Example
of Plate Elements

M. Salviato

William E. Boeing Department of Aeronautics and Astronautics

Seattle, Winter 2021

$\sigma_y = 100$ MPa

$D = 100$ mm

y

$R = 5$ mm

x

$D = 150$ mm

$u_x = 0$

$u_y = 0$
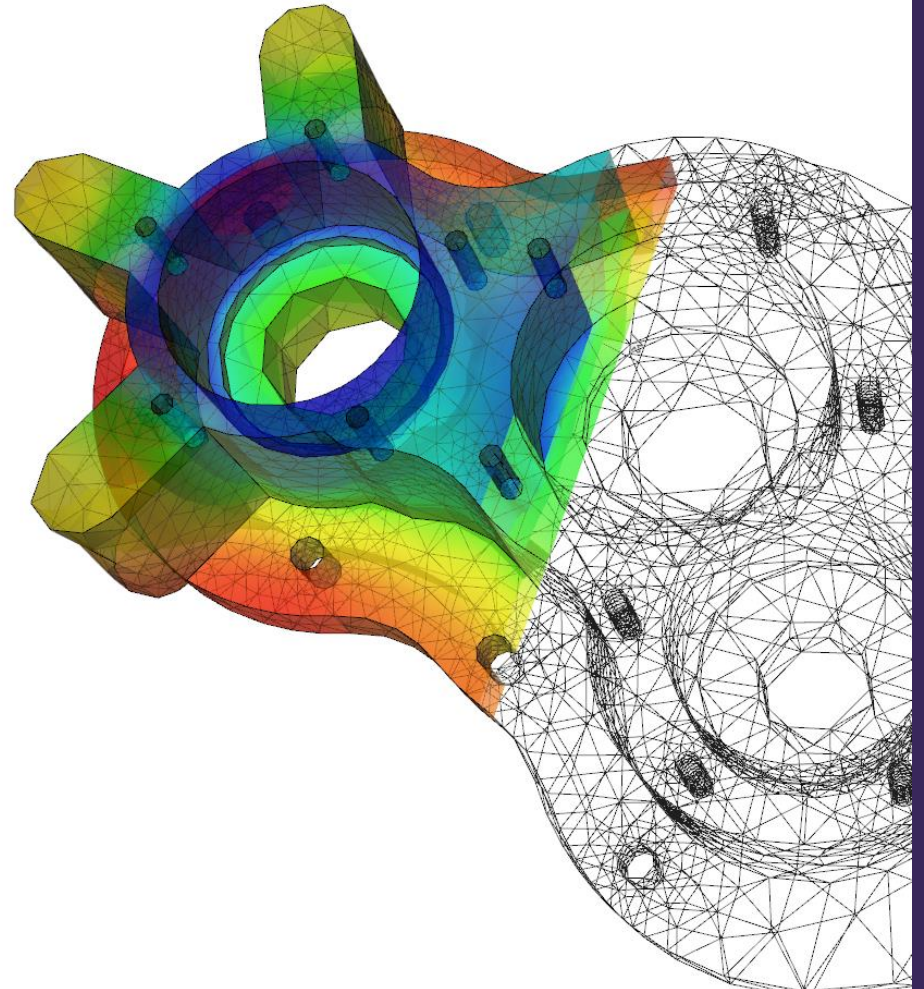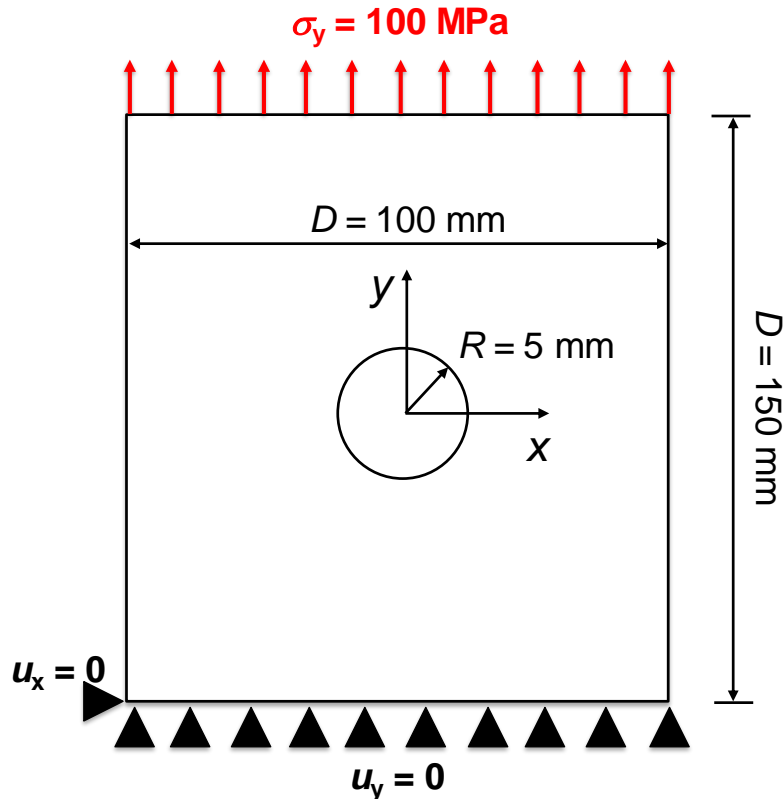
Consider the steel plate weakened by a circular hole in the figure;

The plate is assumed to be **homogeneous**, **isotropic** and **linear elastic.** The thickness of the plate is $t = 2$ mm and a **plane stress** condition is assumed.

Material properties:

Young's modulus $E = 206$ GPa

Poisson's ratio $v = 0.3$

Use ABAQUS/STANDARD to find:

a) the **maximum stress concentration factor**, $K_t = \sigma_{1Max}/\sigma_{nom}$ where σ_{1Max}= max value of the 1st principal stress and σ_{nom} is the nominal stress at the net section

b) the **maximum von Mises stress in the plate**. If the yielding stress of the plate is 350 MPa, will the plate fail?

WILLIAM E. BOEING DEPARTMENT OF AERONAUTICS & ASTRONAUTICS

UNIVERSITY *of* WASHINGTON

ABAQUS/CAE

- ABAQUS CAE is the Graphical User Interface of ABAQUS. It is used for pre-processing operations such as: *creating/importing geometries, defining material properties, assigning element types and meshing or applying BCs*
- Start ABAQUS/CAE. The program can be found at START/PROGRAMS/ABAQUS/ABAQUS CAE
- ABAQUS/CAE is divided into **modules**, where each module defines an aspect of the modeling process;
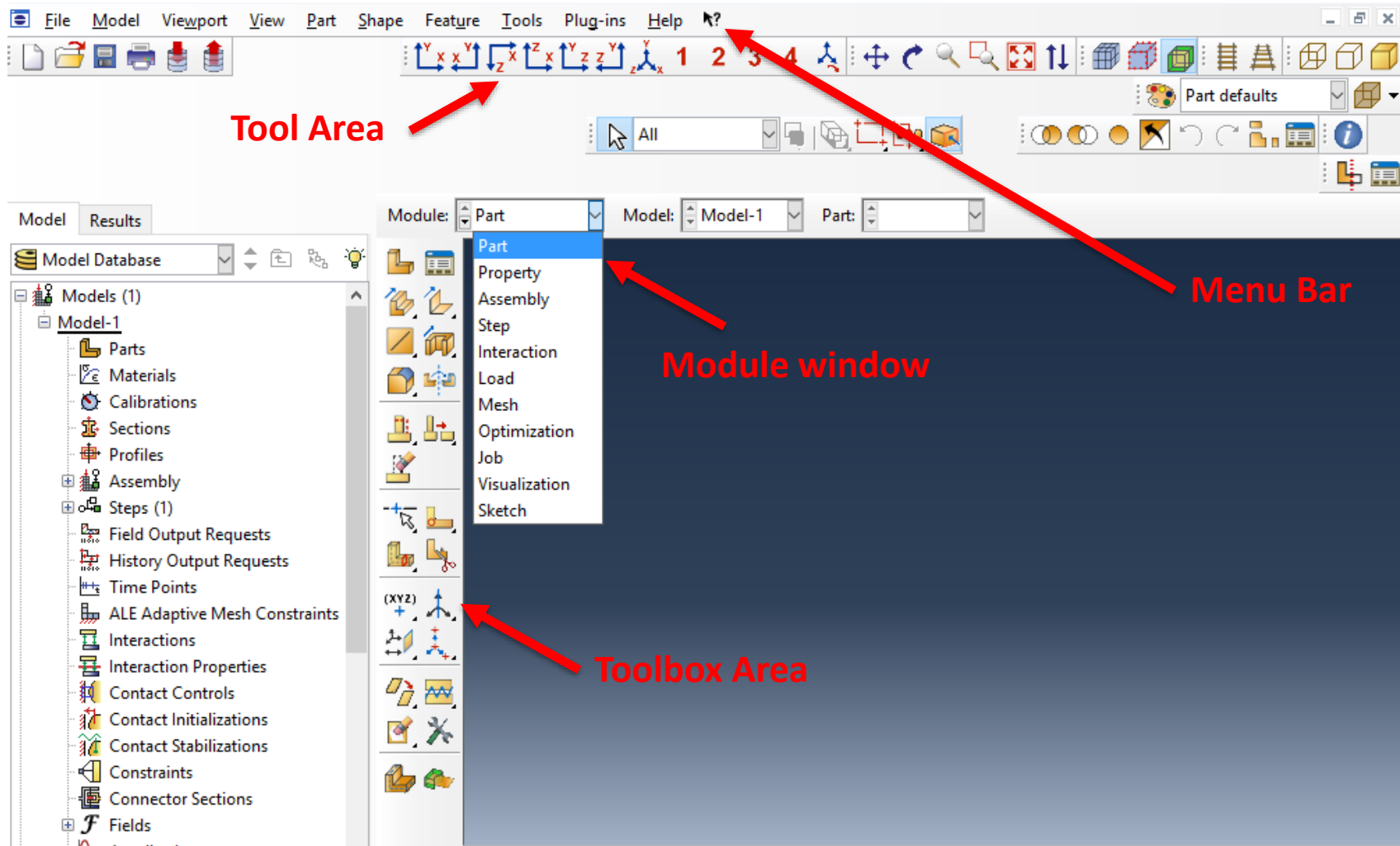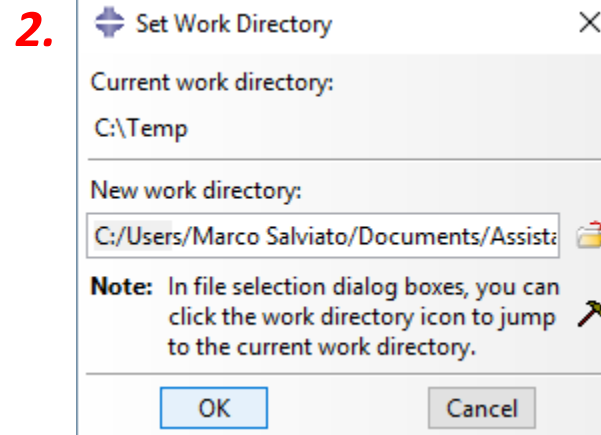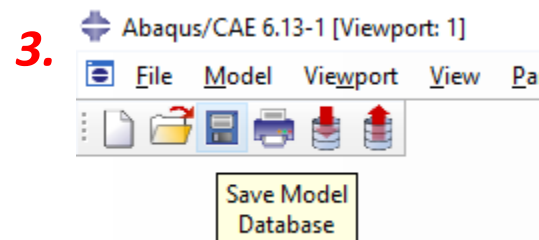
- Before creating the model is important to **set up a work directory**. **All the files related to the model and the simulation will be saved in this folder.** By default, ABAQUS will select C:/TEMP as work directory

1.
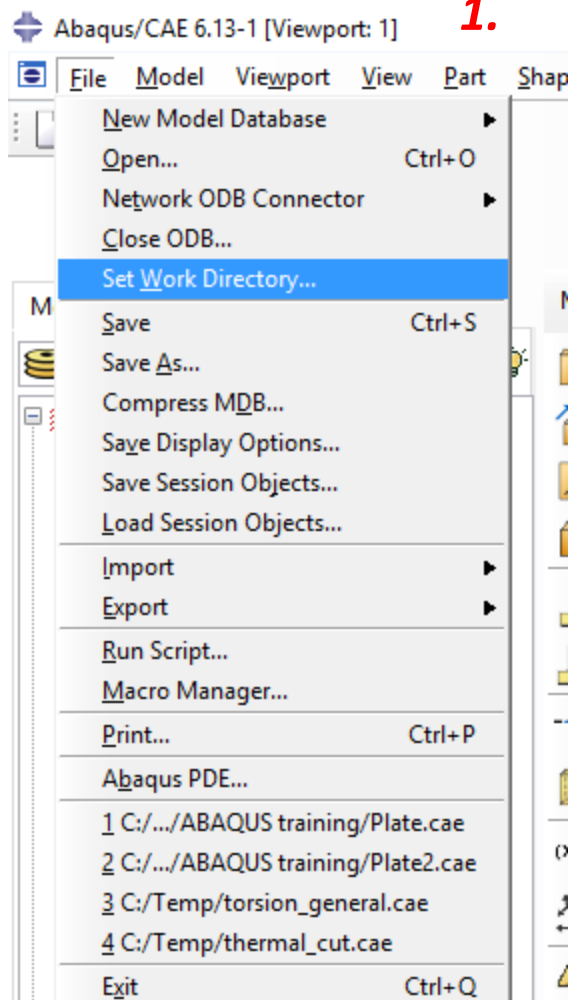
- **Click File/Set Work Directory**

2.

- **Select the path of the desired directory and click OK**

3.

- **Save the model using the "Save" icon.** From now on, all the files related to the model and analysis will be saved in the work directory

- The first step of the preprocessing consists in defining the geometry to be analyzed;

- **The geometry can be imported from e.g. an IGES** file created by other CAE software such as Solidworks Autocad, CATIA etc **or it can be created using ABAQUS/CAE**. The CAE software provided in ABAQUS is not as advanced as dedicated software such as CATIA or Solidworks so:

  - **COMPLEX GEOMETRIES:** create the parts in your preferred CAD software and import in ABAQUS

  - **SIMPLE GEOMETRIES:** use ABAQUS/CAE

- **<u>Since the part geometry is simple, ABAQUS/CAE will be used.</u>**
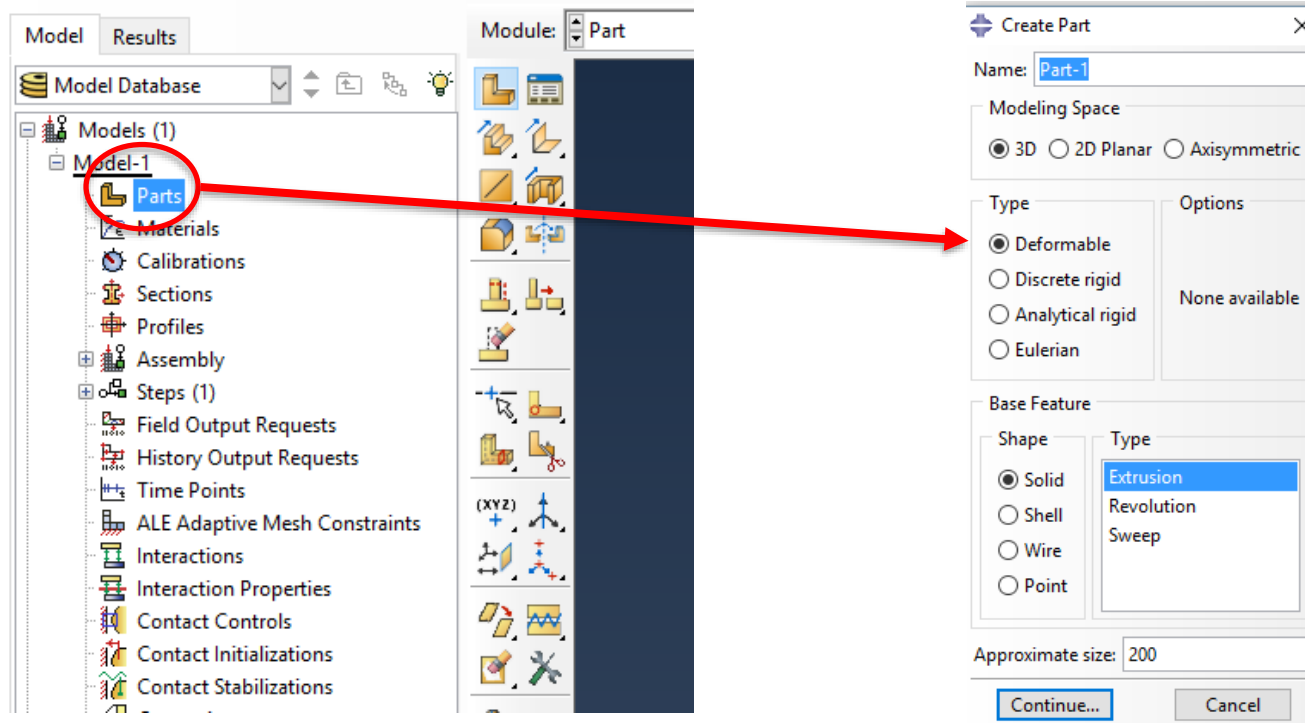
- In the module tree, click on **Parts** to open the **create part menu**;

- This menu is used to set up the main features of the simulation e.g. if the simulation is **2D, 3D** or **Axisymmetric**, if the body is **deformable** etc.

- This menu is used to set up the main features of the simulation e.g. if the simulation is **2D, 3D** or **Axisymmetric**, if the body is **deformable** etc. **It is the analyst's duty to find the correct approximation of the problem and to plan the type of schematization ahead of time**;

- Specify the Part name, e.g. "Plate", in the name field

- The modeling space feature is used to introduce assumptions on the type of problem to solve. The options are "**3D**" for three dimensional simulations, "**2D planar**" for two dimensional simulations (**plane stress** or **plane strain**) and "**Axisymmetric**" for bodies of revolutions (the axis of the body being the y-axis). *Let's select 2D planar since are assuming a plane stress condition*;

- In "type" you can decide if you want your body to be **deformable** (the body will be subjected to a displacement field due to external loadings), **Discrete rigid** (the body will be modeled as an assembly of rigid –non deformable- elements), **Analytical rigid** (the body is treated as a rigid surface). **The plate is linear elastic so… select "deformable".**

- The model can be a "**shell**" if it a 2D geometry, a "**wire**" if it is a 1D geometry in 2D space or a material **point** (e.g. if it is going to be a concentrated mass). In our case we have a 2D geometry is 2D space so… **select shell**.

- This is used to provide a rough estimate of the greatest dimension of the part. It is used by ABAQUS/CAE to prepare the sketch plane. **Keep 200 then click Continue…**

- This module allows the creation of the part. **The main commands can be found in the toolbox area:**
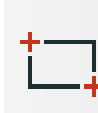
- Creates a point at desired coordinates

- Creates connected lines

- Creates a circle

- Creates a rectangle selecting 2 opposite vertexes

- Creates an ellipse selecting the center and two points in the perimeter

- Creates an arc tangent to an adjacent curve

- Creates an arc from center and the two endpoints

- Creates an arc through 3 points

- Creates a fillet between two curves

- Creates a spline through points

- Creates a construction line (this is not a part feature)

- Converts a line to a construction line

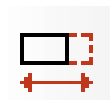- This module allows the creation of the part. **The main commands can be found in the toolbox area:**

- Trims unwanted lines

- Imposes geometrical constraints (e.g. perpendicular lines, parallel lines etc)
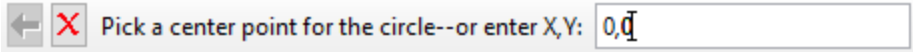
- Adds a dimension (very similar to Solidworks)

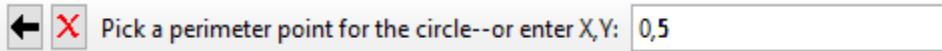- Modifies a pre-existing dimension

- **The commands above are the most used ones. For a detailed description consult the ABAQUS/CAE user's manual**

- Let's proceed with the creation of the plate. **We need to create a circle centered in (0,0) of radius R = 5 mm**

2. ← X Pick a center point for the circle--or enter X,Y: 0,0

1. (1)

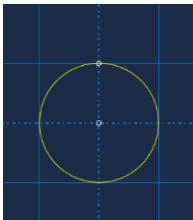**3.** ← X Pick a perimeter point for the circle--or enter X,Y: 0,5
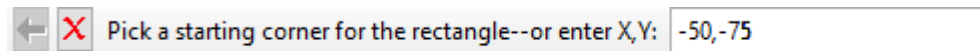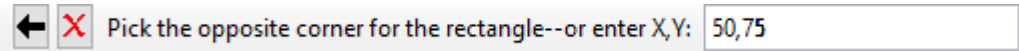
1. Click on the circle icon in the toolbox area;

2. Write the coordinates of the center point at the bottom left of the work space. Each coordinate should be separated by a comma;

3. Write the coordinates of a point at the perimeter of the circle, e.g. 0,5

- Let's create the plate.

