Spring
2018

Abaqus Tutorial (V2.0)

FE Analysis of Trusses and Frames

- **Contact:**
  shahi@gatech.edu
  www.sshahi.com

Shahrokh Shahi

Spring 2018

Introduction

Abaqus is a well-known software package for finite element analysis and computer-aided engineering consisting of five core software products [3]:

- Abaqus/CAE or "Complete Abaqus Environment", which is a Graphical User Interface (GUI) for both modeling and analysis of mechanical components, and visualizing the finite element analysis results.
- Abaqus/Standard, which is a general-purpose Finite Element Solver module.
- Abaqus/Explicit, which is a special-purpose Finite Element Analyzer for solving highly nonlinear systems with complex contacts under transient loads.
- Abaqus/CFD, which is a Computational Fluid Dynamics software application
- Abaqus/Electromagnetic, which is a Computational Electromagnetics analyzer.

This tutorial aims at quickly introducing the Abaqus/CAE to the students in the Finite Element Methods course. This brief tutorial is mainly based the Abaqus documentations [1], which is the best reference to understand the simulation procedure and also an invaluable source of examples. These documentations are usually installed with Abaqus/CAE. A web-based version of Abaqus documentations is also available in the following link:
http://abaqus.software.polimi.it/v6.14/index.html
The Abaqus/CAE User's Guide can directly be accessed through the following link:
http://abaqus.software.polimi.it/v6.14/books/usi/default.htm

In this tutorial, at first, main components of Abaqus/CAE will briefly be introduced. Then, the whole procedure of a finite element analysis will be explained step-by-step through analyzing a simple truss and a simple frame.

Components of the Graphical User Interface

In this section, the main components of the Abaqus/CAE are briefly reviewed. (You can find detailed information in the following link: http://abaqus.software.polimi.it/v6.14/books/usi/pt01ch02s02.html)

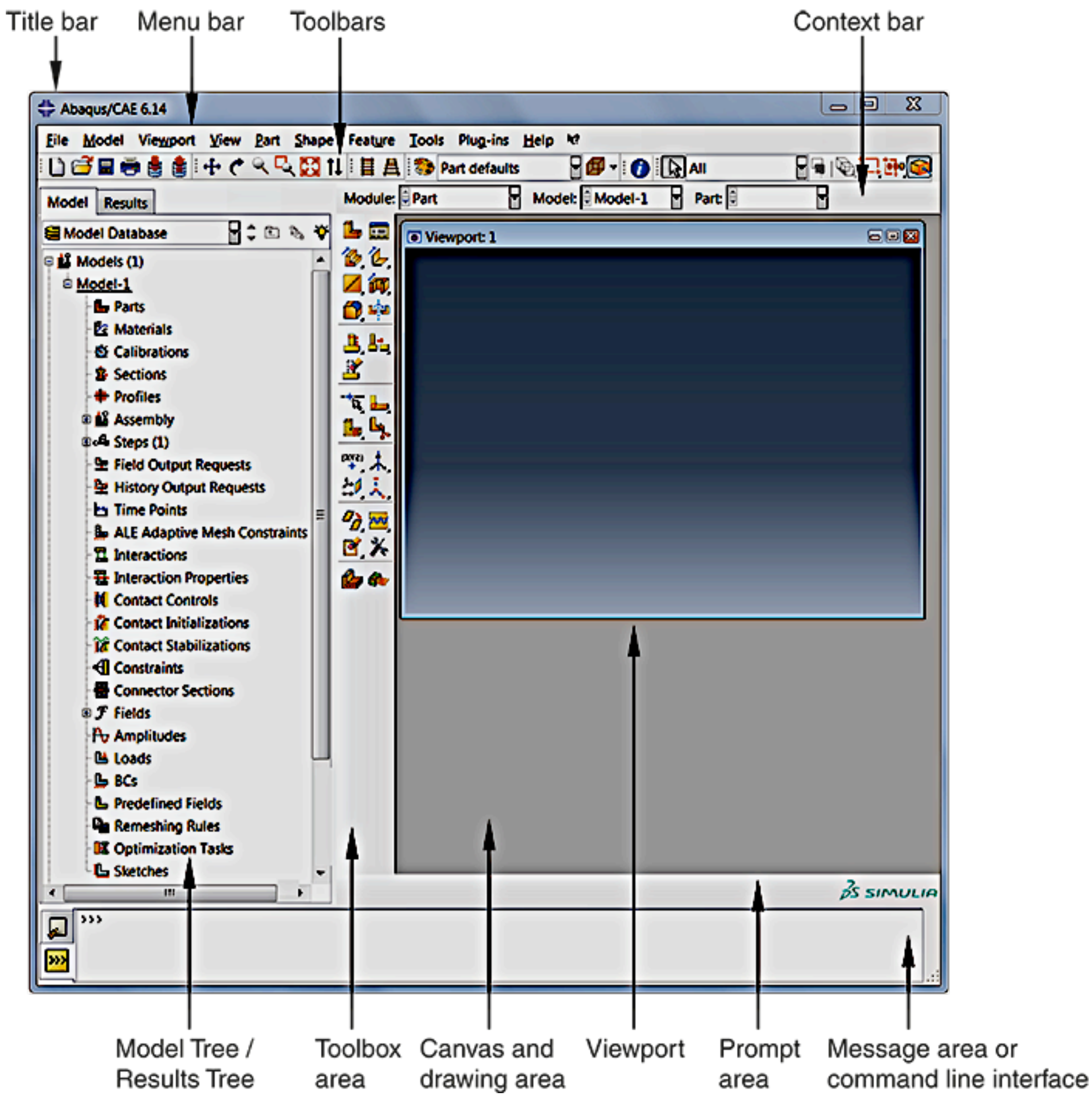


Figure 1 Main Components

Menu bar:
The menu bar contains all the available menus providing access to all the functionalities within Abaqus/CAE. Depends on the current module that the user selects in the context bar, different menu items may be appeared in the menu bar.

Toolbars:
The toolbars provide quick access to the items that are also available in the menus

Context bar:
Abaqus/CAE consists of a set of modules which each of them is associated with one specific aspect of the model. The **Module list** in the context bar allows the user to move between these modules. Other items in this bar are functions of the current module.

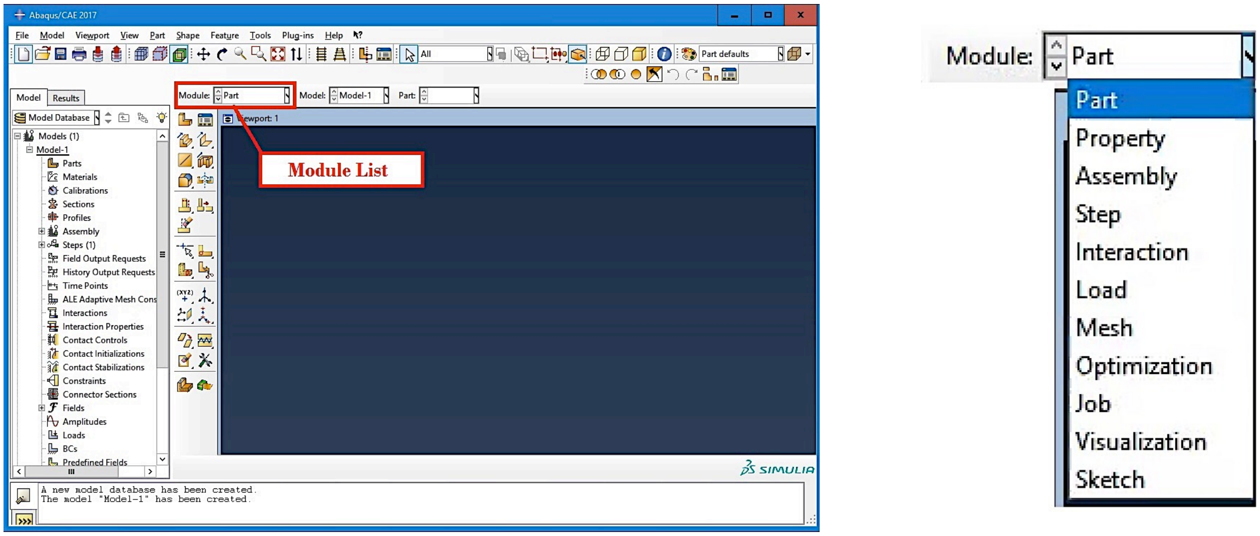


Figure 2 Context bar and the Module list

Model Tree:
The Model Tree is a quick way to access the model database and provides a graphical overview of the models and their objects.

Result Tree:
The Result Tree provides a graphical overview of the output database. It can be considered as a quick way to perform most of actions in the Visualization module.

Toolbox area:
The toolbox area displays and provides a quick access to the tools related to the current module. These tools are also accessible through the menu bar.

Viewport:
Viewports are windows on the canvas and drawing area in which models are displayed.

Prompt area:
The prompt area displays instructions to follow during procedures.

Message area:
Status information and warnings are printed in the message area

Command line interface:
This part of the Abaqus/CAE can be very helpful during creating a model. The command line interface can be used to type **Python** commands and evaluate mathematical expressions using the Python interpreter. (More information: http://abaqus.software.polimi.it/v6.14/books/cmd/pt02.html)

Main Modeling Approach

A complete Finite Element Analysis in Abaqus consists of three main distinct steps: pre-processing, processing, and post-processing [1,3]:

- **Pre-processing:** In this step, an Abaqus input file is created, which defines the model of a physical problem including geometry, material, mesh, etc. This model can be created graphically by employing Abaqus/CAE or some other pre-processors, such as Hypermesh. Moreover, the input file can directly be generated by using a text editor like Notepad or Sublime Text.

- **Processing or Finite Element Analysis:** This step involves numerically solving the problem defined in the model and usually is run as a background process. The processing time depends on the complexity of the model and the computation power. Abaqus/Standard, Abaqus/Explicit or Abaqus/CFD are responsible for accomplishing the processing step.

- **Post-processing:** This step is a visual rendering step to evaluate the simulation results and is generally done interactively within the Visualization module of Abaqus/CAE or another post-processor.

Finite Element Analysis of a Truss

In this section, the step-by-step procedure to analyze a simple triangular truss structure subjected to a vertical force in Abaqus/CAE is presented.

Problem: Consider the 2D plain truss structure shown in Figure 3 [2]. This Structure consists of three planar truss members. A vertical downward force of 1000 N is applied at node 2. Table 1 demonstrates the material properties and dimensions of the truss members in the structure.

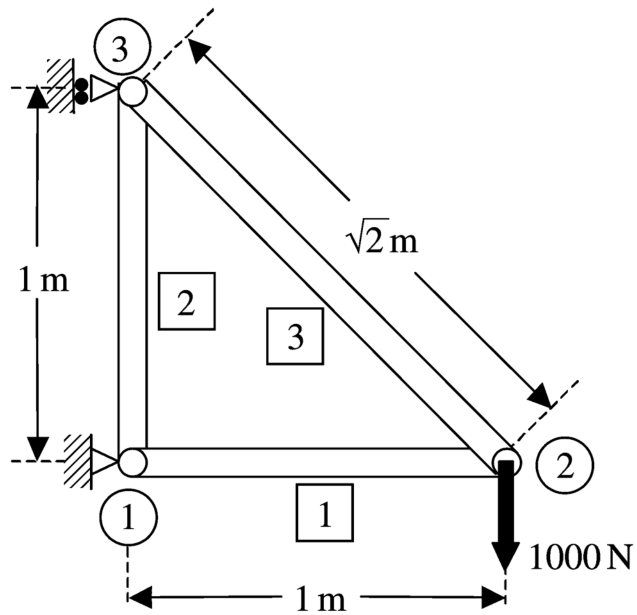


Figure 3 A three member truss structure [2]

Table 1 Dimensions and properties of truss members

Element number	Cross-sectional area A (m^2)	Young's modulus E (N/m^2)	Poisson ratio υ
1	0.1	70e9	0.3
2	0.1	70e9	0.3
3	0.1	70e9	0.3

Modeling Procedure:

- On the main menu, click on the **File** and choose the **New Model Database**, and then **With Standard/Explicit Model**. It is a good practice to save your work regularly, and save the database at the beginning. Click on the **Save** button (or **File/Save**) and choose the working directory. Name the database file and click on **OK** (Figure 4).

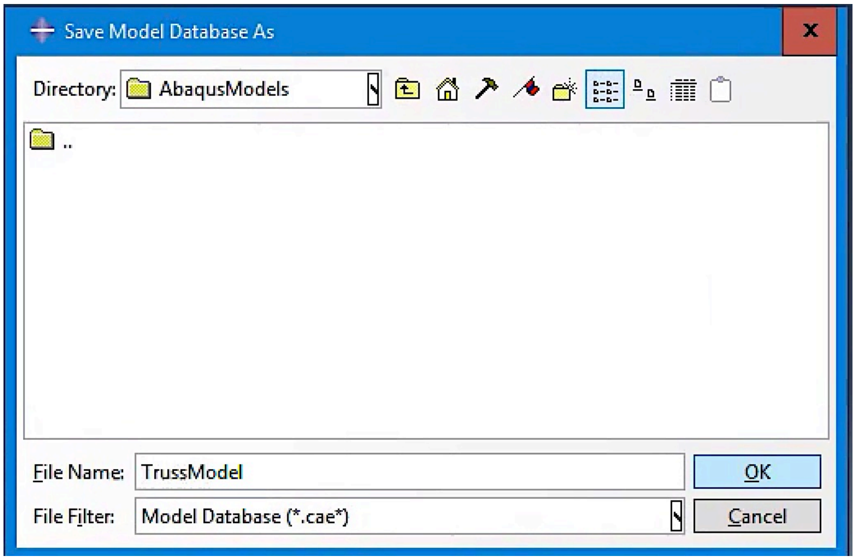


Figure 4 Saving the model database file

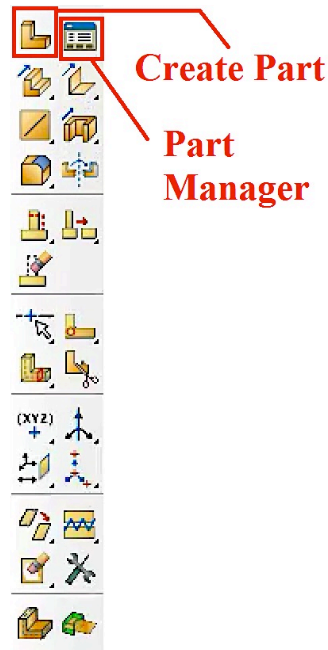


Figure 5 Module Part toolbox

- Select **Part** from Module List on the context bar
- Parts are building blocks of an Abaqus/CAE model. In general, the first step of each modeling is creating the parts. Then, you can use Assembly module to assemble instances of the parts. There are several ways to create a part in Abaqus/CAE, such as using the tools available in the Part module, importing from another database or a third-party format, for instance SolidWorks, and also merging or cutting instances in the Assembly module.

