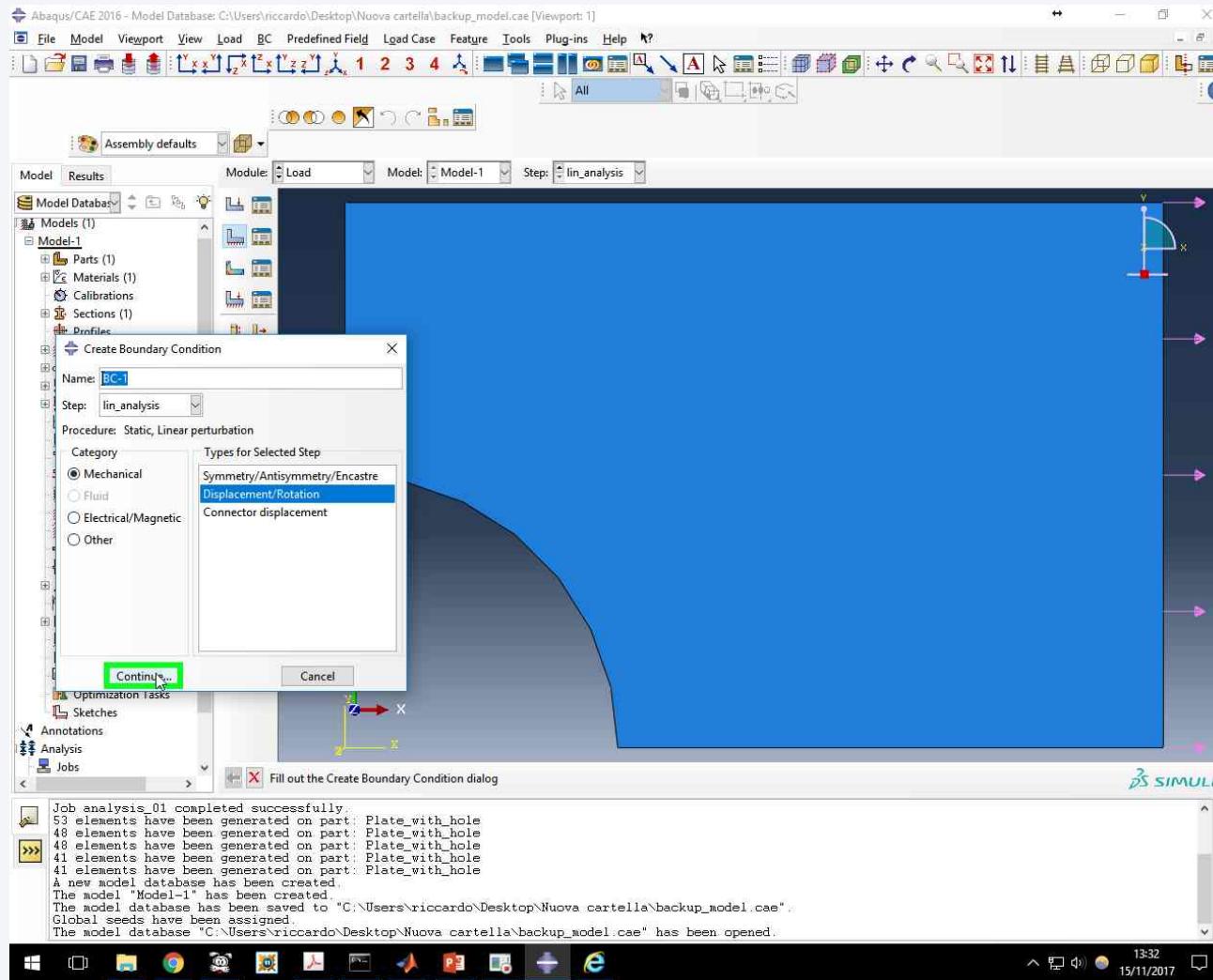
Confirm.

Definition of boundary conditions



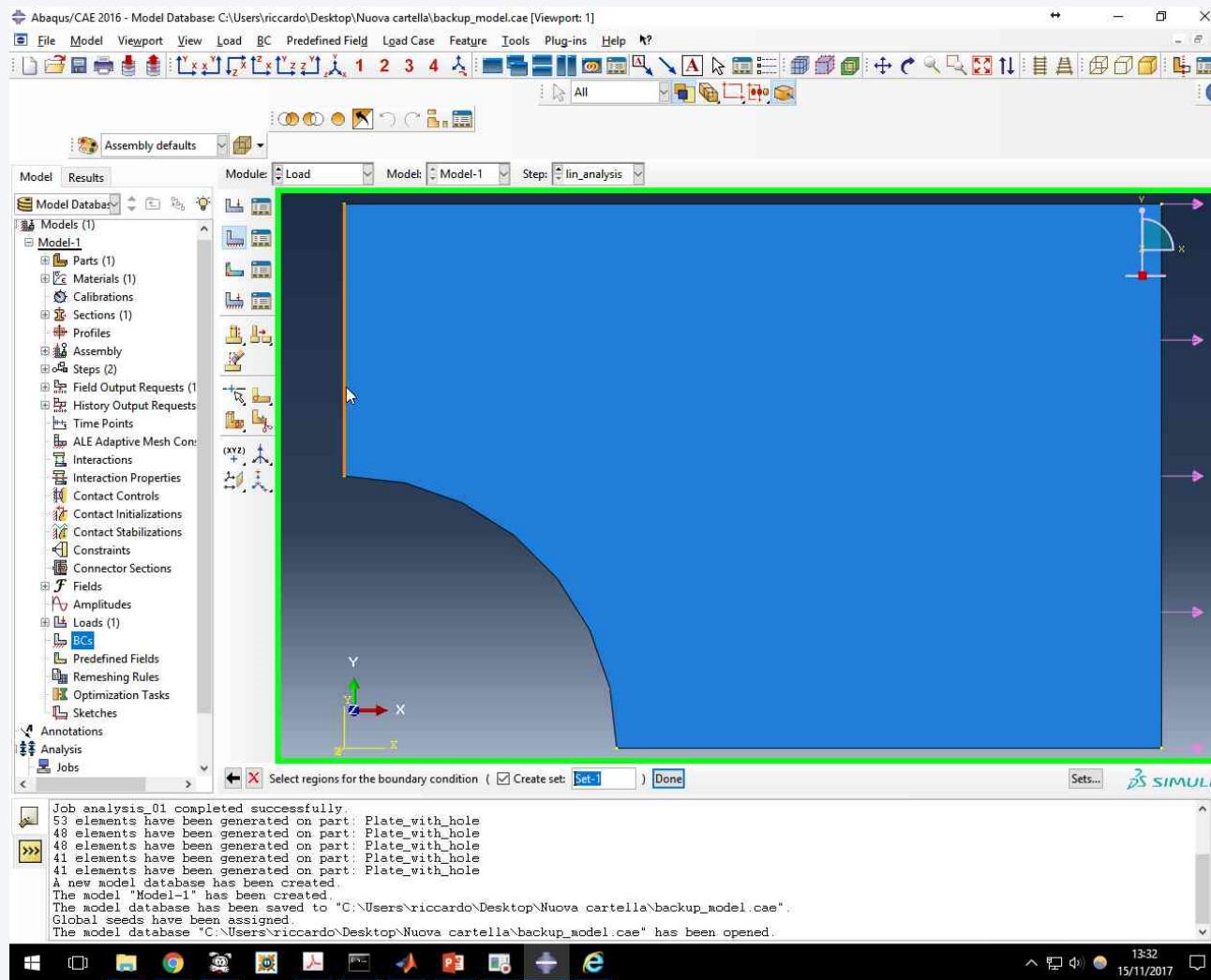
It is now necessary to specify the boundary conditions.

Definition of boundary conditions



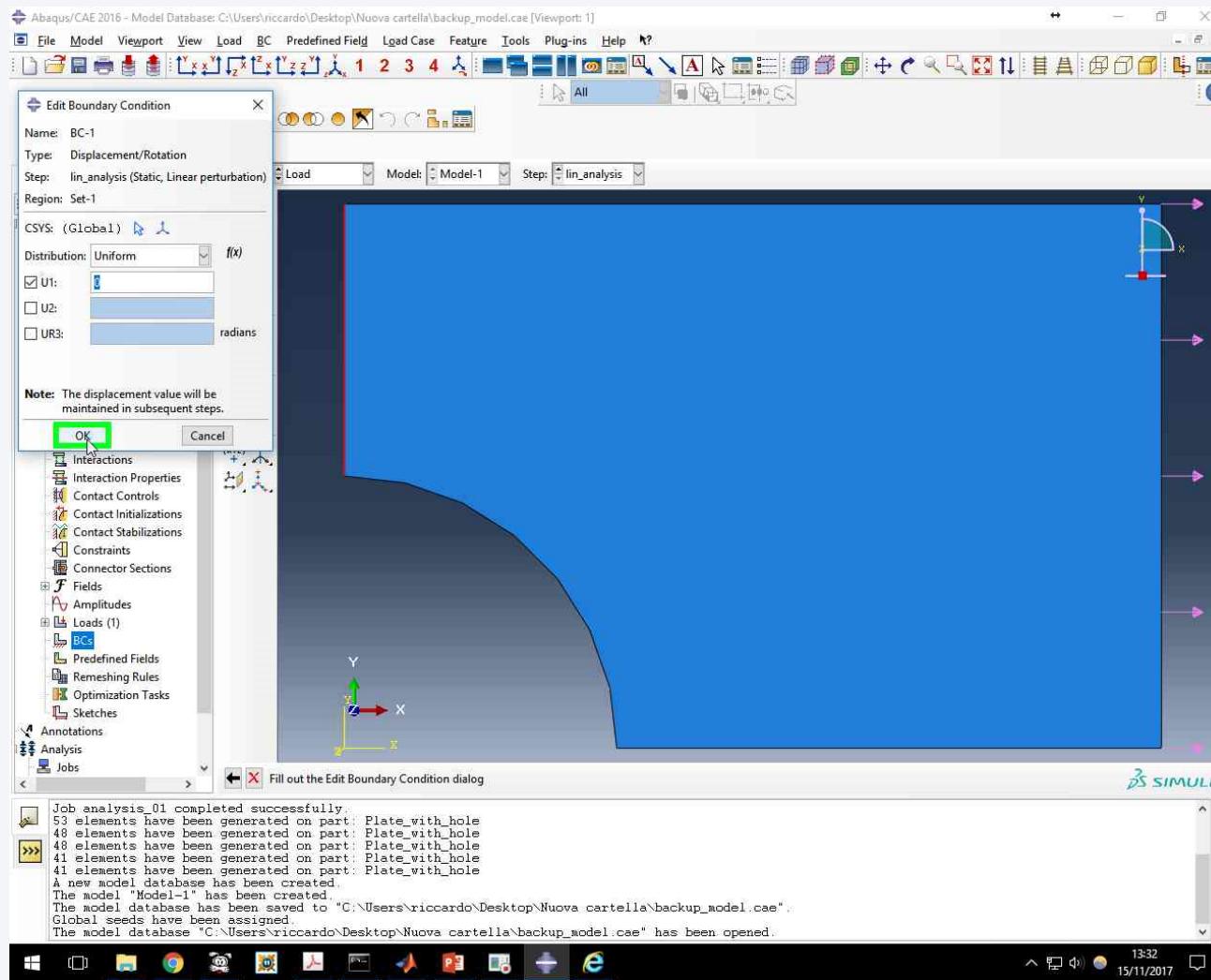
Select conditions of mechanical type and "Displacement/Rotatio n"

Definition of boundary conditions



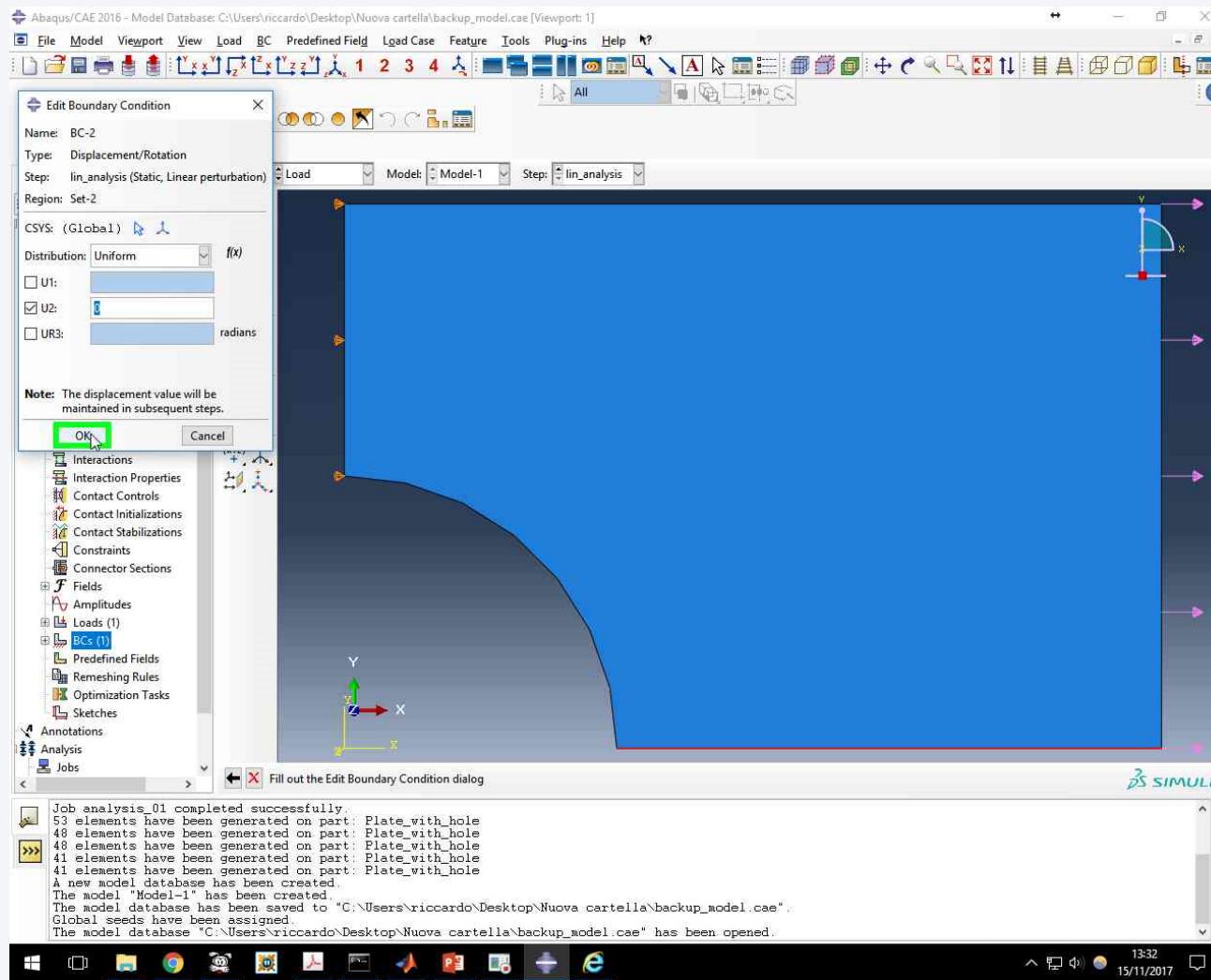
Confirm and select the first edge to be constrained.

Definition of boundary conditions



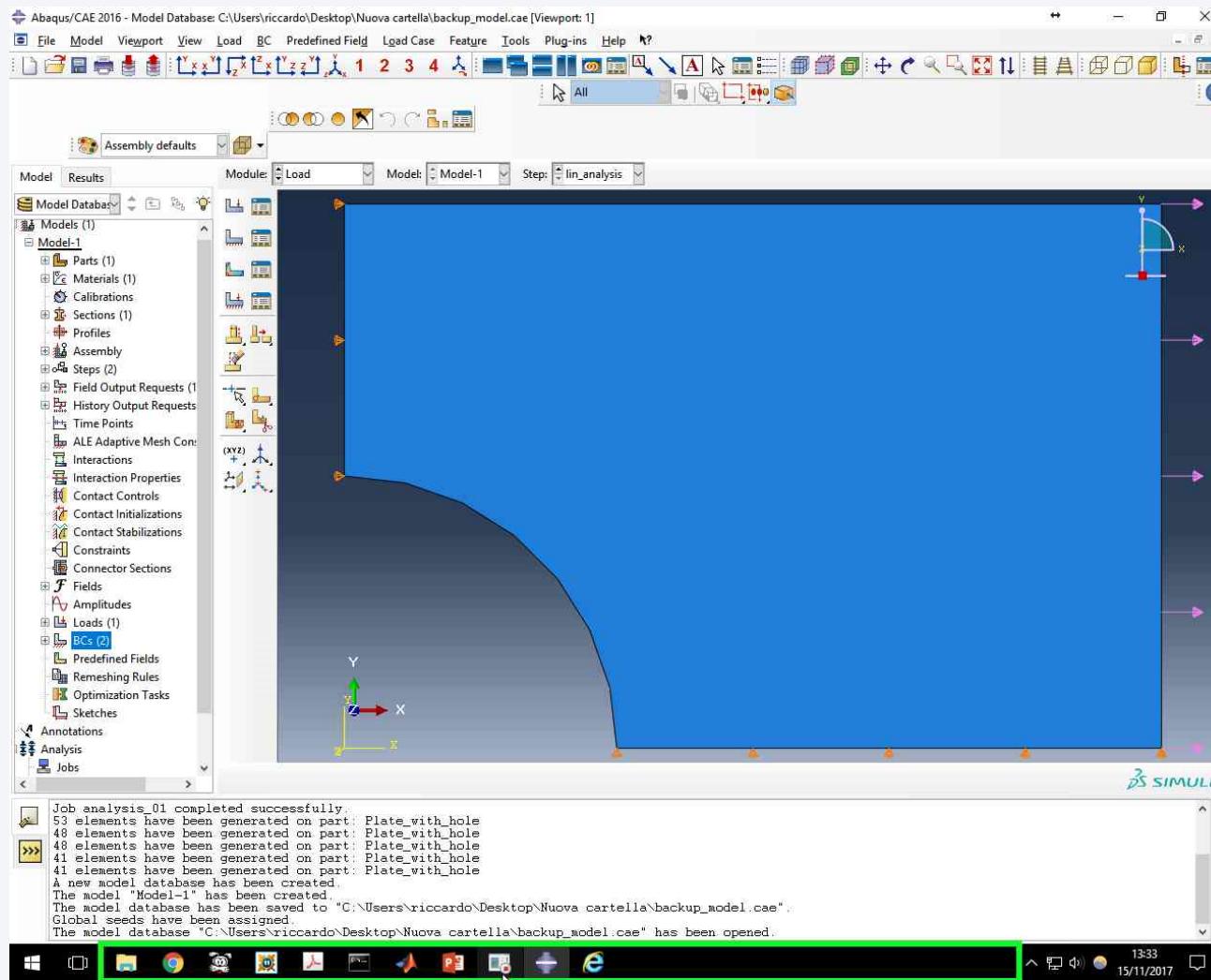
Confirm and specify the components of the displacement to be constrained.

Definition of boundary conditions



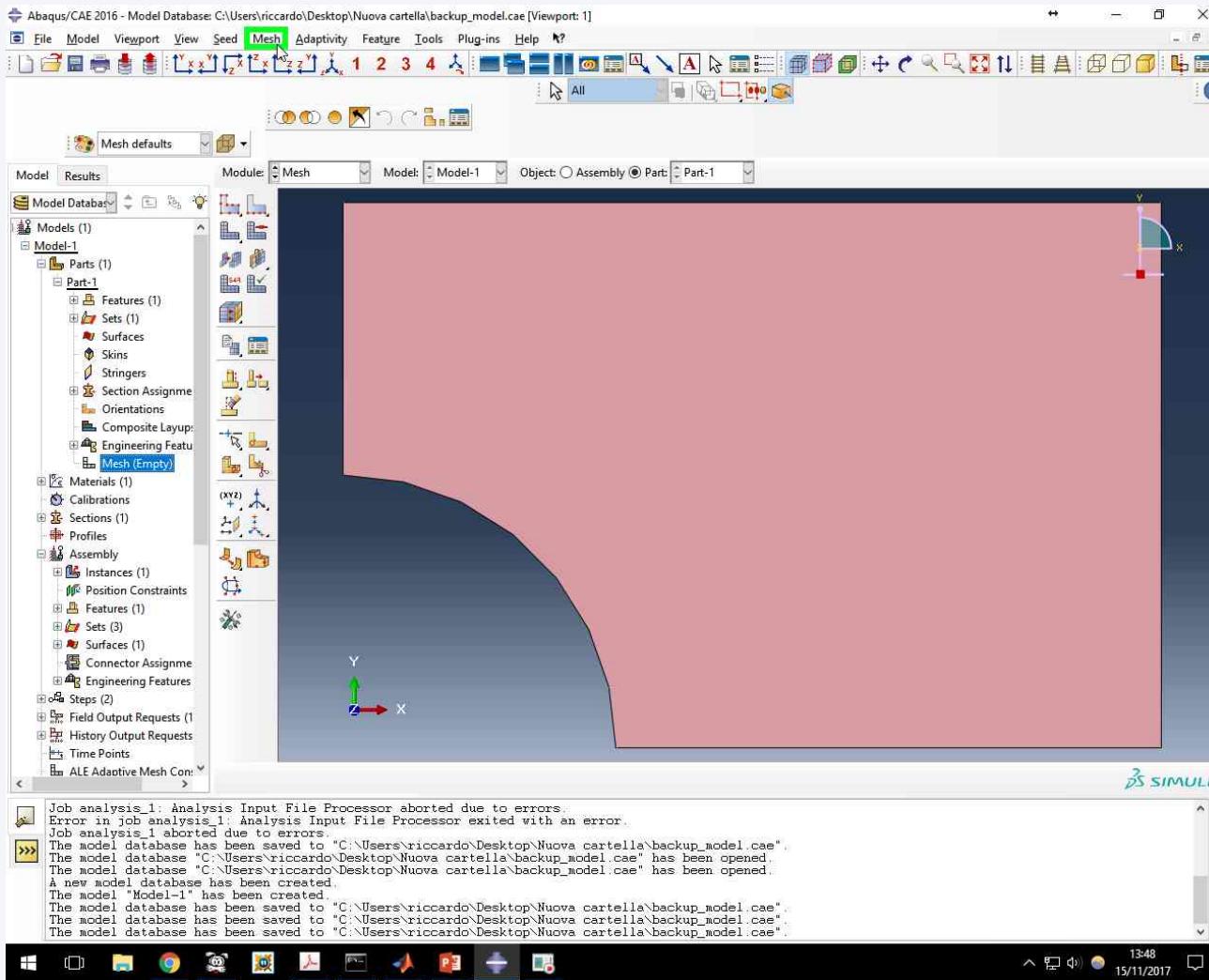
Repeat the same steps for the second edge to be constrained.