2. ← X Pick a starting corner for the rectangle--or enter X,Y: -50,-75

1.

**3.** ← X Pick the opposite corner for the rectangle--or enter X,Y: 50,75

1. Click on the rectangle icon in the toolbox area;

2. Write the coordinates of the bottom-left vertex at the bottom left of the work space.

3. Write the coordinates of the top-right vertex at the bottom left of the work space.

- Click on "done' at the bottom left of the working space. Your part is created and showed in grey.
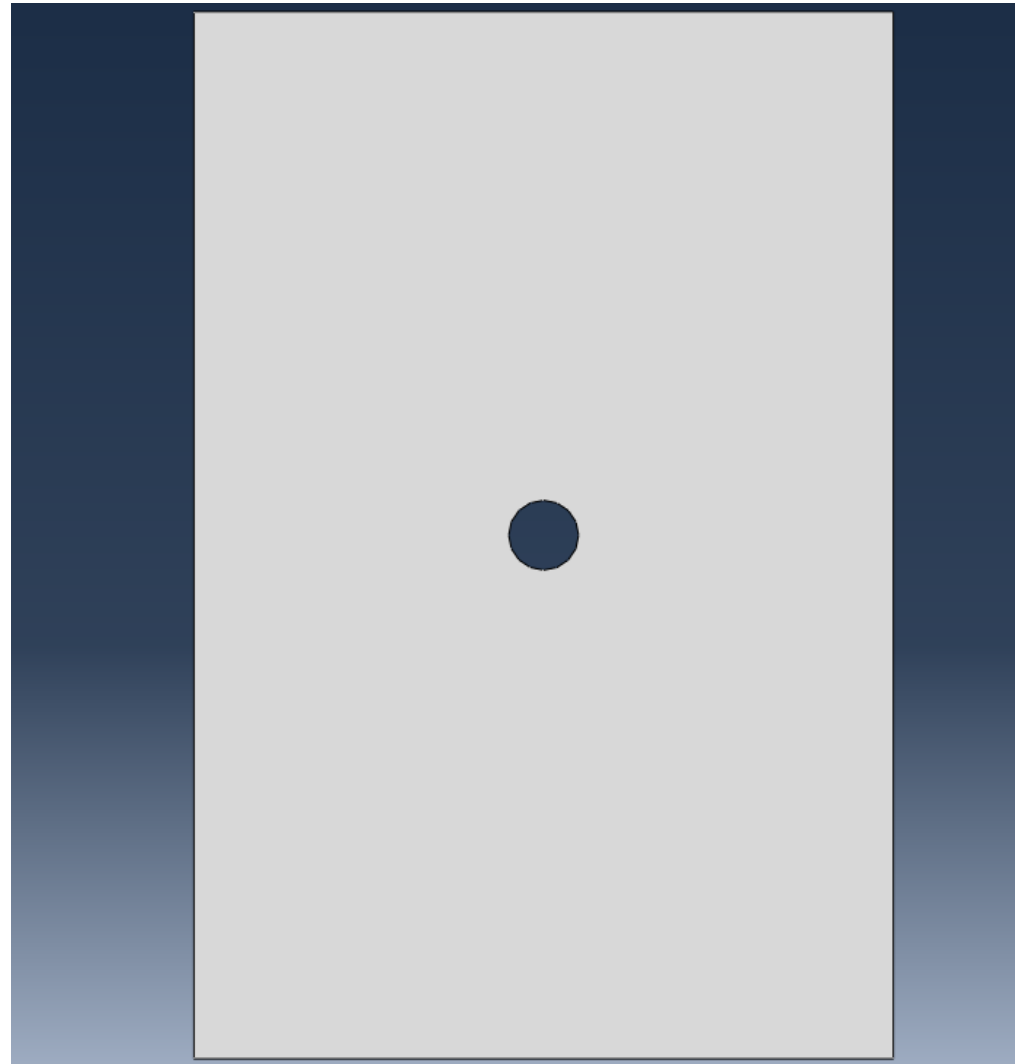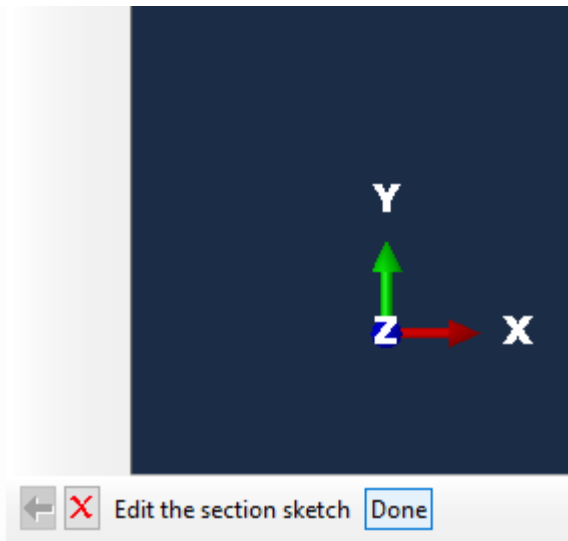
- We now want to assign the constitutive behavior of the material. Click on the **materials** module. This will open the edit material window.

- We want to do a structural analysis so we need to assign mechanical properties. **Click Mechanical/Elasticity/Elastic to open the material library for linear elastic materials**
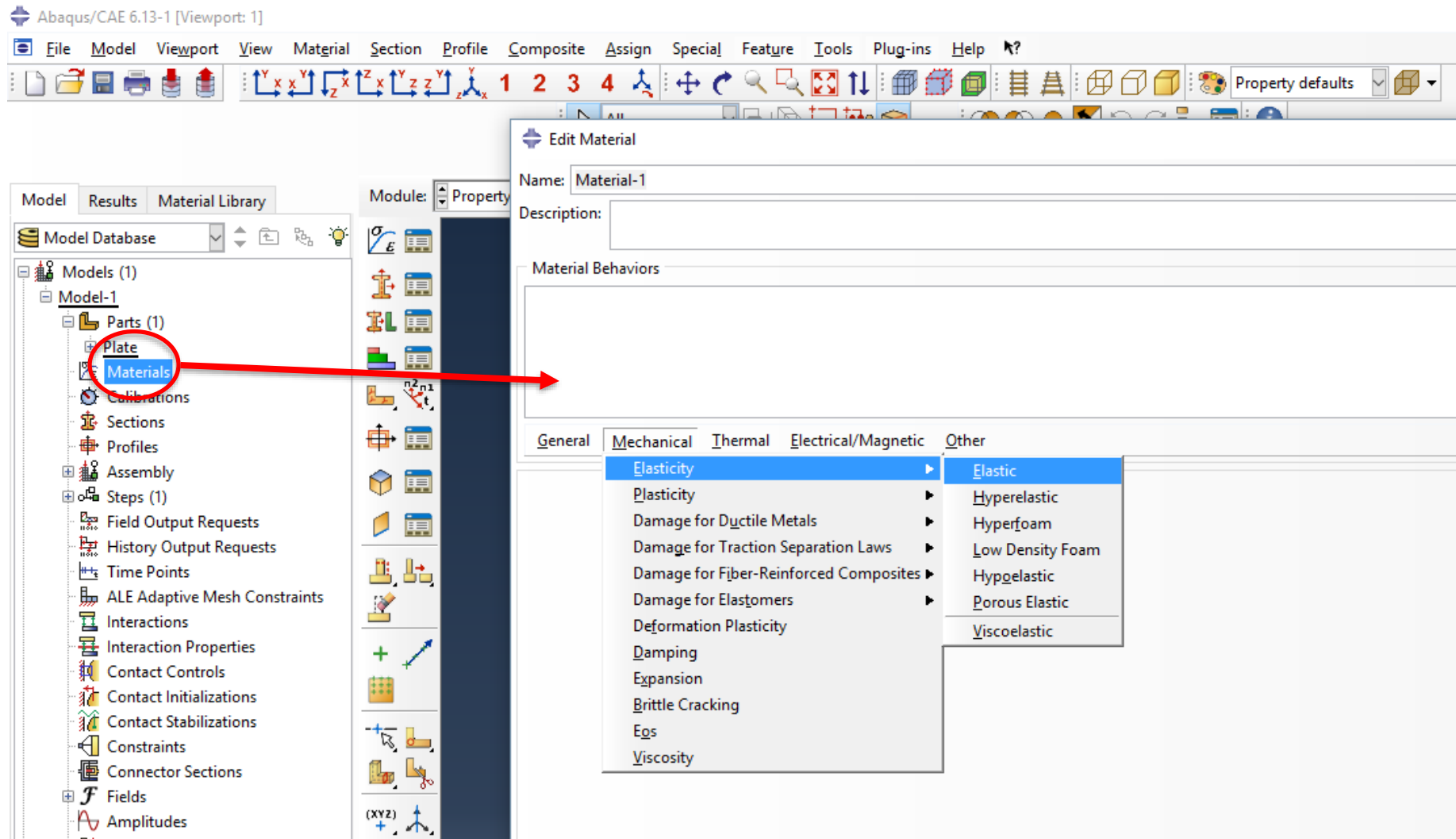
- In this field you specify the material symmetries (e.g. select **isotropic** if the material is spherical symm., select **orthotropic** if you have 3 symmetry planes etx). **Select isotropic as type (the plate is isotropic);**

- If the elastic properties of the material are temperature-dependent you can provide this information selecting related box. Leave it empty;

- In the data field, **input the Young's modulus and the Poisson's ratio in the chosen units**. (Since we are using mm for lengths and N for forces, E should be provided in MPa)

- Click OK at the bottom left of the card to create the material

- After creating the material, we need to assign it to the geometry. This can be done by creating a Section.

- **Click on Sections in the module tree**, this will open the "create section" menu

- In the "create section" menu, **assign the name "Plate"** to the section by completing the Name field

- In category you can select the type of geometry to which assigning the section. **Solid** refers to 2D solid elements, **Shell** to shell structures (translational dof + rotations), **Beam** refers to 1D components (Bars, Beams) other opens the library for thermal problems. **Select Solid**;
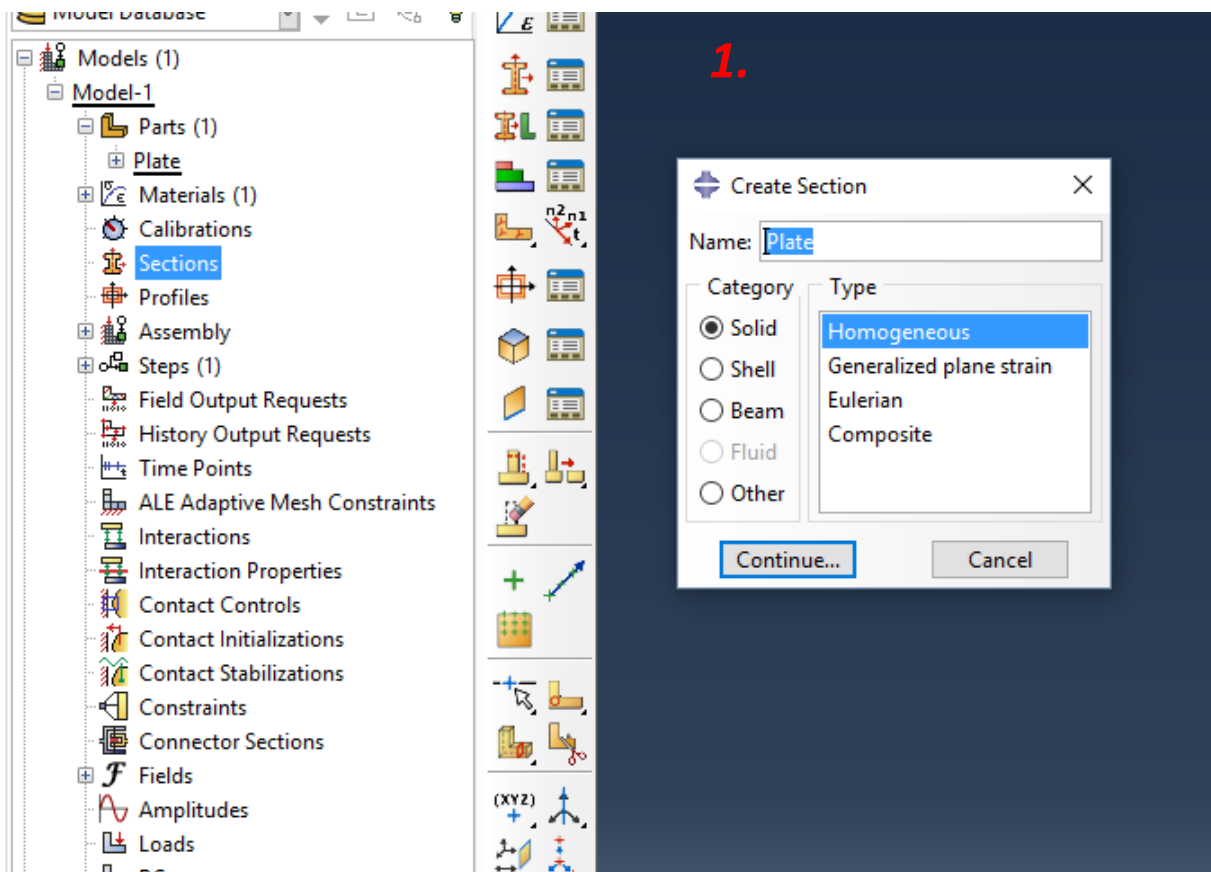

- In type you can specify if the material is homogeneous or if it is a composite material composed by various layers. **Select homogeneous**