  In this example, the geometry is created using an important tool on the toolbox by clicking on the **Create Part** button (Figure 5). Moreover, to edit or delete created parts the **Part Manager** tool can be used.

- The "Create Part" dialog box appears on the screen (Figure 6). Name the part "truss_all_parts". Check **2D Planar** because it is a planar truss. Then, choose **Deformable** in the type and check **Wire** as the base feature. Then, enter 10 as the approximate size, which is used by Abaqus/CAE to calculate the size of the **Sketcher** sheet and the spacing of its grid. (For more information about these options see: http://abaqus.software.polimi.it/v6.14/books/usi/pt03ch11s19.html). Click on the **Continue** to go to the Sketcher view.

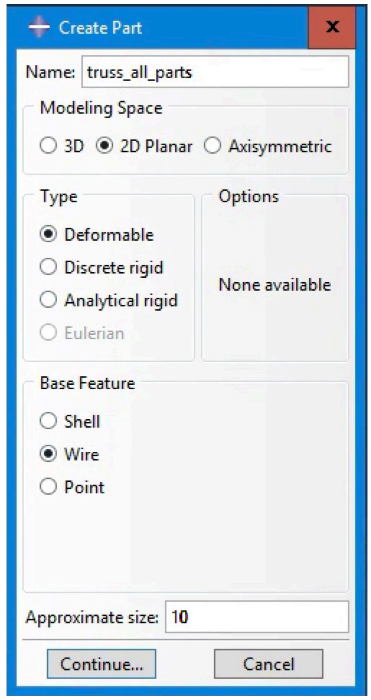


Figure 6 Creating parts

- In the Sketcher toolbox, choose the **Create-Lines: Connected** button to begin drawing the geometry of the truss. The coordinates of the cursor are shown on the top-left corner.
- Begin drawing the truss. You can enter the coordinates of a point or pick it directly (See the prompt area). Once finished, click on **Done** button to exit the sketcher.

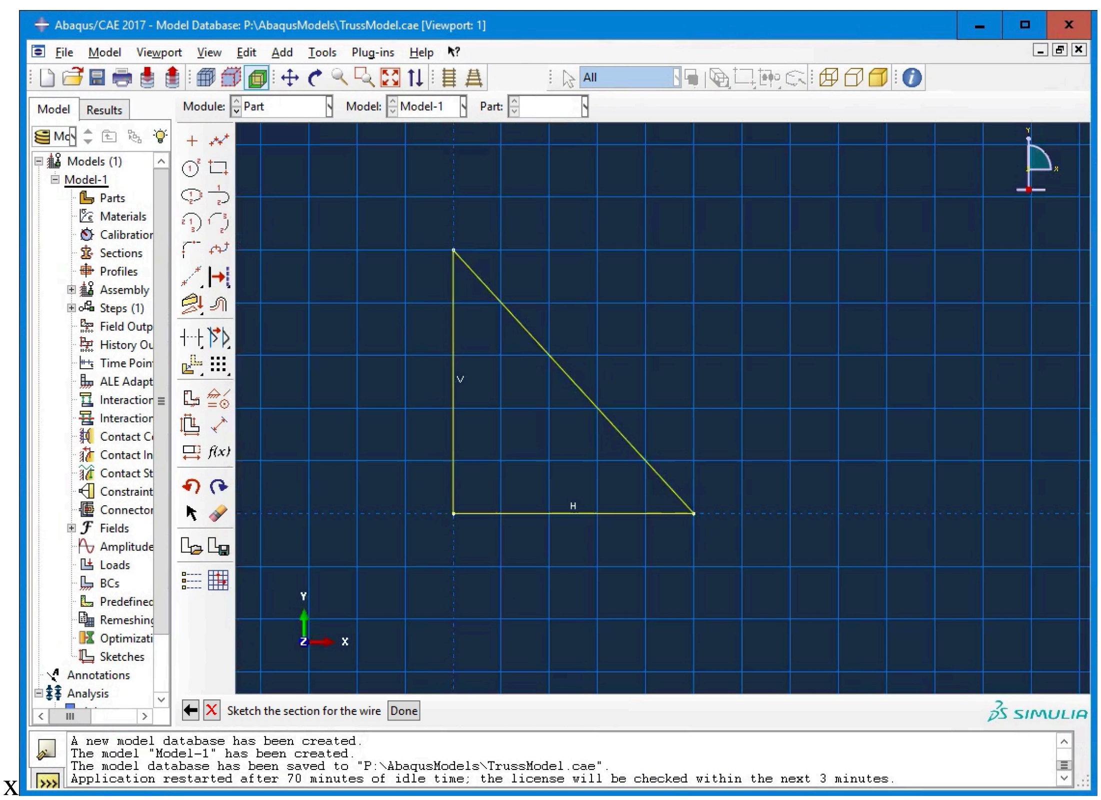
x 

Figure 7 Draw the geometry in Sketcher

- The completed part is demonstrated in the following figure.

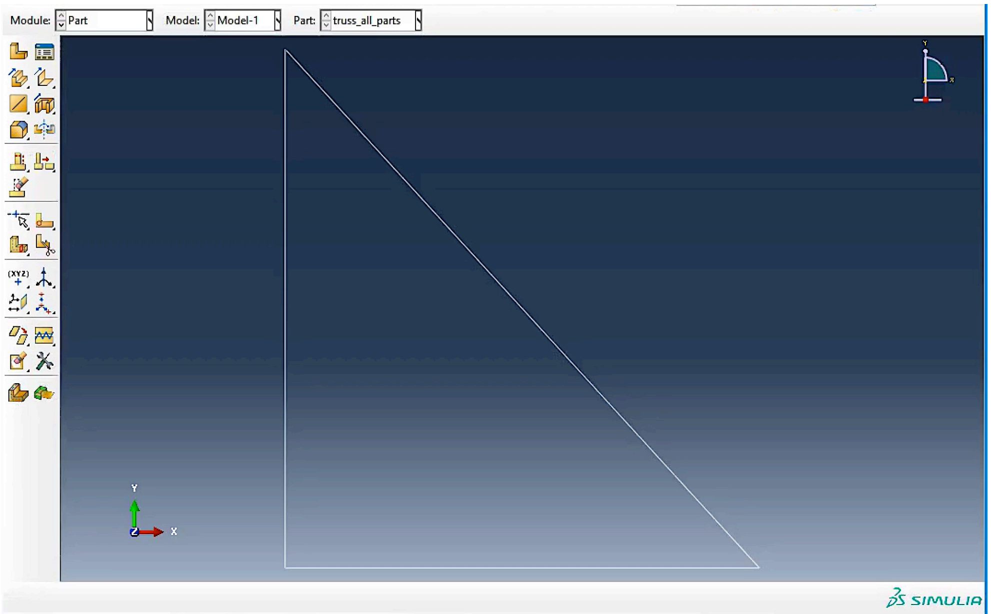


Figure 8 Completed part

- Select **Property** module from Module List on the context bar. The most functional tools of this module are demonstrated in Figure 9.

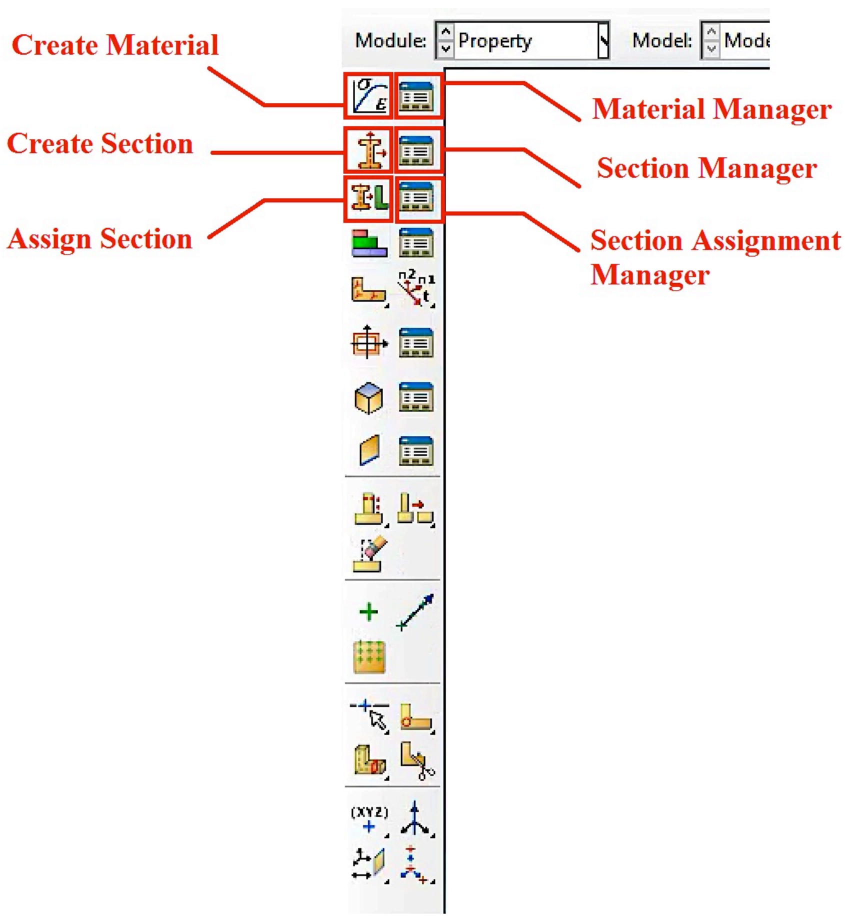


Figure 9 The Functional tools in Property module

- Click on **Create Material** button to appear the Edit Material dialog box. Since all the members of the truss are made of the same material, one material is enough to be defined. It is also enough to define the mechanical properties. First, name this material truss_mat. You can also add some description about this material. Then, click on the **Mechanical** tab, then **Elasticity**, and **Elastic** (Figure 10).

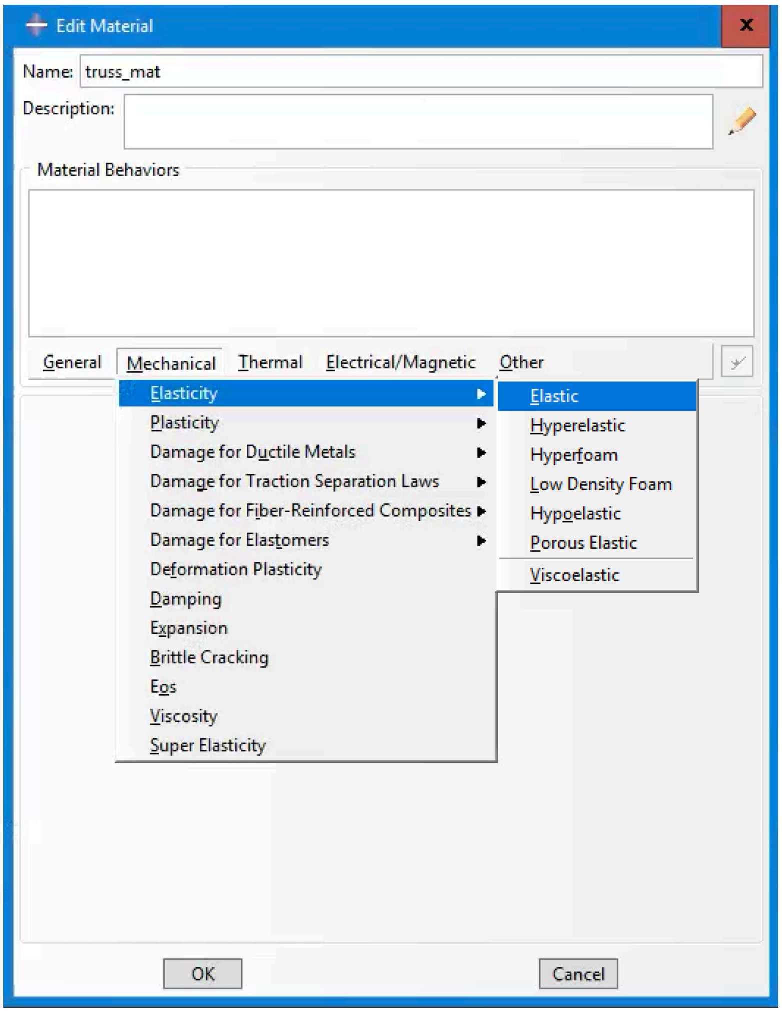


Figure 10 Creating material properties

- Remember to be consistence in the units. For instance, enter all the values in SI unit system. Enter 70e9 N/m^2 for the Young elasticity modulus, and 0.3 for Poisson's ratio. Then, click on **OK**.

  (Note: we know that the Poisson's ratio is not applicable for this analysis; however, it is a good practice to enter this value.)