Definition of boundary conditions



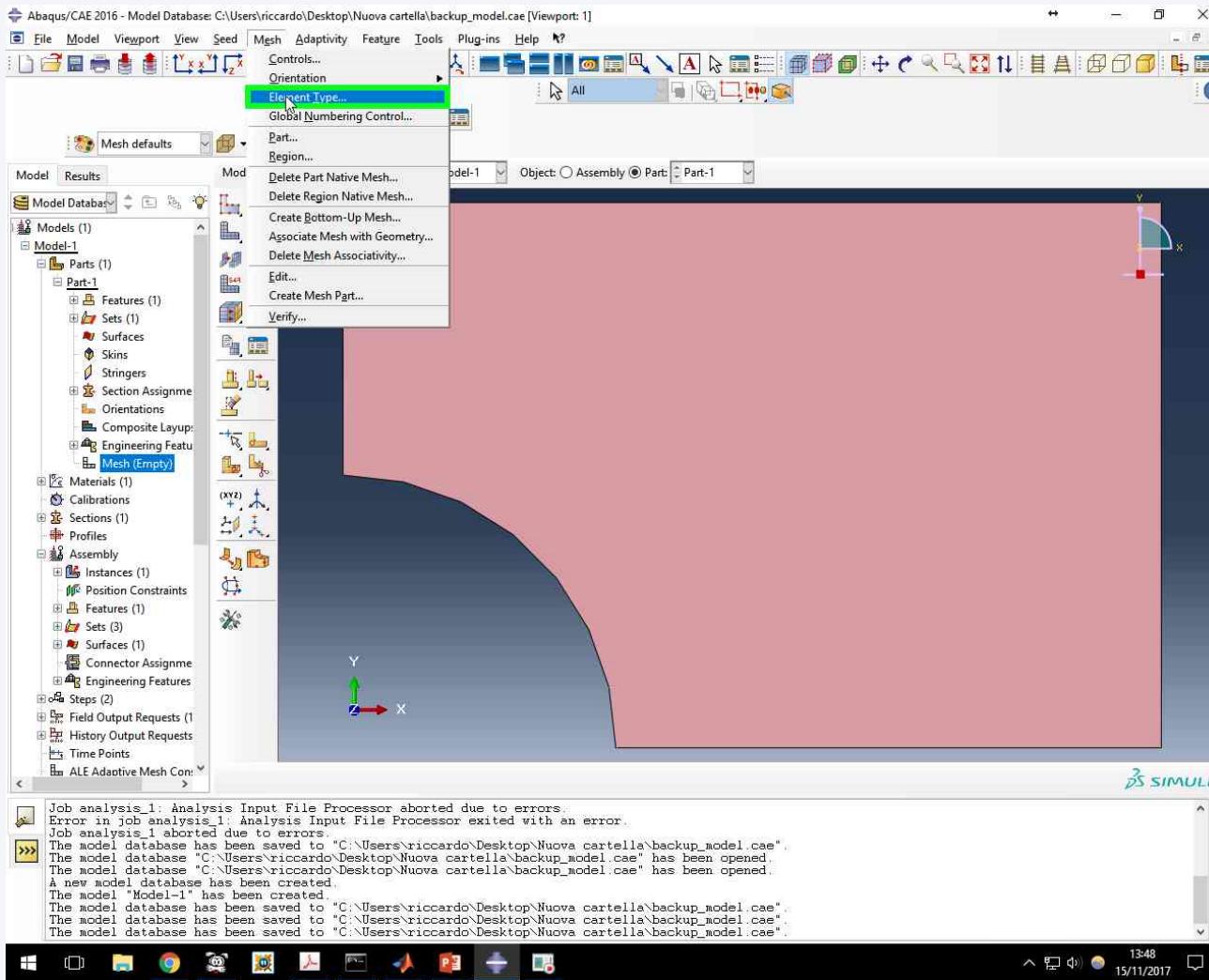
The model should now appear as reported.

Creating the mesh



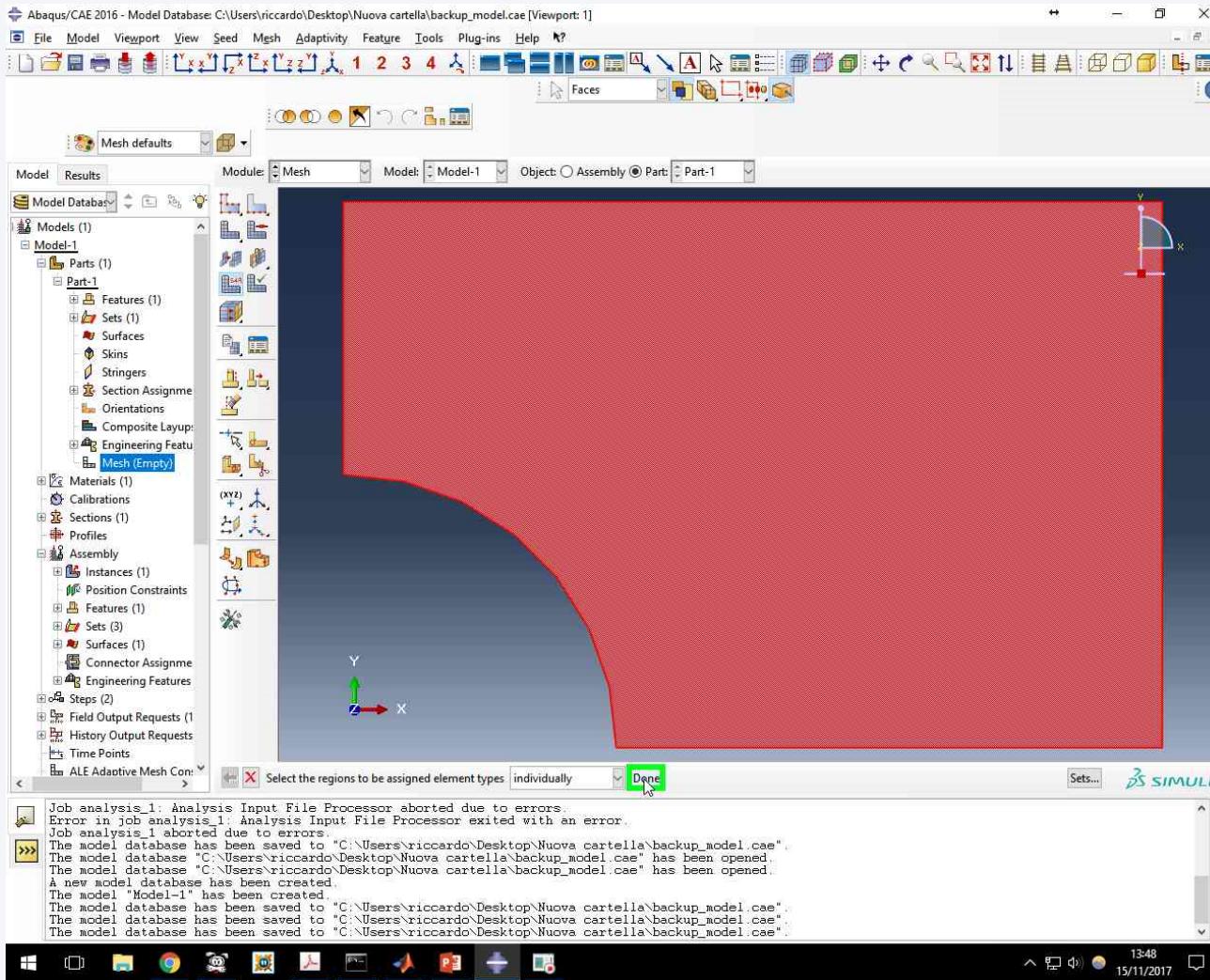
Move now to the mesh menu.

Creating the mesh



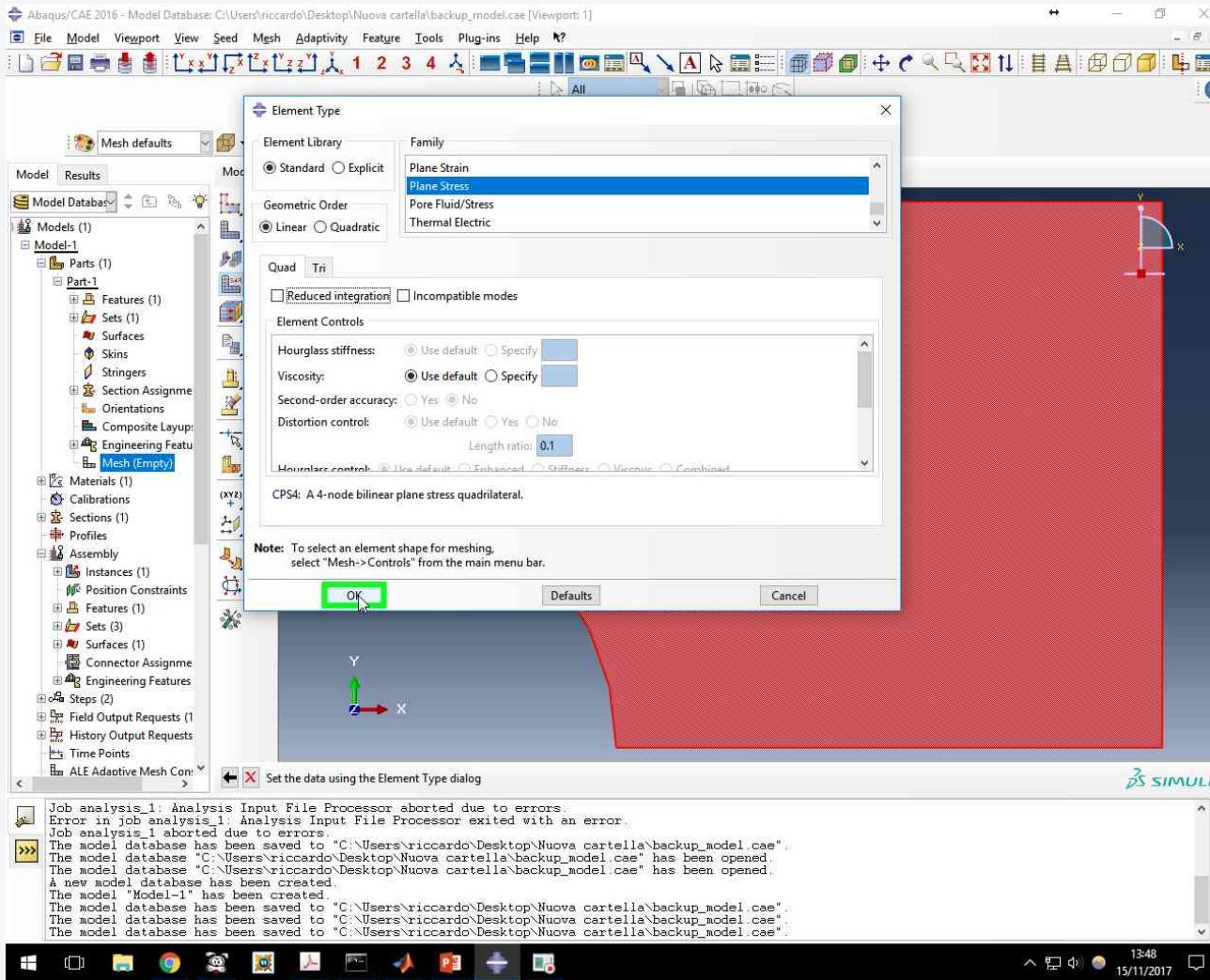
Select "Element type" to specify the kind of element to be used.

Creating the mesh



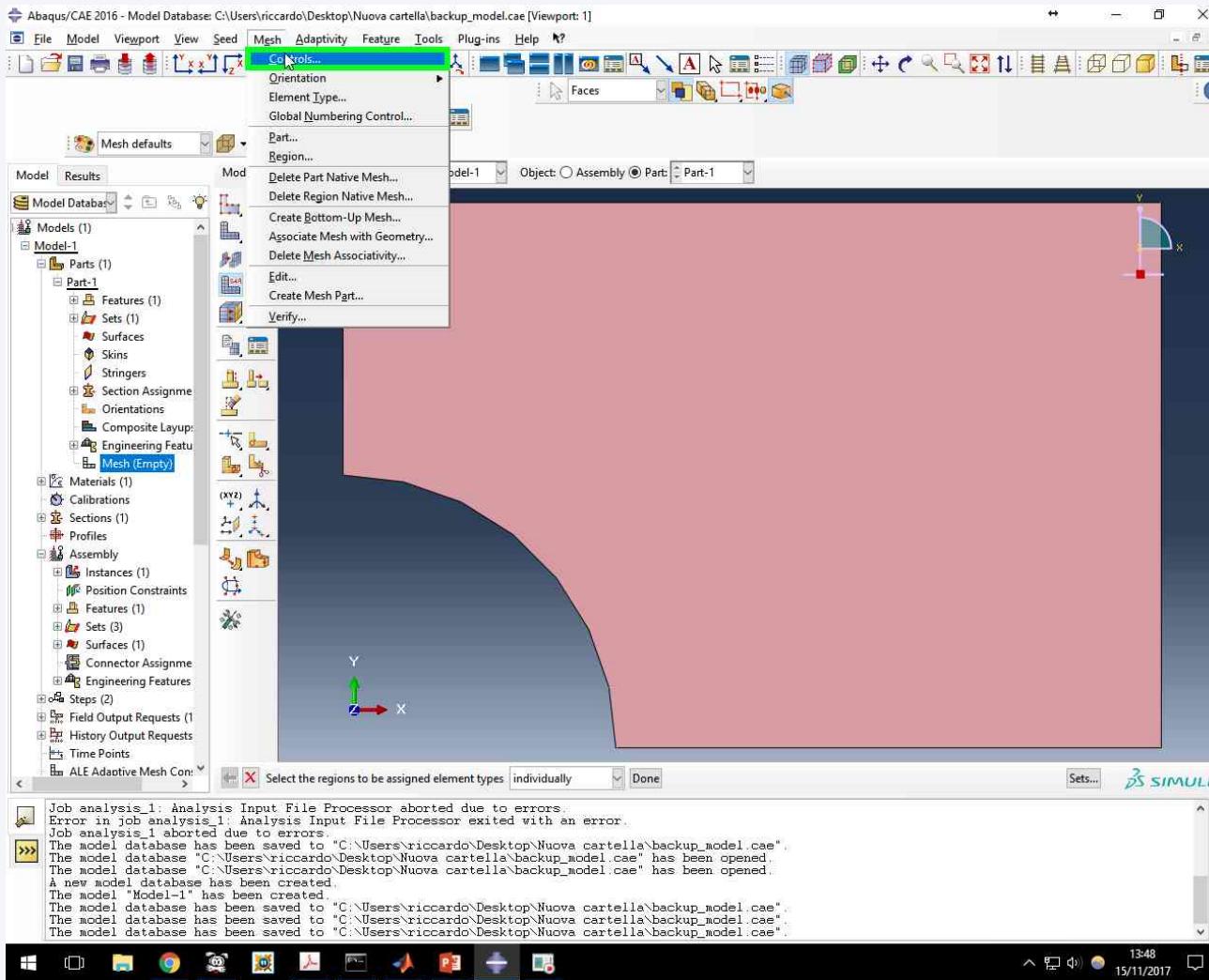
Select the part that will be associated to the specified kind of finite element.

Creating the mesh



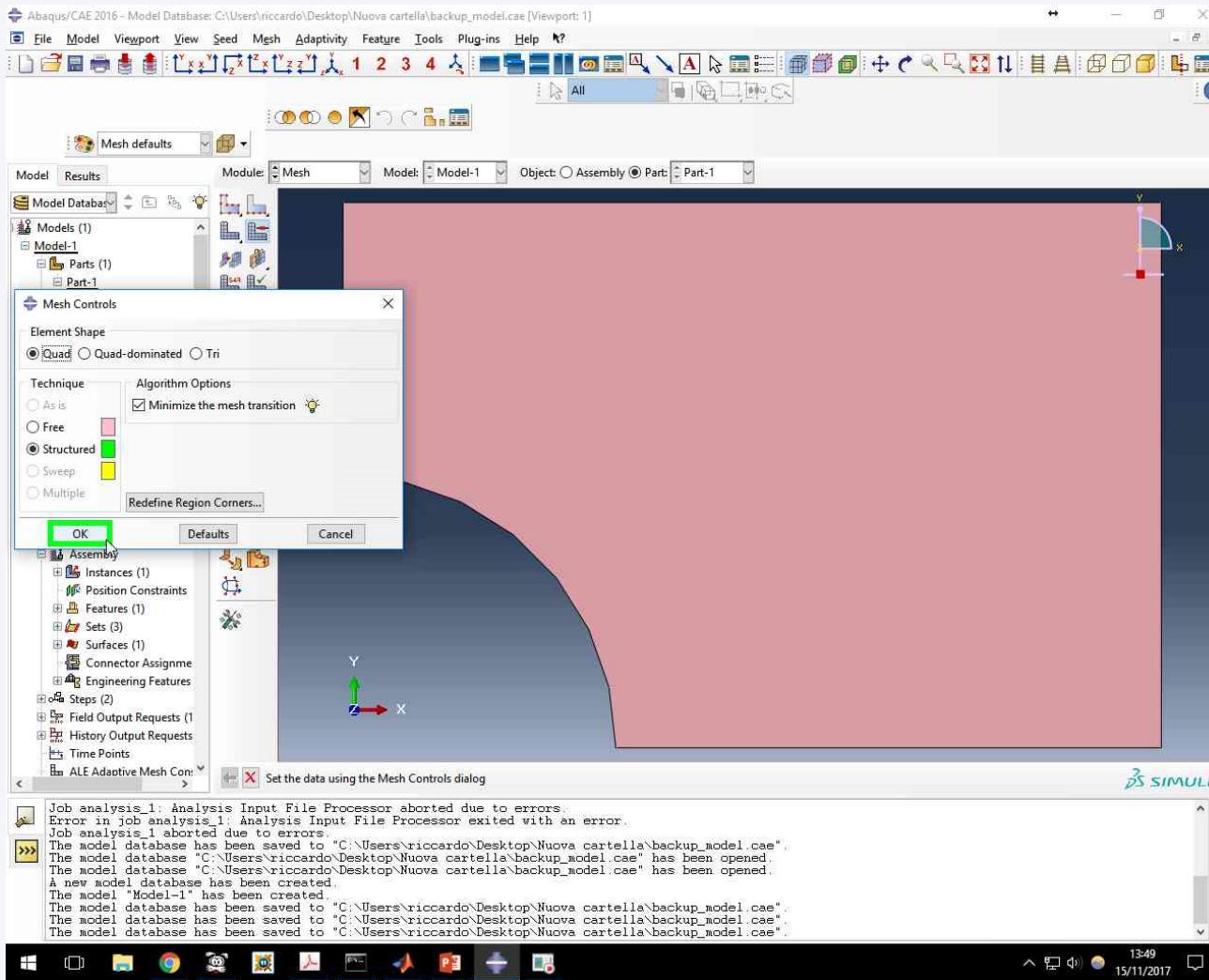
Many possibilities are available. In this case a membrane plane stress element is adopted.