- **Click on continue**

- In the new window **select Material-1 as material and 2 as plate thickness. Click OK.**
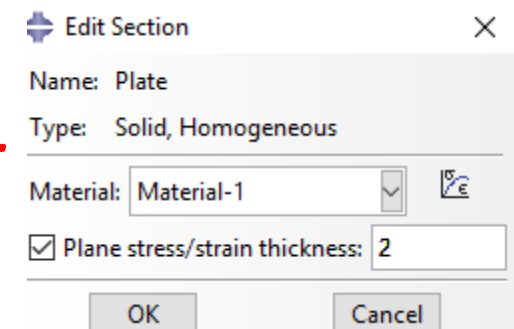
- After creating the section, we need to assign it to the geometry.
- Click on the "**Assign Section**" icon in the toolbox area and select the whole part. **Click Done at the bottom left.**
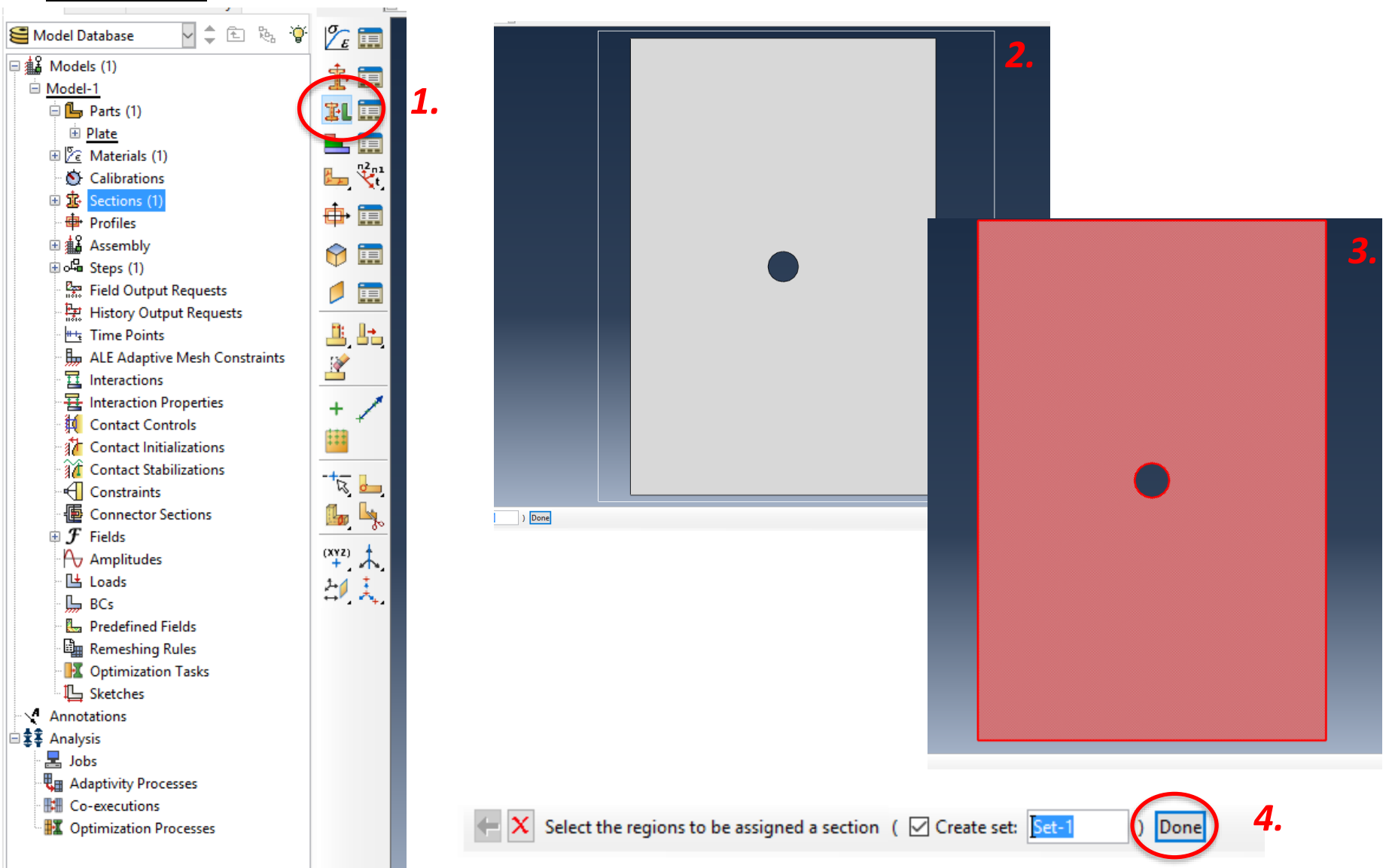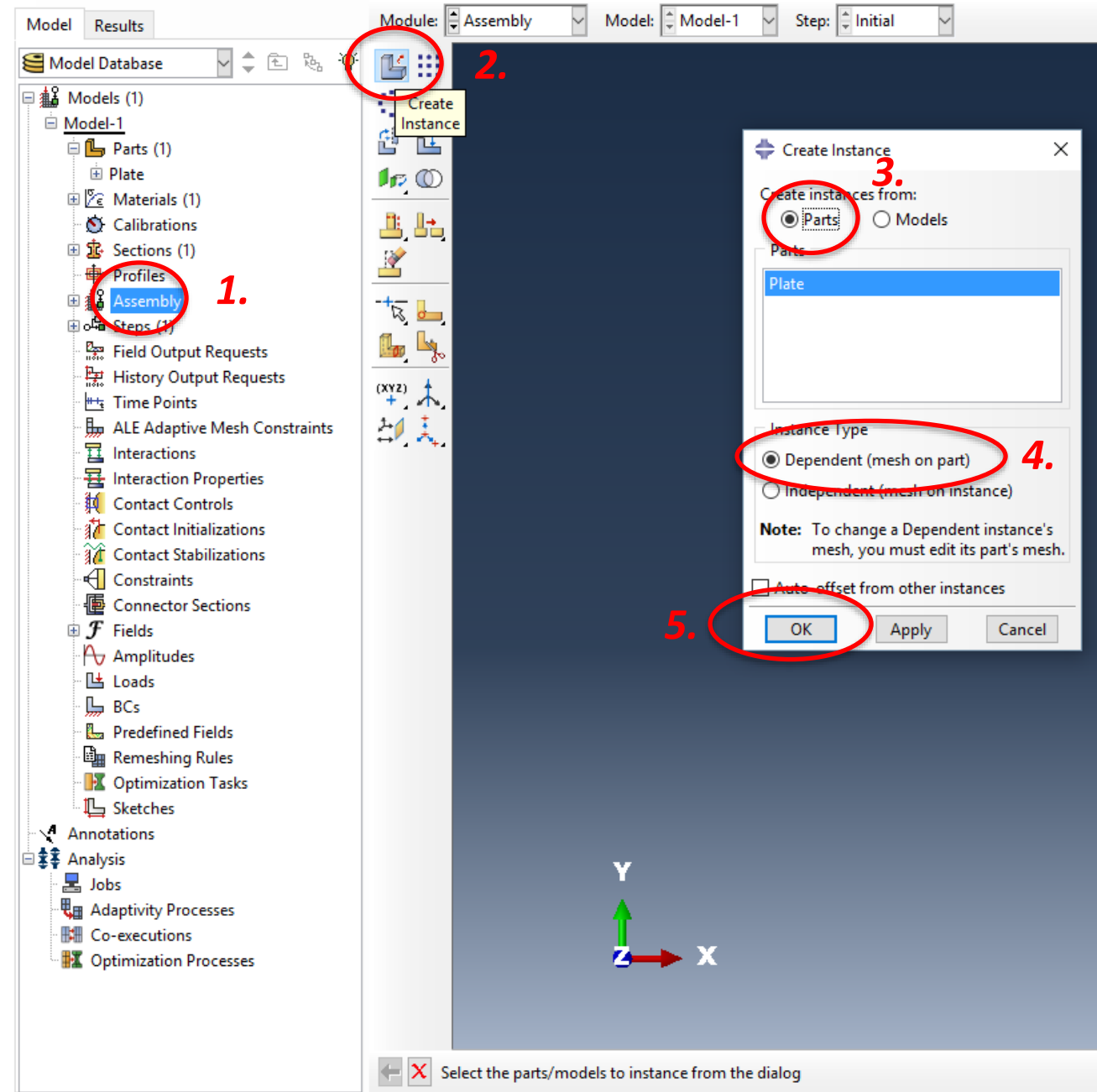
- **Select "Plate" the section to assign and click OK**. The geometry will be now in green showing that a section has been assigned.
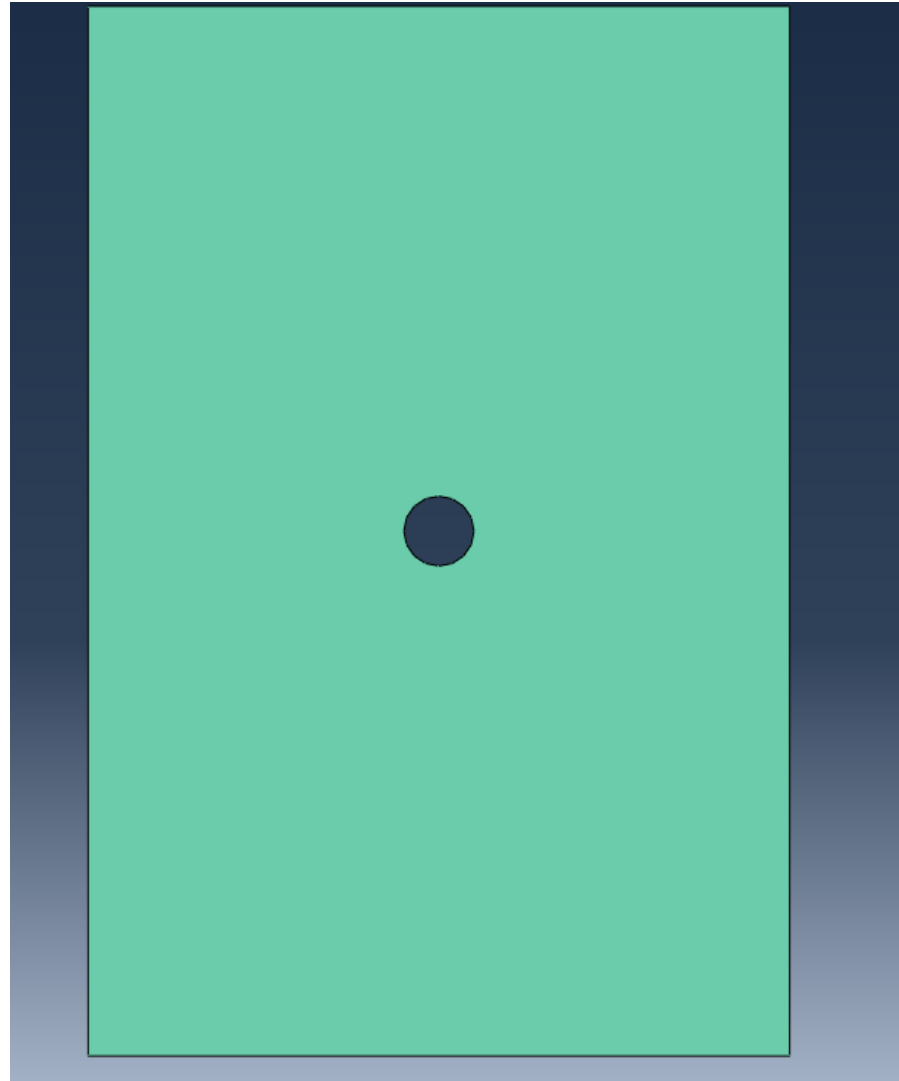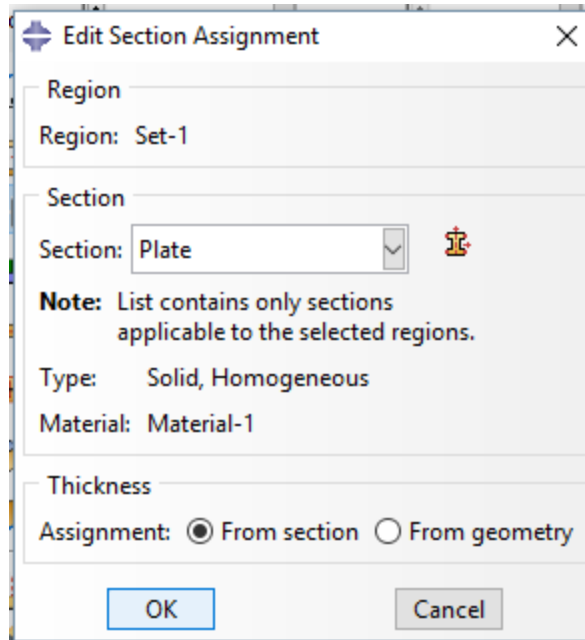
- After all the parts are created, we need to assemble them in the "**Assembly**" module**. In the current case, only one part is considered.**

- **Click on Assembly in the module tree then click on the part instance icon**

- **In the instance menu follow the instruction in the figure**

- This will create an assembly composed by part "Plate" only. The mesh will be created on the part rather than on the instance. This way, any change on the part mesh will be automatically reflected to the assembly
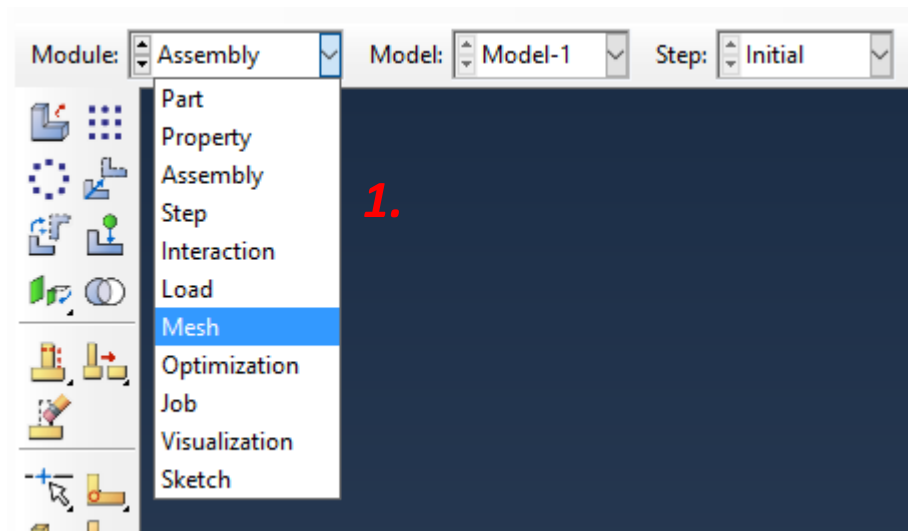
- The assembly should appear in blue in the Assembly module.
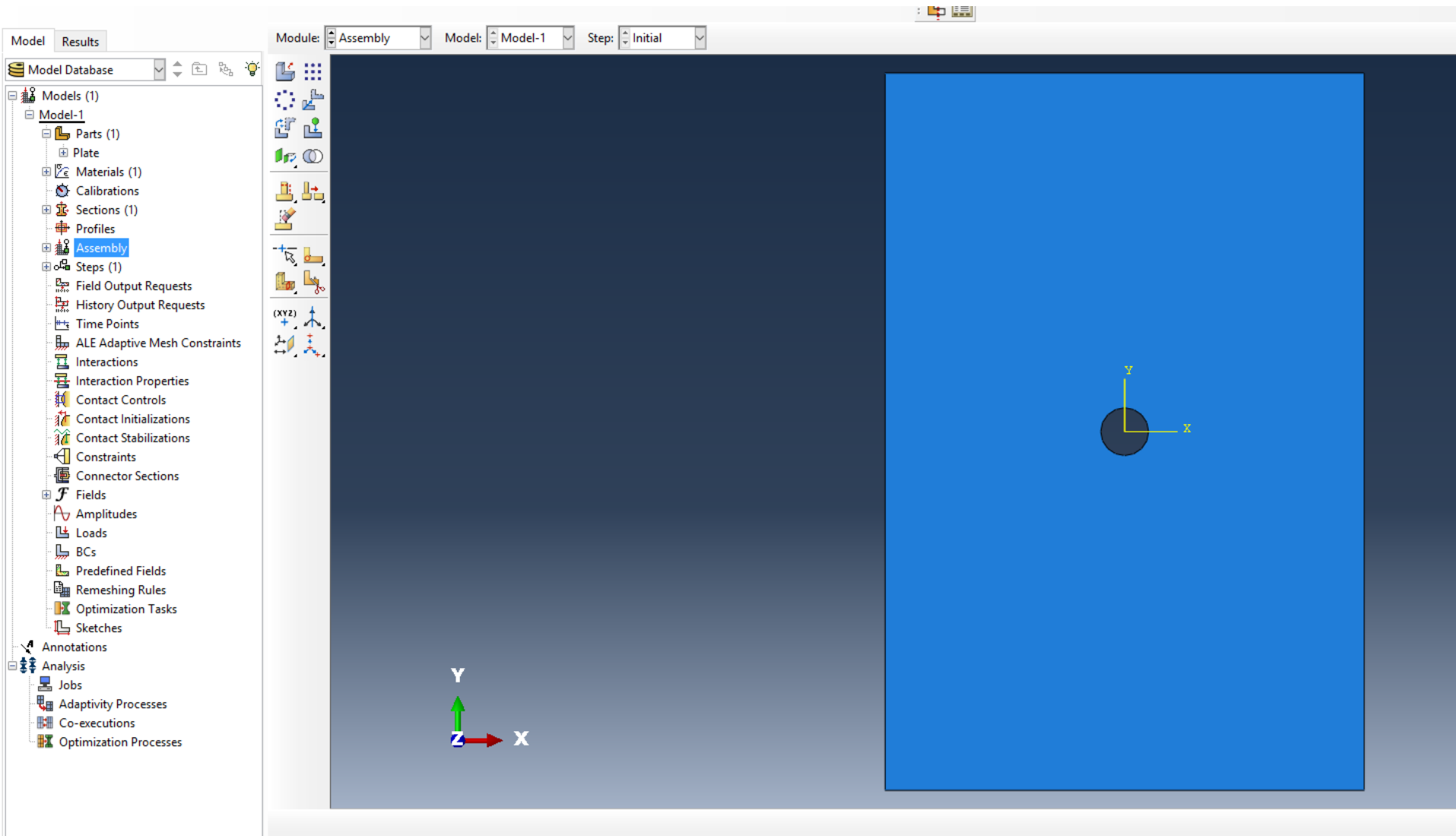
- We now want to create a mesh of elements to simulate the plate. The first step is to open the mesh module. **Click on the mesh module from the "module window"**. **Then select Part.**

- This will open the mesh module which allows you **a) selecting the type of element for the discretization of the problem b) defining the main parameters of the mesh c) checking the validity of the created mesh**
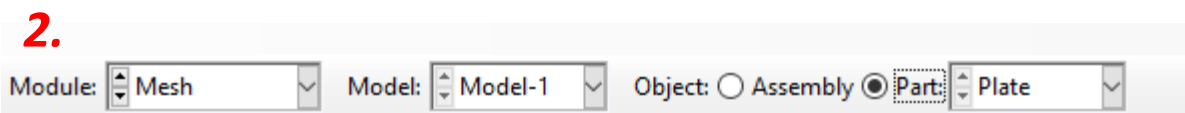
WILLIAM E. BOEING DEPARTMENT OF AERONAUTICS & ASTRONAUTICS
UNIVERSITY *of* WASHINGTON

- Let's assign an element type. We want to use a **8-node iso-parametric quadrilateral element (CPS8).**

- With reference to the following figure:

1. Click on the "element type" icon then Select the whole geometry and press "done"

2. **In the element type card, <u>select the library for standard simulation.</u>** Use this library when you plan to use the standard solver. The Explicit solver is mainly used for dynamic simulations. Since our simulation is not dynamic, we will use the standard solver**.**

3. **<u>In family, pick the plane stress option</u>**. In case you want to run a plane strain simulation, you just need to select the related family of elements.

4. **<u>Select the QUAD card.</u>** This will give you access to the options for quadrilateral elements. In case you want to use triangular elements, click on the TRI card**. <u>Select quadratic as the order of the element shape functions and keep the default options for the element</u>.**
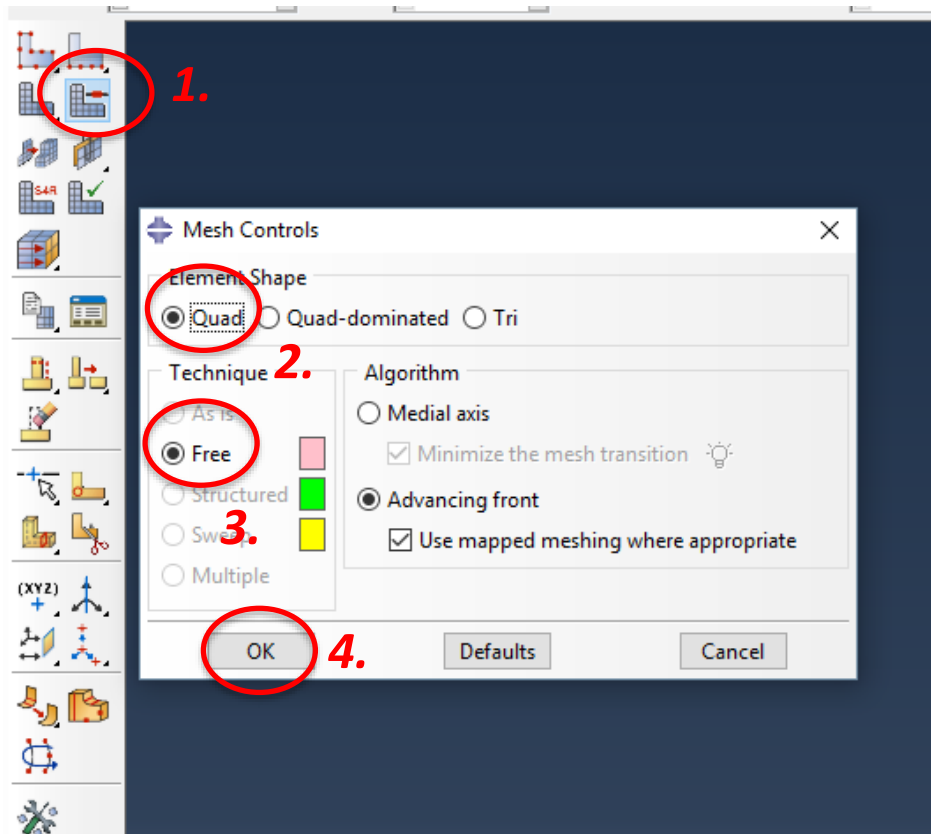
5. **<u>Press OK</u>**

- Let's assign an element type. We want to use a **8-node iso-parametric quadrilateral element (CPS8).**
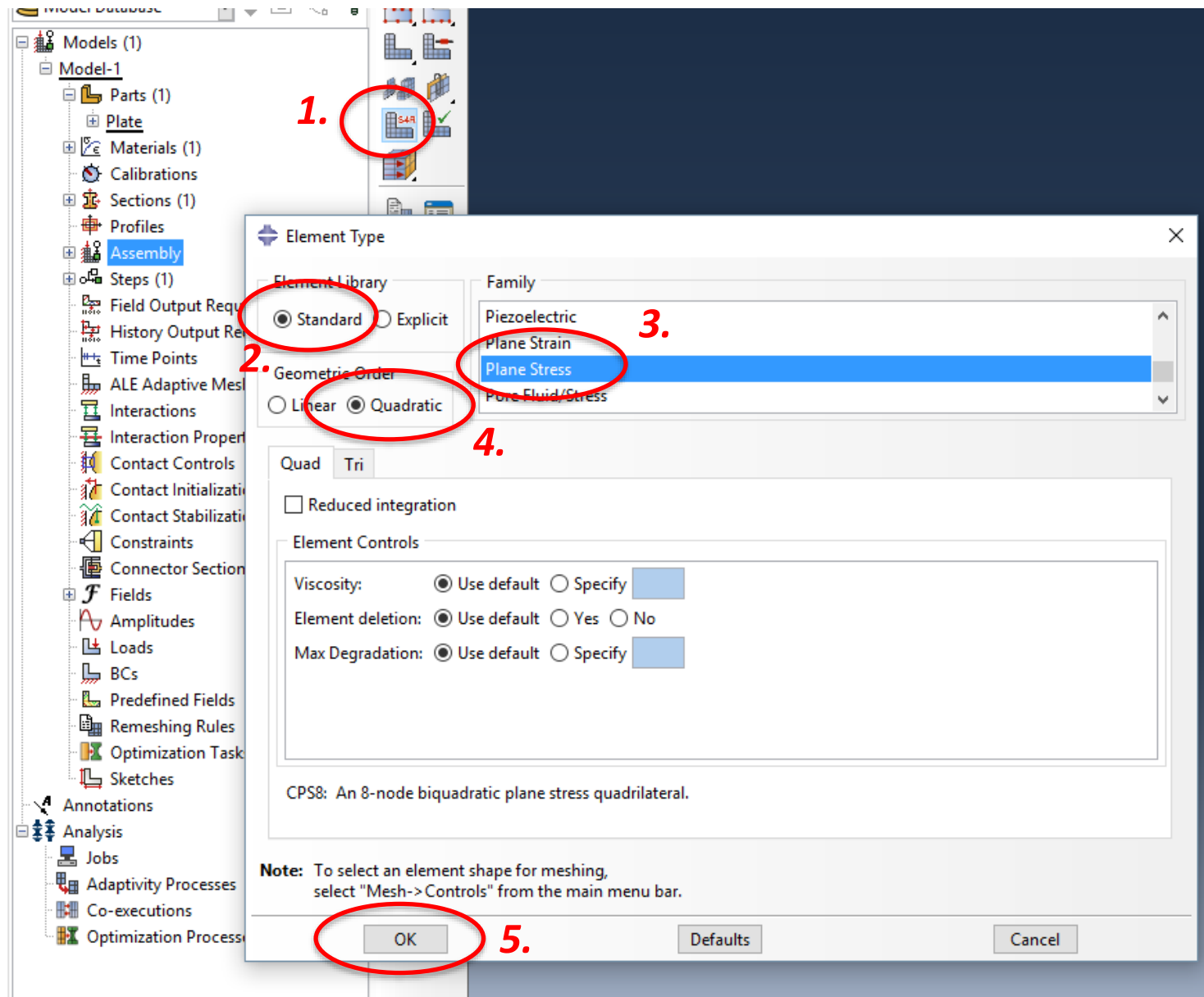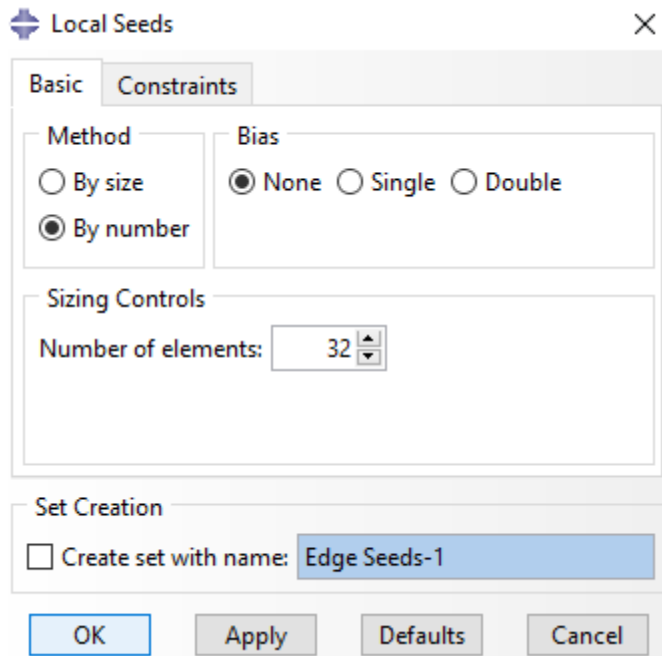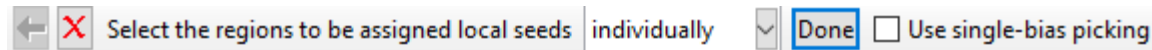
- Let's define some mesh controls.