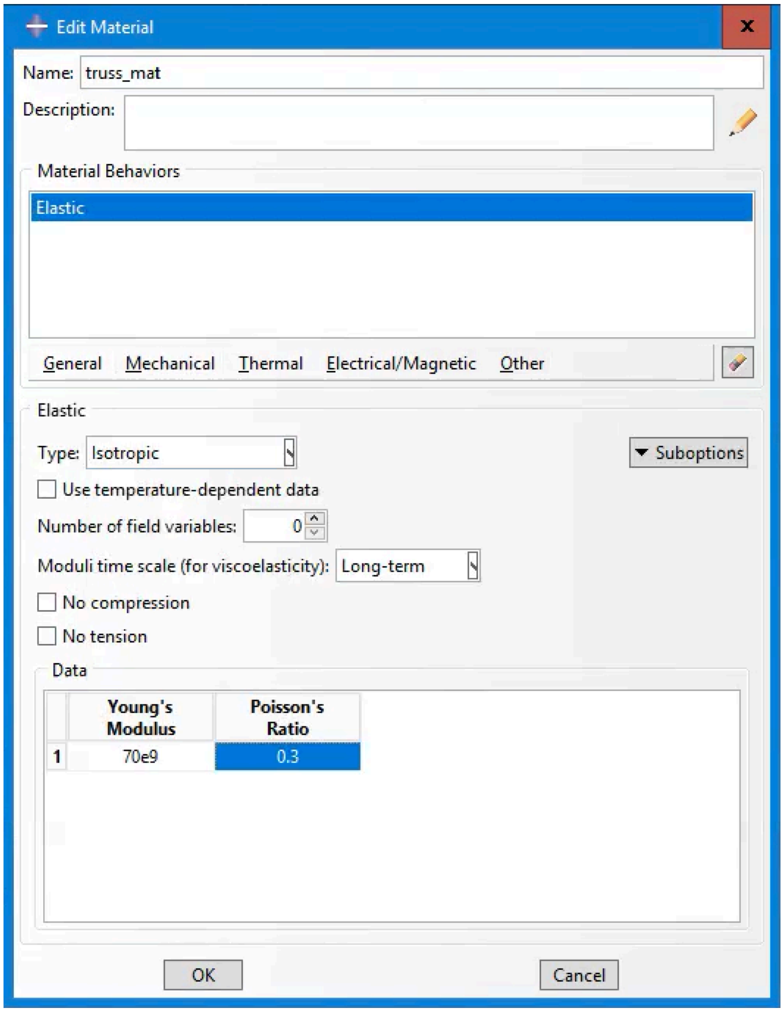


Figure 11 Entering mechanical properties values

- Next, click on the **Create Section** button. When the Create Section window appears, name the section truss_sect. Then, check **Beam** in the Category and choose **Truss** as the Type. (Note: It does not mean that Truss members are beam elements; however, in the Abaqus they are considered in the same section category)

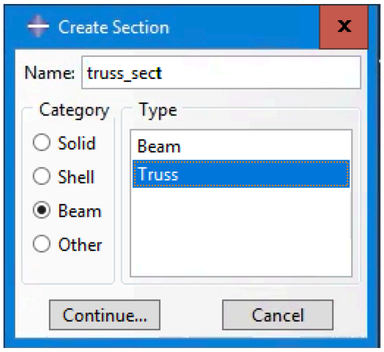


Figure 12 Create new section

- Click on the **Continue** button. In the next window, choose the already created material (truss_mat) to assign it to this section. You can also create new materials in this window. Then, enter the **Cross-sectional area** 0.1 m^2 and click on **OK**.

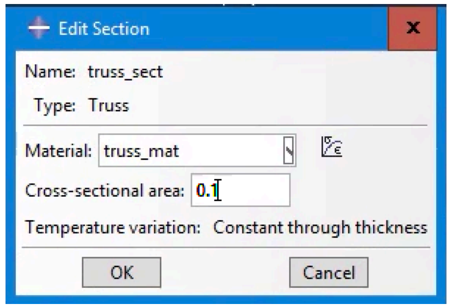


Figure 13 Create new section

- If the problem includes various sections (i.e. different material properties and/or cross-sectional area), the same procedure should be followed to define all other materials and/or sections.
- Then, the defined section should be assigned to the corresponding members (in this case all members). By clicking on the **Assign Section** button, the message **Select the regions to be assigned a section** will be appeared in the prompt area. You can select multiple members by either keeping the **Shift key** and clicking on the elements or drawing a box including all the members. When you select the elements with similar sections (in this case, all elements), click on the **Done** button. In the Edit Section Assignment window, choose the truss_sect section (Figure 14) and click on **OK**. In the case of various element sections, the same procedure should be repeated until all the defined sections are assigned to their corresponding members.

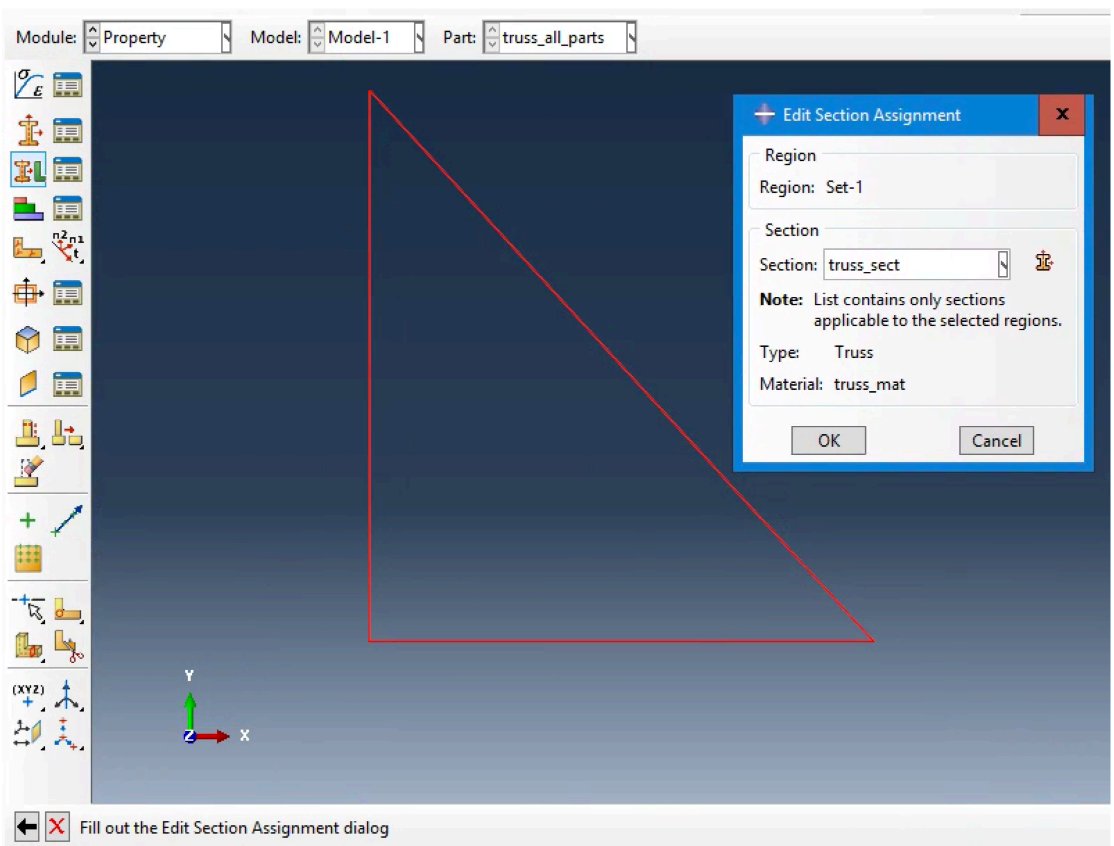


Figure 14 Assigning sections to elements

- Next module is **Assembly**. Generally in Abaqus, each part is oriented in its own coordinate system, independently. Various parts should be assembled to form the model geometry. This will happen by creating instances of defined part(s) and then positioning them relative to each other in the global coordinate system.

  An instance may be independent or dependent. Independent part instances are meshed individually, while the mesh of dependent instances is associated with their original part meshing. (Further information can be found in:
  http://abaqus.software.polimi.it/v6.14/books/usi/pt03ch13s03.html)

- Creating instances from original part is very helpful and avoid drawing all geometry at a time. In particular in the modular geometry, it is a good idea to draw one of the members and then created instances of the first one and finally assemble them. There are many functional tools in the associated toolbox that can help define the pattern of geometry.

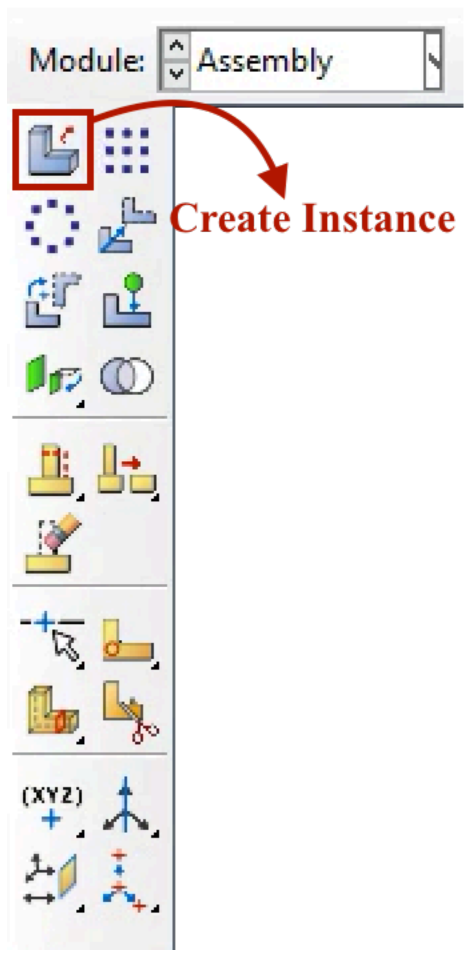


Figure 15 Assembly toolbox

- By clicking on the **Create instance** button, the corresponding dialog box will be appeared (Figure 16). In this example, the whole geometry has been created as one part; therefore, only one instance is enough to be created. Therefore, choose truss_all_parts in the **Parts** section and click on **OK**.

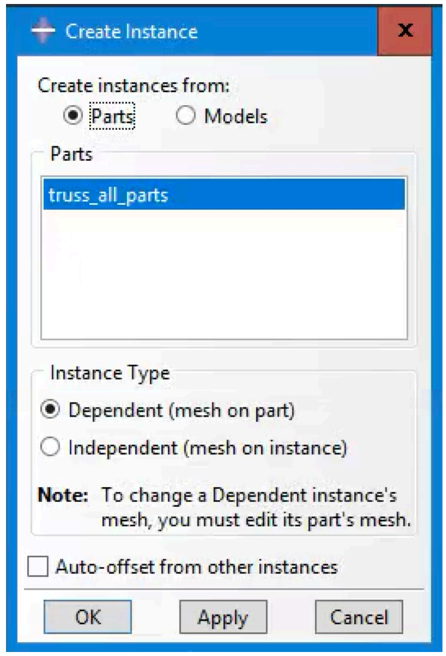


Figure 16 Create an instance

- Next, analysis configuration should be set. Choose **Step** module and click on the **Create Step** button.

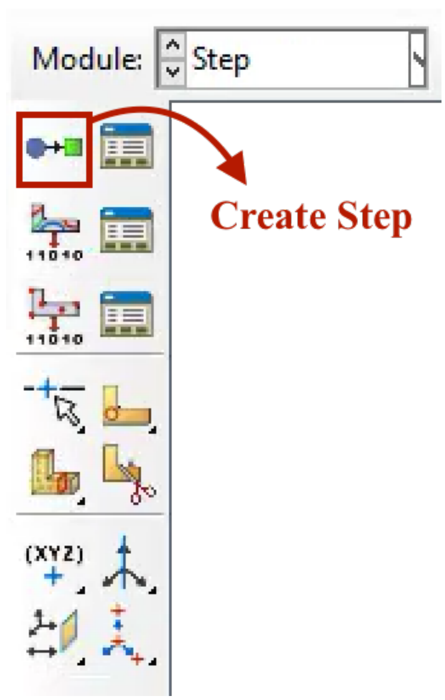


Figure 17 Step toolbox

- In this simple analysis, the load is applied in one step and a static analysis can solve the problem. Therefore, in the Create Step dialog box, select **Static, General**, and click on **Continue**. In the **Edit step** dialog box, leave all the details as they are, and click on **OK** (Figure18).

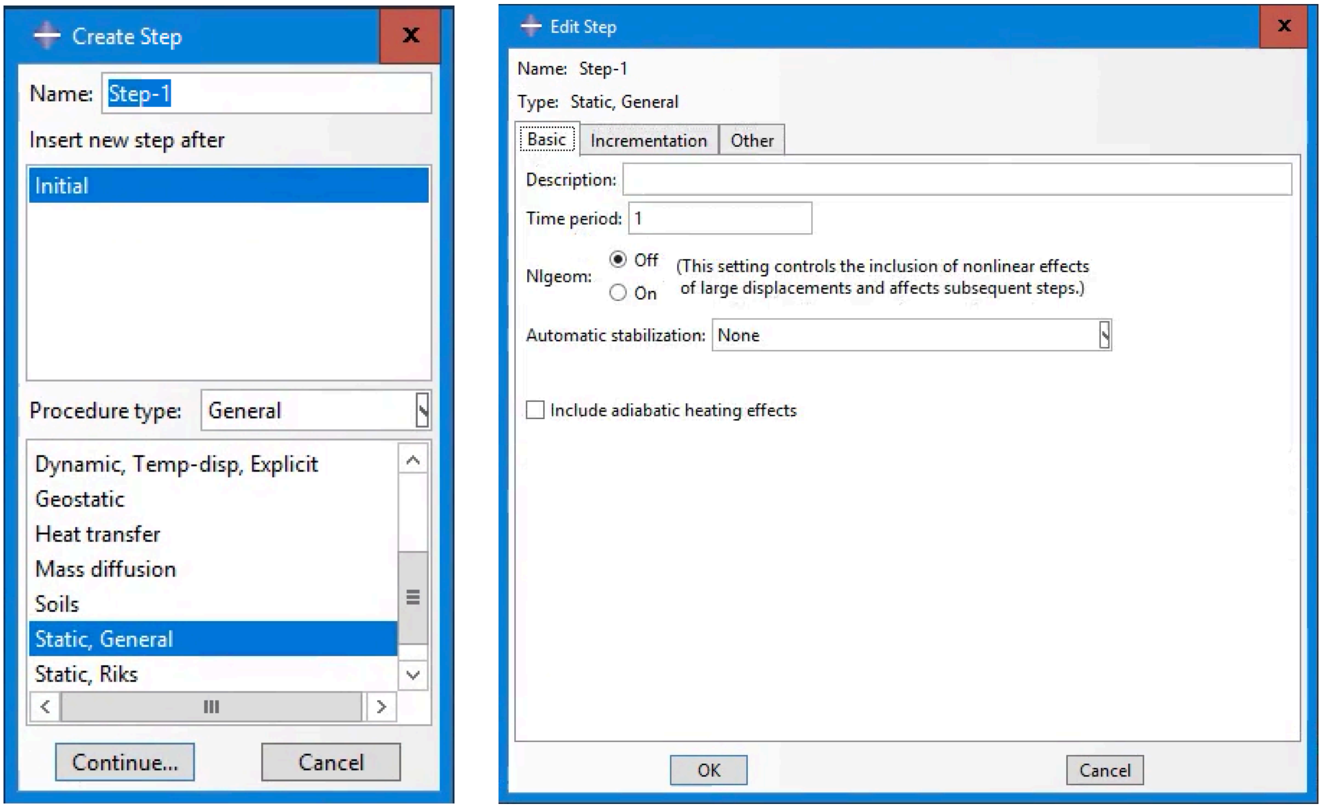


Figure 18 Configuring analysis: Creating a step (left) and setting its configuration (right)

- Next important module is **Load**. In this module, loads and boundary conditions should be defined.