Creating the mesh



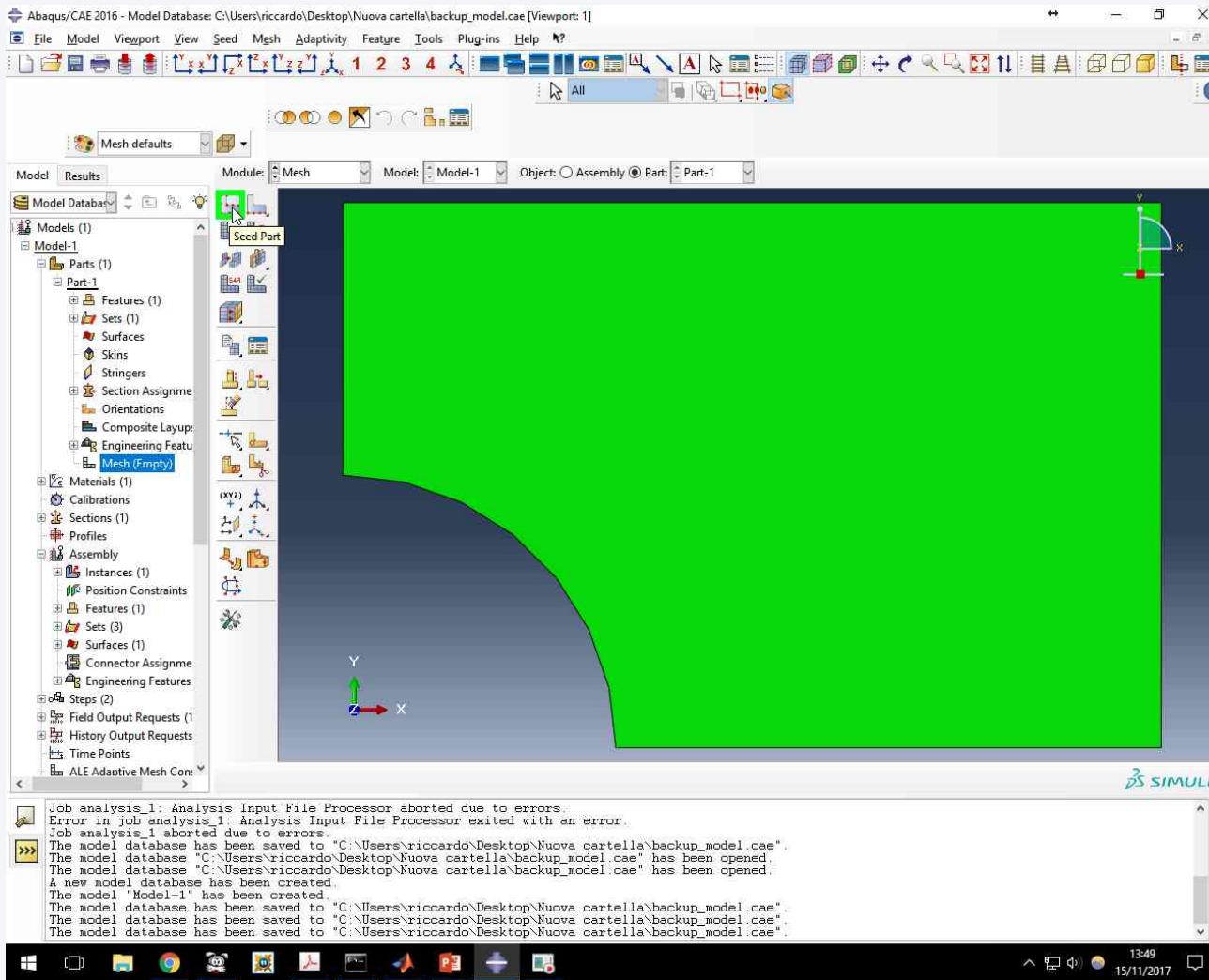
Specify some controls over the meshing rules.

Creating the mesh



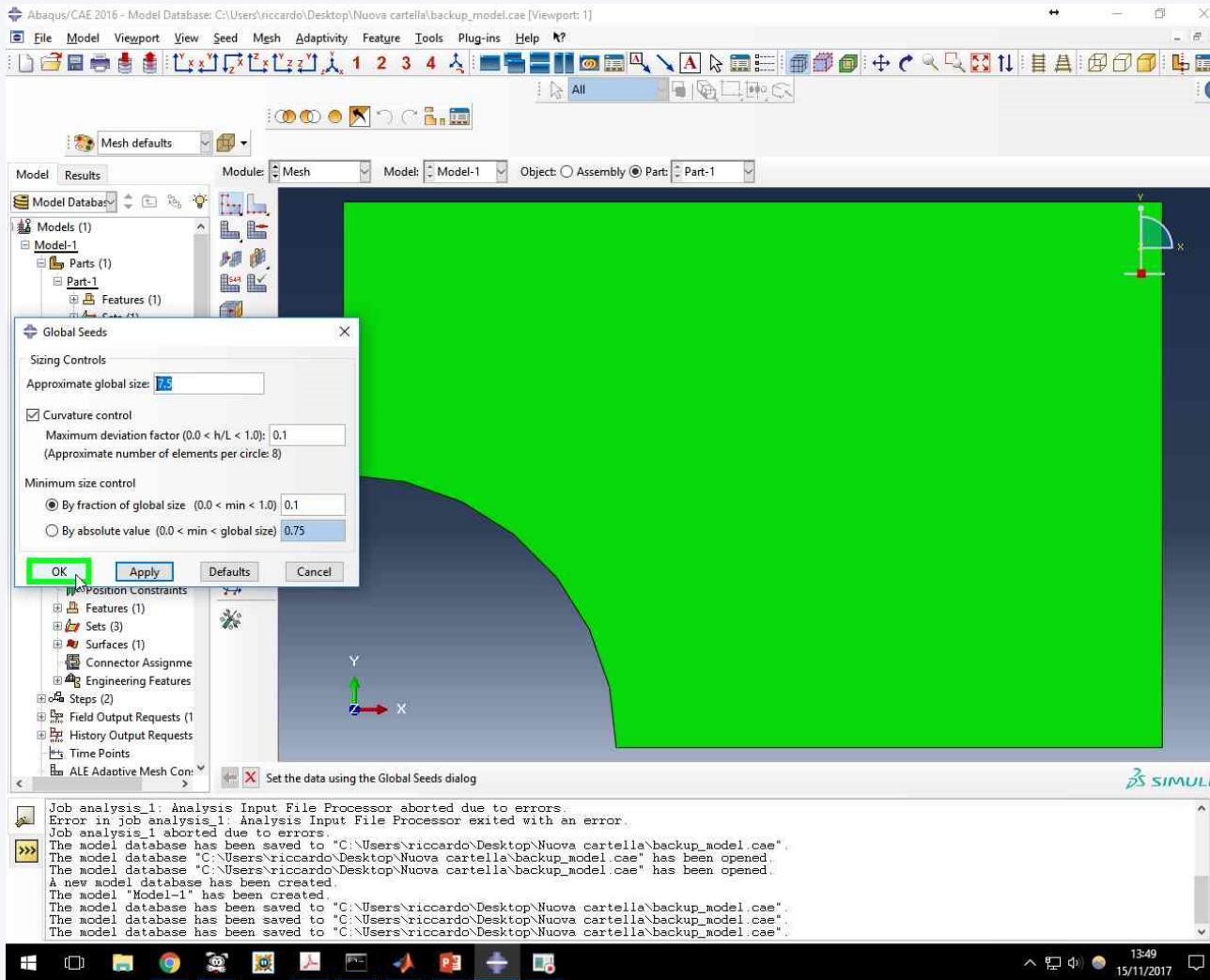
Select the options illustrated.

Creating the mesh



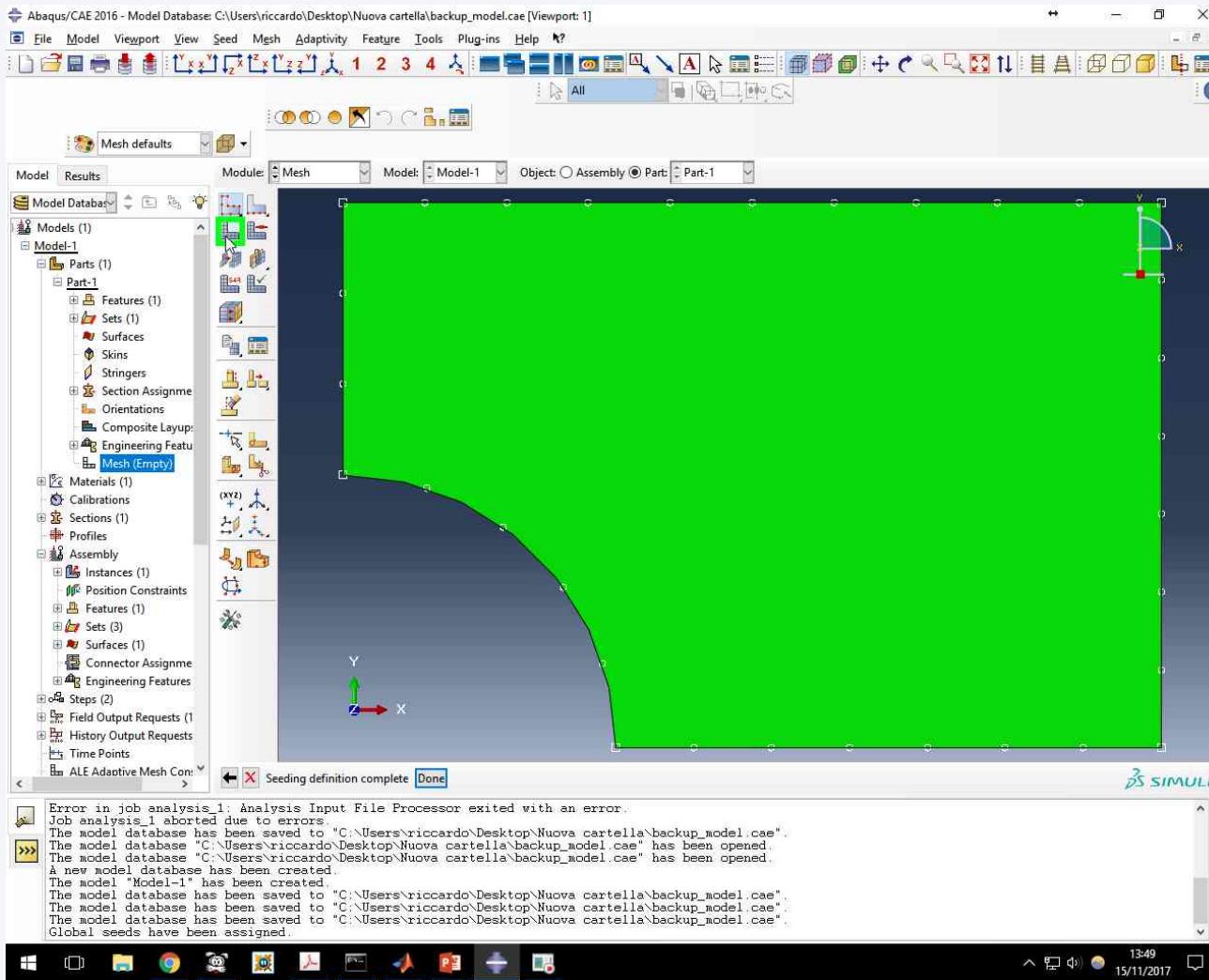
Specify the number of elements and nodes, for now, with the seed command.

Creating the mesh



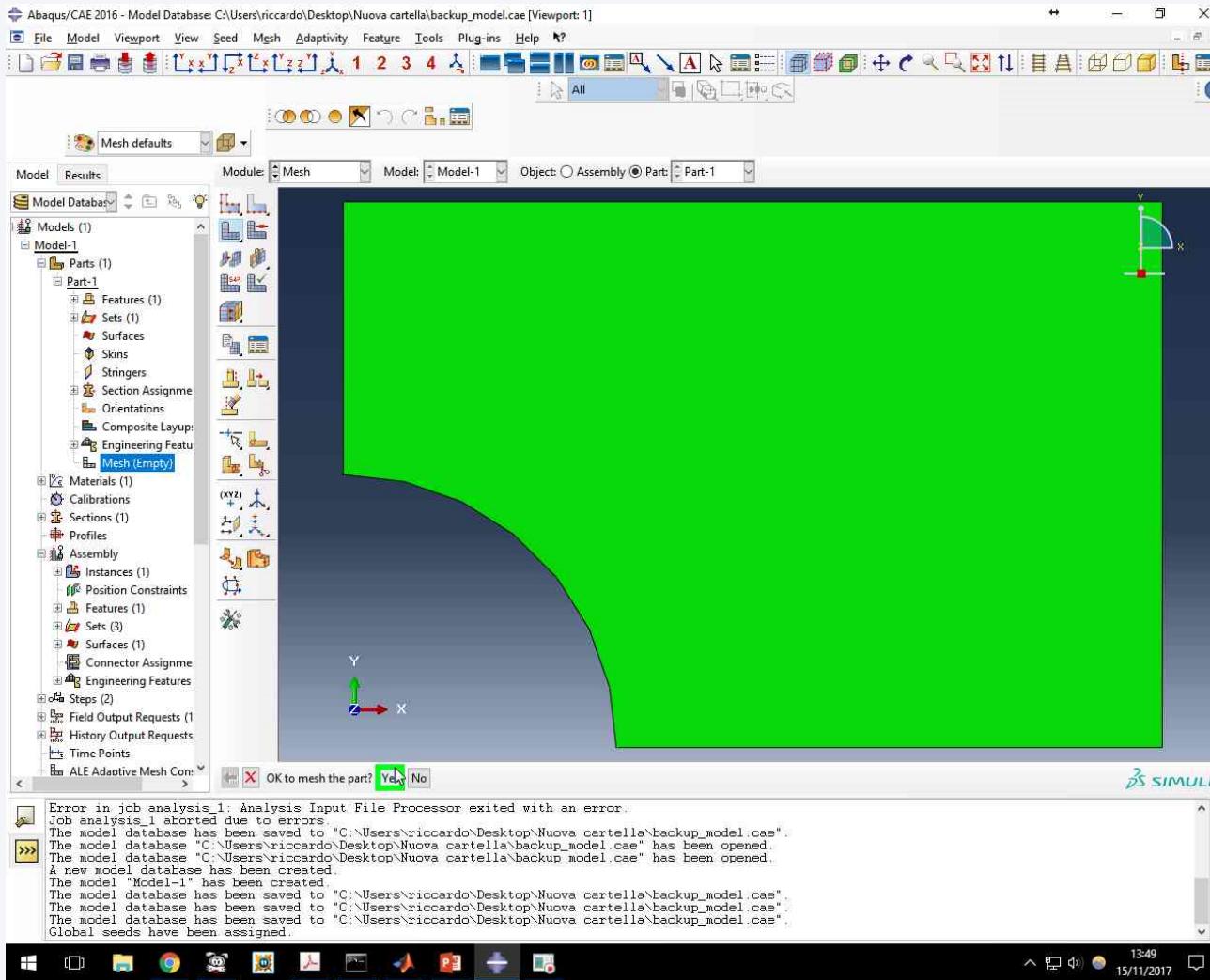
Accept the default options.

Creating the mesh



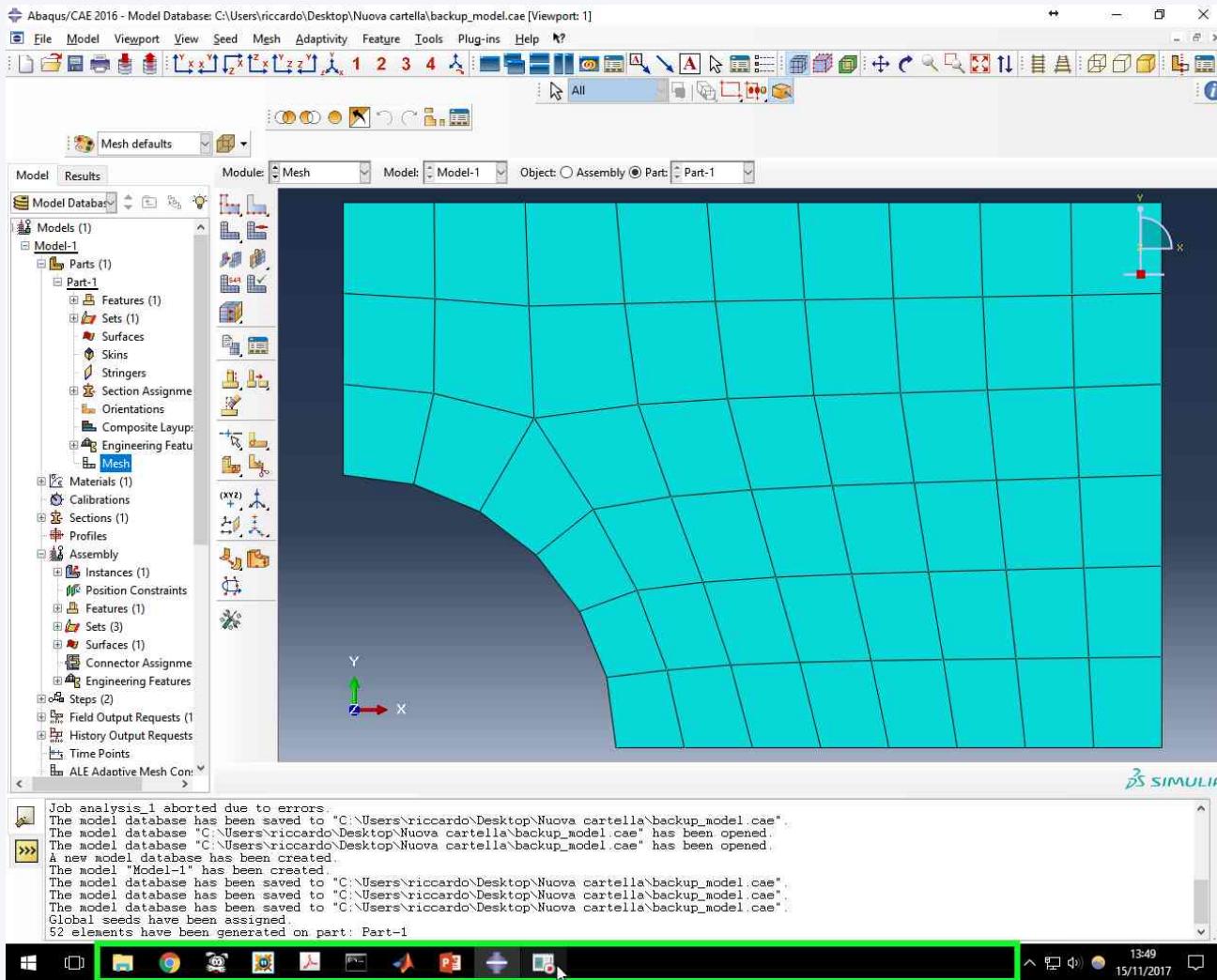
Realize now the mesh by clicking on "Mesh Part Instance".

Creating the mesh



Select the part and confirm.

Creating the mesh

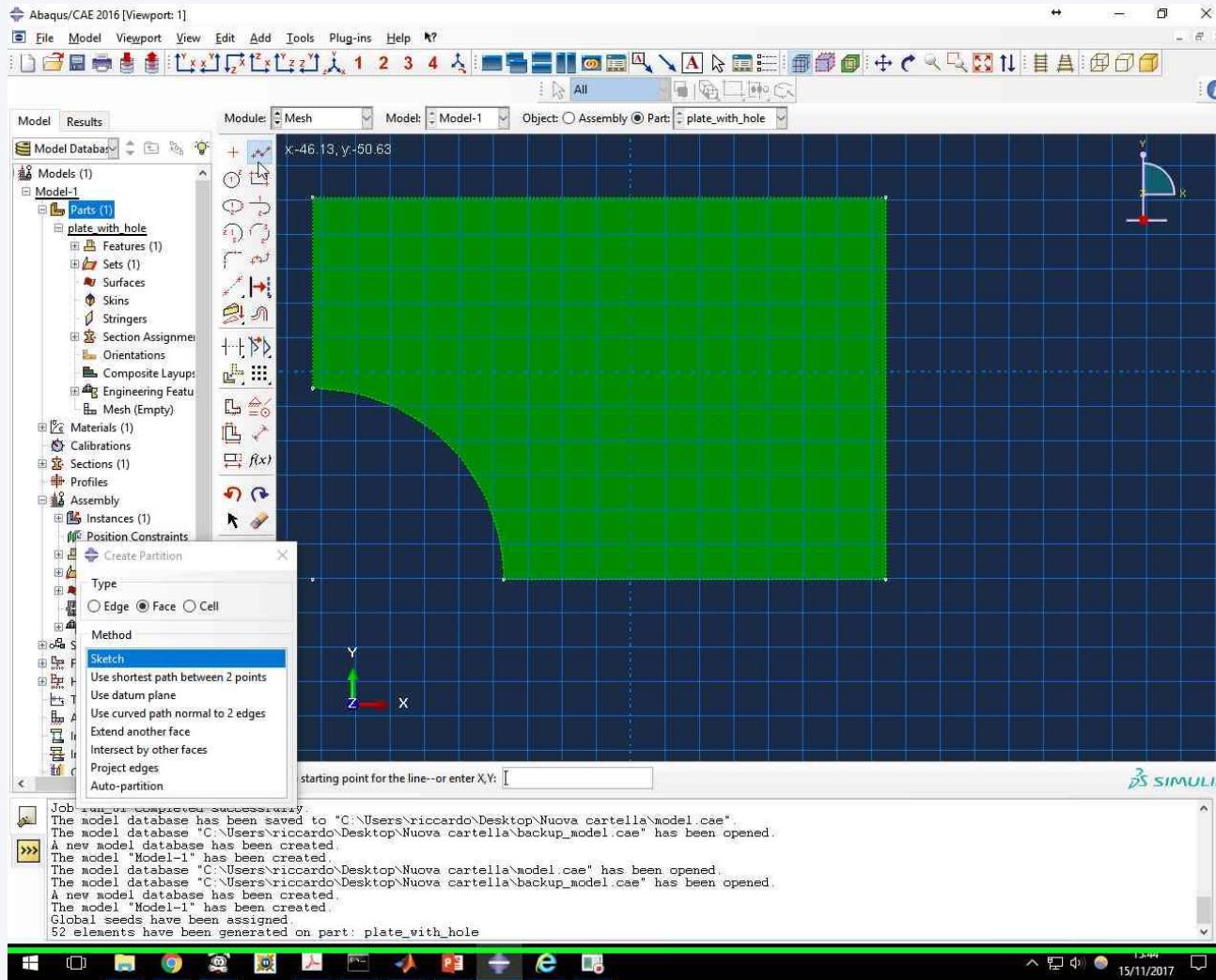


The mesh appear as reported.