- **<u>Click the "Mesh controls" icon</u>**

- We want quad elements only. **<u>Select Quad as element shape option</u>**

- We need to select a meshing technique. **<u>Let's select Free mesh.</u>** The free mesh allows the meshing of complex geometries. However, the mesh is not as regular as a structured mesh. **<u>Select Advancing front as algorithm</u>**.

- **<u>Press OK</u>**

- Let's define how many elements we want in the structure.
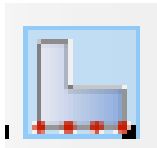
  - **Click on the "edge seed" icon then select the boundary of the circle in the part;**

  - **Click on Done at the bottom left;**

Select the regions to be assigned local seeds | individually | Done ☐ Use single-bias picking

Local Seeds ✕

Basic | Constraints

Method
- ○ By size
- ◉ By number

Bias
- ◉ None ○ Single ○ Double

Sizing Controls
Number of elements: 32

Set Creation
☐ Create set with name: Edge Seeds-1

OK | Apply | Defaults | Cancel

- In the local seeds card, **select "by number" as method**. This will allow to define how many elements we want along the boundary of the circle. Alternatively, one can select an element size.

- **select "None" as bias.** This will make sure the distribution of elements along the boundary is uniform.

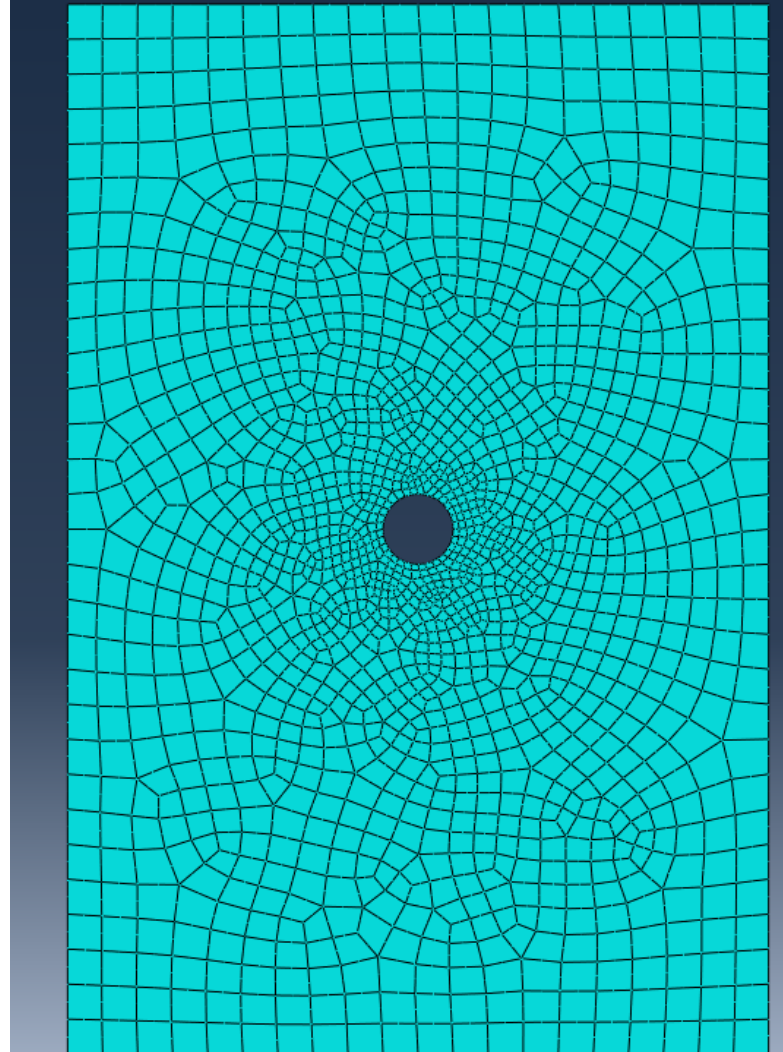- **input 32 as number of elements.** ABAQUS will place 32 elements along the boundary of the circle

- **Click OK.**

- Repeat the same procedure for the top and bottom edges and assign 20 elements. Your model should look as follows:
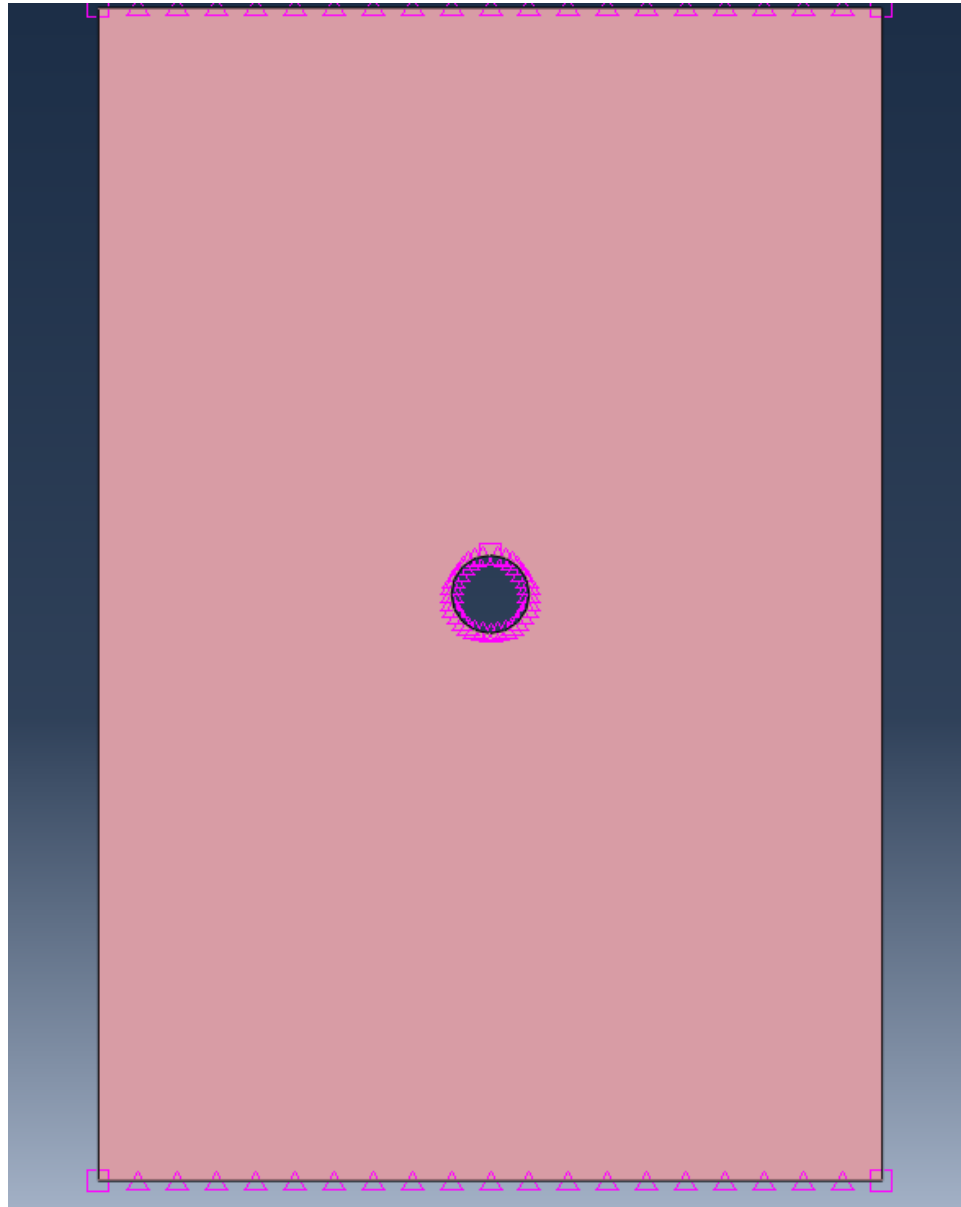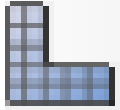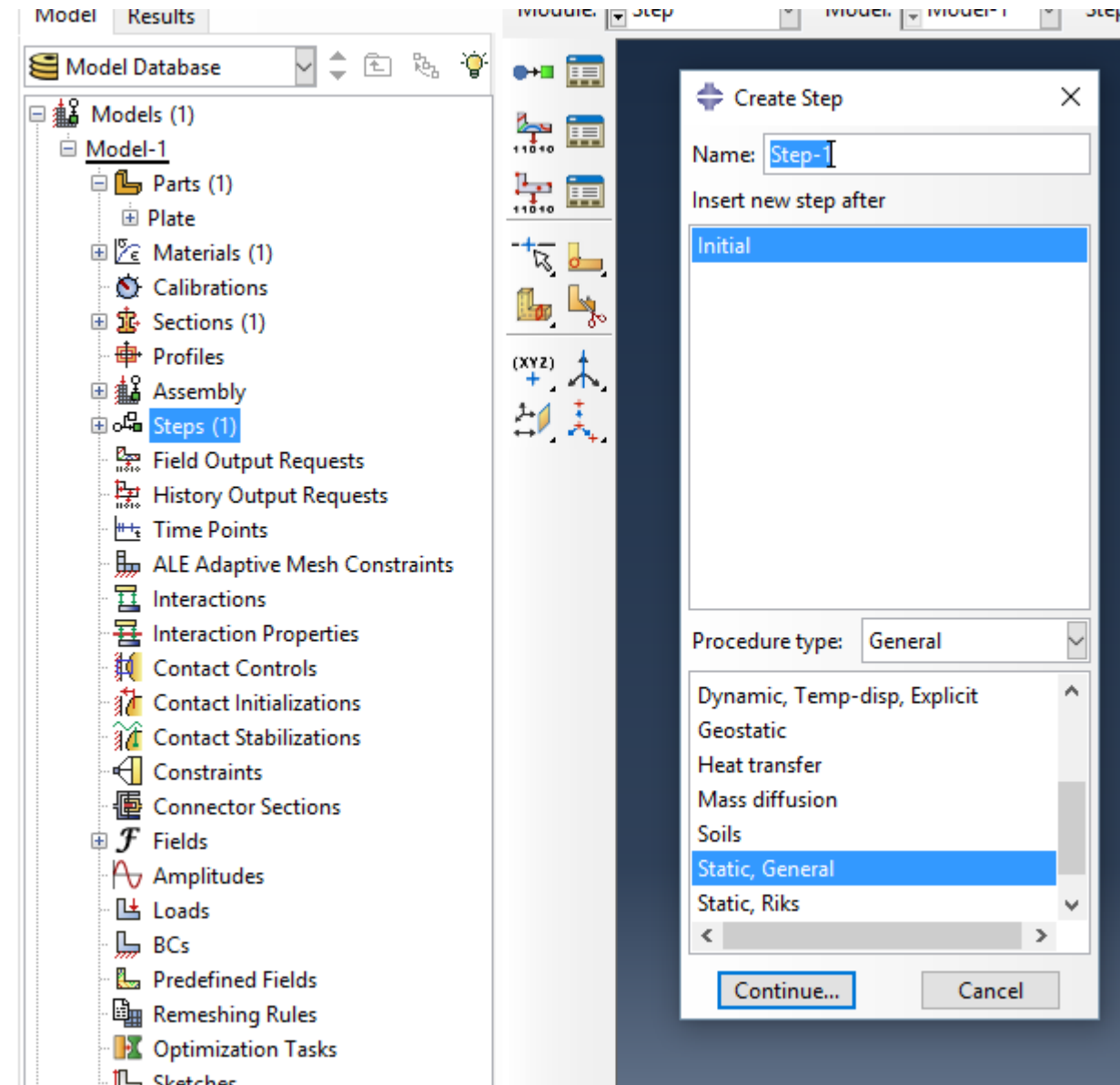
- Click on the mesh icon and press YES. Your model should look as follows.

- The next step is now defining what type of simulation we want to carry out. This is done in the **step module**

- **Click on Steps in the "module tree"**

- **Select a name for the step (e.g. Step-1)**

- **Select after which step you want to create it** (usually after the initial)

- In the procedure type, **select Static, General**. This will assign ABAQUS/Standard as solver. Since the type of problem to solve is static, this is the right choice for the current problem.

- **Press Continue…**

- In the edit step window, keep the default values (we will describe their use in the next tutorials)
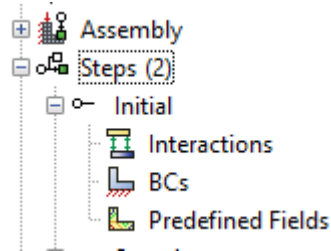
- We need to provide **Boundary Conditions**. Click on the + icon as below to expand the Initial Step card in the module tree. ***In the initial step we need to provide all the BCs provided by the constraints.***

- Double click on BCs and, in the new window, select the type of BC you want to apply. In this case we want to impose that $u_y = 0$ along the bottom edge. **Select "Displacement/Rotation" then press continue…**
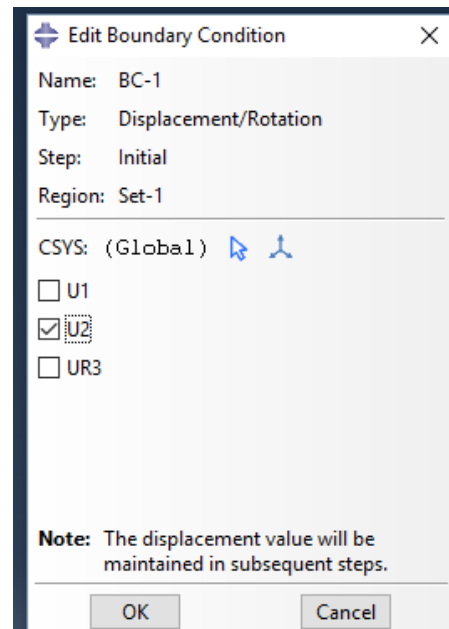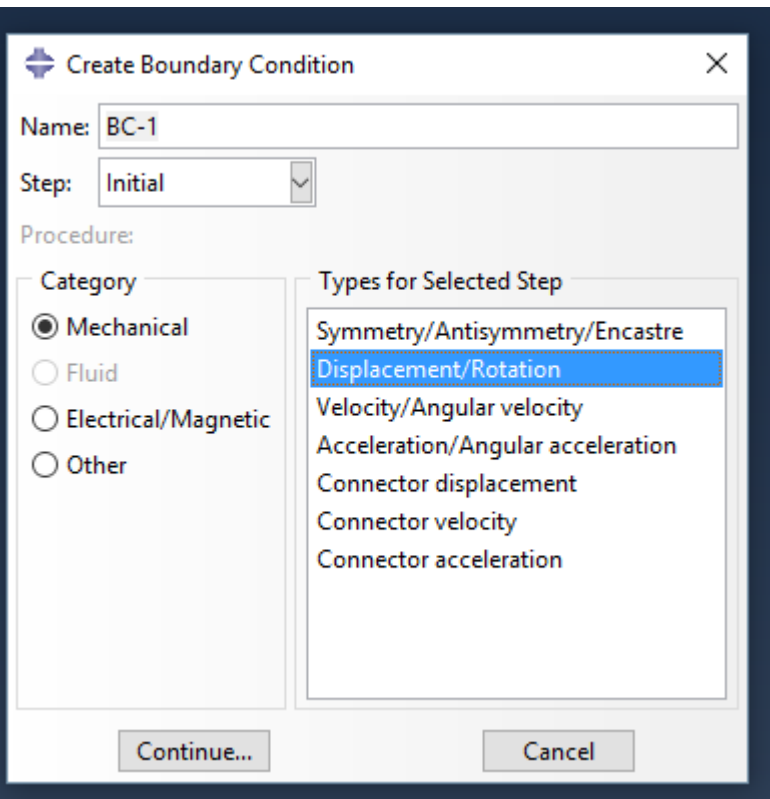
- **Select the bottom edge then select the displacement component to set to zero. In our case is u_2.** In ABAQUS x->1, y->2, z->3

- **Repeat the procedure to assign $u_x = 0$ to the bottom-left vertex**

- Now, we need to assign the external loading. This is done in the step which we created i.e. Step-1
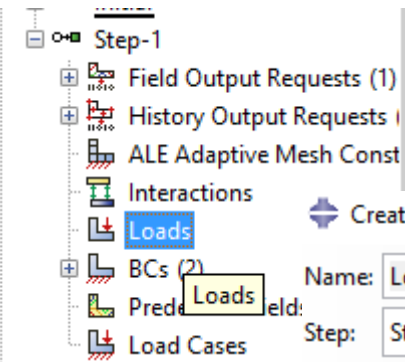
- **Expand the Step-1 tree and click on loads**
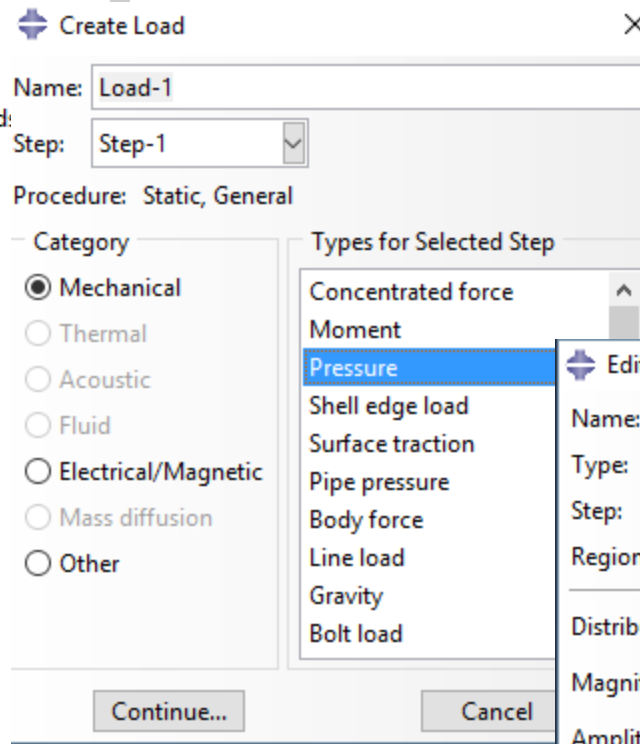
- **Select Pressure as type of load and click continue...**

- **Assign a uniform distribution of pressure of -100 MPa as shown**

- Note that, since we are defining a pressure, we need the – sign.