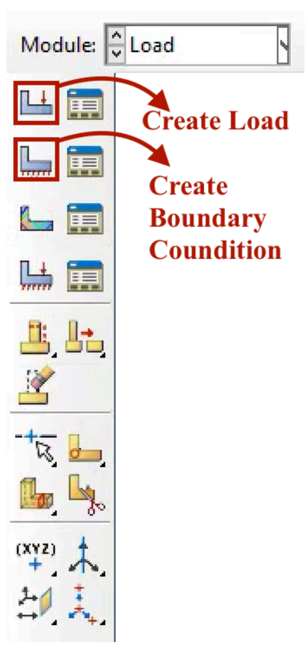


Figure 17 Load toolbox

- To define applied loads, click on the **Create Load** button. In the appeared dialog box (Figure 18), choose **Mechanical** as the category, and select the **Concentrated force** due to the problem definition (Figure 3). Then click on **Continue**. Note that the other types of mechanical load can be selected, if needed. For instance, Line load or Pressure can be chosen to define distributed forces in 2D and 3D problems, respectively.

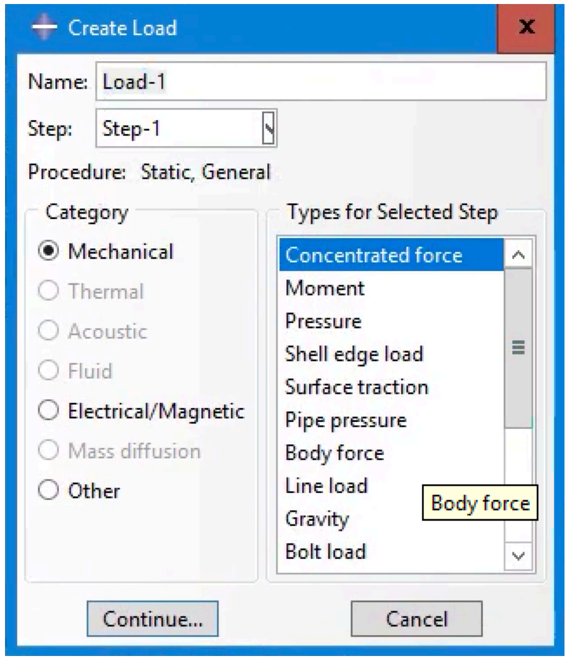


Figure 18 Create Load dialog box

- Then, Abaqus asks you to select points to apply the concentrated force. Click on the point that the concentrated force should be applied on. The selected point will be marked in red (Figure 19). Click on **OK**.

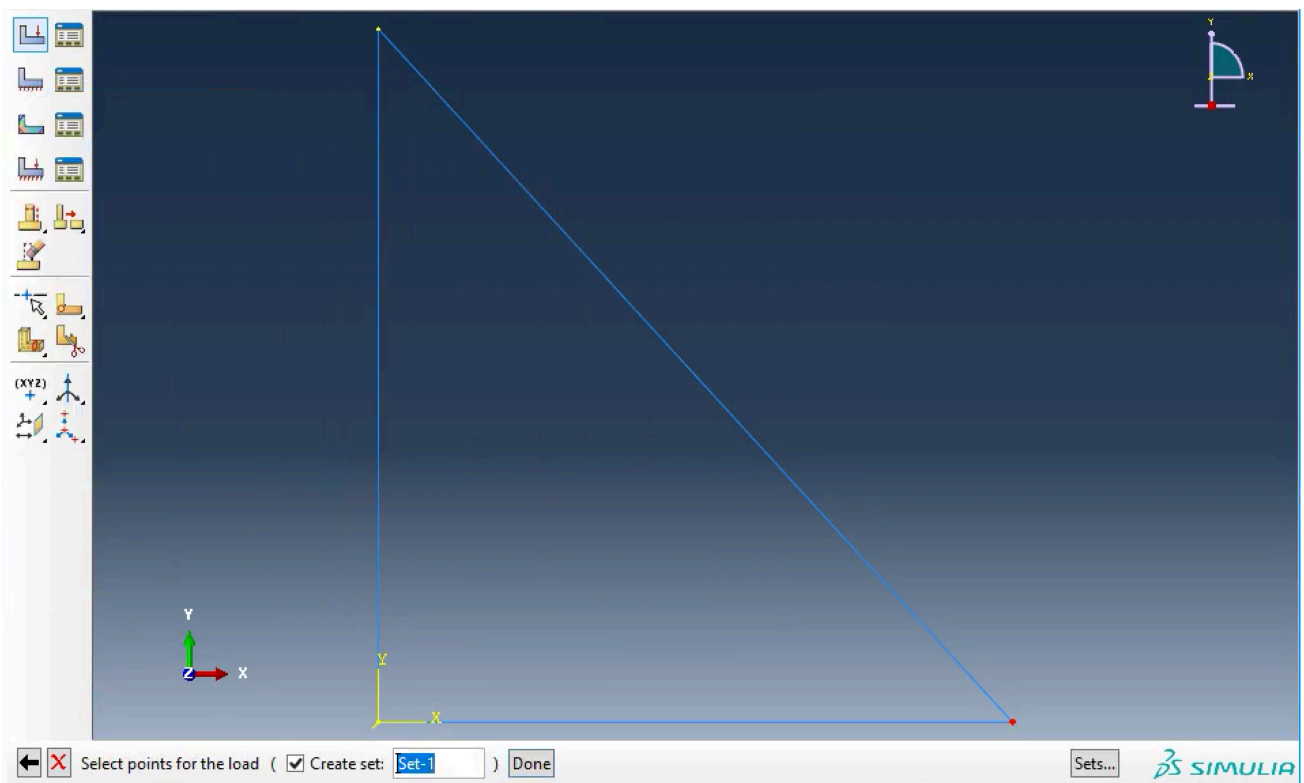


Figure 19 Picking the point for applying the concentrated force

- Afterwards, the **Edit Load** dialog box is appeared in which **CF1** and **CF2** means the amount of concentrated force in the direction of X and Y, respectively. In this example, these values are 0 and -1000 (Note the position direction of X and Y axis).

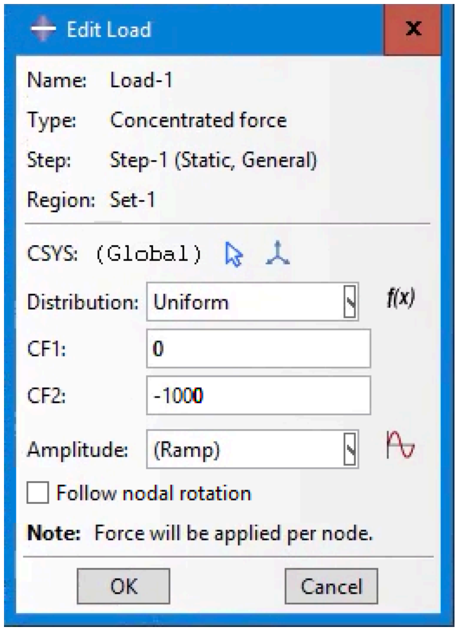


Figure 20 Entering the values of the concentrated force

- After clicking **OK**, the defined load(s) will be demonstrated on the viewport in yellow color (Figure 21).

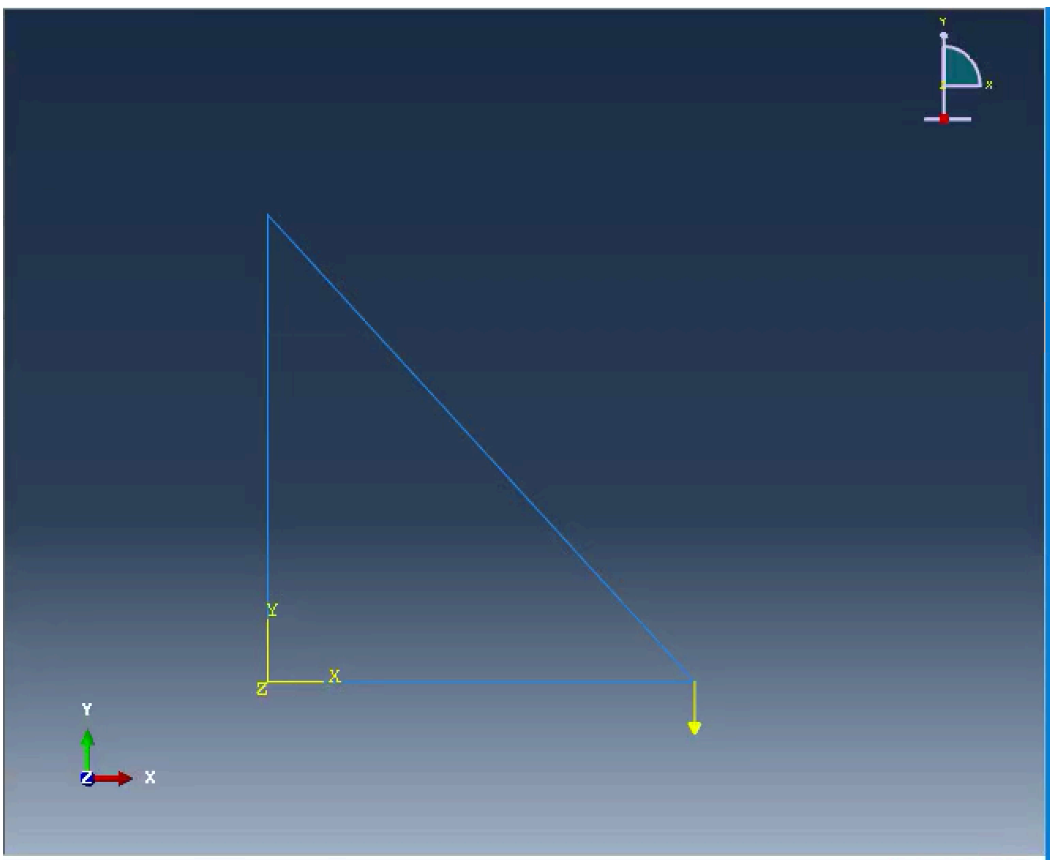


Figure 21 Demonstrating the defined concentrated load on the viewport

- In this example, there are two types of support at two different points: pin and roller. Therefore, two boundary conditions should be defined. To define the pin at node 1, click on the **Create Boundary Condition** button to define the boundary condition. In the appeared dialog box, choose **Mechanical** as the category, and **Displacement/Rotation** as the type, and click on **Continue** (Figure 22).

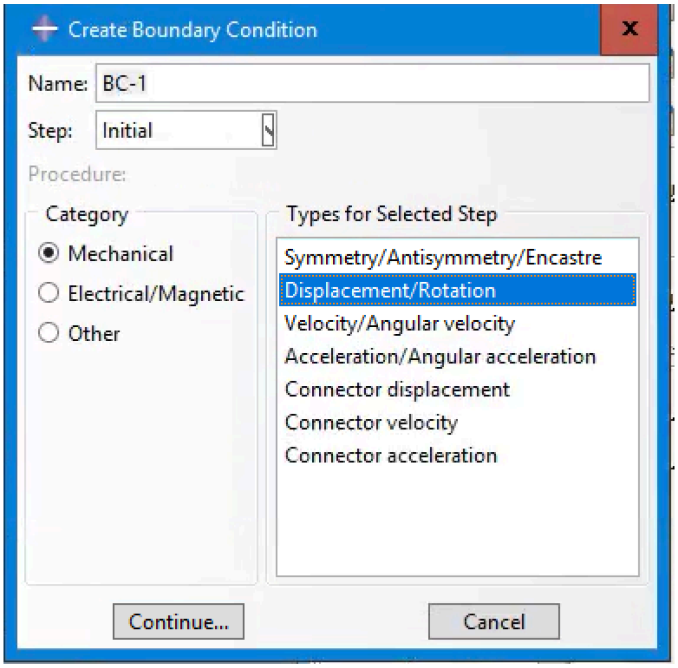


Figure 22 Create Boundary Condition dialog box

- Click on the region (point) imposed to the boundary condition, which is point 1 at the coordinate center. The selected point is marked in red color (Figure 23).