In general, the direct meshing of the part may lead to unsatisfactory meshes. For instance, some elements could be excessively distorted.

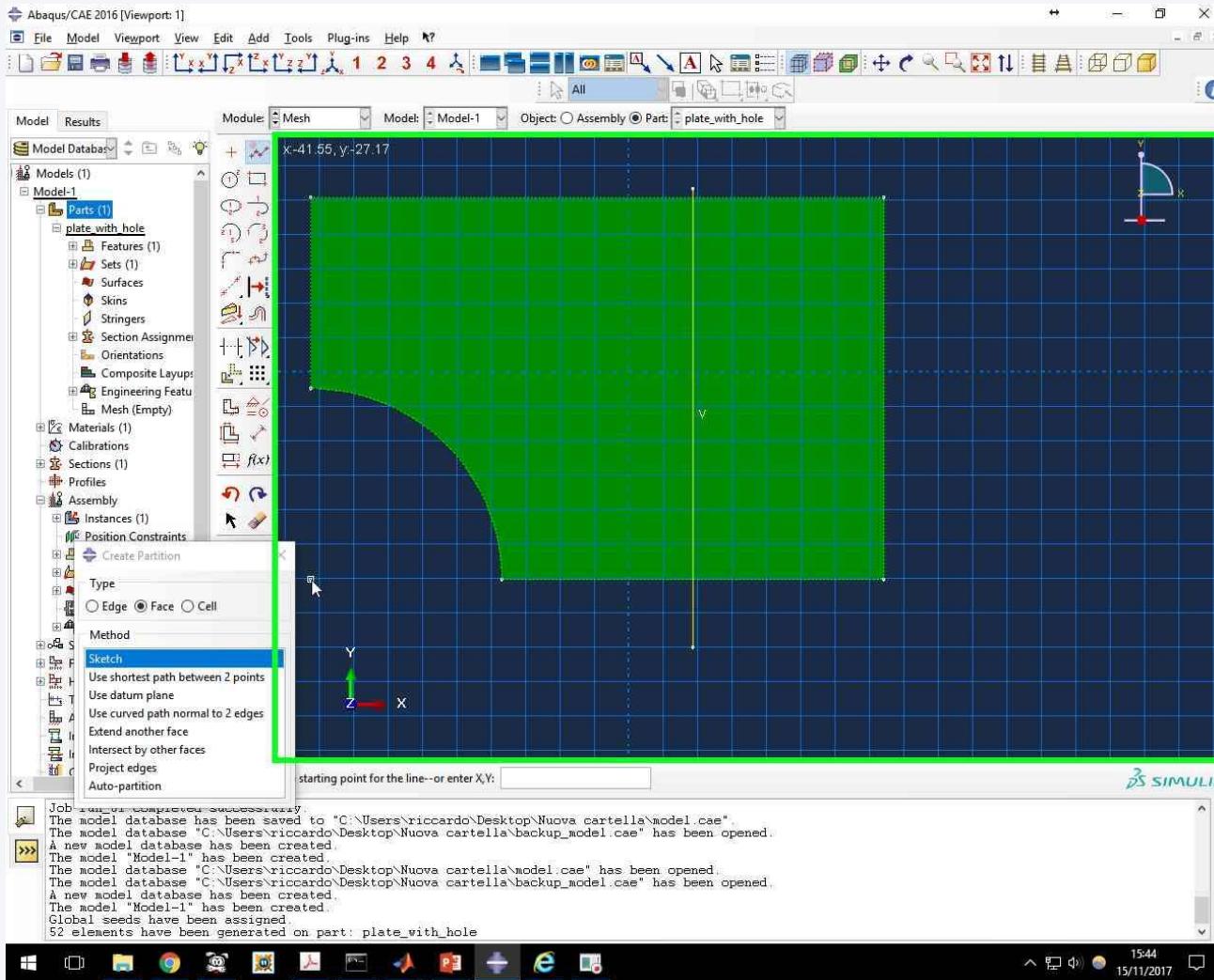
To this aim a good approach is to divide the part into a number of simpler sub-parts to be meshed independently (see later).

Remeshing using partitions



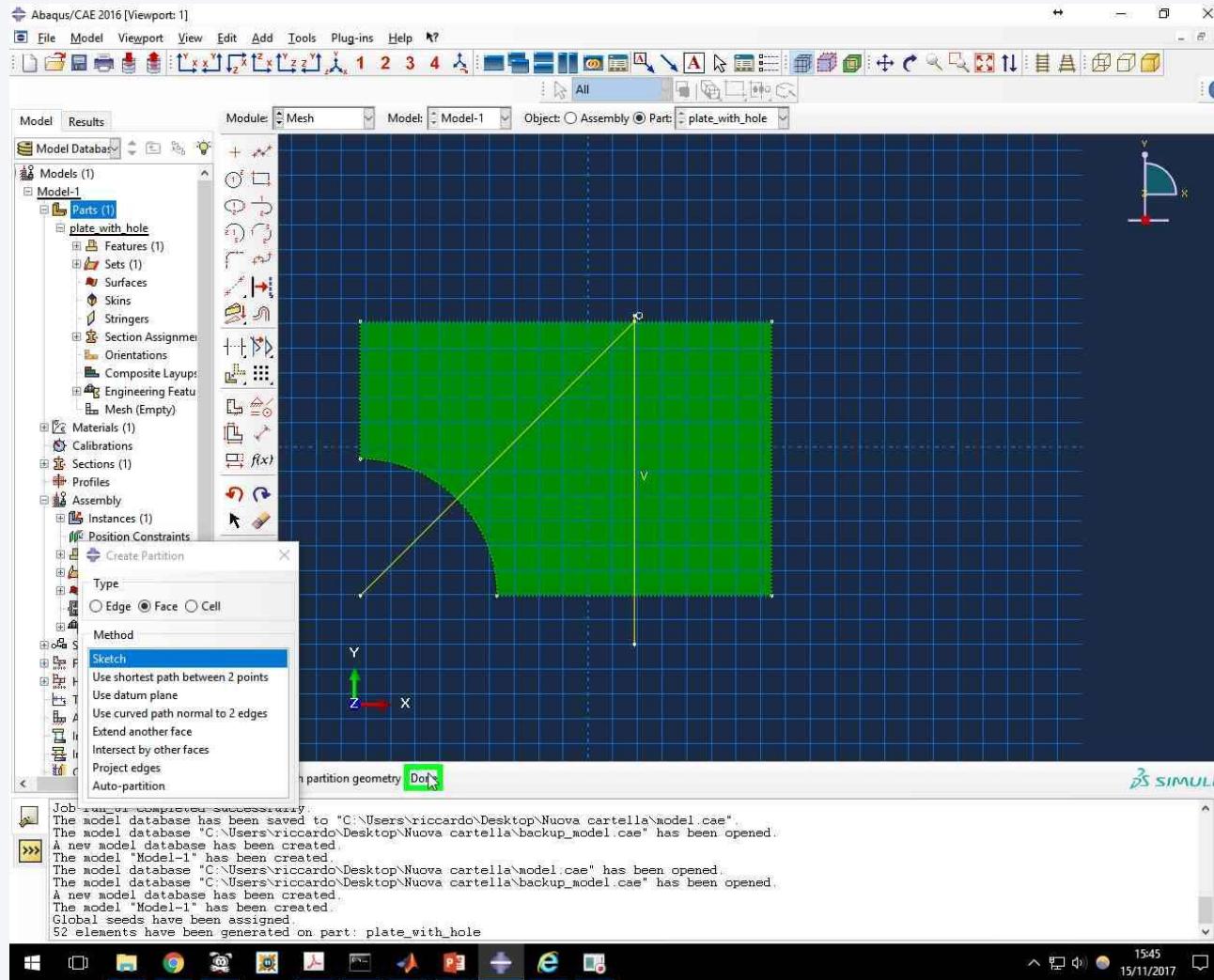
Using the command Tools → Partition the part can be divided into simpler entities.

Remeshing using partitions



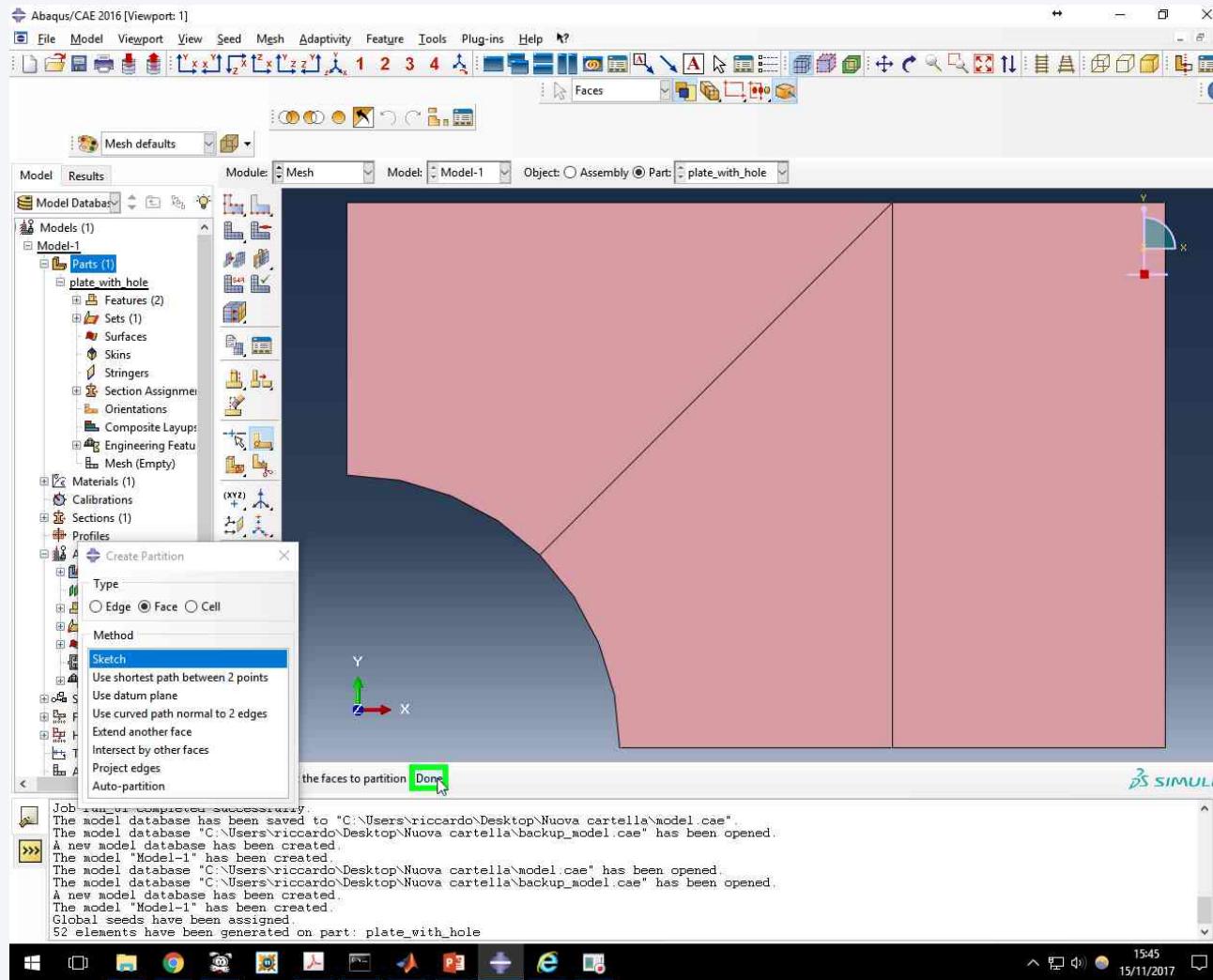
In this case a natural choice consists in dividing the plate into three subdomains.

Remeshing using partitions



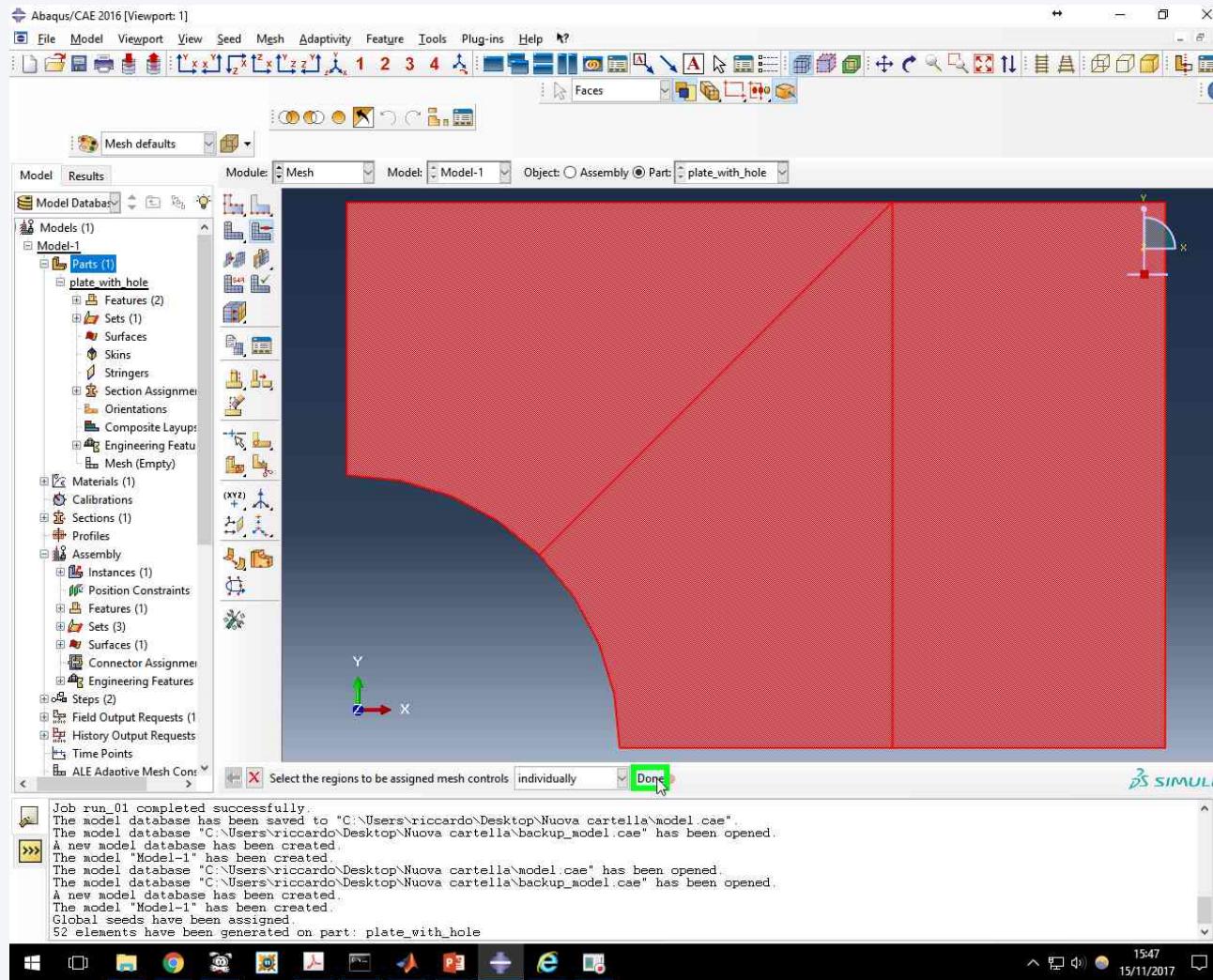
In this case a natural choice consists in dividing the plate into three subdomains.

Remeshing using partitions



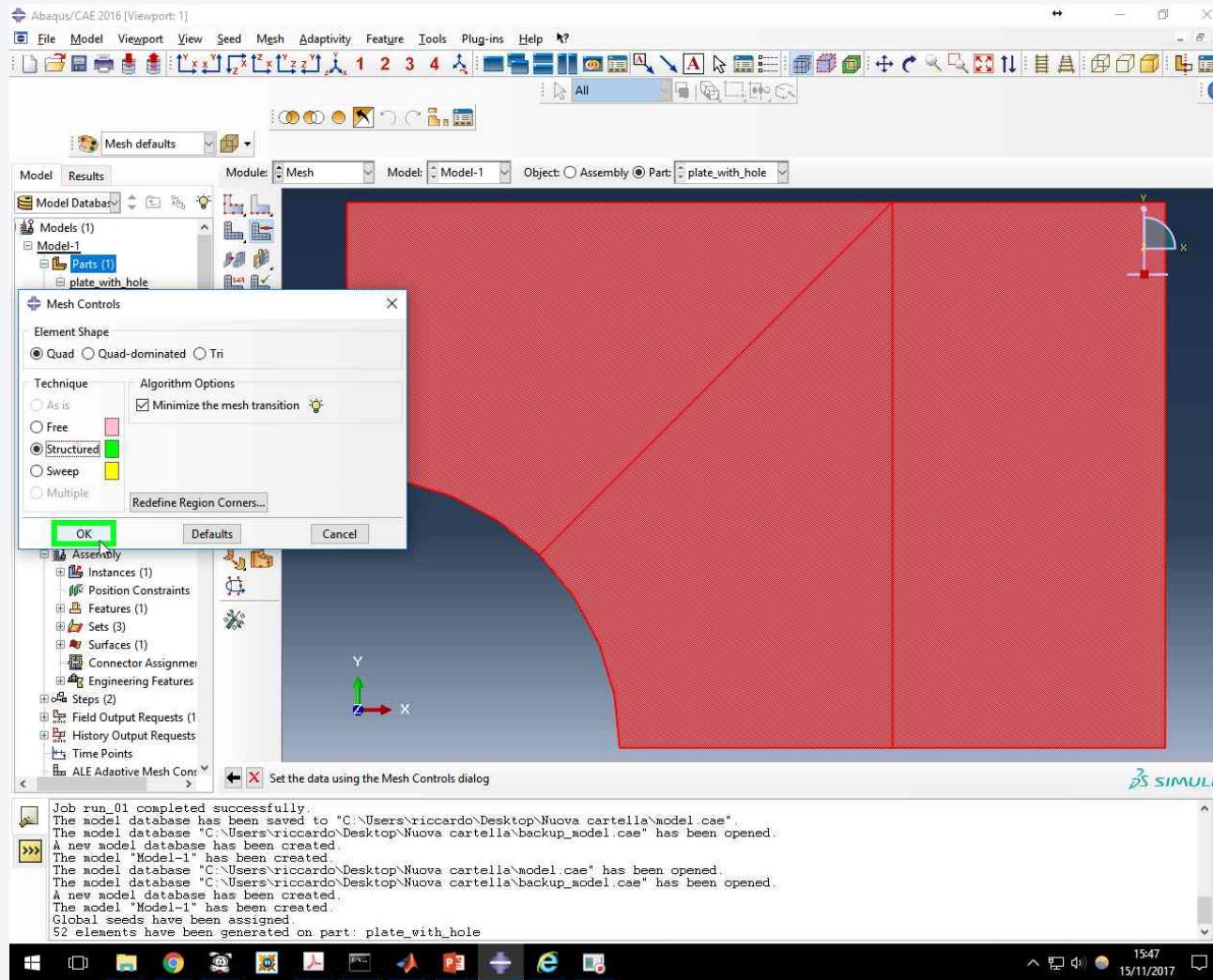
The subdivision looks now as reported.