- **Press OK**

- This is how the model should look.
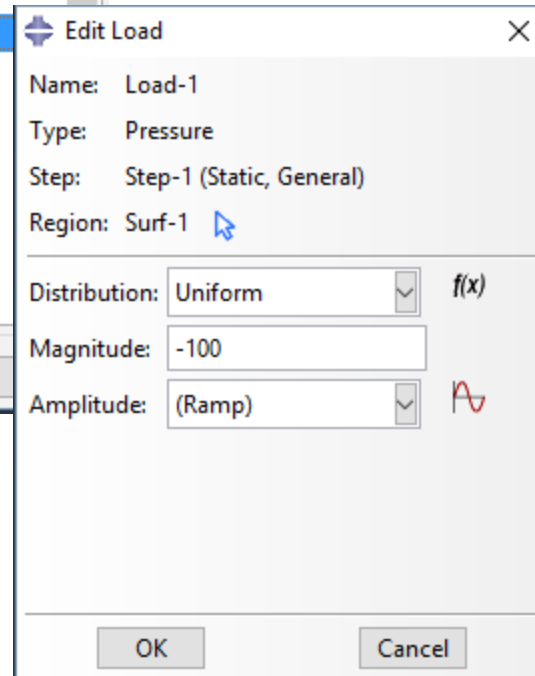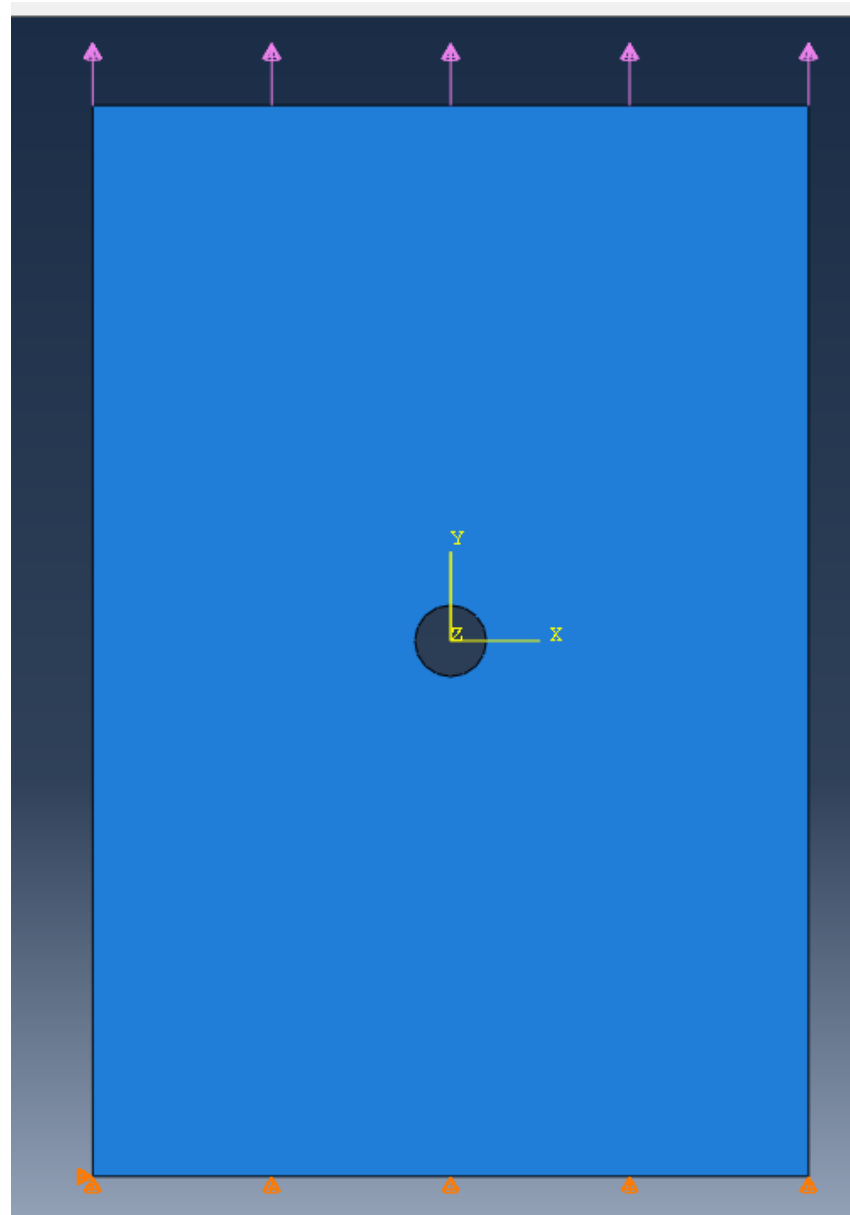
- The model is now ready for the analysis. **We need to create a job to submit to the solver.**
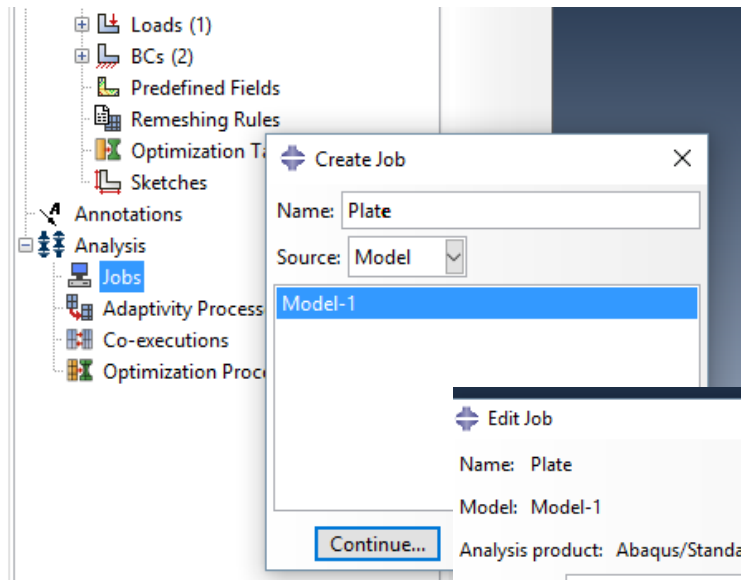
- Under the Analysis module, **click on Jobs**

- In the new window, **insert plate as name of the job and select the model you just created (Model-1) as source. Press Continue...**
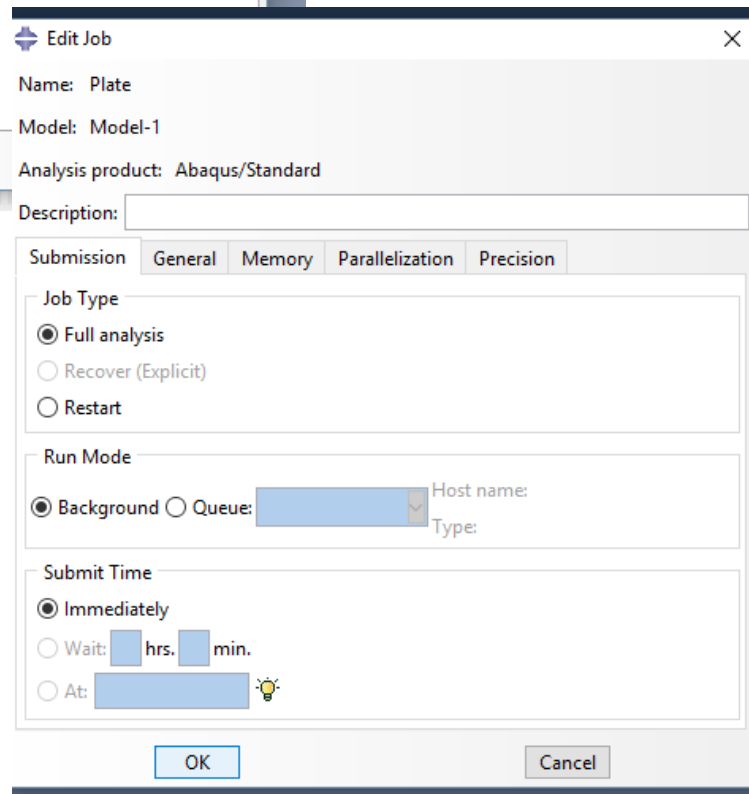
- **Keep the default values in the edit job window**. We will cover this part in the following tutorial

- The job is ready for submission. **<u>Right click on the job name and press submit!</u>**
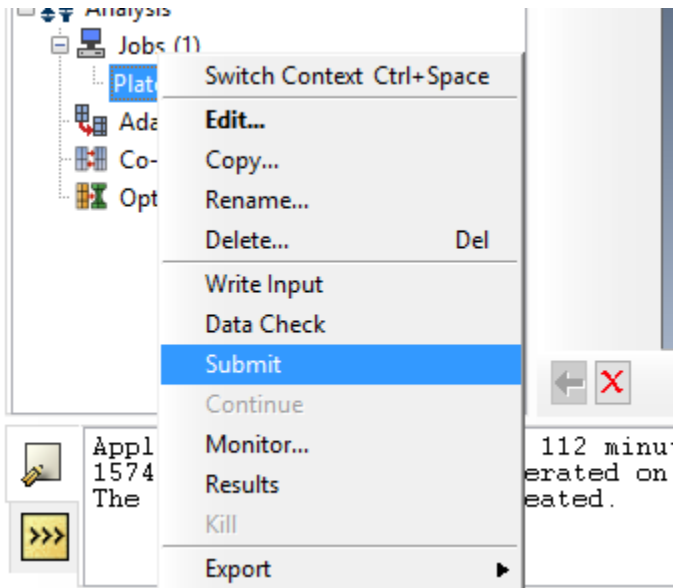
- Once the analysis is completed, the status of the job will change to completed
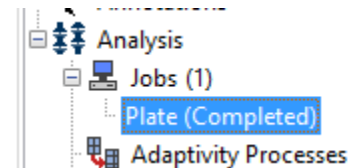
- To visualize the results we need to open the .odb file created by ABAQUS/CAE with the ABAQUS/viewer
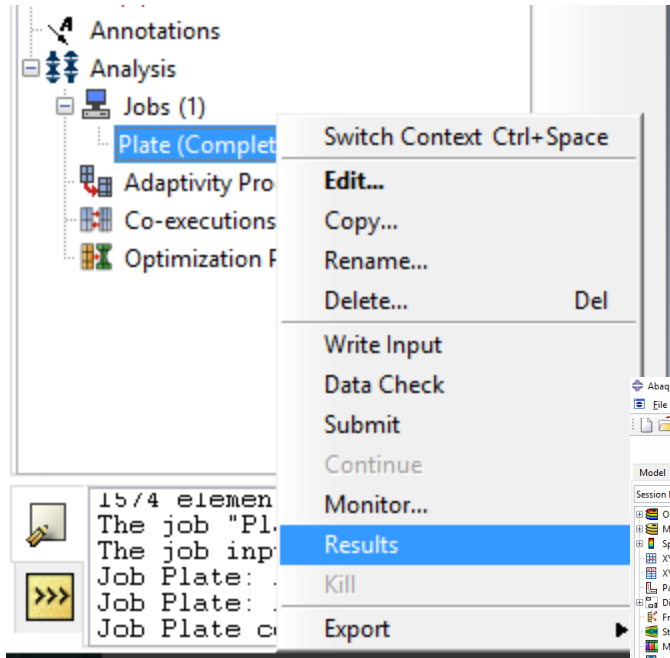
- **Right click the job and select "Results"**. This will open the viewer
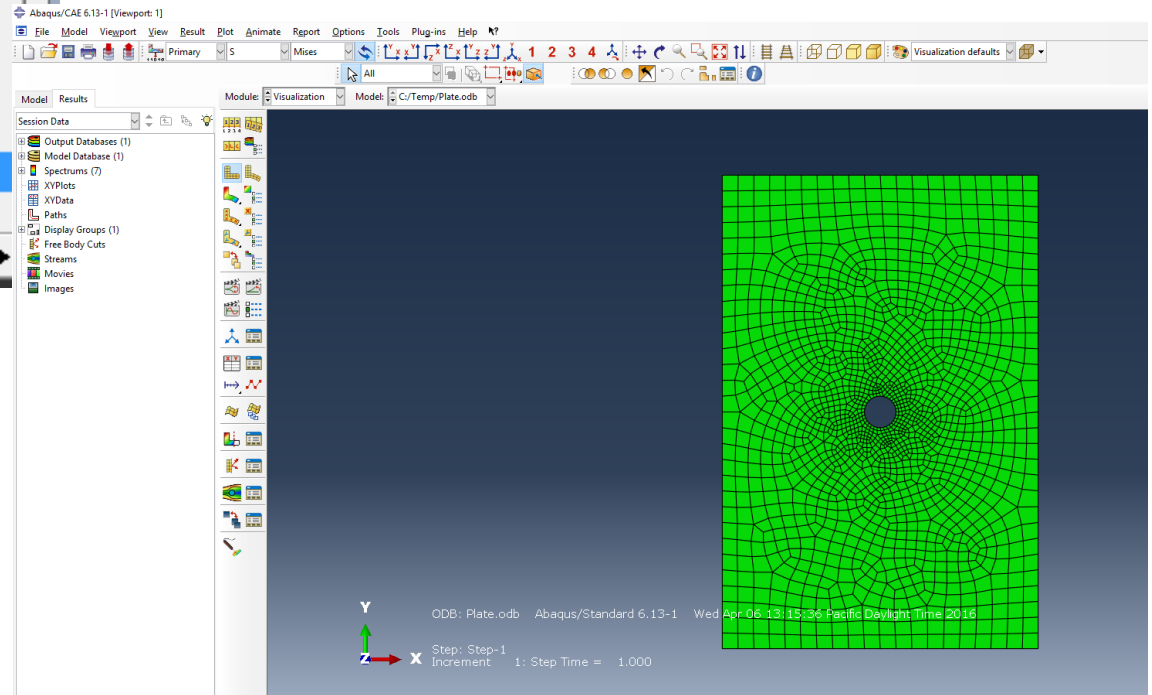
- Let's plot the deformed shape of the body.
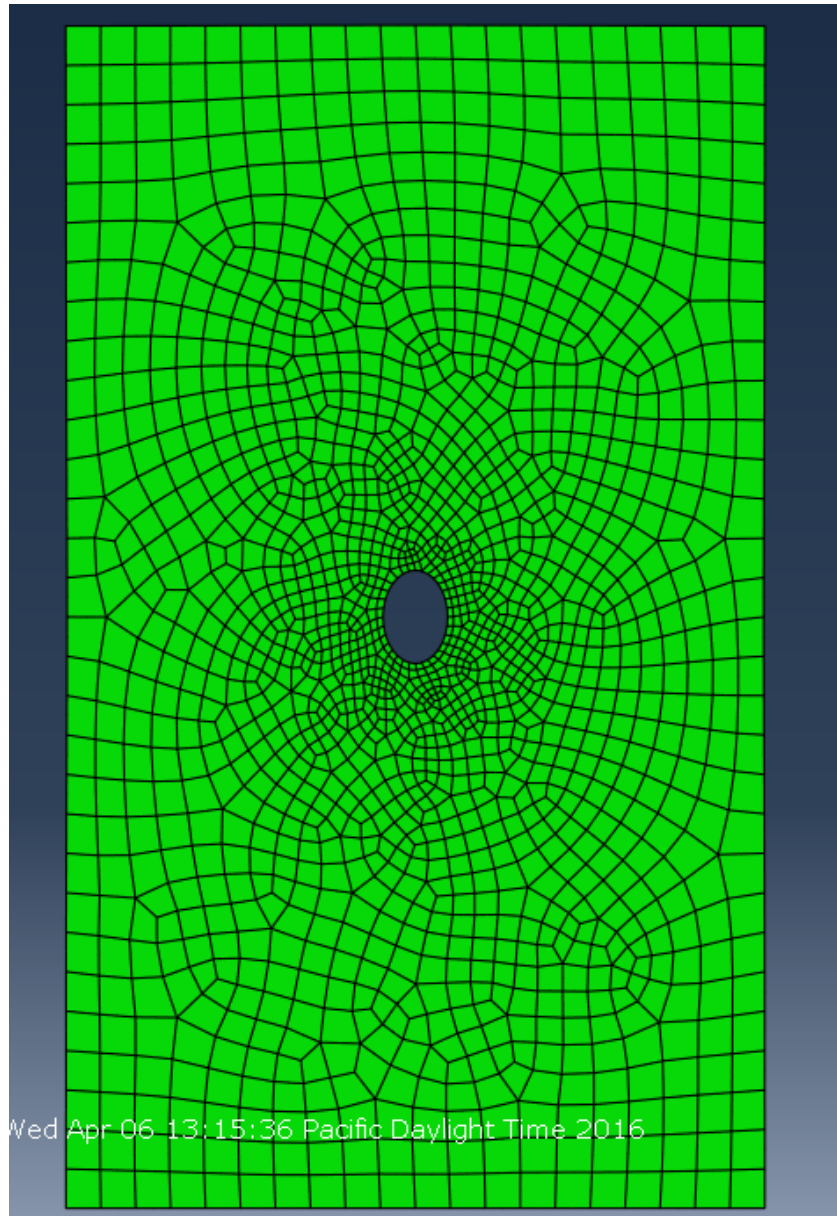
- **Click on the "Deformed Shape" icon**. This will provide you with the plot of the deformed shape.

  - **Use this icon if you want to change the scale factor to e.g. 100**
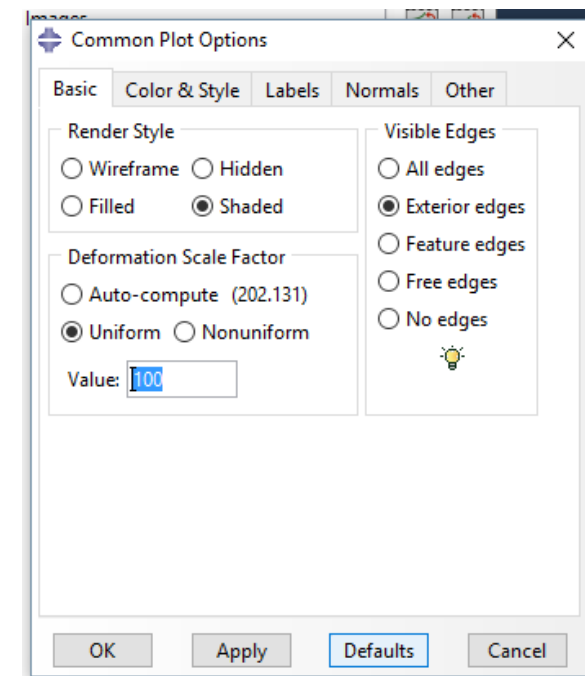
- Let's plot the maximum principal stress in the body.
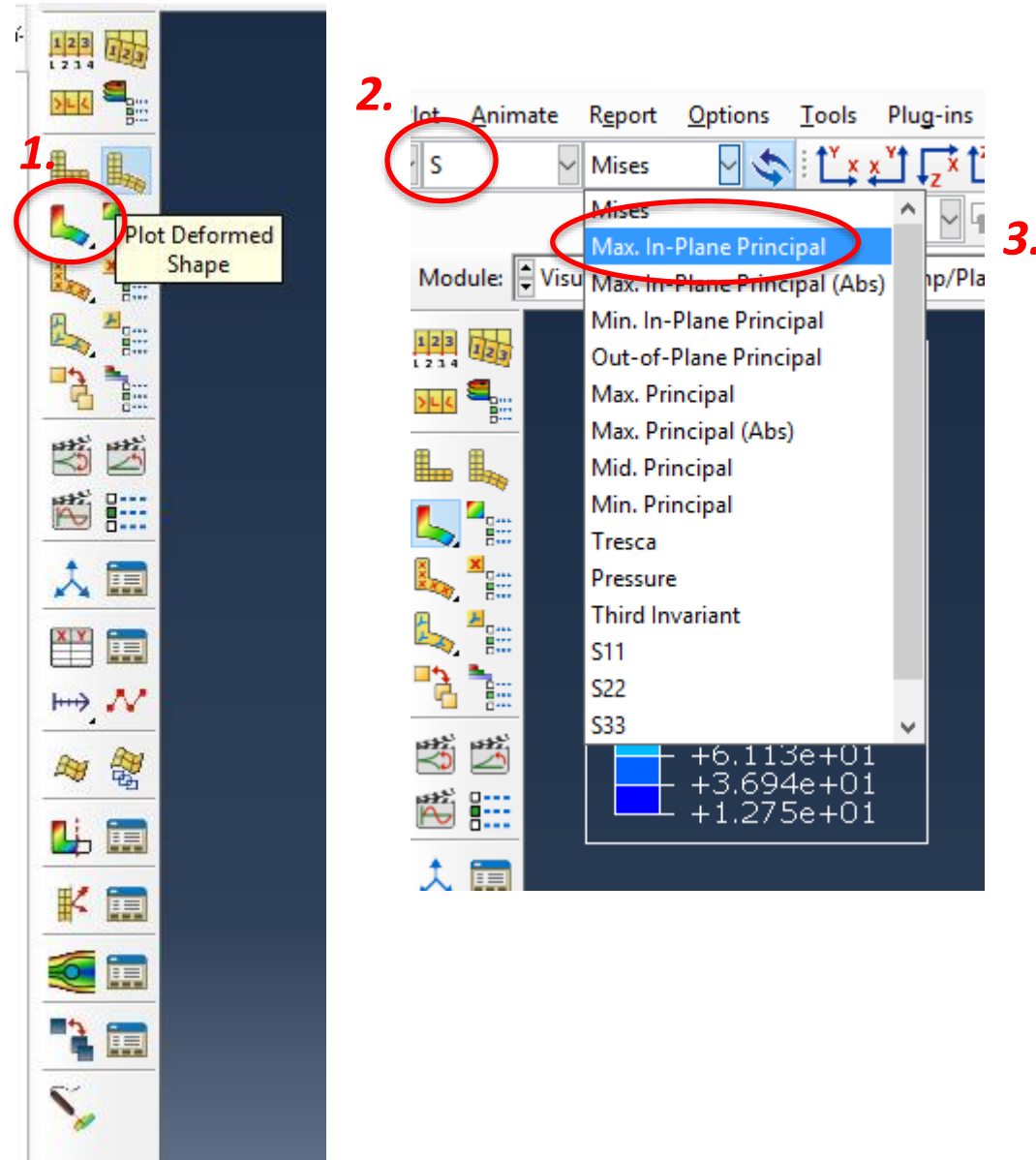
- **<u>Click the contour plot icon</u>**