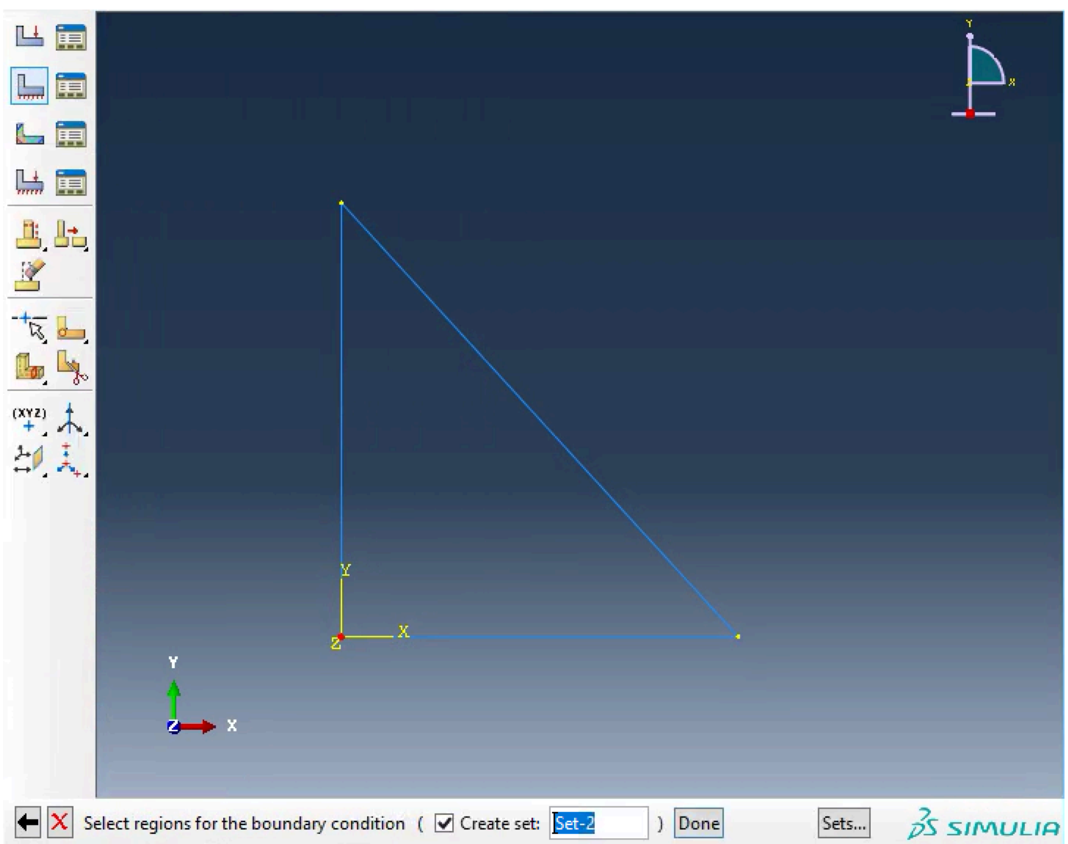


Figure 23 Selecting region imposed to the boundary condition

- Click on the **Done** button to appear the **Edit Boundary Condition** dialog box in which the corresponding degree of freedom should be checked. In a 2D setup, U1 and U2 are translations in direction X and Y, respectively, and UR3 represents the rotation. Toggle on U1 and U2 since the support at this point is a pin and all translational degree of freedom need to be constrained (Figure 24).

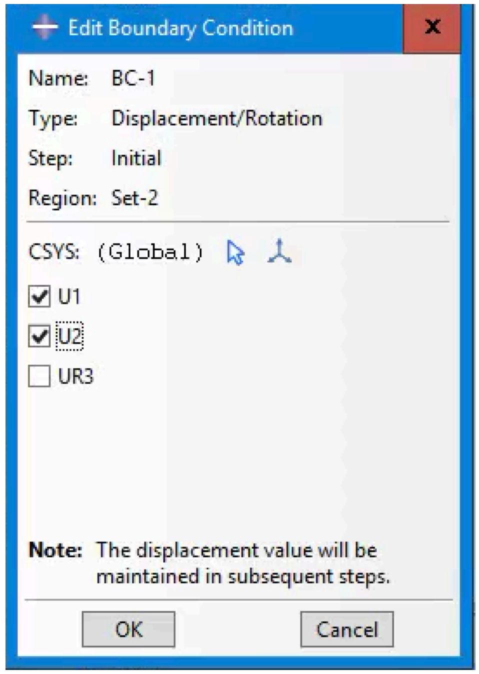


Figure 24 Edit Boundary Condition dialog box

- Repeat the same procedure for the roller support. In this case, only U1 should be toggled on (Figure 25 and 26).

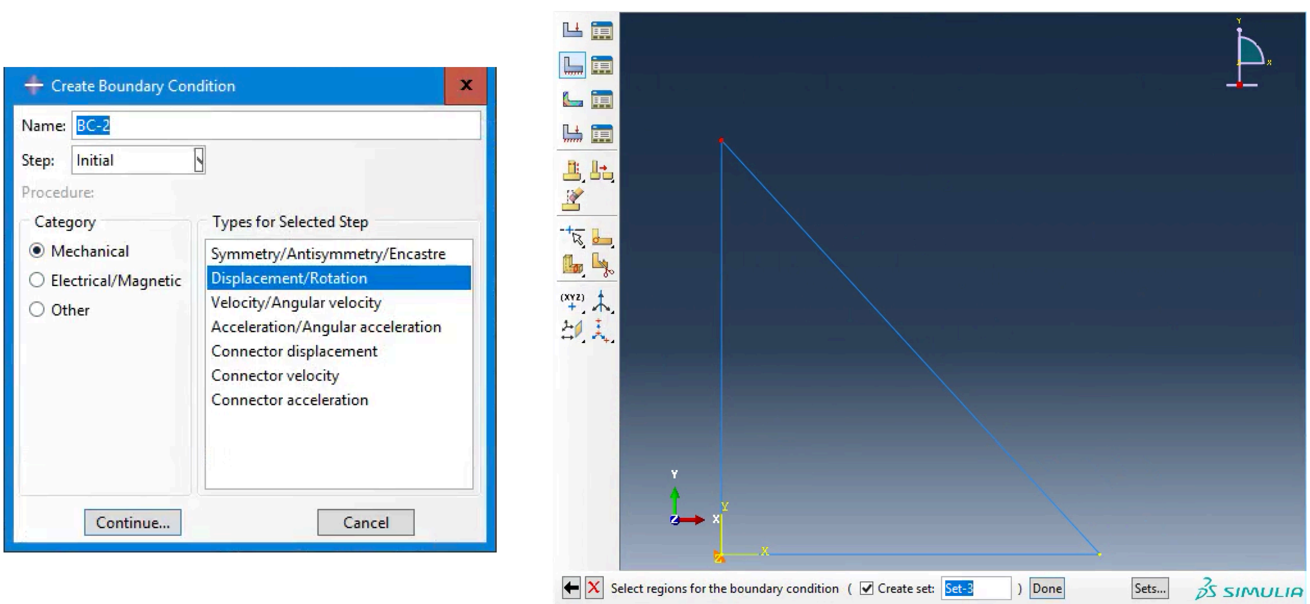


Figure 25 Defining the roller boundary condition

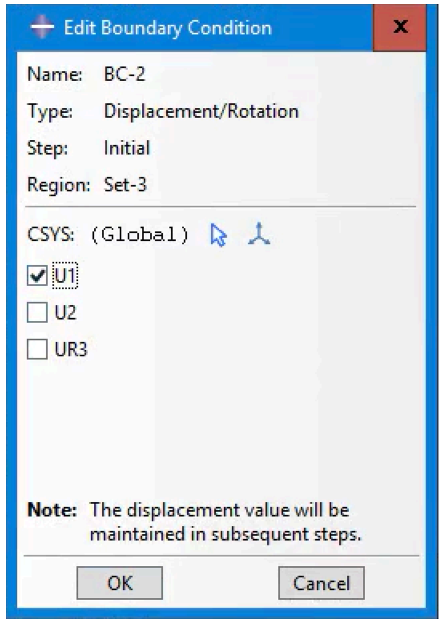


Figure 26 Defining the roller boundary condition

- Abaqus/CAE displays small orange arrowheads in the direction of constrained degrees of freedom (Figure 27).

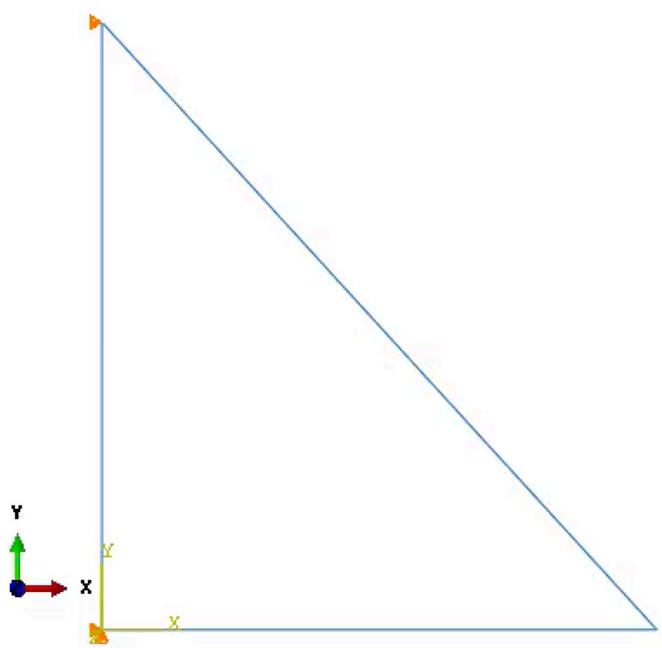


Figure 27 Displaying constrained degrees of freedom in the viewport

- Next, the elements should be defined. Choose **Mesh** module from the Module List. Since the instance has been defined dependent, click on the Part on the Context bar to mesh the corresponding part (in this case, truss_all_parts).

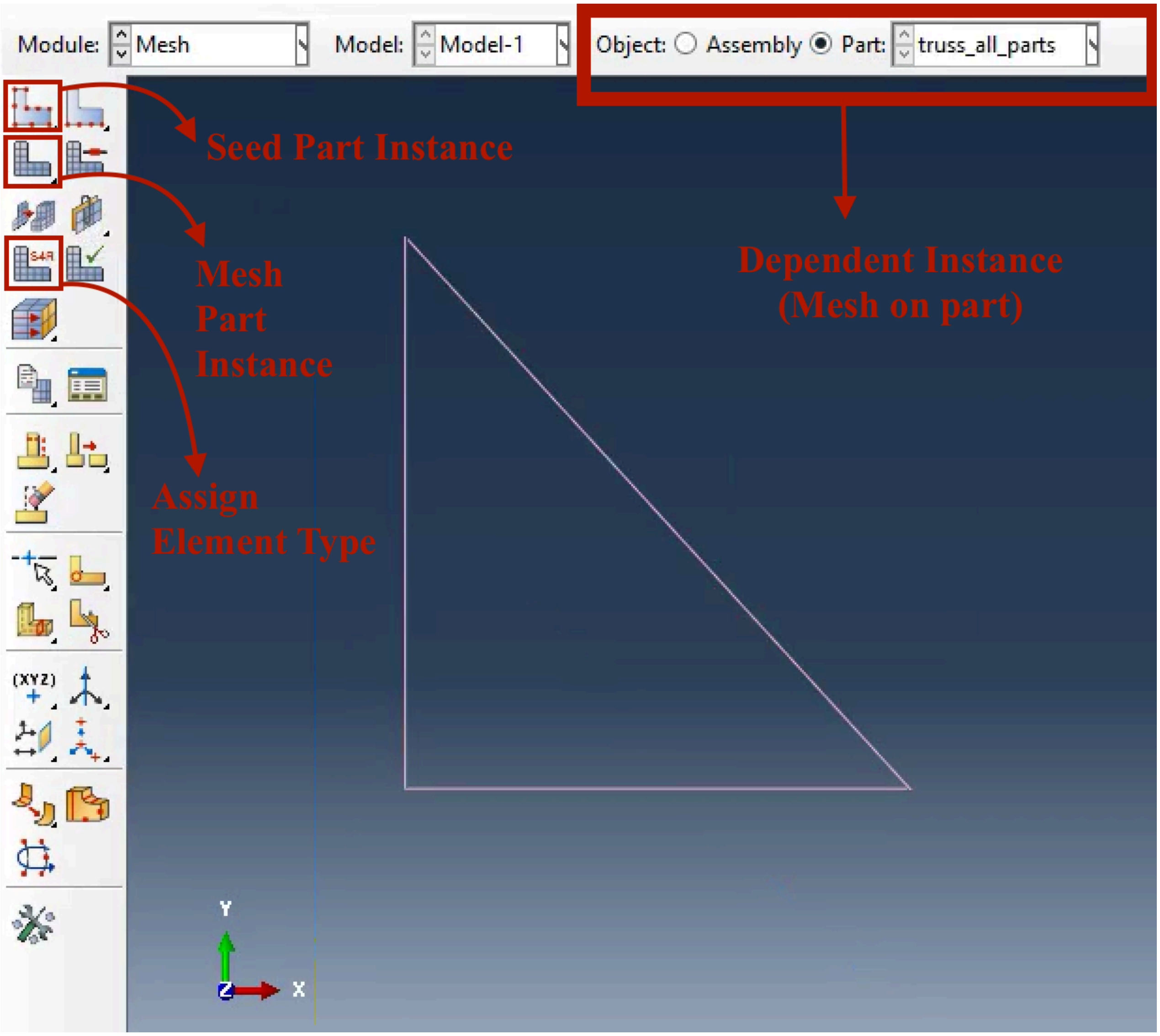


Figure 28 Mesh toolbox

- It is a good practice to assign element types at the beginning. Click on the **Assign Element Type** button, and select the whole part. In the appeared dialog box, choose **Truss** element (Figure 29). As you can see, this element recognized as **T2D2** element in Abaqus (You can find more information about Abaqus pre-defined elements in the following link: http://abaqus.software.polimi.it/v6.14/books/usb/pt06.html)