Remeshing using partitions



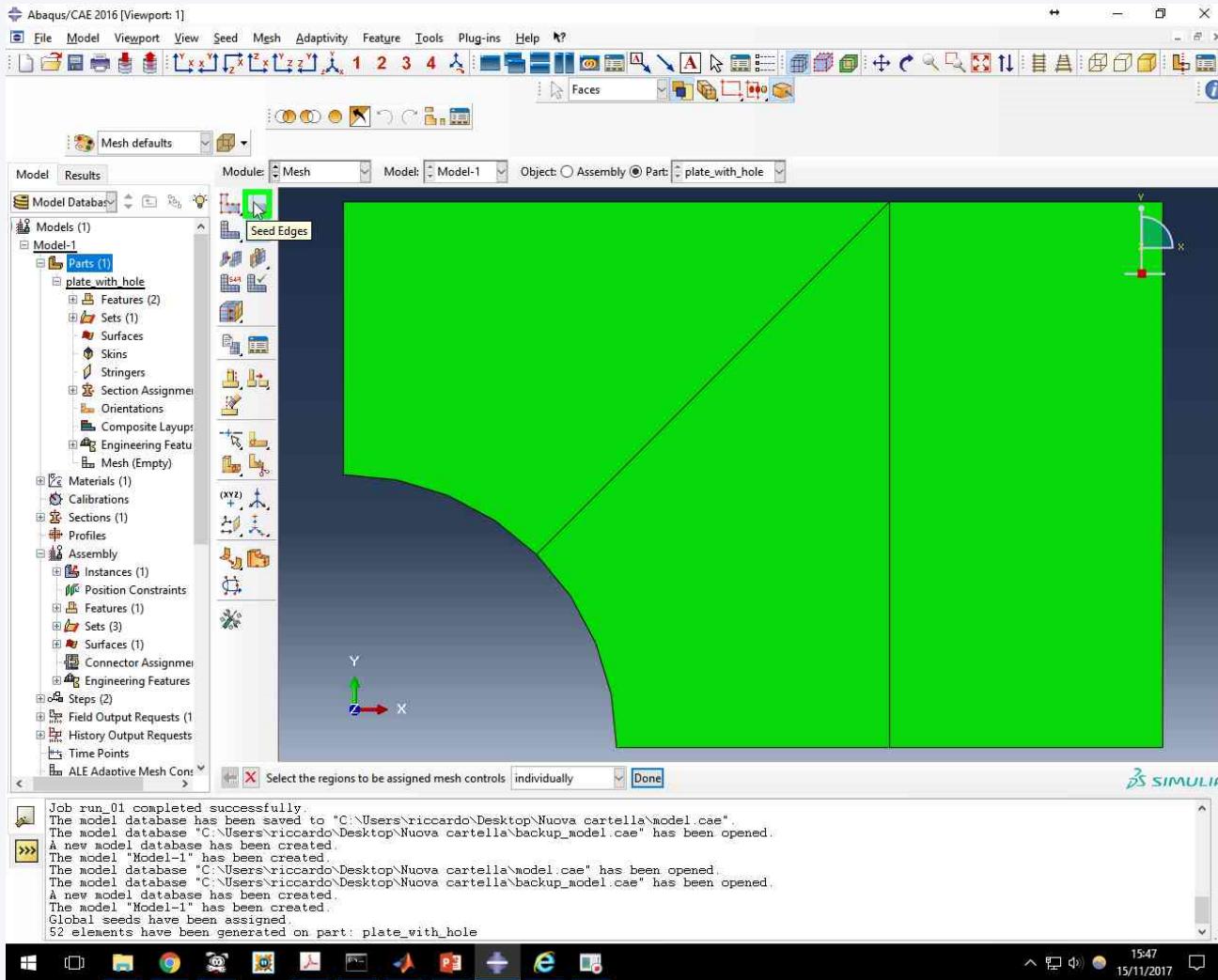
As far as the part has been modified, the mesh controls should be re-defined

Remeshing using partitions



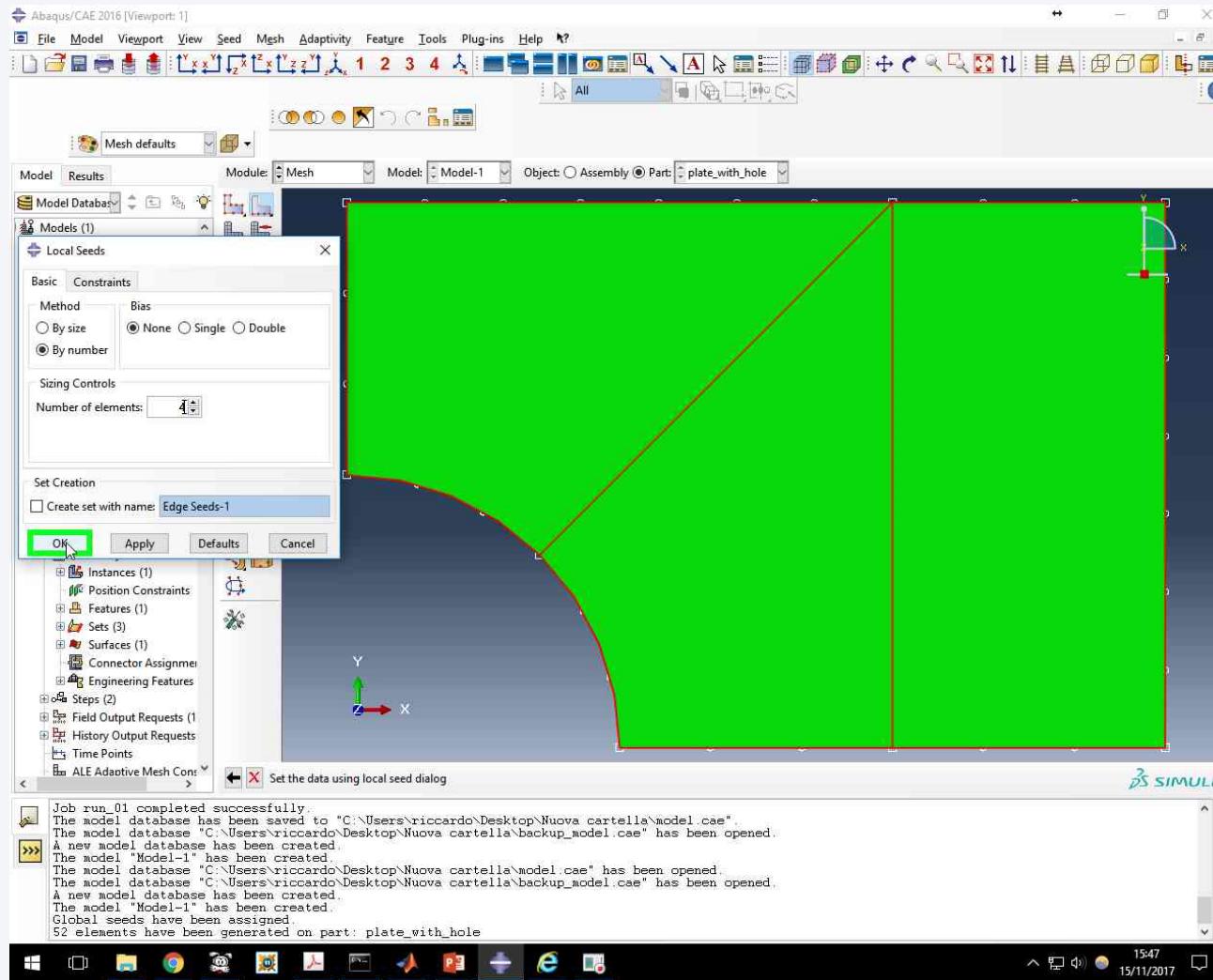
Select again the same options.

Remeshing using partitions



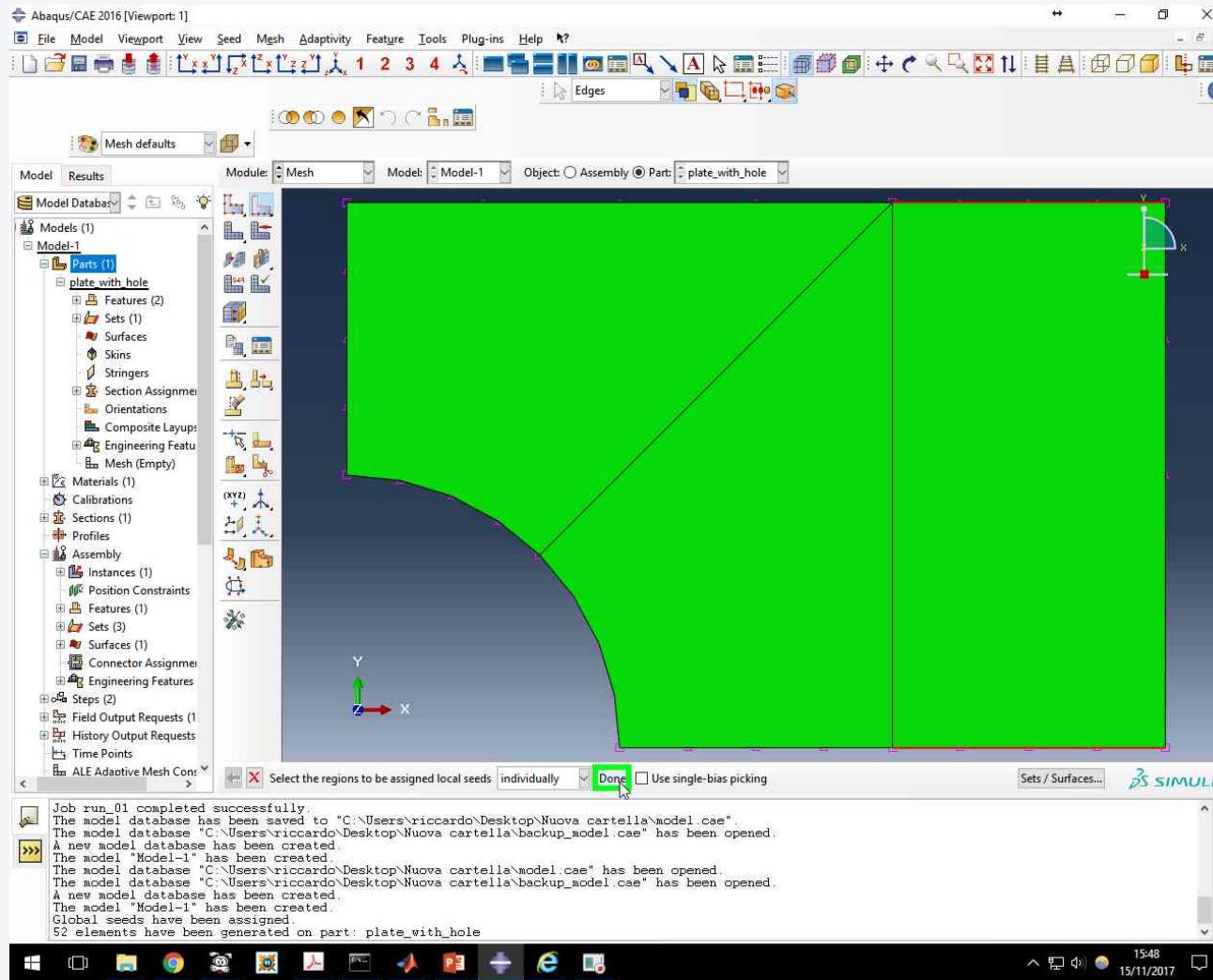
Define now the number of nodes along each of the lines composing the boundaries of the three domains.

Remeshing using partitions



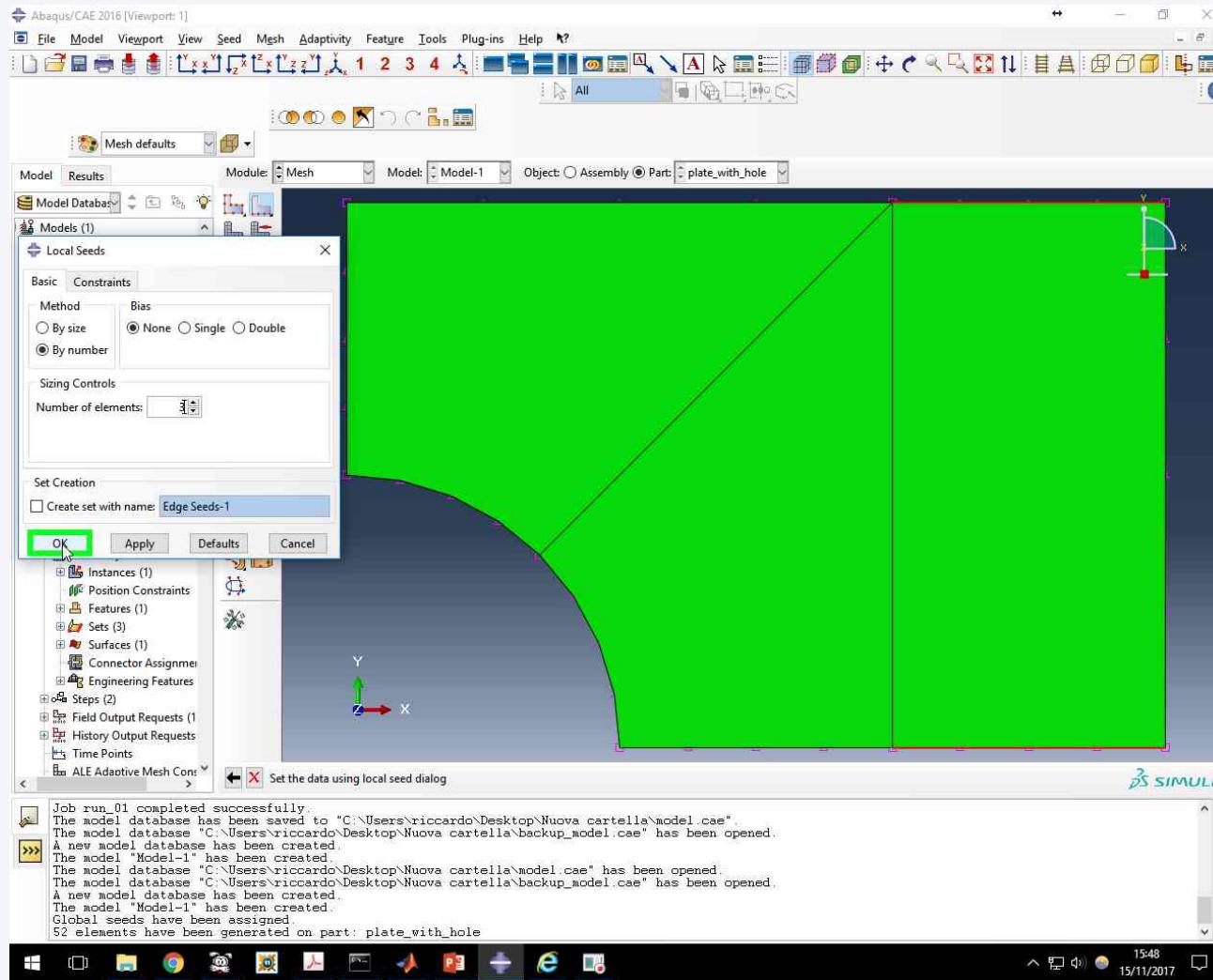
Select 4 elements for edges here highlighted.

Remeshing using partitions



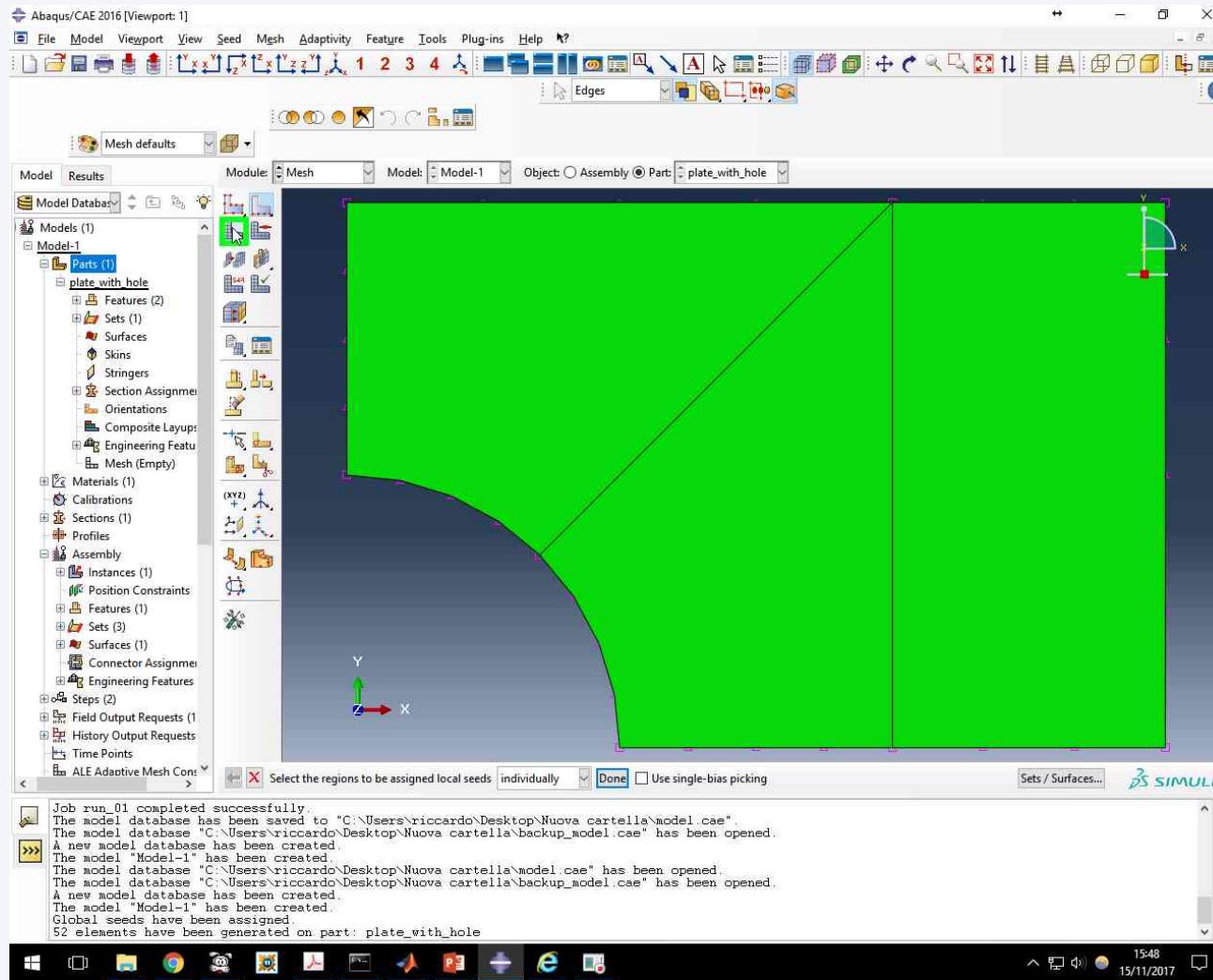
Select 3 elements for the remaining two edges.

Remeshing using partitions



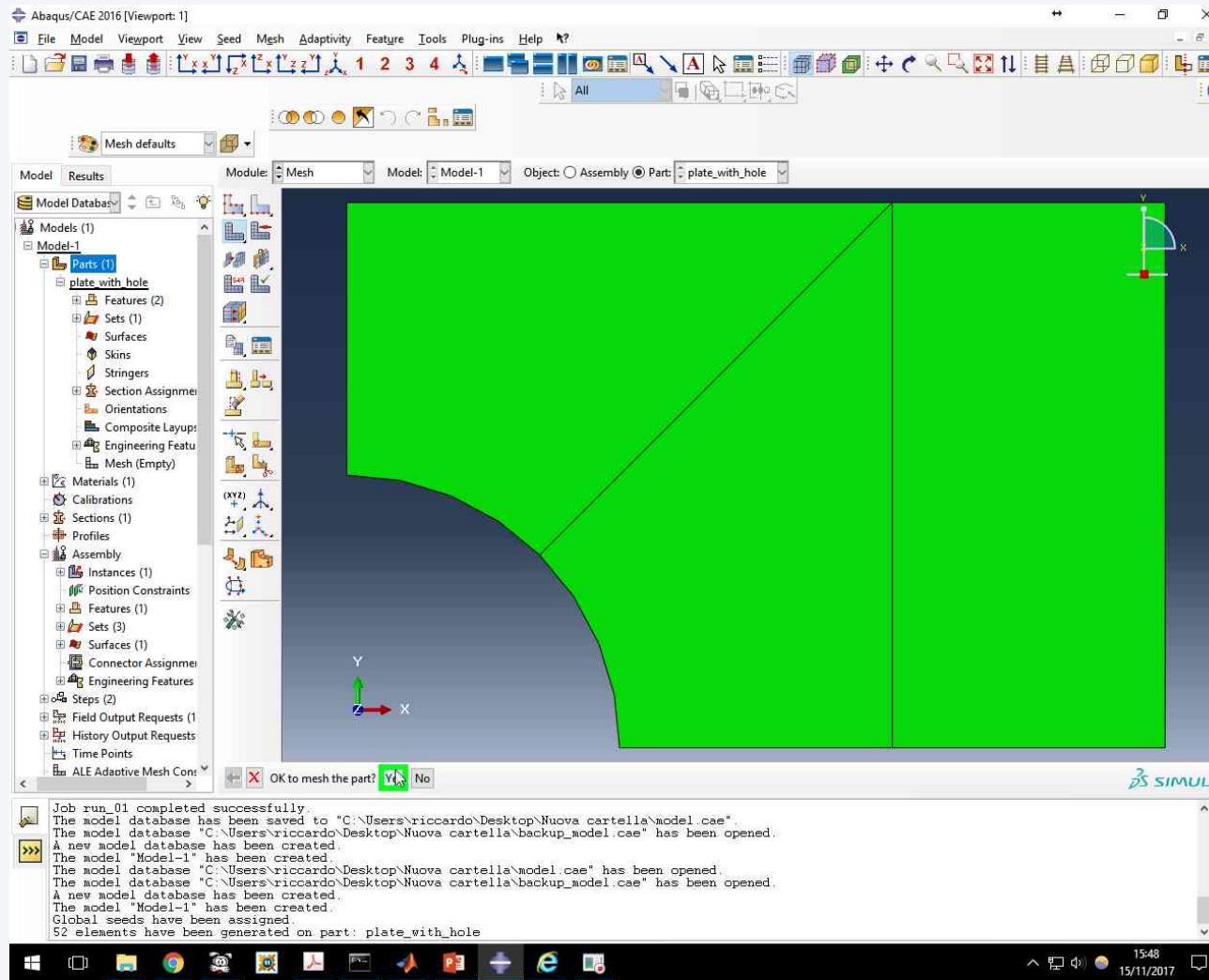
Select 3 elements for the remaining two edges.