- **<u>In the tools area, select S for stresses and Max In-Plane Principal</u>** to visualize the maximum principal stress;

- **<u>Try to plot other quantities such as von Mises stress, maximum principal strain etc....</u>**

- This is the contour plot of the maximum principal stress

S, Max. Principal
(Avg: 75%)
```
+3.058e+02
+2.803e+02
+2.548e+02
+2.293e+02
+2.039e+02
+1.784e+02
+1.529e+02
+1.274e+02
+1.019e+02
+7.645e+01
+5.096e+01
+2.548e+01
+0.000e+00
```
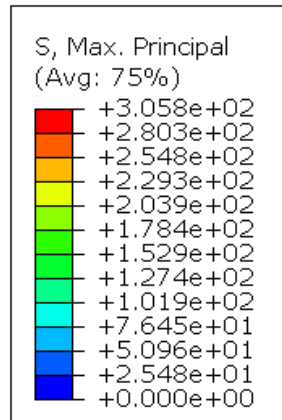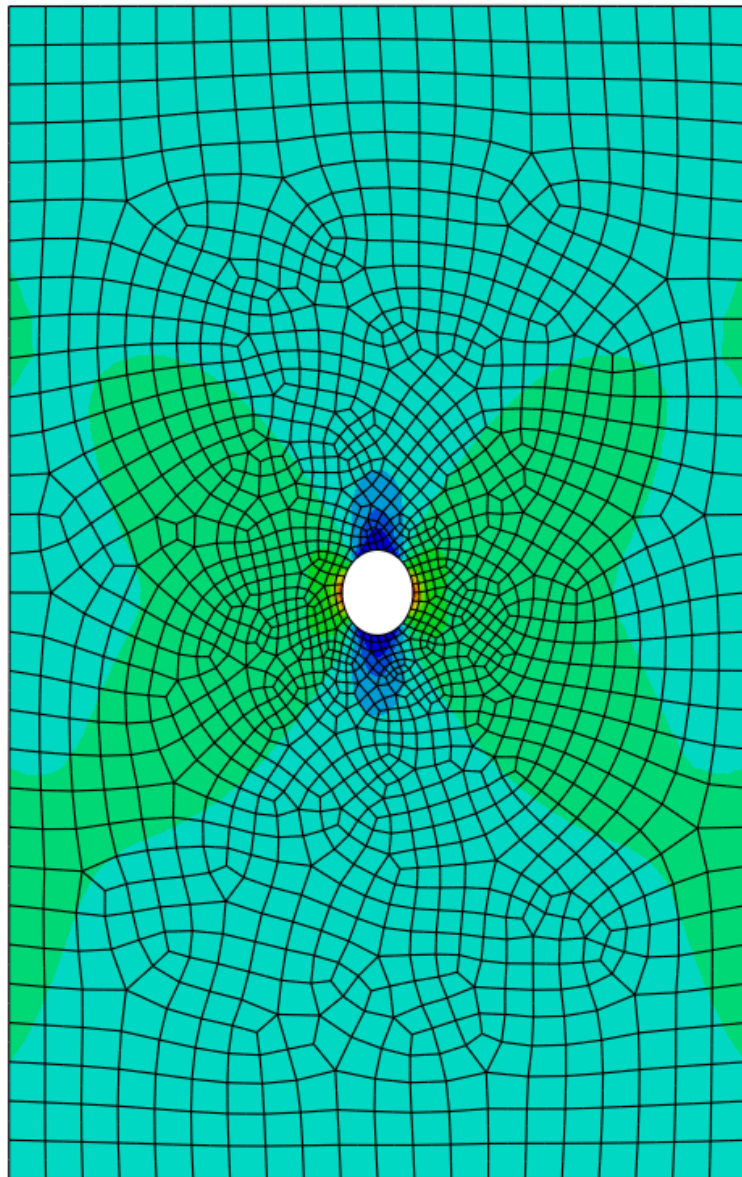
- This plot shows that the maximum value of the principal stress is 305.8 MPa

- The stress concentration factor is $K_t = \sigma_{1Max}/\sigma_{nom}$ = 305.8/111.1 = 2.752

  where σ_{nom} is calculated at the net section as $\sigma_{nom} = D/(D-d)\,\sigma_\infty$

with d = hole diameter, D = plate width, σ_∞ = remote applied stress

- Following Peterson, the solution is:

$$K_t = 3.000 - 3.140\frac{d}{D} + 3.667\frac{d^2}{D^2} - 1.527\frac{d^3}{D^3} = 2.721$$

- **The error is just 1.14%** with the current coarse mesh. (Try to improve the results by creating a finer mesh close to the circular hole)

Pilkey, W. D., Peterson's Stress Concentration Factors, Wiley, New York, 1997

- This is the contour plot of the von Mises stress

S, Mises
(Avg: 75%)

+3.030e+02
+2.788e+02
+2.547e+02
+2.305e+02
+2.063e+02
+1.821e+02
+1.579e+02
+1.337e+02
+1.095e+02
+8.532e+01
+6.113e+01
+3.694e+01
+1.275e+01
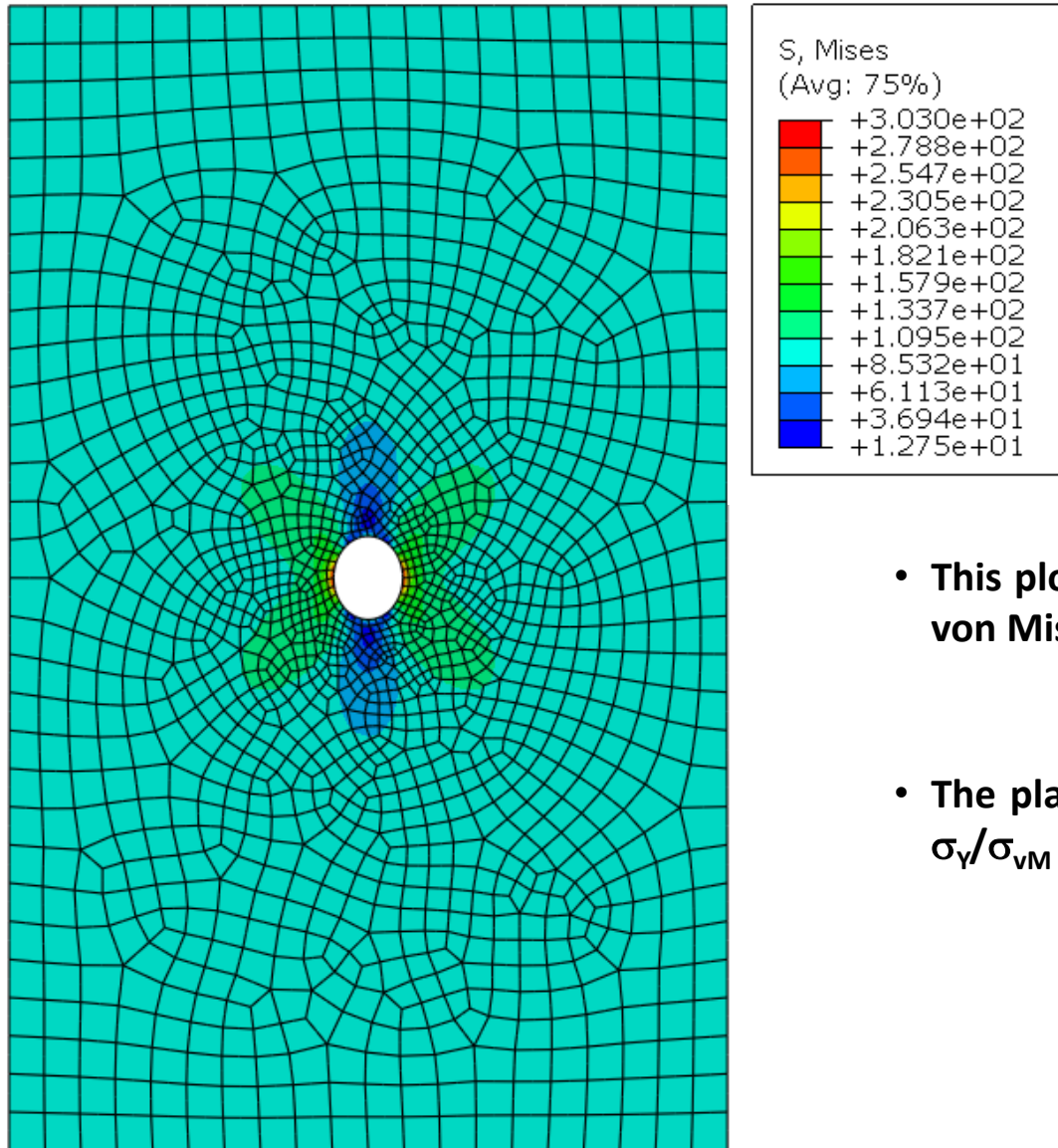
- **This plot shows that the maximum value of the von Mises stress is 303.0 MPa**

- **The plate will not yield. The safety factor is $N_t = \sigma_Y/\sigma_{vM} = 350/303 = 1.16$**

Pilkey, W. D., Peterson's Stress Concentration Factors, Wiley, New York, 1997

- Sometimes it is useful to plot stresses, strains or other physical quantities in a different coordinate system (e.g. cylindrical or spherical coordinate system). This can be done in ABAQUS in a very simple way. **Let's create, for example, a cylindrical coordinate system centered at the center of the hole**.
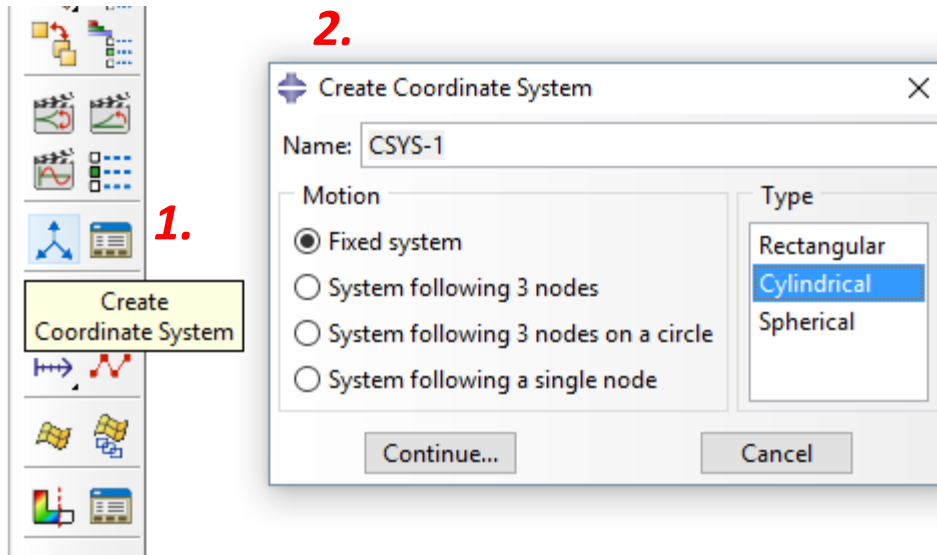
- **Click on "Create a system of reference" icon**
- **Select e.g. CSYS-1 as coordinate system name**
- **Select fixed system** (we do not want a system moving with the body)
- **Select the option cylindrical system then continue…**
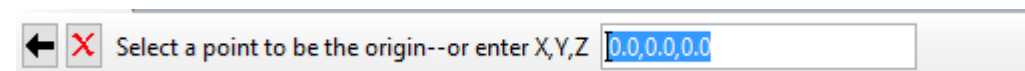
- **Select 0,0,0 as the coordinates of the origin of the coordinate system and press enter**
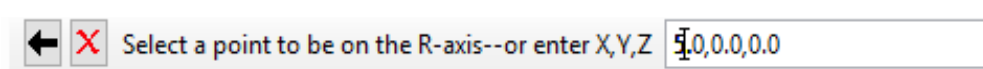
- **Select 5,0,0 as the coordinates of a point along the R-axis** (however, you can pick any point in the x-y plane. ABAQUS uses this point and the following point just to identify the $u_R - u_\theta$ plane). **Press enter.**
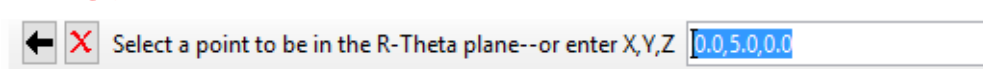
- **Select 0,5,0 as the coordinates of a point in the R-Theta plane. Press enter.**

- Once we have created a new system of reference, we can use it to calculate the e.g. stress or strain components in the structure. To do so:
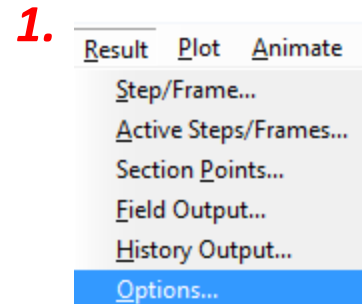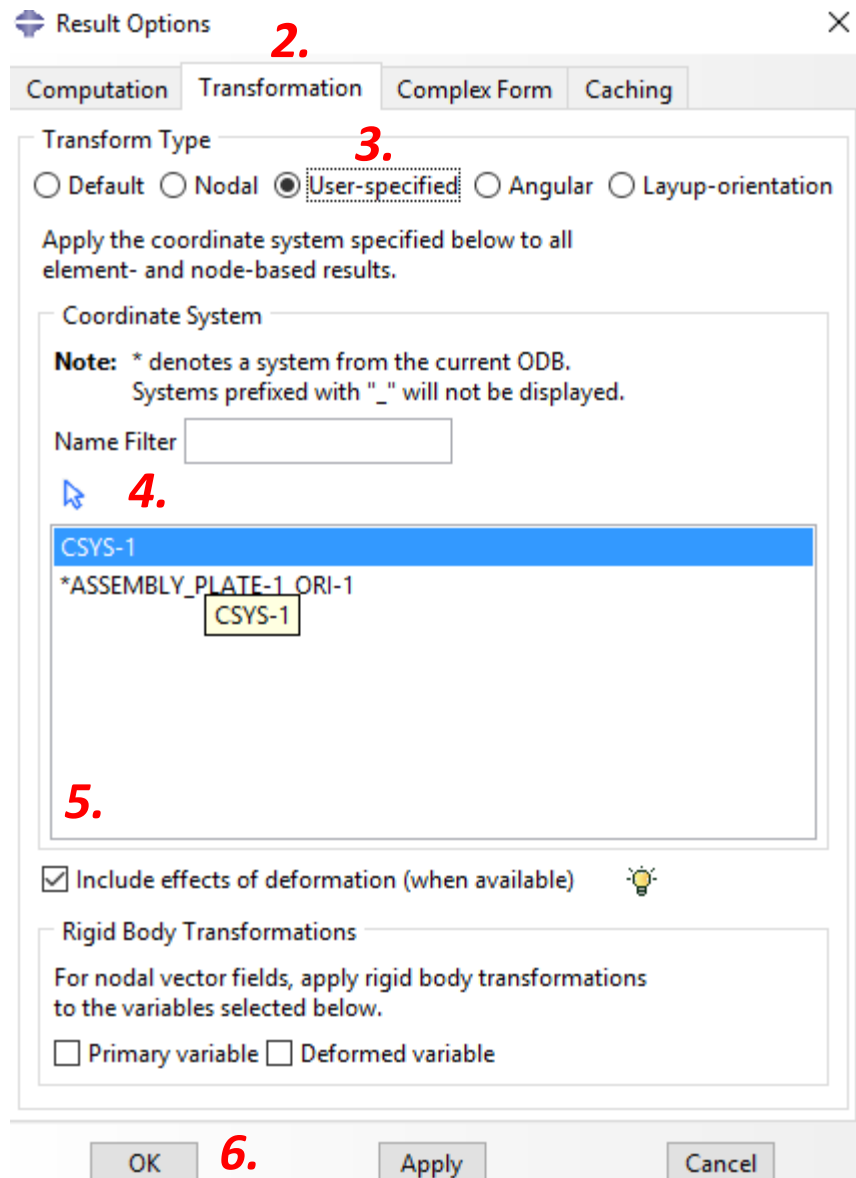
1.

- **Click on Result/Options from the menu bar**

2.

- **Select the Transformation card.**

- **In "Transform type" select "User-specified"** since we want to use a user-defined coordinate system.