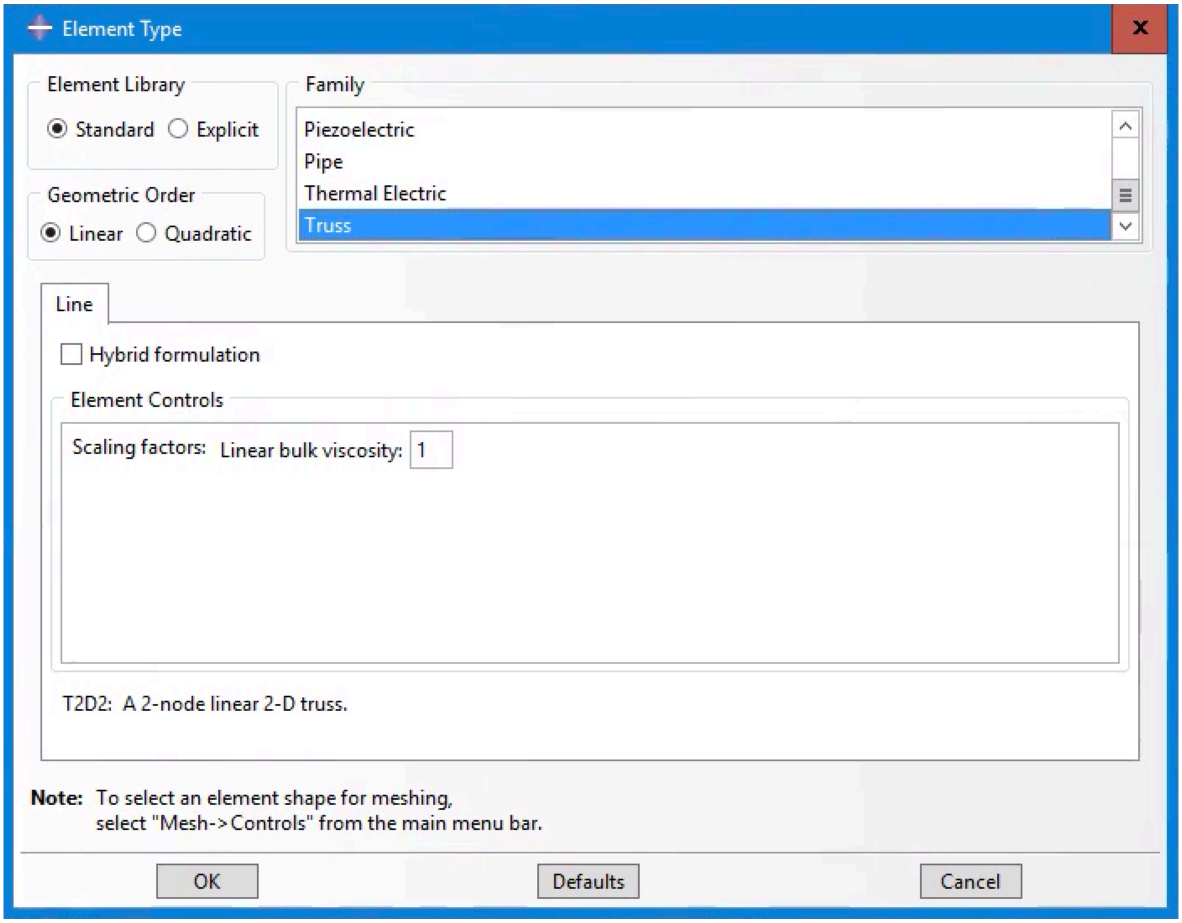


Figure 29 Assigning element type

- Next, click on the **Seed Part Instance** button to define global seeds. In the appeared dialog box, enter 2 (or more) for **Approximate global size**. Seeds are actually the nodes of the finite elements defining the approximate element size for all edges of a part or part instance. The toolbox provides a variety of tools for defining seeds. In this example we want to create one element per each truss member. Entering a number bigger than the maximum member length, which in this example is $\sqrt{2}$, for approximate global size is a naïve way to dictate the program to create one element for each member (Figure 30).

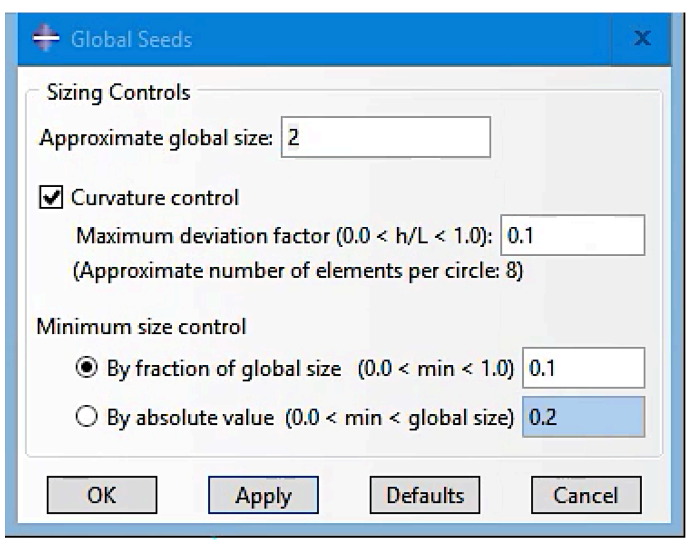


Figure 30 Seeding dialog box

- A more elegant way is using **Seed Edges**. You can click on the button, then select all truss members, and click on **Done** button. In the appeared dialog box, in the Method section, toggle on **By number**. Then, in the Sizing Controls, enter 1 as **Number of elements**. (Please note that you may prefer to run an analysis with having more elements. In that case, you can define more seeds to have more elements). The defined seeds will be displayed by small white squares (Figure 31).

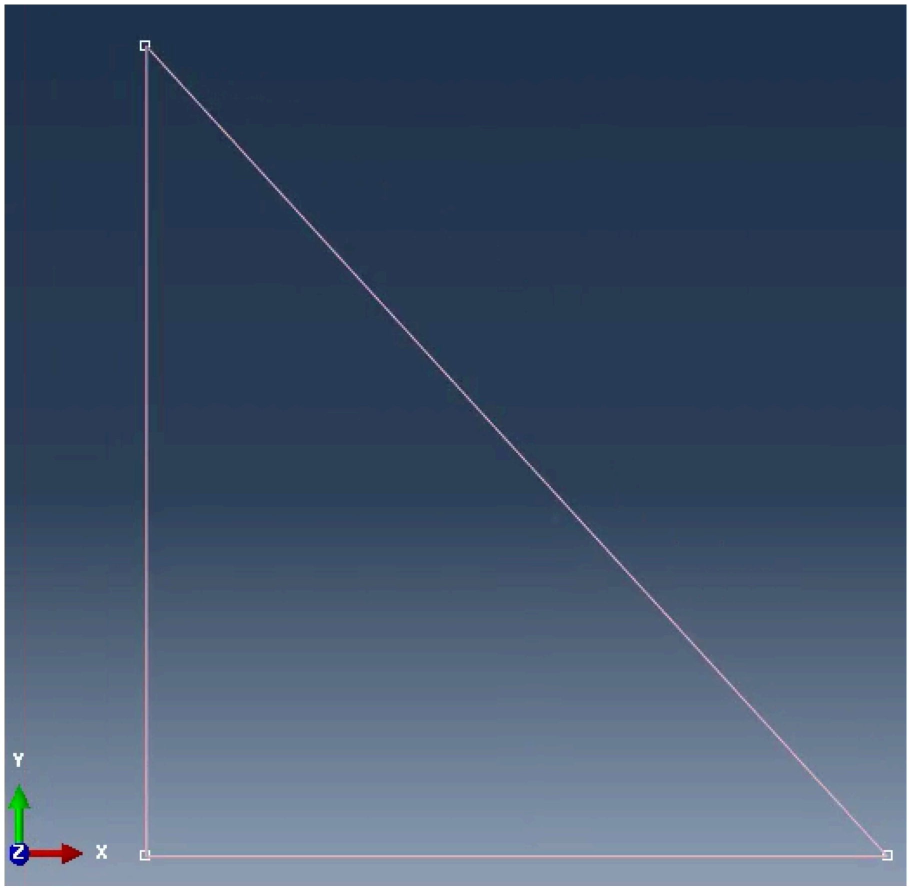


Figure 31 Displaying seeds

- Next, click on the **Mesh Part** button, and then, click on **Yes**. Abaqus will show you a message on message area about successfully generating the mesh and number of generated elements. For further inspection, you can use **Query** toolset from **Tools** menu. In general, this toolset can provide you with useful information in each module. Query toolset in the Mesh module allows you to obtain information about the nodes and elements in the mesh. For instance, by clicking on Element and then selecting an element, the Query toolset gives you some information element id, type, and nodal connectivity (Figure 32)

  (More information on querying a mesh:
  http://abaqus.software.polimi.it/v6.14/books/usi/pt03ch17s06s02.html)

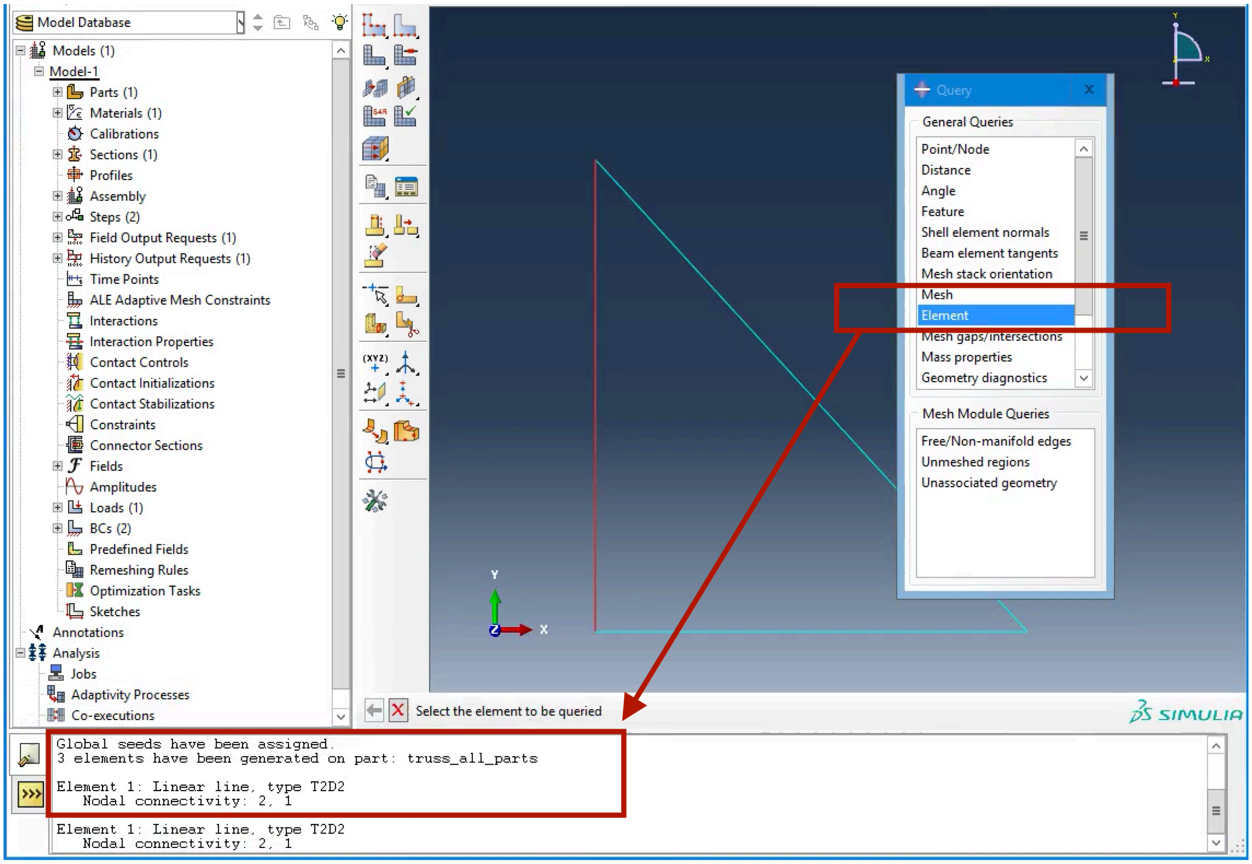


Figure 32 Query elements

- Next step is creating a job that is associated with the model. Select **Job** on the Module list. Then, click on the **Create Job** button. In the **Create Job** dialog box, name your job or just leave the name as it is. Remember that the input and output files and also the other temporary files will be generated under this name. Then, click on the **Continue** button (Figure 33).

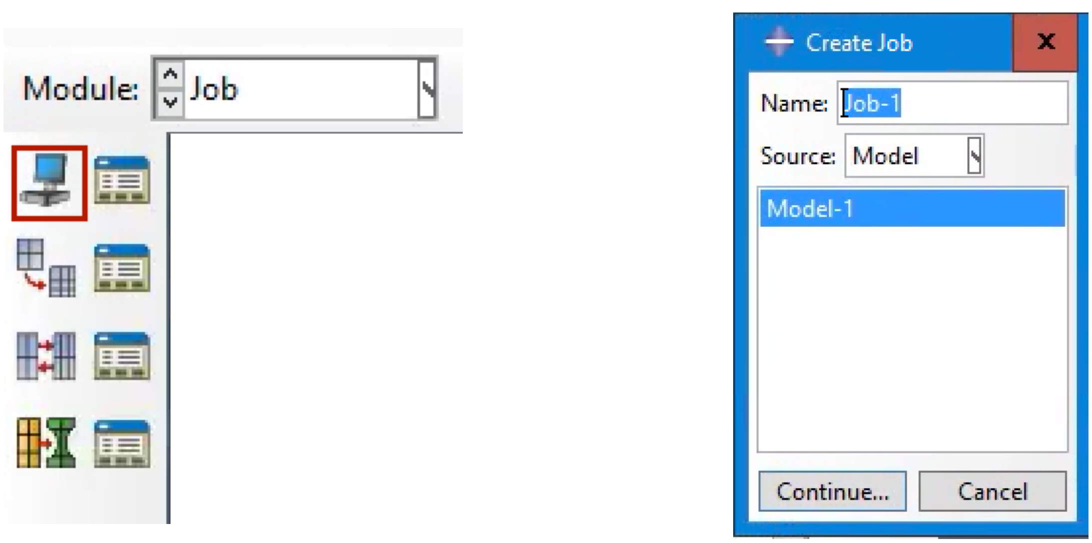


Figure 33 Creating job

- The **Edit Job** dialog box provides many settings regarding the analysis, such as memory, parallelization, and precision settings (Figure 34). For this example there is no specific settings. You can also add some description about the analysis.