Remeshing using partitions



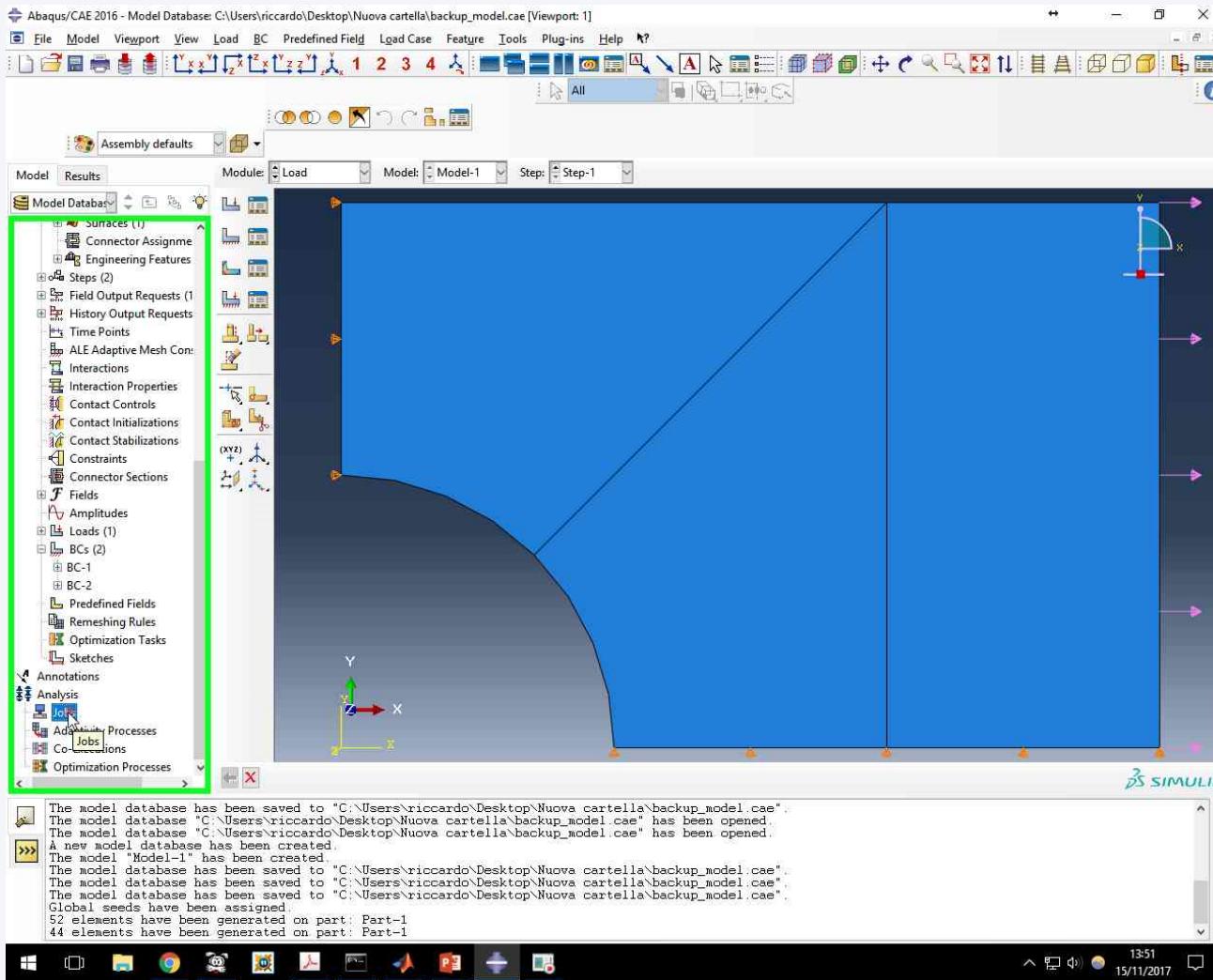
Mesh again the part.

Remeshing using partitions



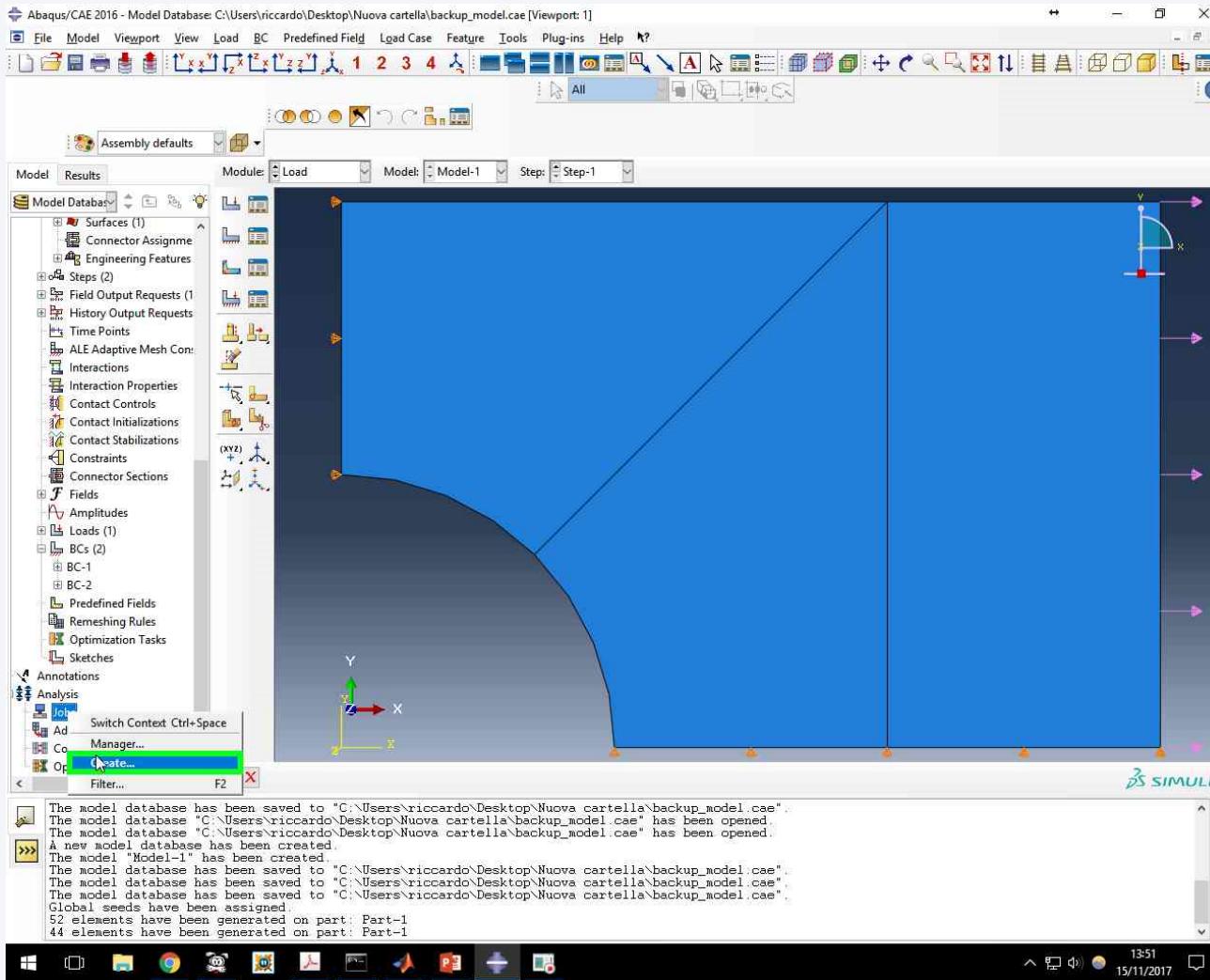
Confirm.

Create job



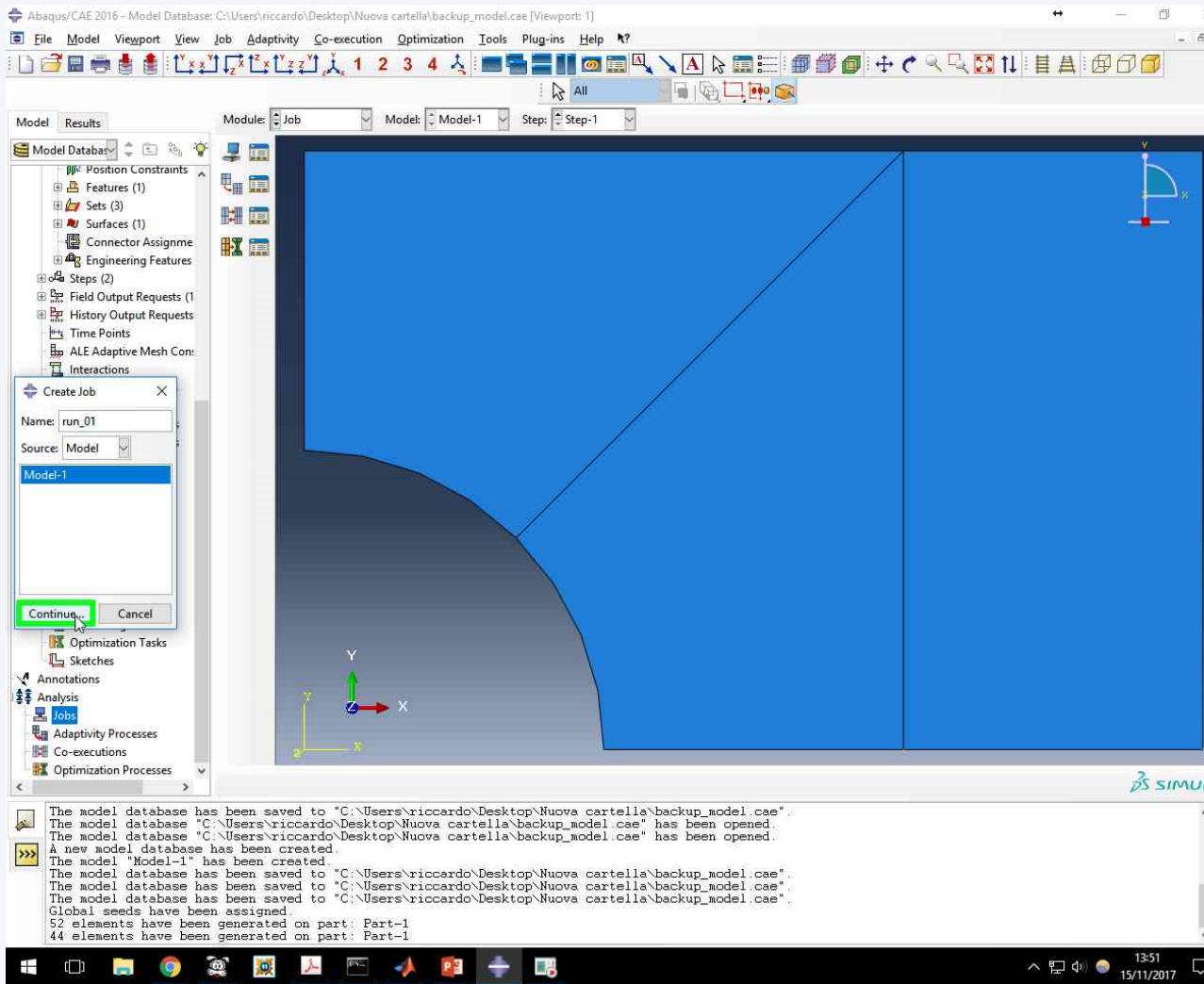
Creating the job is the last step before launching the analysis. This operation consists in assigning a name to the analysis, writing the input file and submitting the analysis.

Create job



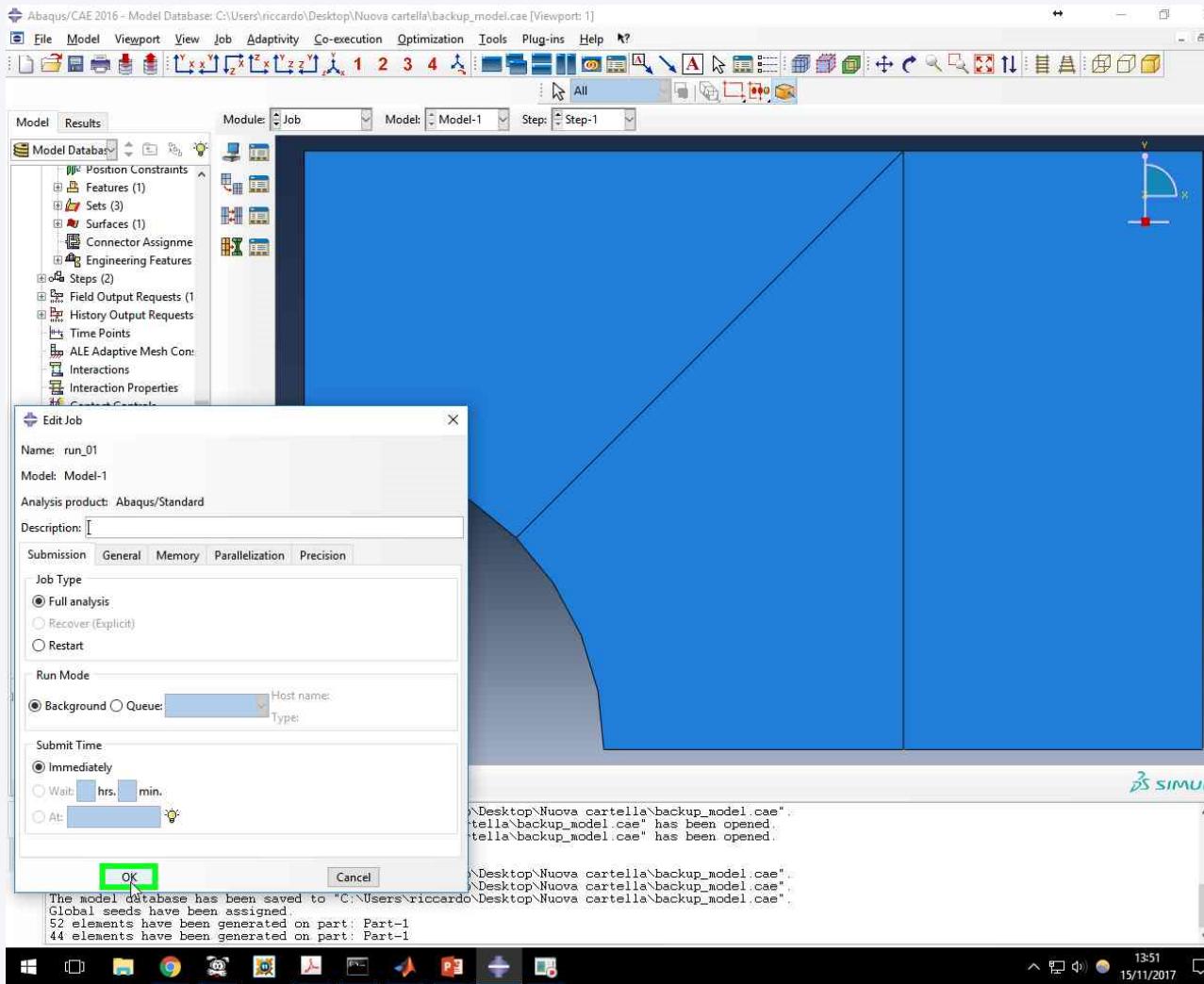
Right-click on job and select "Create".

Create job



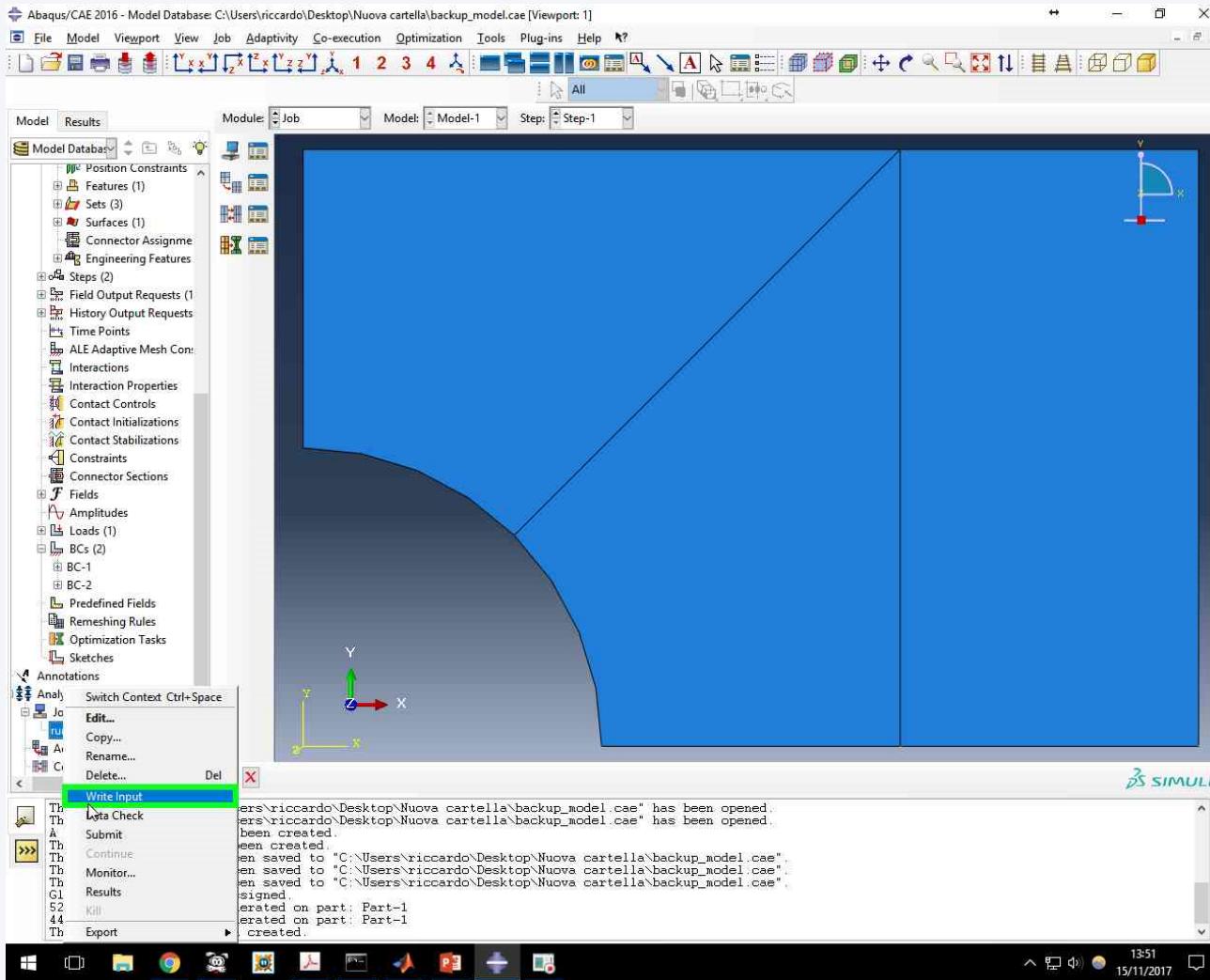
Confirm.

Create job



Confirm.

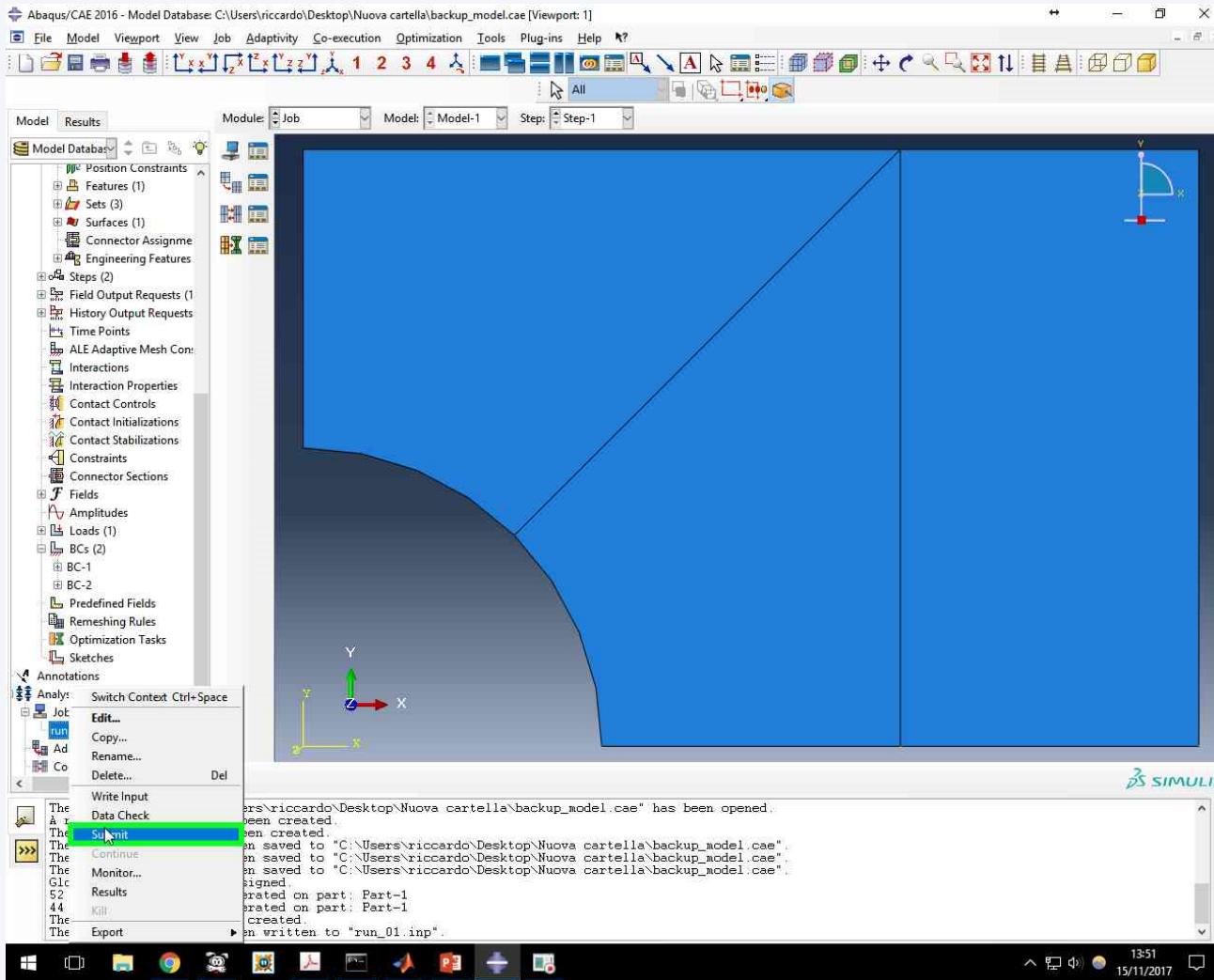
Create job



Although not strictly necessary, write the input file. The file is written with .inp extension into the folder where Abaqus cae was launched. The name of the file is the name selected for the job.