- **Select the name of the cylindrical coordinate system previously created (CSYS-1)**

- **Keep the option "Include effects of deformation".** This will move the system with the deformed body (this option has effects only in case of large deformations)

- **Click OK**

- From now on, all the physical variables considered in ABAQUS will refer to the cylindrical coordinate system just created. Below is a plot of the radial component of stress in the plate:
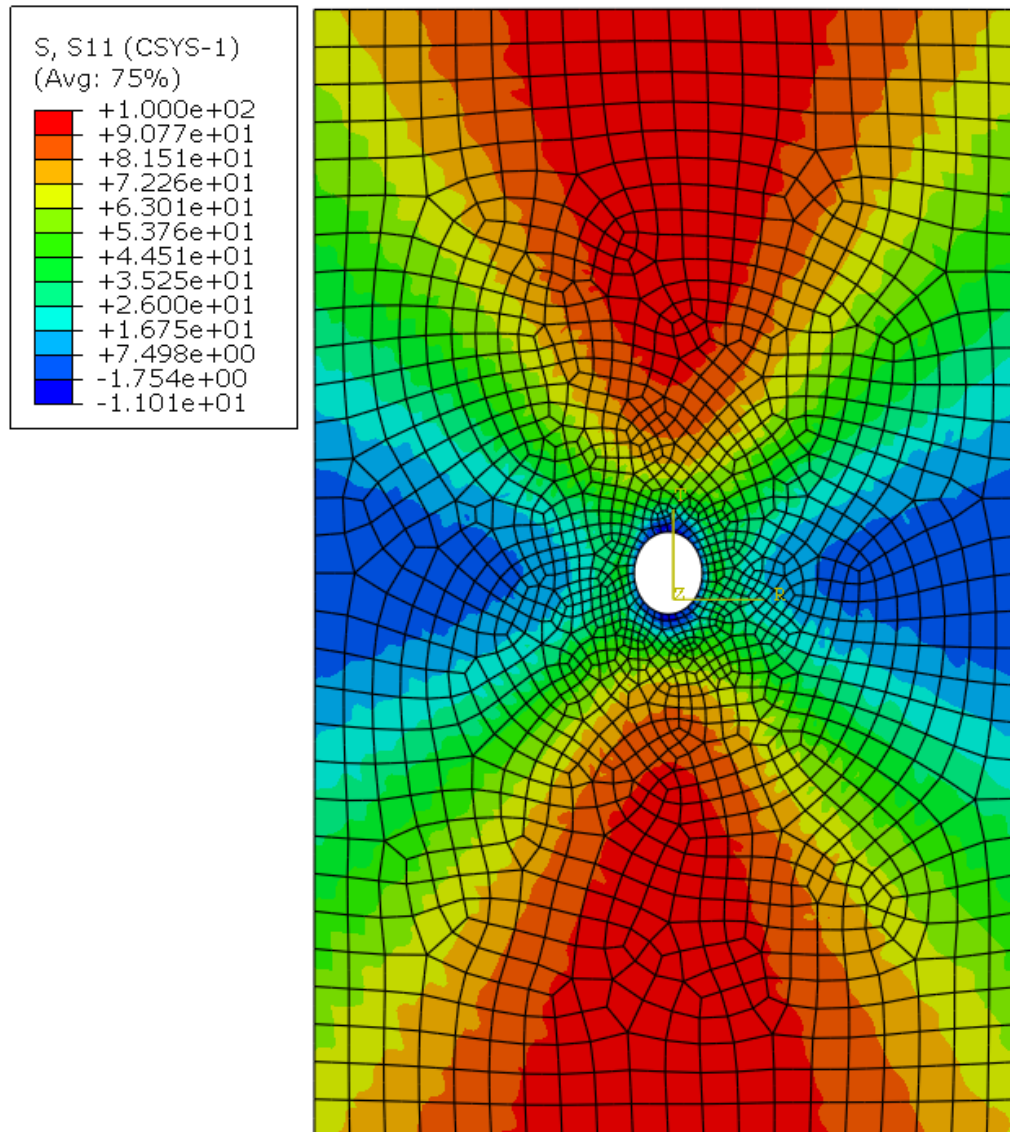
- **Note that now r->1; θ -> 2; Z->3 so, to plot σ_R, we need plot S11!**

- **Note that, due to equilibrium, σ_R = 0 along the boundary of the circle!**

- Sometimes it is useful to plot stresses, strains or other physical quantities along a path. To exemplify, let's compare the radial and tangential stresses along the boundary of the circle with the exact solution formulated by Kirsch in 1898:

$$\sigma_{rr} = \frac{\sigma}{2}\left(1 - \frac{a^2}{r^2}\right) + \frac{\sigma}{2}\left(1 + 3\frac{a^4}{r^4} - 4\frac{a^2}{r^2}\right)\cos(2\theta - \pi)$$

$$\sigma_{\theta\theta} = \frac{\sigma}{2}\left(1 + \frac{a^2}{r^2}\right) - \frac{\sigma}{2}\left(1 + 3\frac{a^4}{r^4}\right)\cos(2\theta - \pi)$$

$$\sigma_{r\theta} = -\frac{\sigma}{2}\left(1 - 3\frac{a^4}{r^4} + 2\frac{a^2}{r^2}\right)\sin(2\theta - \pi)$$

- Along the boundary $r = a$ so:

$$\sigma_{rr} = 0$$

$$\sigma_{\theta\theta} = \sigma\left[1 - 2\cos(2\theta - \pi)\right]$$

$$\sigma_{r\theta} = 0$$

1.

2.

3.

4.
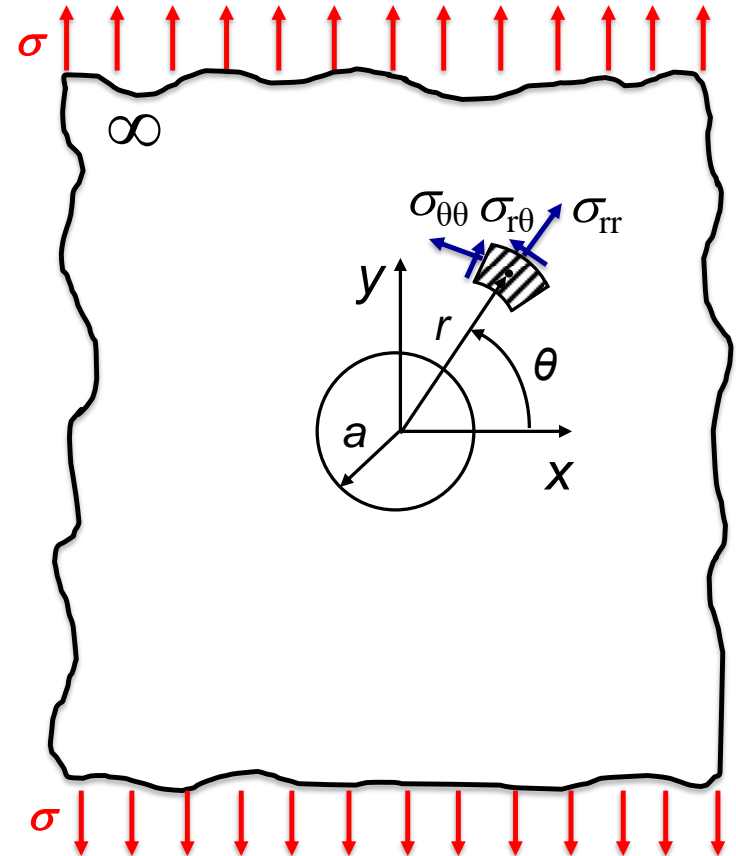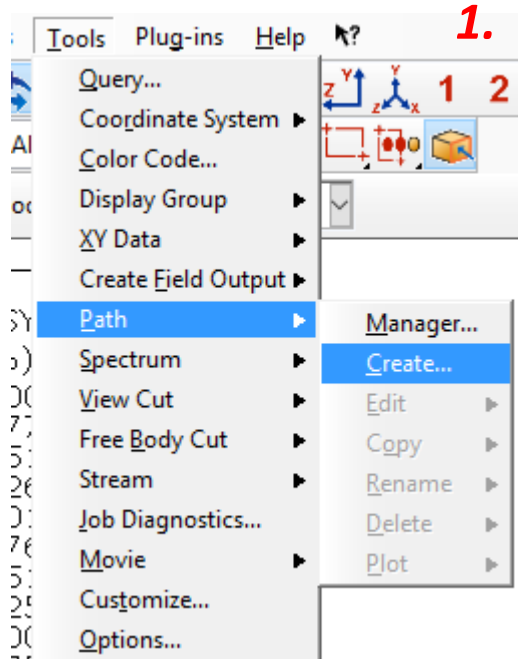
- Let's create a path along the boundary of the circle and use it to plot the stresses. *Select Tools/Path/Create… from the menu bar*
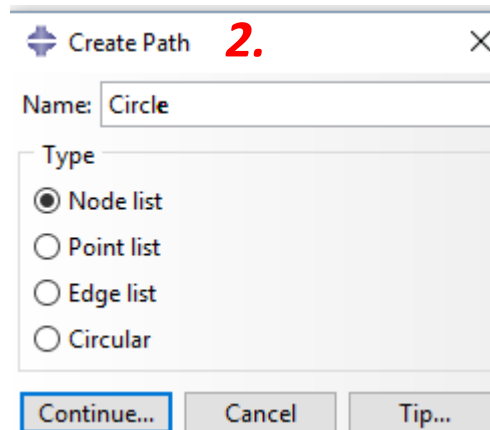
- *Rename the path as Circle*

- *Select Node list as type of path then click Continue…* This allows the selection of the path through a list of nodes. This is the most precise way to define the path (highly recommended).
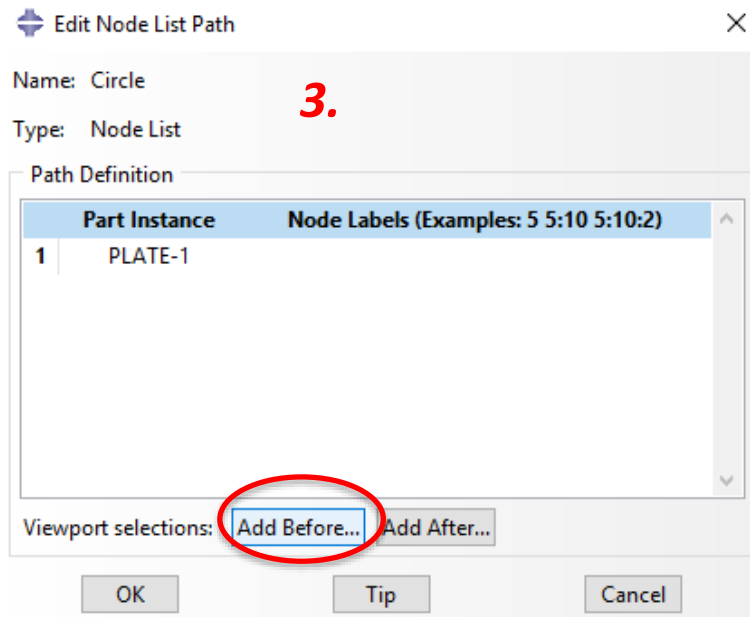
- *Select Add Before to select the nodes which will constitute the path. Use the mouse to select all the nodes at the boundary of the circle. Click Done at the bottom left*

Edit Node List Path

Name: Circle

Type: Node List

Path Definition

	Part Instance	Node Labels (Examples: 5 5:10 5:10:2)
1	PLATE-1	36, 12, 17, 18, 1, 19, 2, 20, 21, 3, 22, 23, 4, 24, 5, 25,

Viewport selections: Add Before... Add After...

OK   Tip   Cancel
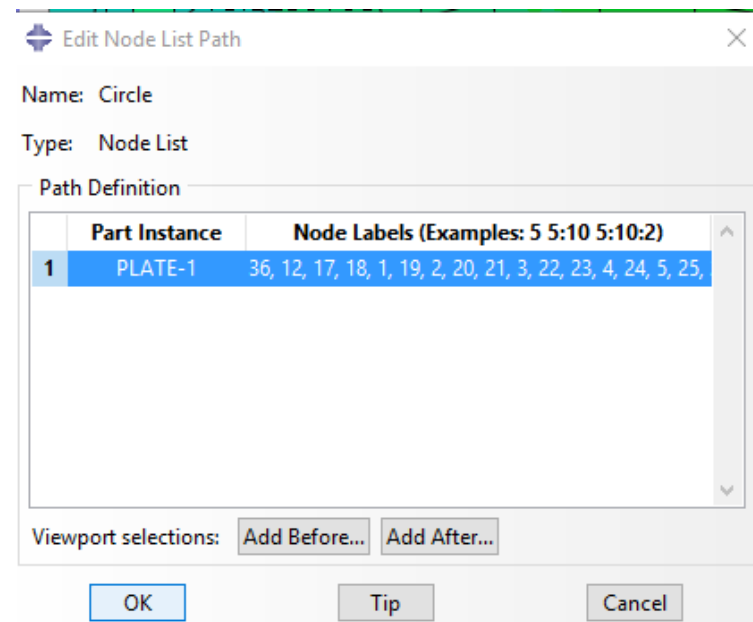
- The following list should now appear. **Press OK**

1.

Tools Plug-ins Help ⸙?

Query...
Coordinate System ▶
Color Code...
Display Group ▶
XY Data ▶ → Manager...
Create Field Output ▶ → Create...
Path ▶   Edit ▶
Spectrum ▶   Copy ▶
View Cut ▶   Rename ▶
Free Body Cut ▶   Delete ▶
Stream ▶   Plot ▶
Job Diagnostics...   Copy to ODB... ▶
Movie ▶
Customize...
Options...

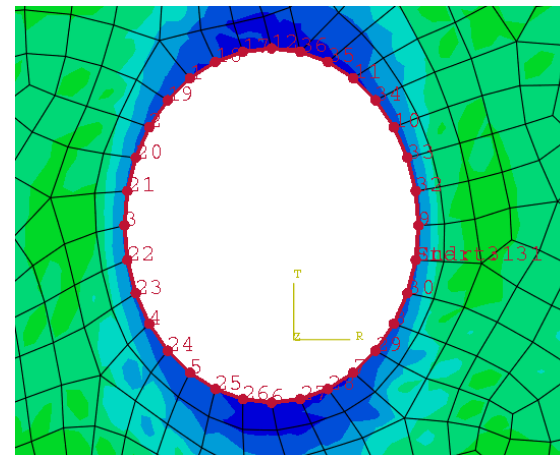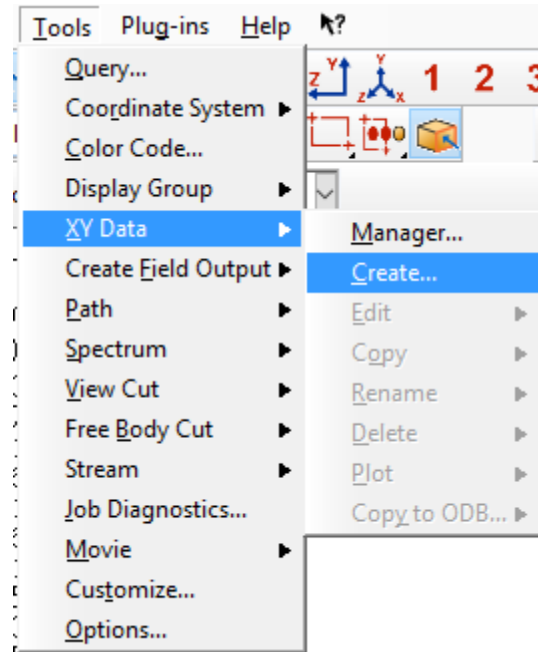- The next step is creating a database of the quantity of interest along the created path. ***Select Tools/XY Data/Create...***

Create XY Data

2.

Source
○ ODB history output
○ ODB field output
○ Thickness
○ Free body
○ Operate on XY data
○ ASCII file
○ Keyboard
● Path

Continue...   Cancel
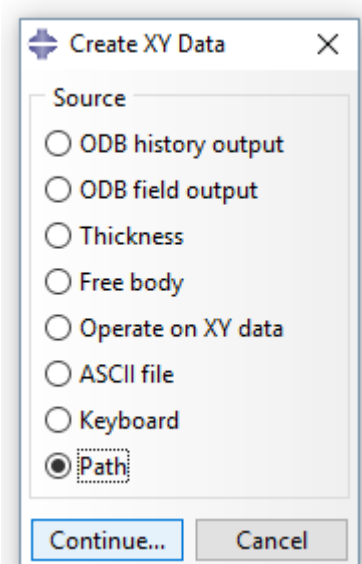
- ***Select Path as the source of the data and press Continue...***

- The next step is defining the main parameters for data extraction. **Select the path previously created (e.g. Circle)**

- **_Select Deformed as Model shape_**. In this way the path will follow the deformed shape of the body.

- **_Select Path points in point locations_**. In this way the desired variable will be calculated exactly on the selected points. You can use uniform spacing if you do not necessarily need the function to be calculated at path points and you want a uniform spacing.

- **_Select True distance_**. In this ABAQUS will calculate the curvilinear distance of the desired points. Alternatively you can select X distance etc to have the Cartesian coordinates of the points

- **_Click on Field Output to select the variable to be plotted. Choose S11 to plot the radial stress._**

- **_Click on Save as… to save the database._**

- **_Save as Radial stress and click OK._**

- **_Repeat steps 5-7 to save the tangential and shear stresses_**
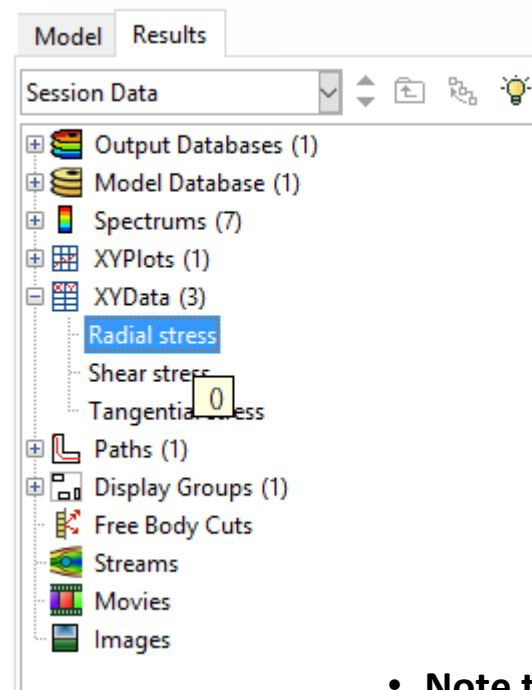
- The saved data are accessible from the Results tree. ***Clicking the left button of the mouse on a saved variable will automatically plot it. Below are the plots of the (a) radial stress, (b) tangential stress and (c) shear stress.***

(a)
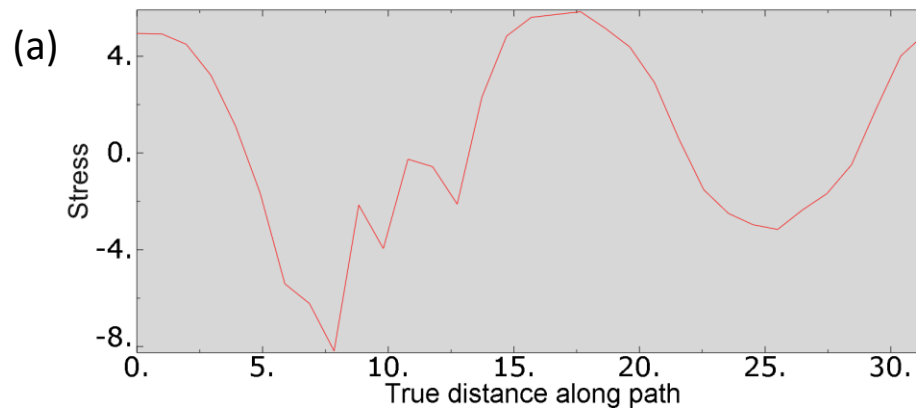
- **Note that the radial and shear stresses is not exactly zero (try with a finer mesh)**

(b)

(c)
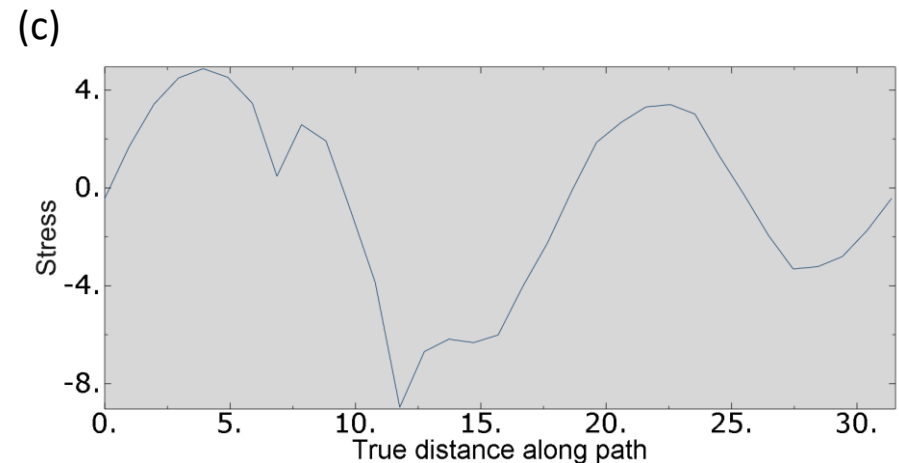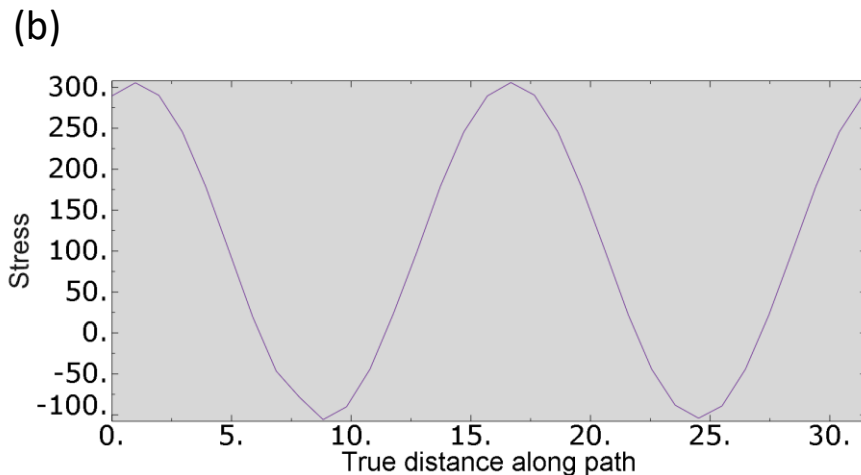
- The saved data can be used to make plots in Matlab, Excel etc. It is possible to copy/paste the data directly to the preferred software by **_clicking Tools/XY Data/Manager_**