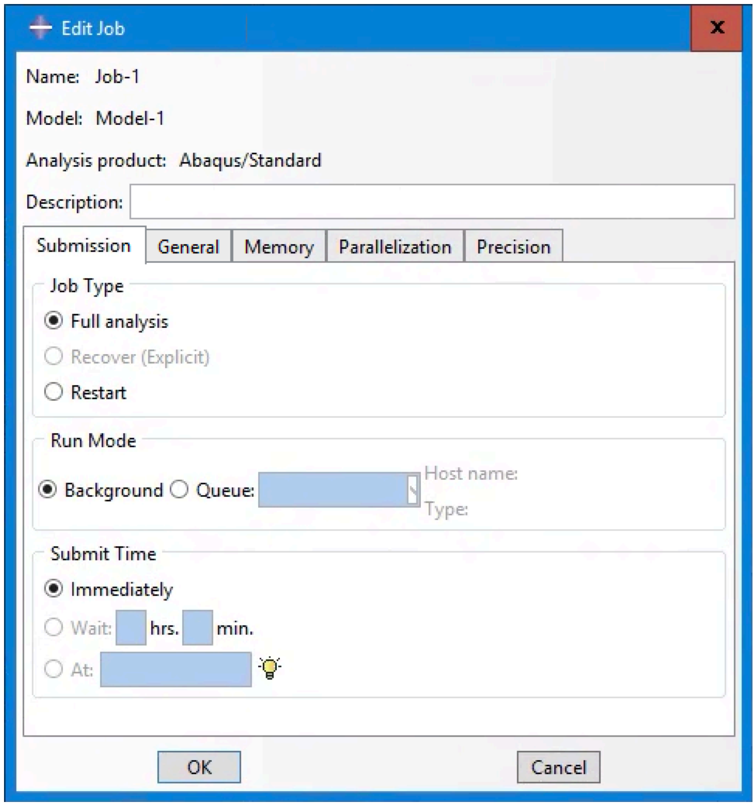


Figure 34 Edit Job dialog box

- Then, click on **Job Manager** button. In the appeared dialog box (Figure 35), first, click on the **Write Input** button. This generates an Abaqus Input file (*.inp) with the name of the job that you have just created.
- In general, when you submit a job associated with a model for analysis, Abaqus/CAE generates an input file representing the model and then analyze the input file by calling Abaqus/Standard, Abaqus/Explicit, or other related solvers. The input file is an ASCII format file that will be read by these solvers. You can also write an input file by yourself or editing an existing input file in a text editor.

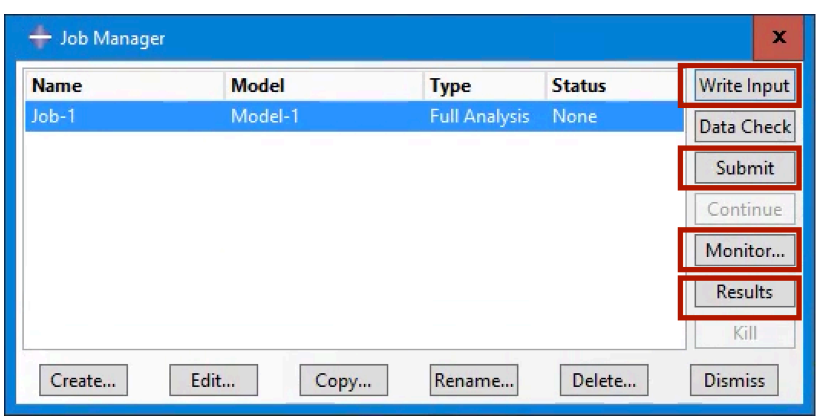


Figure 35 Job Manager: Producing the input file and submitting analysis

- Now, the model is ready to be analyzed. It is also a good idea to run a **Data Check** to make sure that there is no error in the model because of incorrect or missing data. Then, click on **Submit** to run the analysis. During the analysis, Abaqus/Standard sends some updates about the analysis status to Abaqus/CAE to allow you to monitor the progress of the job. Such information is accessible by clicking on the **Monitor** button to appear the Monitor dialog box (Figure 36). Besides, the program provides you with some updates about the status of the job in the message area (Figure 37)

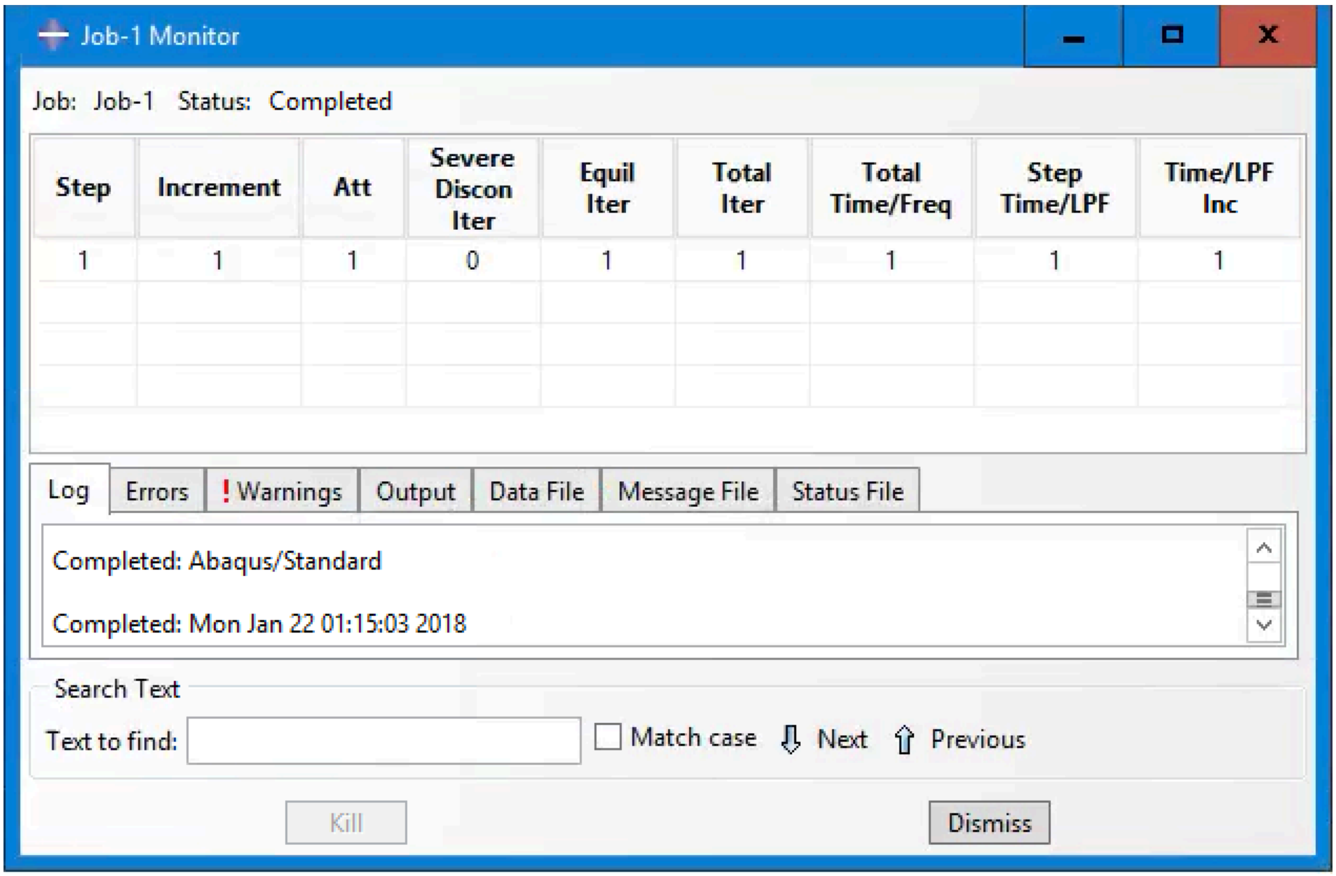


Figure 36 Monitoring the job analysis

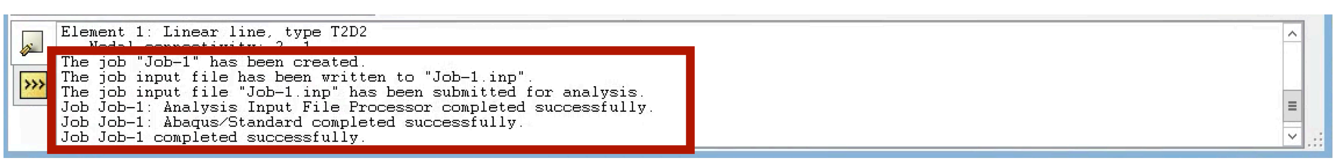


Figure 37 Messages on message area notifying a successful analysis

- In general, when you start a session and begin defining a model, Abaqus/CAE generates some files. Furthermore, when you submit a job for analysis, Abaqus/Standard and Abaqus/Explicit create a set of other files. These files are typically generated and stored in the default work directory, which is **"C:\Temp"**. This work directory can also be changed by selecting **Set Work Directory** in the **File** menu.

As an example, Figure 38 demonstrates the generated files for analyzing this problem. For more information about the files generated during creating and analyzing a model, see the following links:
http://abaqus.software.polimi.it/v6.14/books/usb/usb-link.htm#usb-int-dfileextensions and
http://abaqus.software.polimi.it/v6.14/books/usi/pt02ch09s04.html

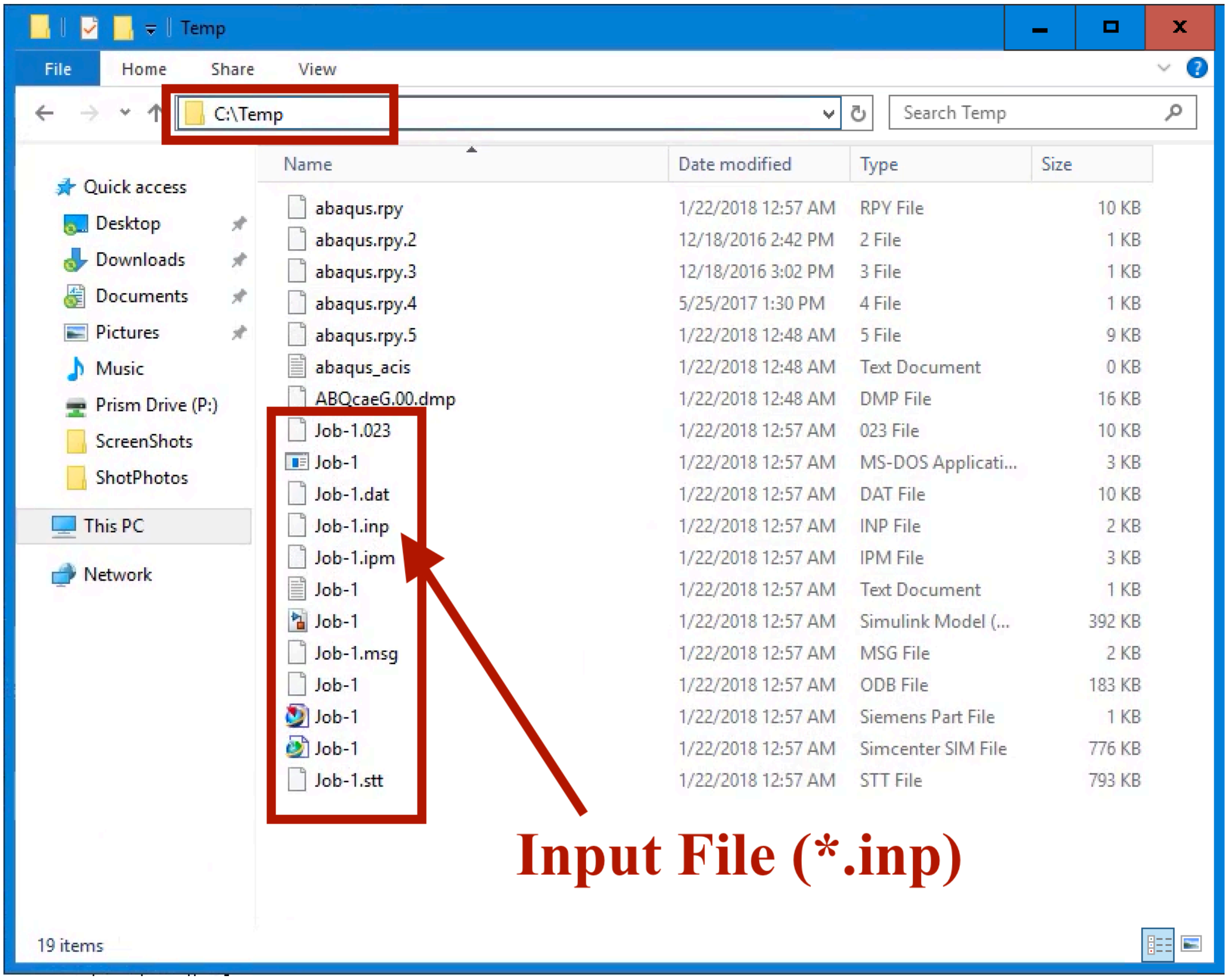
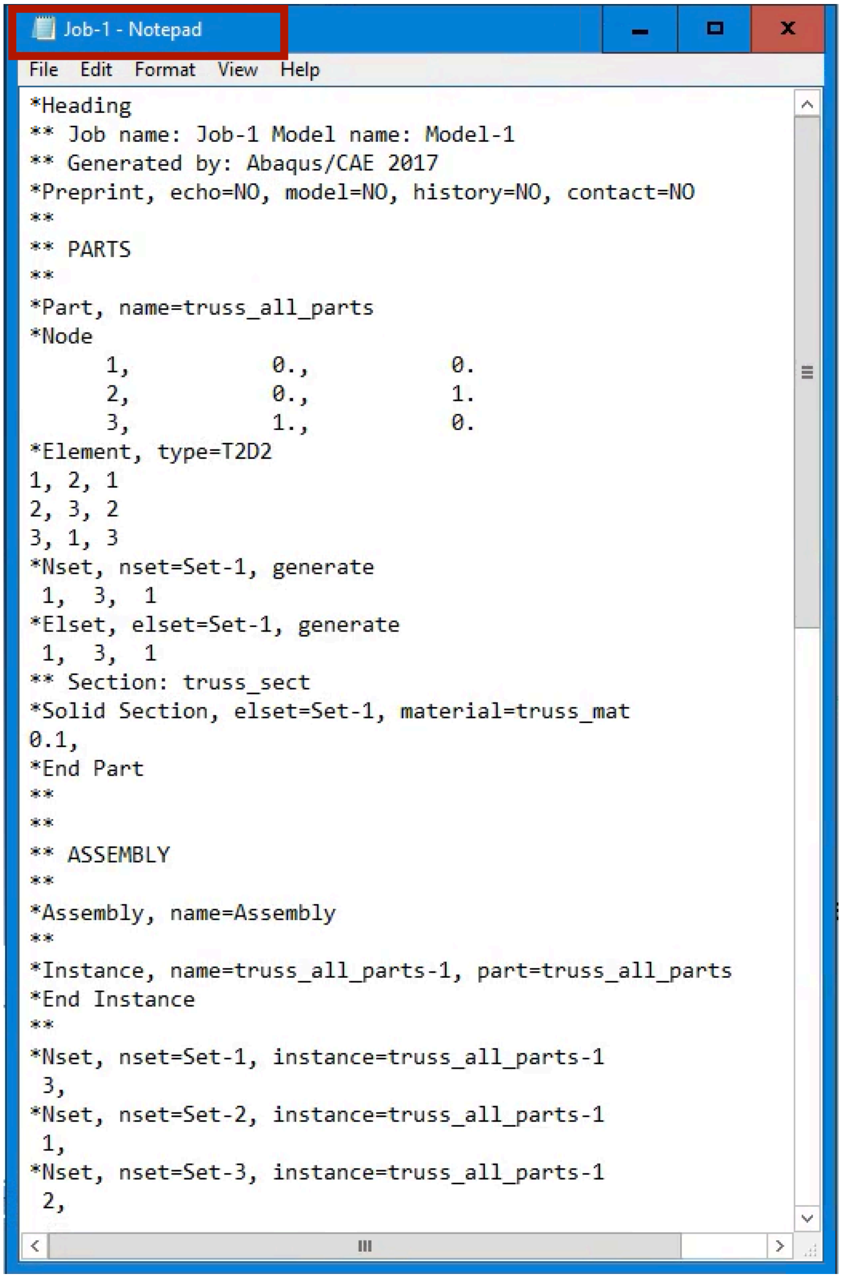