The .inp file can be edited with any text editor.

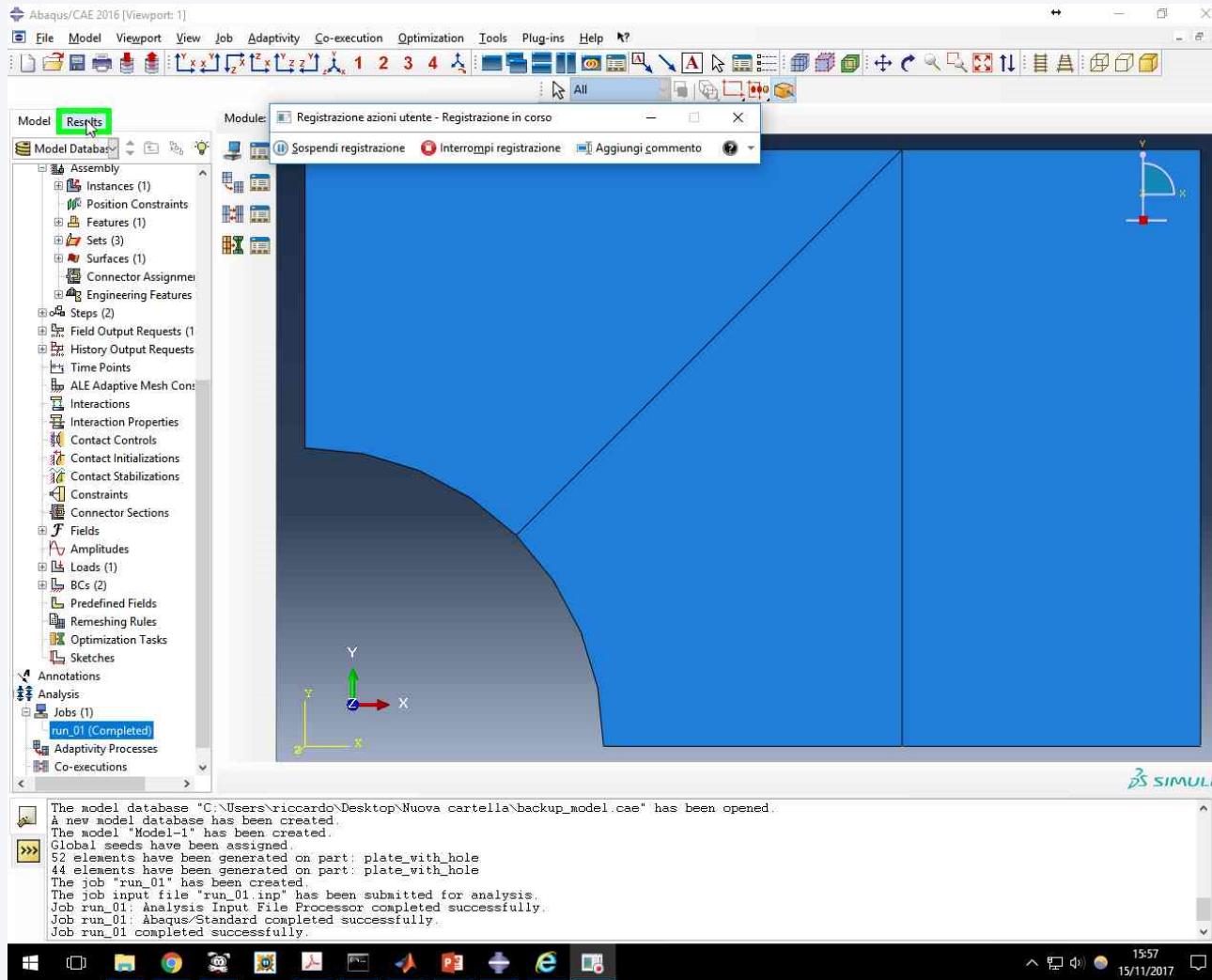
Create job



Click on "Submit" for running the analysis. This operation is reported here for completeness but is not recommended. In general, a better practice consists in:

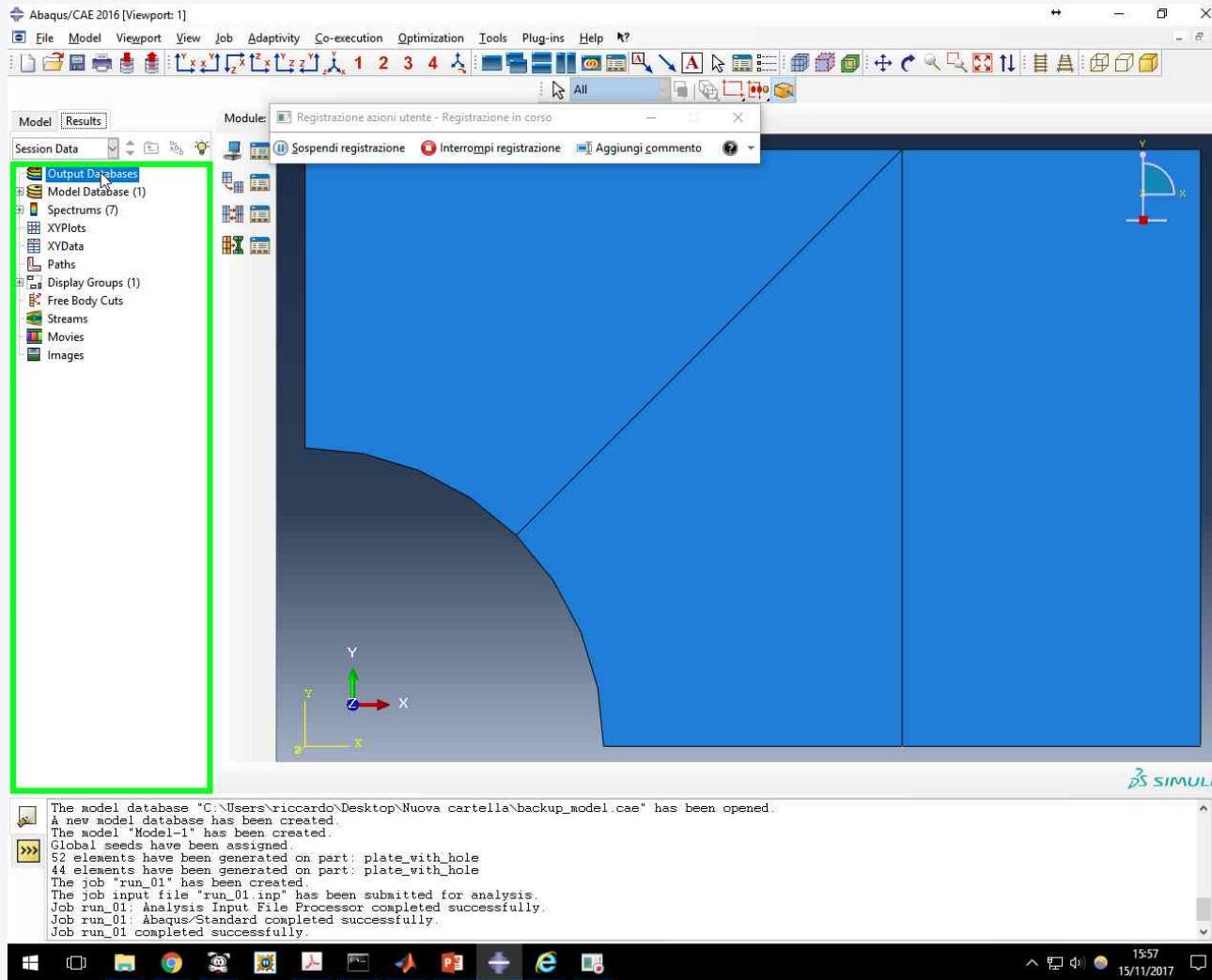
1. editing the .inp file to check the model
2. Running the analysis via command prompt

Accessing results



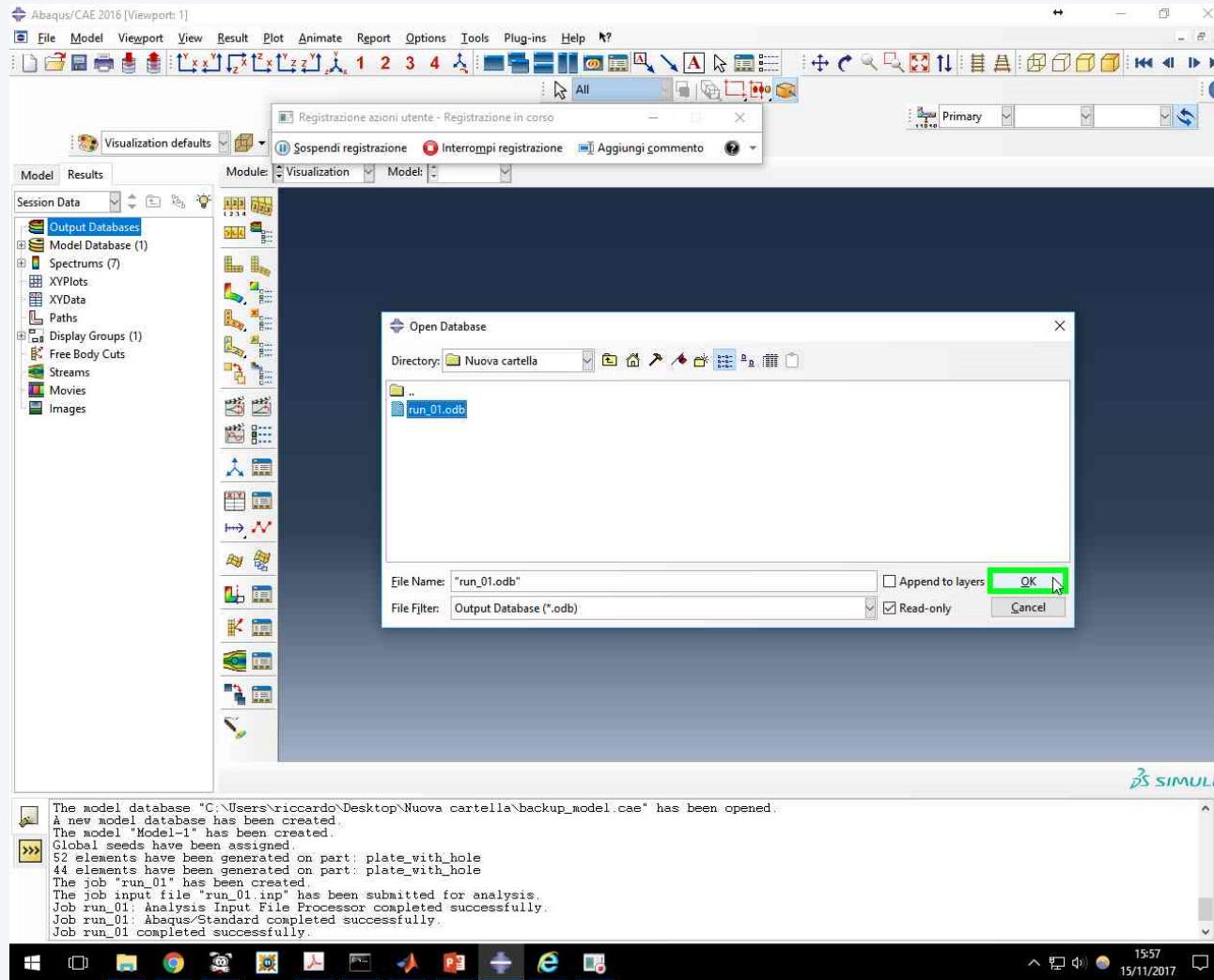
Once the submission status is "Completed" it is possible to open the result file. This file has extension .odb, and is a binary file.

Accessing results



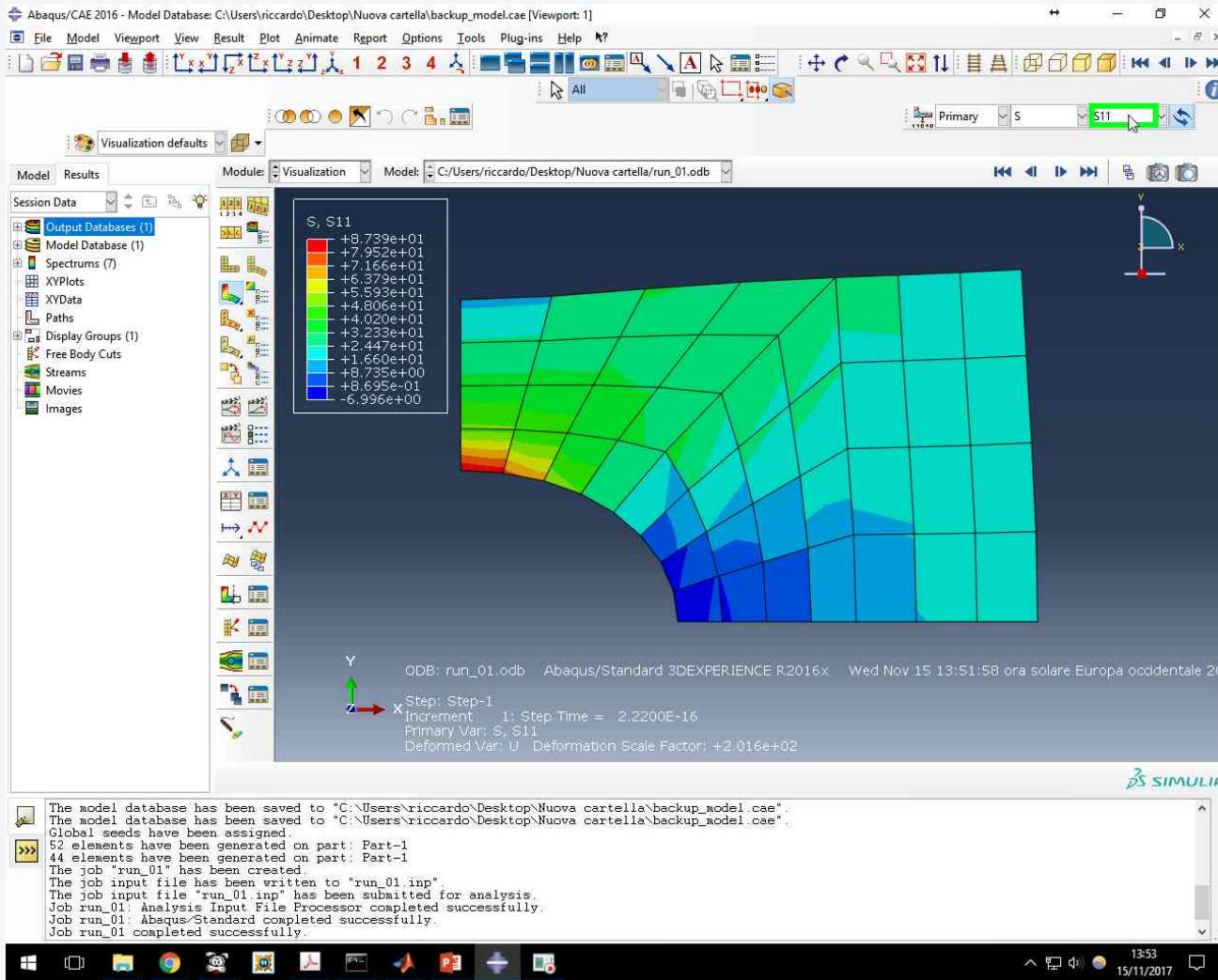
Once the submission status is "Completed" it is possible to open the result file. This file has extension .odb, and is a binary file.

Accessing results



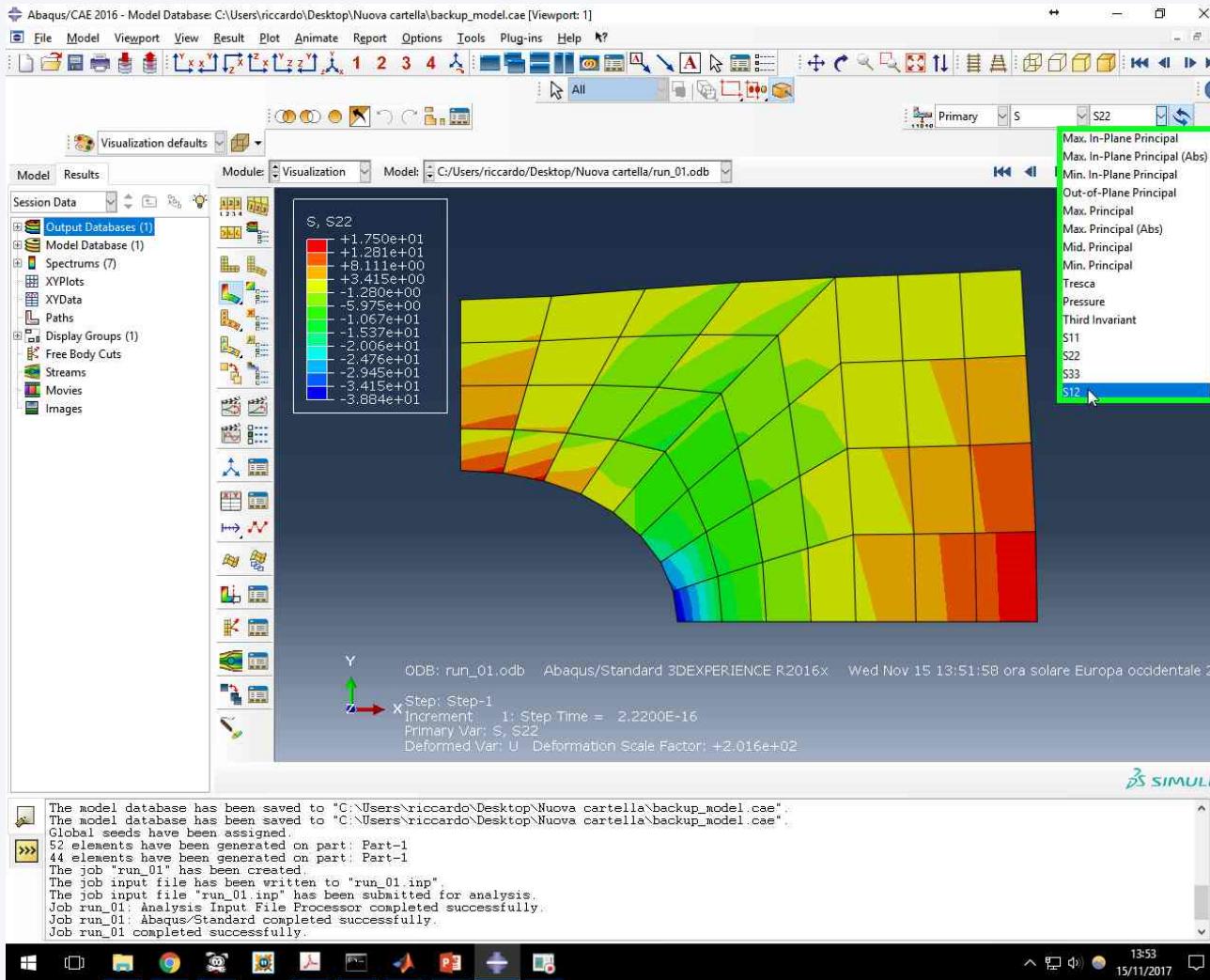
Open the result file.