- **_Click the left button of the mouse twice on the desired variable_** to visualize the data. You can **_copy the variable and paste it in the preferred software_** (e.g. Excel)
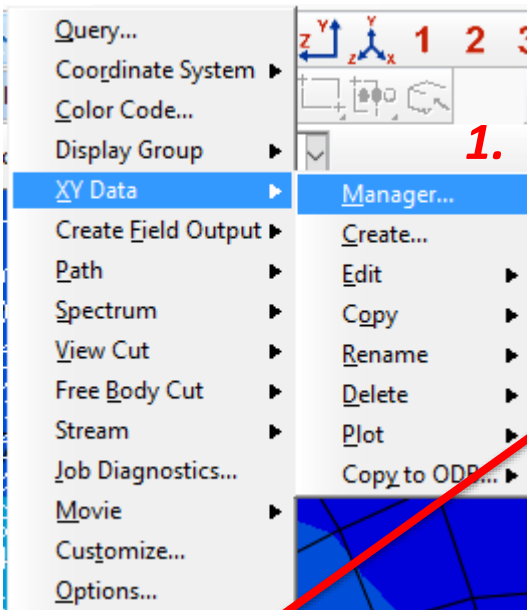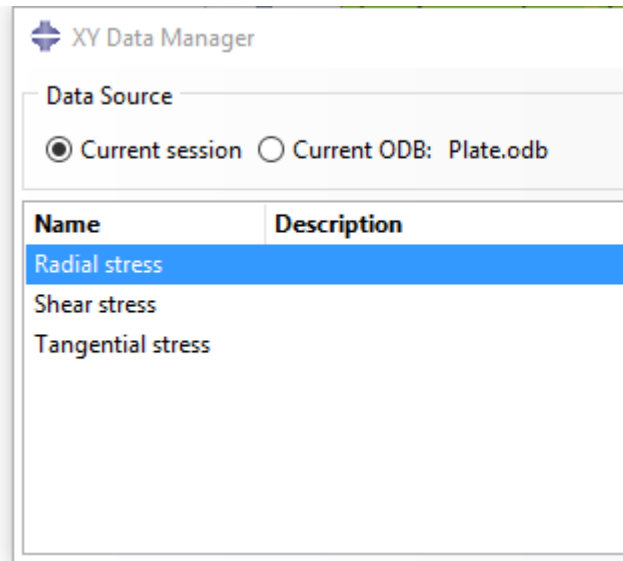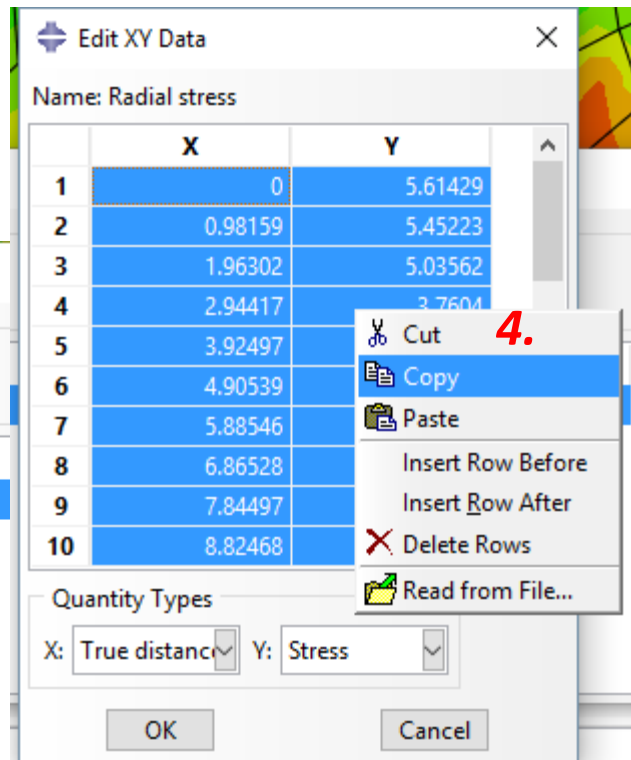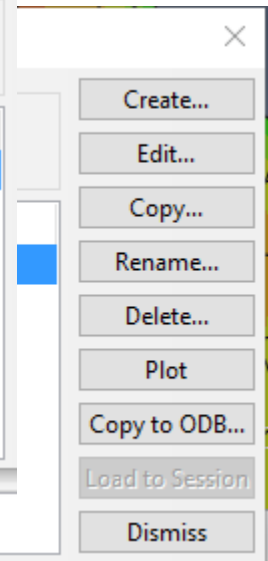
- With the copied variables we can create a plot in e.g. Excel of the stresses along the boundary and compare them with the analytical solution
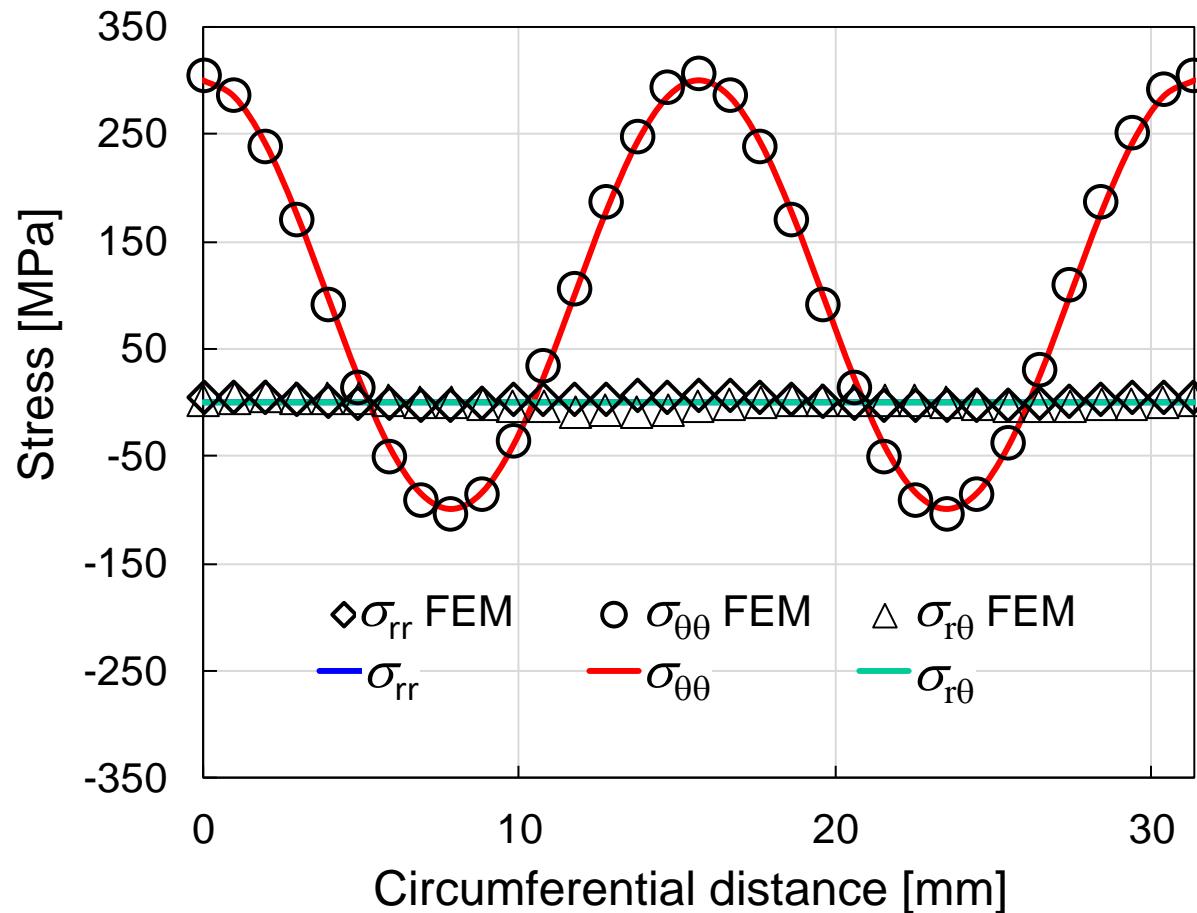
- **Note that the FE solution does not match exactly Kirsch's solution (but it is very close)**

- **This is due to the fact that (a) Kirsch solution is for an infinite plate, (b) the mesh is not fine enough**

- In some cases the amount of data is to high and copying/pasting the data is not feasible. In such cases, we can write a report in ABAQUS. This will create a .rpt file in the work folder containing the requested output.
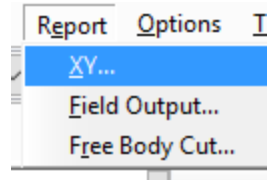
- **<u>Go to Report/XY… in the menu bar</u>**
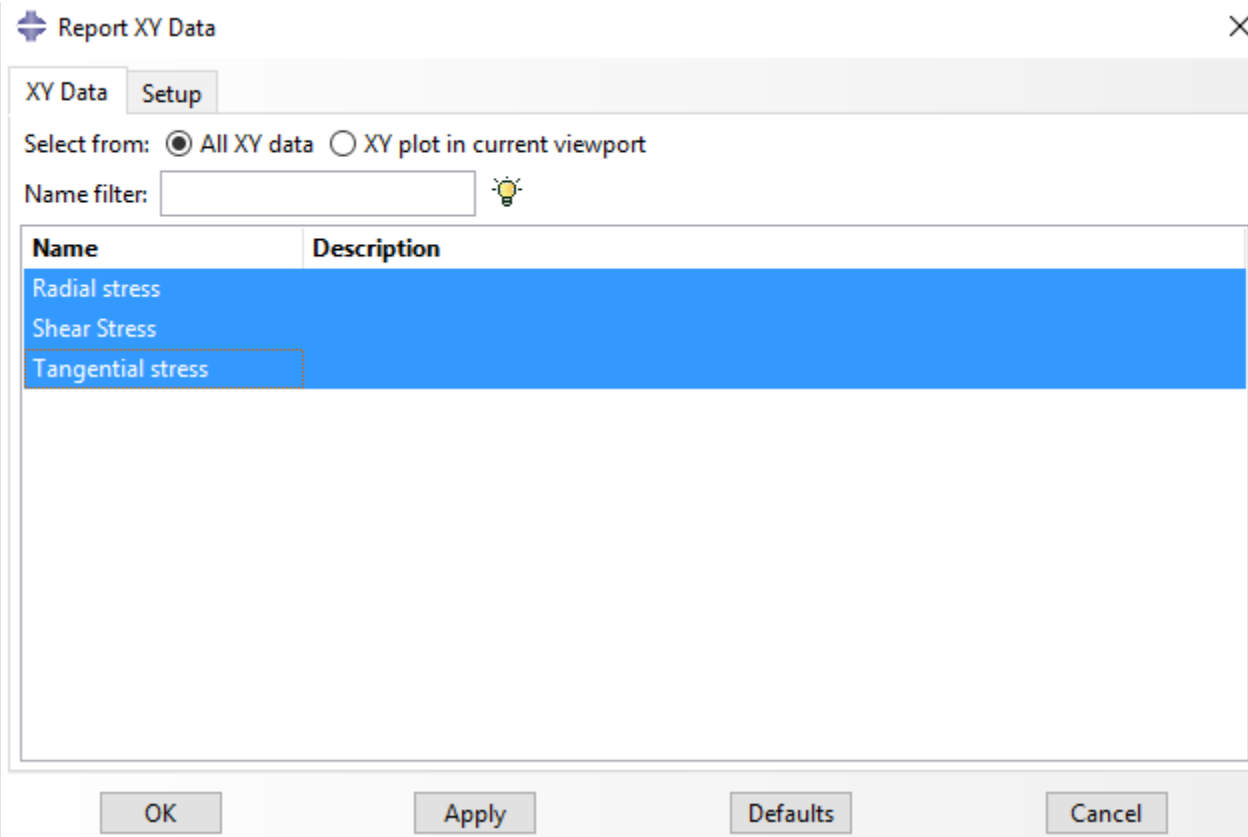
- **<u>Select the variables you want in the report. You can make multiple selections by using shift + left button</u>**

Report XY Data

XY Data | **Setup**

File

Name: Stress.rpt [Select...]

☐ Append to file

Output Format

Layout: ⦿ Single table for all XY data
- ☐ Interpolate between X values (if necessary)
- ◯ Separate table for each XY data

Page width (characters): ⦿ No limit ◯ Specify: 80

Number of significant digits: 6

Number format: Engineering

Data

Write: ☑ XY data ☐ Column totals ☐ Colum

[OK] [Apply]

- **Go to the Setup card**

- **Select a name for the file (e.g. Stress.rpt)**

- **Make sure append to file in NOT selected.** This will create a new file instead of appending on old files

- **Click OK**

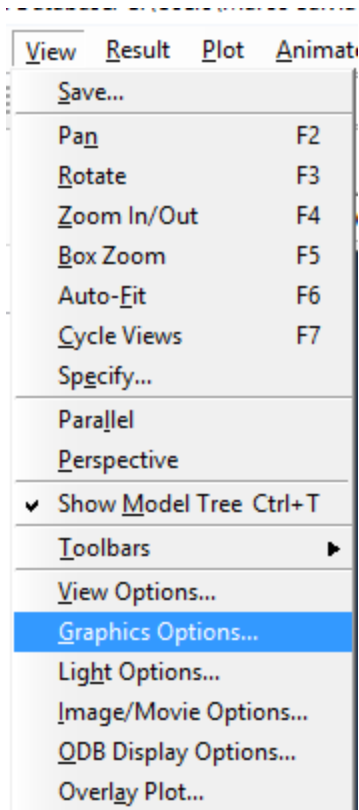| X	Radial stress	Shear Stress	Tangential stress
0.	5.71399	2.28403	306.055
981.59E-03	5.16922	3.77341	285.018
981.59E-03	5.73524	3.68893	287.277
1.96302	4.71525	4.77547	237.712
1.96302	5.35599	4.8892	240.192
2.94417	3.47373	4.41652	170.074
2.94417	4.04707	4.80645	172.432
3.92497	1.04737	4.00216	90.2959
3.92497	1.69679	4.18013	93.2815
4.90539	-1.02383	3.33346	12.4759
4.90539	-557.032E-03	3.35955	14.945
5.88546	-1.9288	2.35632	-51.4647
5.88546	-1.24267	2.77156	-48.5067
6.86528	-2.3742	669.674E-03	-93.1344
6.86528	-1.56284	1.107	-90.2867
7.84497	-2.33717	-488.782E-03	-105.05
7.84497	-1.92329	-472.654E-03	-104.141
8.82468	-1.08033	-1.85383	-84.3531
8.82468	-1.11374	-1.60851	-86.0134

- **A file Stress.rpt should appear in the work folder. You can view the file using Notepad, excel or Matlab to analyze the data. The file should look as in the figure.**

- **Note that X here is the independent variable of the functions you are plotting, not the Cartesian component!**

- **Let's create a picture of the contour plot of the tangential stress.**

1.

- First, let's change the background to white to save toner. ***Click View/Graphics Option from the menu bar***

2.
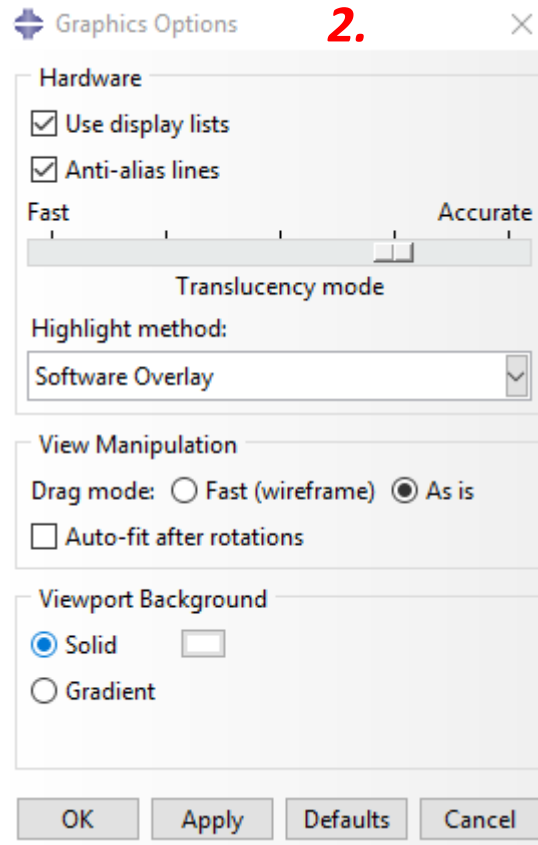
- ***Select Solid in the Viewport Background and select white as color***

- ***Click OK. The background should now be white.***

- ***You can save a picture on the clipboard anytime by clicking Ctrl+C***

- ***Alternatively, you can save the picture as file as shown in the next slide….***

File menu (shown open):

File | Model | Viewport | View | Result

- New Model Database ▶
- Open...   Ctrl+O
- Network ODB Connector ▶
- Close ODB...
- Set Work Directory...
- Save   Ctrl+S
- Save As...
- Compress MDB...
- Save Display Options...
- Save Session Objects...
- Load Session Objects...
- Import ▶
- Export ▶
- Run Script...
- Macro Manager...
- Print...   Ctrl+P
- Abaqus PDE...
- 1 C:/.../ABAQUS training/Plate2.cae
- 2 C:/.../ABAQUS training/Plate.cae
- 3 C:/Temp/torsion_general.cae
- 4 C:/Temp/thermal_cut.cae
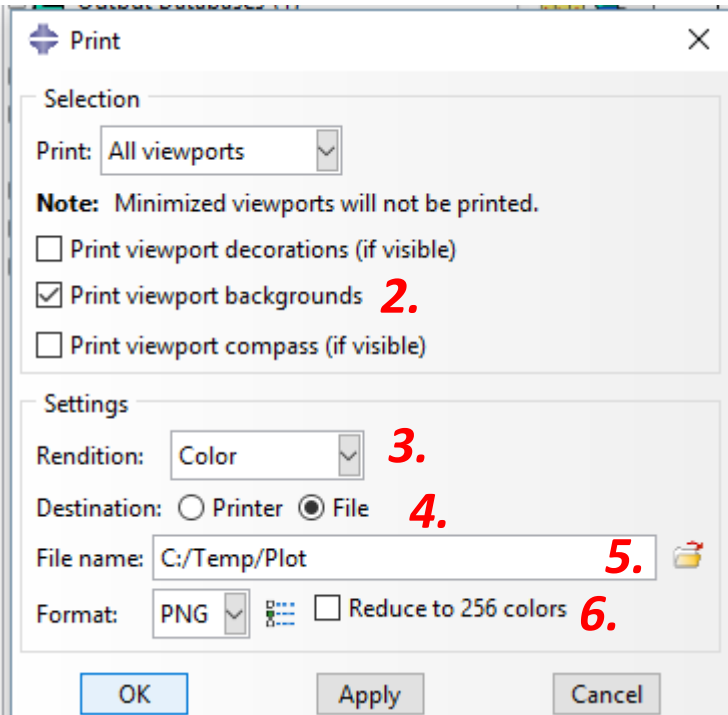- Exit   Ctrl+Q

Print dialog:

Selection

Print: All viewports

Note: Minimized viewports will not be printed.

- ☐ Print viewport decorations (if visible)
- ☑ Print viewport backgrounds **2.**
- ☐ Print viewport compass (if visible)

Settings

Rendition: Color **3.**

Destination: ○ Printer ● File **4.**

File name: C:/Temp/Plot **5.**

Format: PNG ▾ ☐ Reduce to 256 colors **6.**

[OK] [Apply] [Cancel]

1. • **Click File/Print... from the menu bar**

- *Select Print Viewport backgrounds to make sure the white background will be printed*

- *Select Color as rendition (we want colors)*

- *Print to file to save the picture as a file*

- *Select the file name and path (e.g. C:/Temp/Plot*

- *Select a format (e.g. PNG) and unselect the option "reduce to 256 colors" for better quality*

- *Click OK. The picture should now appear in the work folder*

- **This is a picture of the tangential stress!**