Viewing results



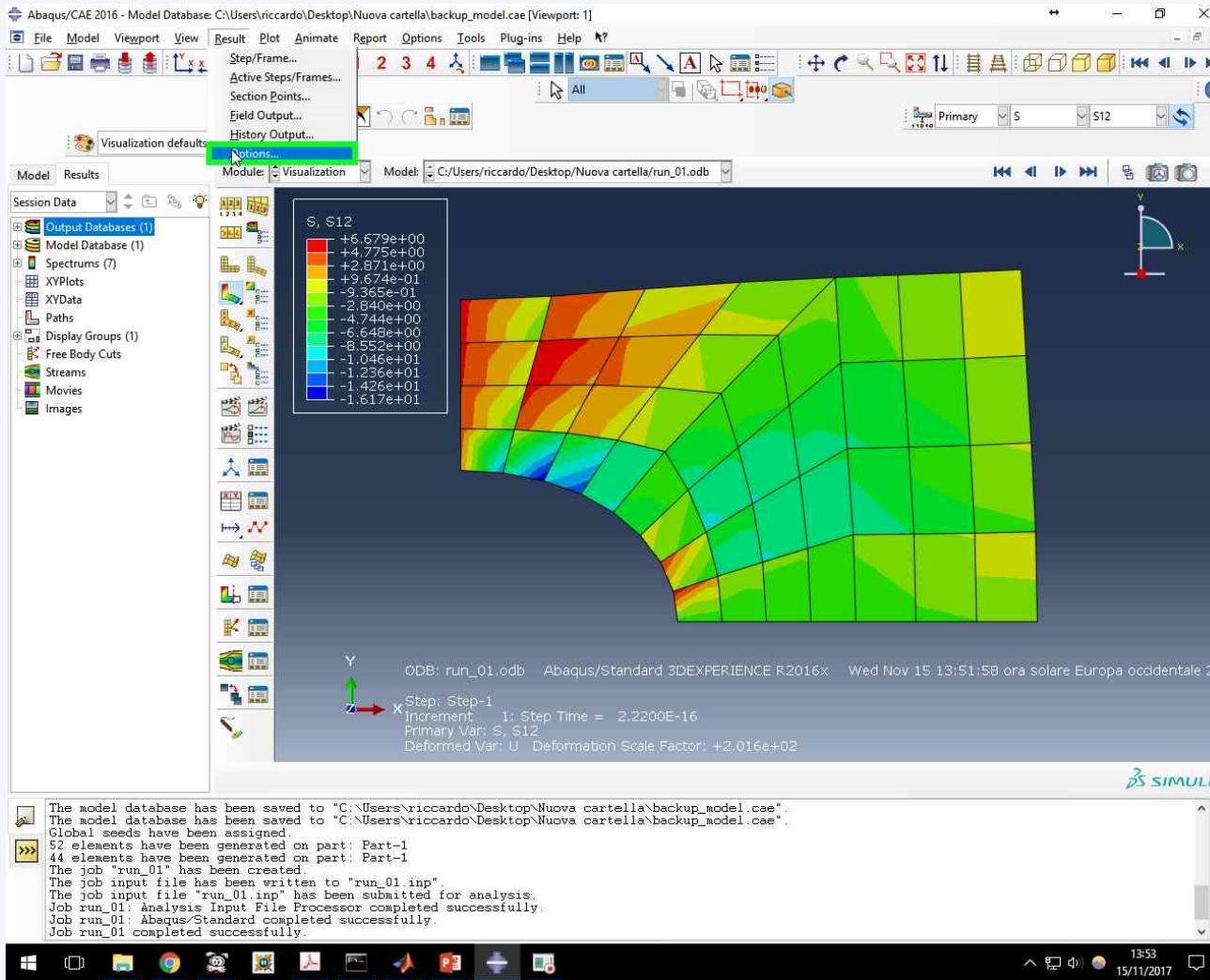
The results are reported in the form of contour plots. On the top right of the viewport it is possible to select the quantity to be used for the contour. In this example reported is the stress component s11 (=sxx).

Viewing results



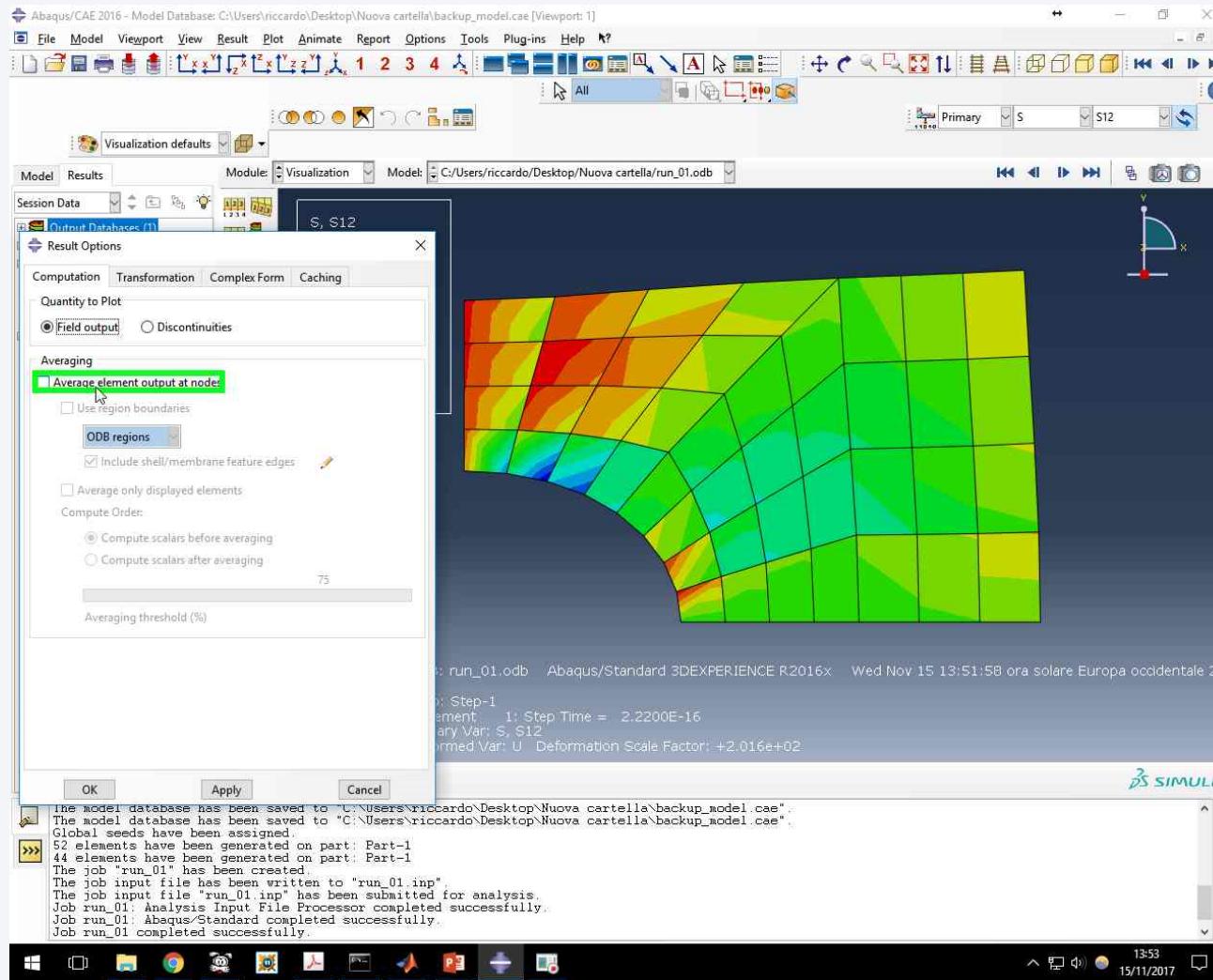
To visualize, for instance, the in-plane shear stresses, select s12.

Viewing results



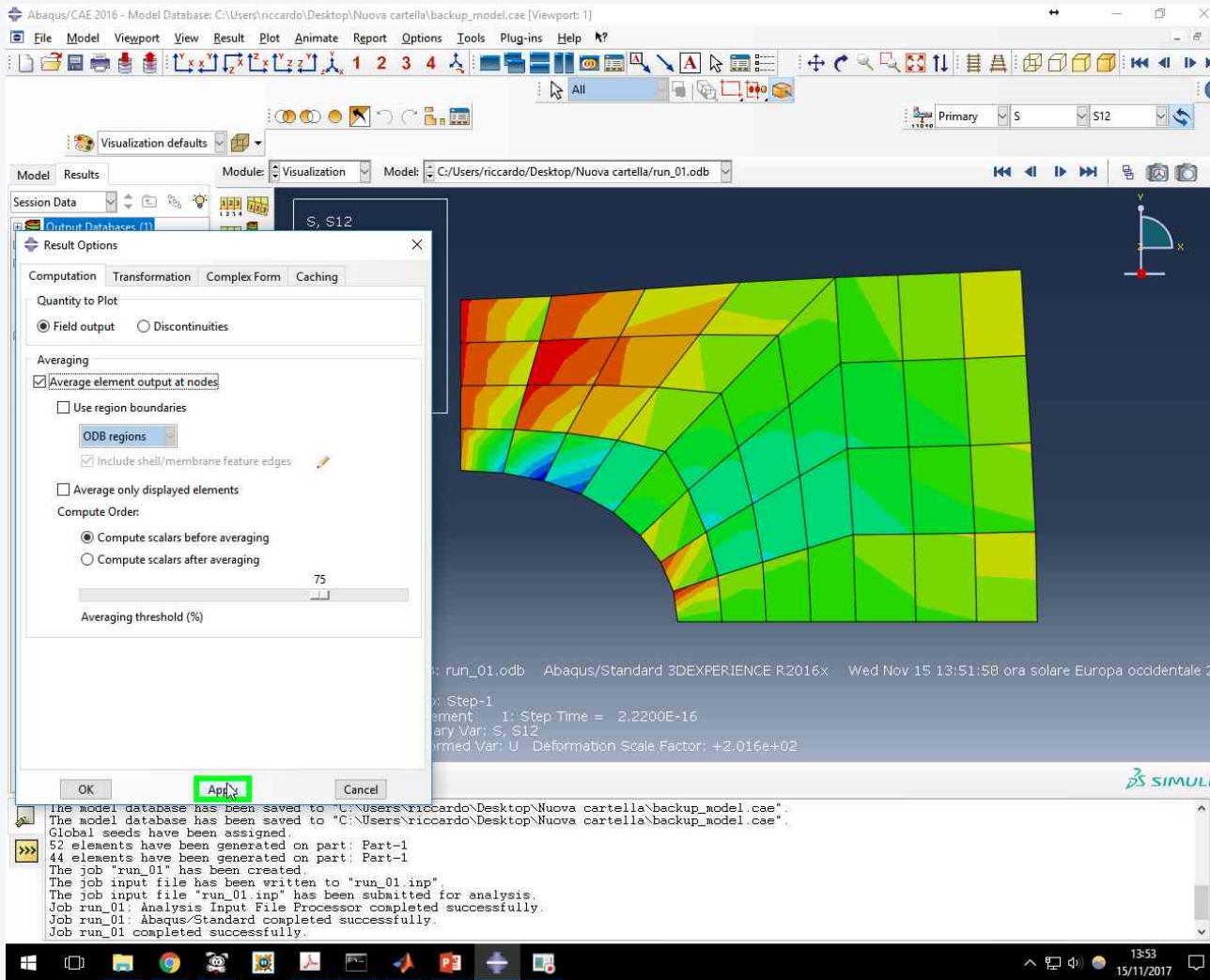
The contour can be realized with or without averaging the nodal values. In the current plot, the stresses are reported without averaging at common nodes of the elements.

Viewing results



The averaging option can be activated as shown.

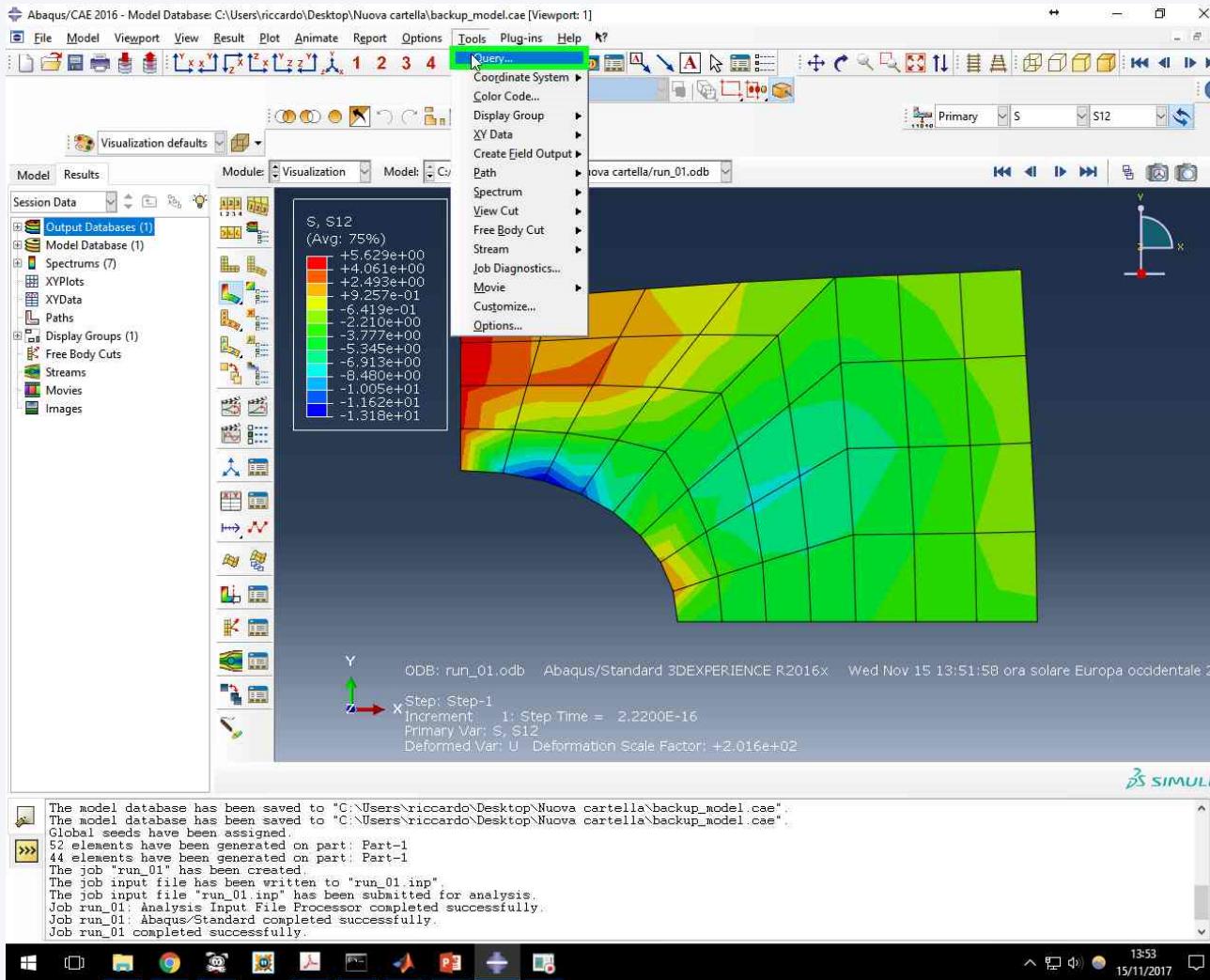
Viewing results



Confirm.

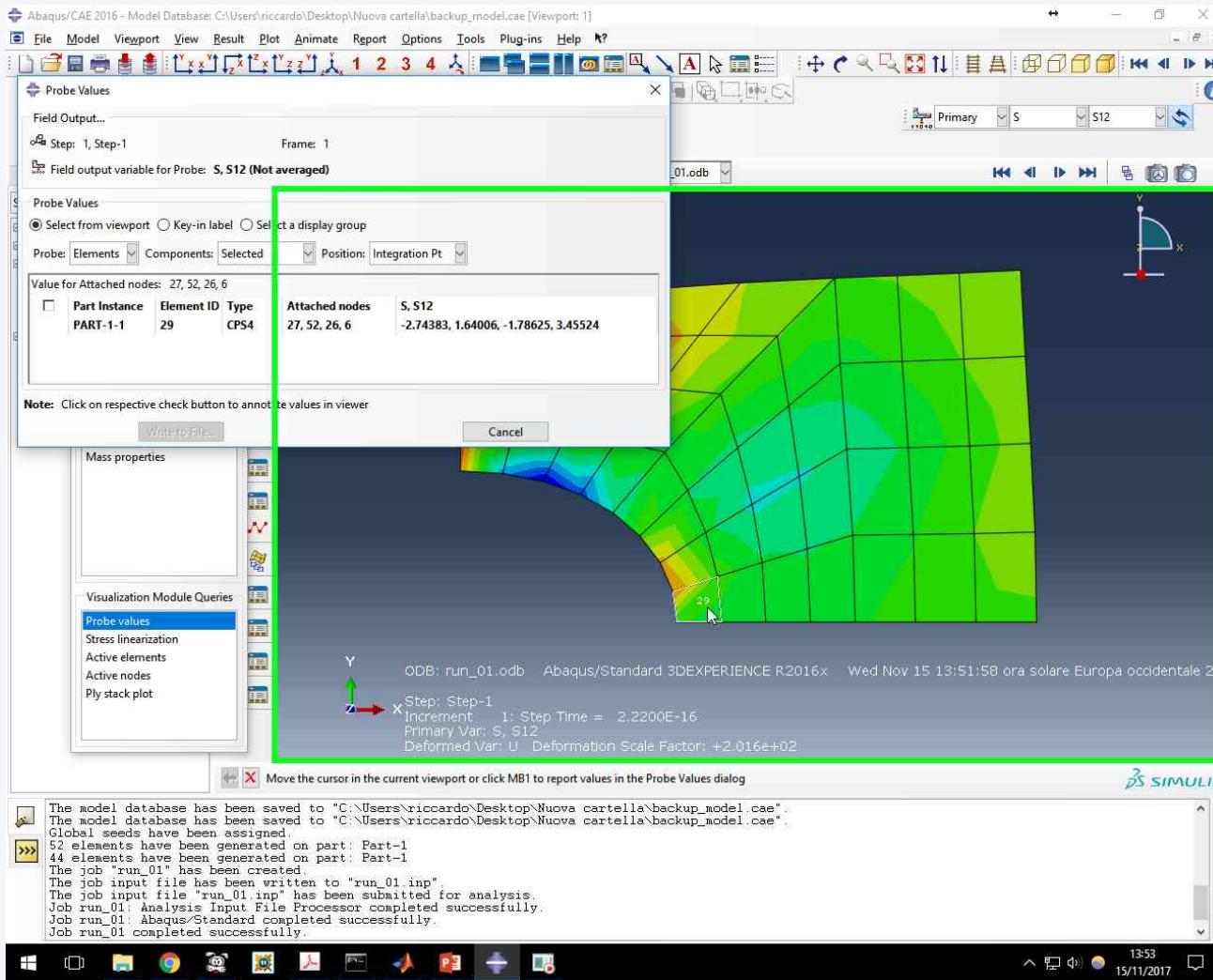
Due to the averaging process the plot is now more smooth. While the averaging process is helpful in providing nicer and more understandable plots, careful should be used to avoid not desired smoothing effects of stress discontinuities, as it happens in the presence of sudden variations of thickness, material properties, ...

Viewing results



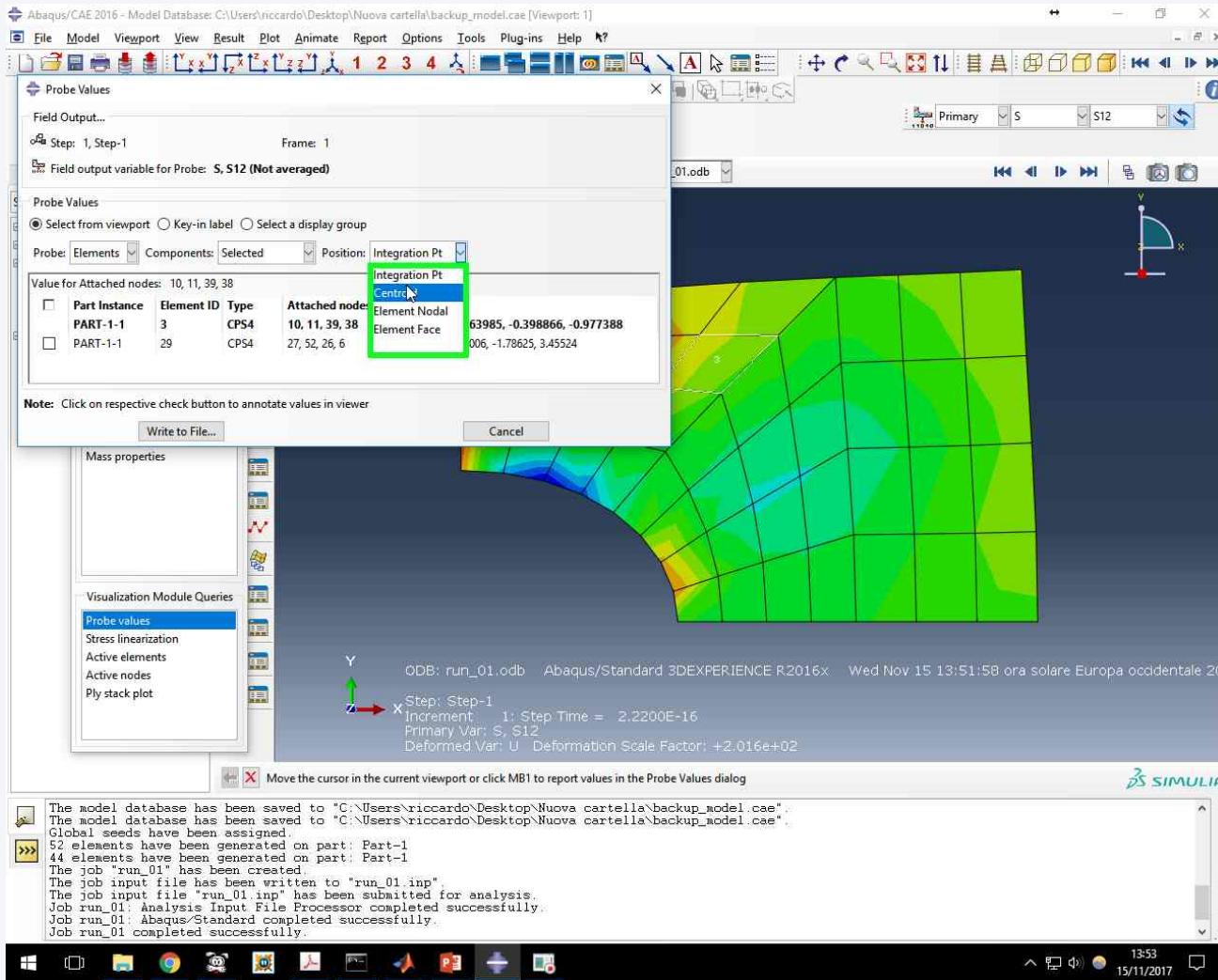
The local values can be required using the Tools → Query functionality.

Viewing results



By clicking on the elements, it is possible to visualize the stresses at the integration points...

Viewing results



... as well as their extrapolation to other positions. The centroid of the elements as well as the nodes.