Figure 38 Generated files during the analysis

- You can also see the contents of the input file (job-1.inp) in Figure 39. If you are interested in writing or understanding an input file contents, you can find the required information with an example in the following link:
http://abaqus.software.polimi.it/v6.14/books/gsk/ch02s02.html



```
*Heading
** Job name: Job-1 Model name: Model-1
** Generated by: Abaqus/CAE 2017
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**
** PARTS
**
*Part, name=truss_all_parts
*Node
      1,           0.,           0.
      2,           0.,           1.
      3,           1.,           0.
*Element, type=T2D2
1, 2, 1
2, 3, 2
3, 1, 3
*Nset, nset=Set-1, generate
 1,  3,  1
*Elset, elset=Set-1, generate
 1,  3,  1
** Section: truss_sect
*Solid Section, elset=Set-1, material=truss_mat
0.1,
*End Part
**
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=truss_all_parts-1, part=truss_all_parts
*End Instance
**
*Nset, nset=Set-1, instance=truss_all_parts-1
 3,
*Nset, nset=Set-2, instance=truss_all_parts-1
 1,
*Nset, nset=Set-3, instance=truss_all_parts-1
 2,
```

Figure 39 Produced Abaqus input file by Abaqus/CAE

Post-processing with Abaqus/CAE

- Once the analysis is completed, you can click on the **Results** button (Figure 37) to move to the **Visualization** module. There are several tools on the associated toolbox to view your model and the results of your analysis. You may click on **plot Deformed Shape** or **Plot Contours on Deformed Shape** to view the displacements and stresses (Figure 40 and 41). For instance, a deformed shape displays the shape of model according to the values of nodal variables such as displacements. (You can find detailed information in the following link: http://abaqus.software.polimi.it/v6.14/books/usi/pt05ch40.html)

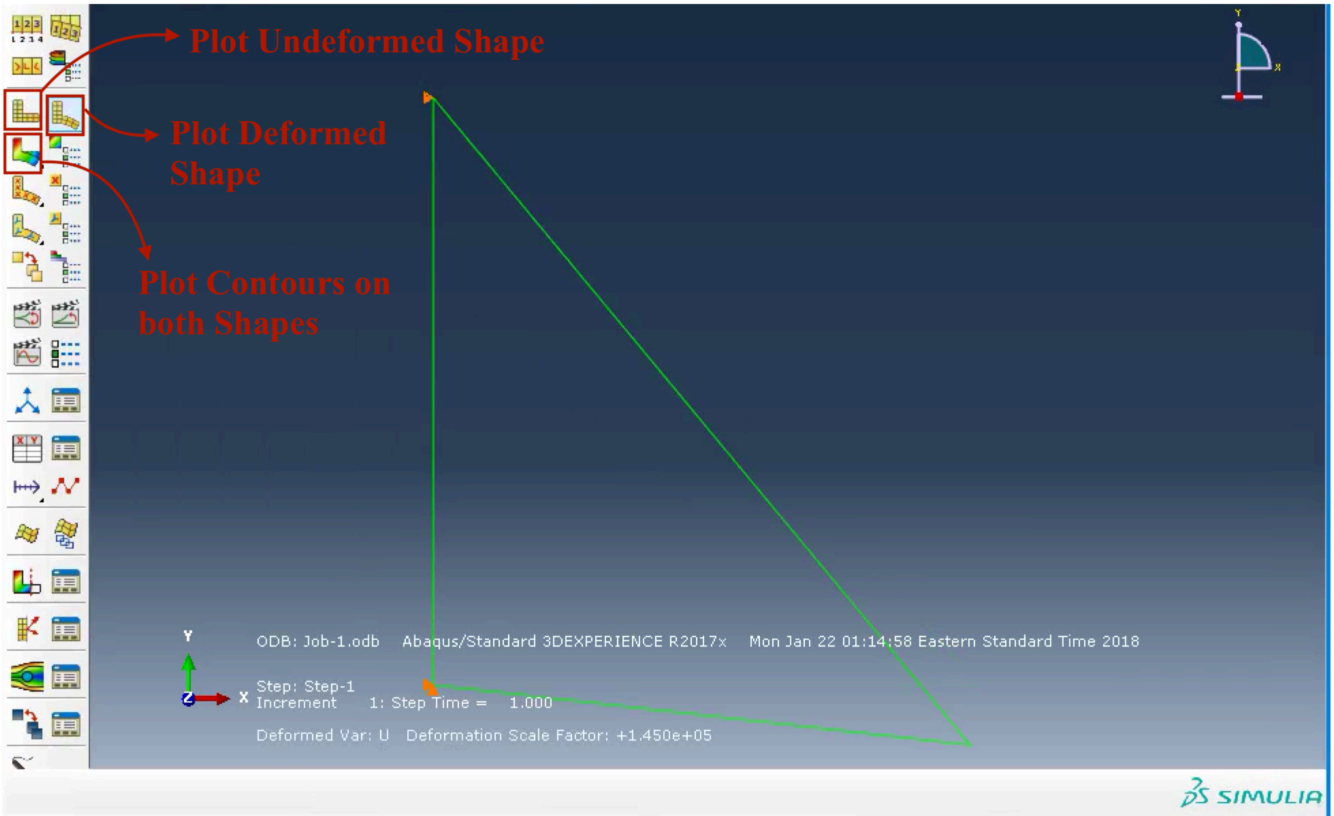


Figure 40 Visualization toolbox and output results (displacement)

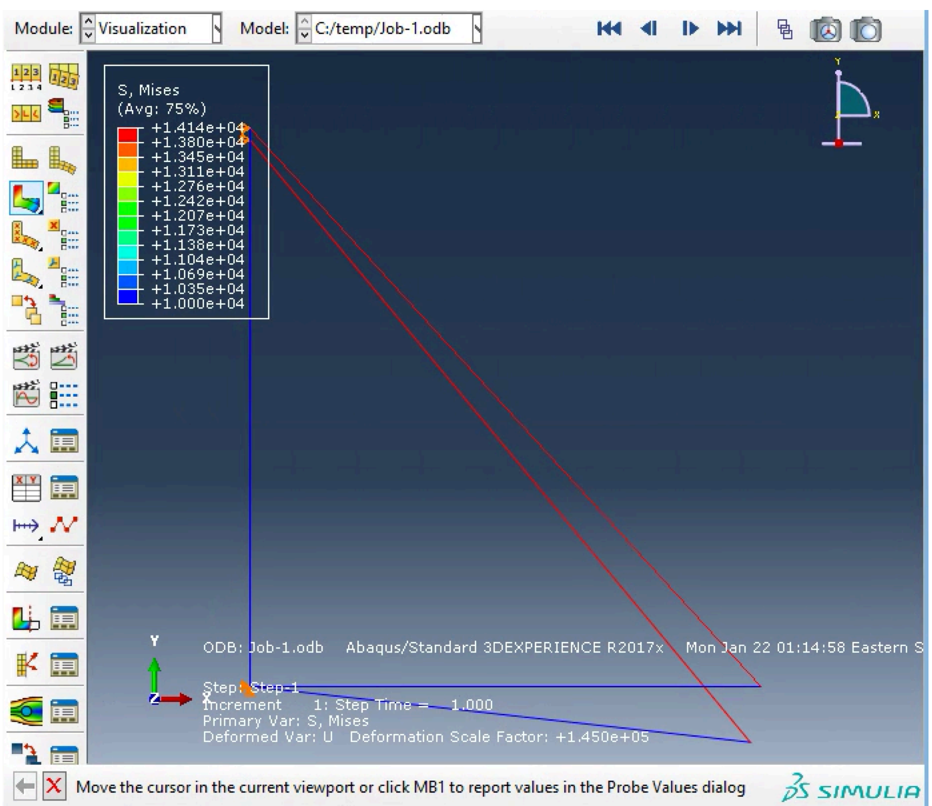


Figure 41 Output results (stress)

- To check the result values in all or some specific elements, you can use the **Query** toolset in the Visualization module. Again, you can access the Query toolset through **Tools/Query**. In the appeared dialog box, select **Probe values** from Visualization Module Queries. Then, if you move the mouse pointer over the elements and click on a specific element, the values will be displays on **Probe Values** dialog box (Figure 42).

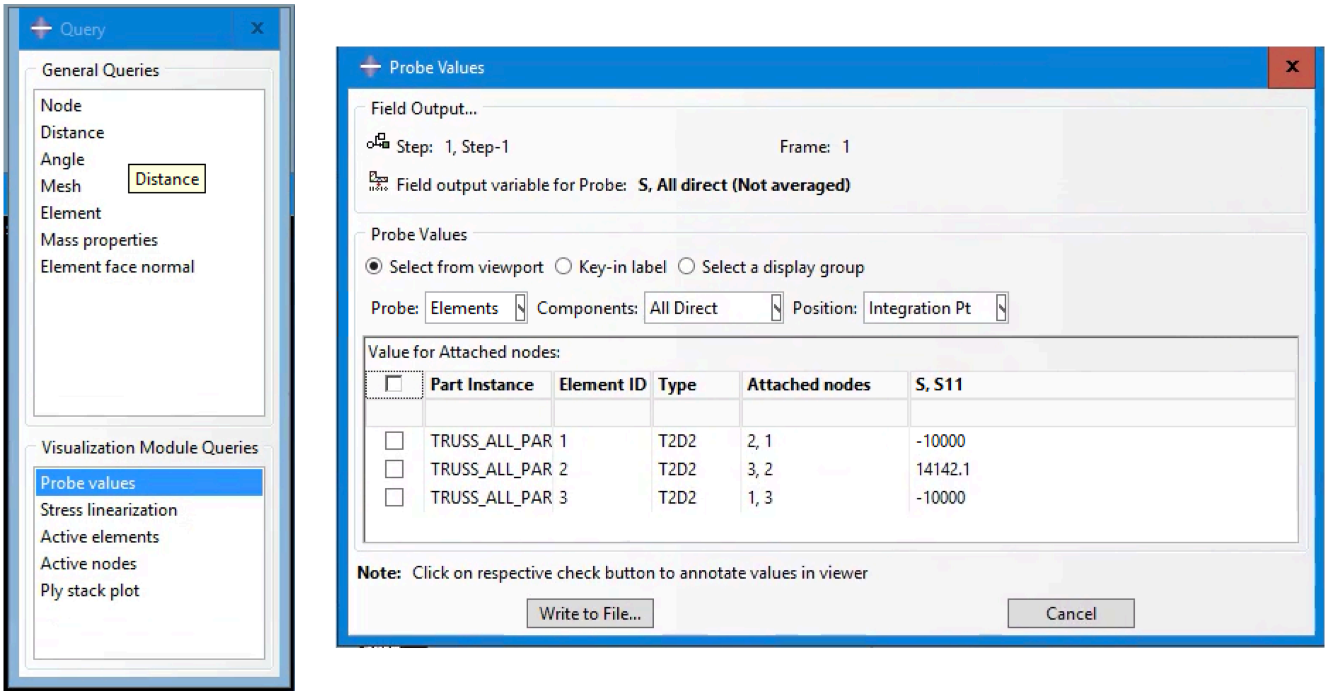


Figure 42 Probing output values

- You can also store these values in a text file by checking the associated checkbox and then click on the **Write to File** button. In the **Report Probe Values** dialog box, you are able to specify the name and location of the file and also the output format to store the output results (Figure 43).

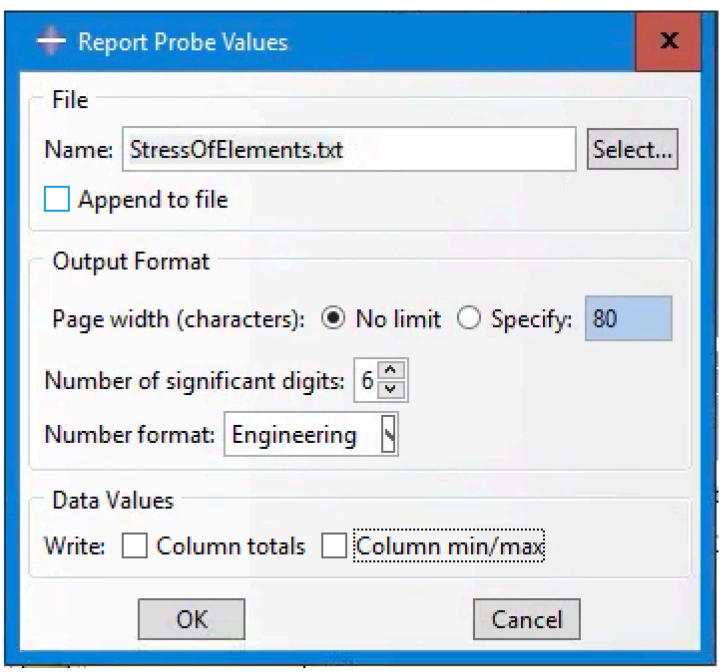
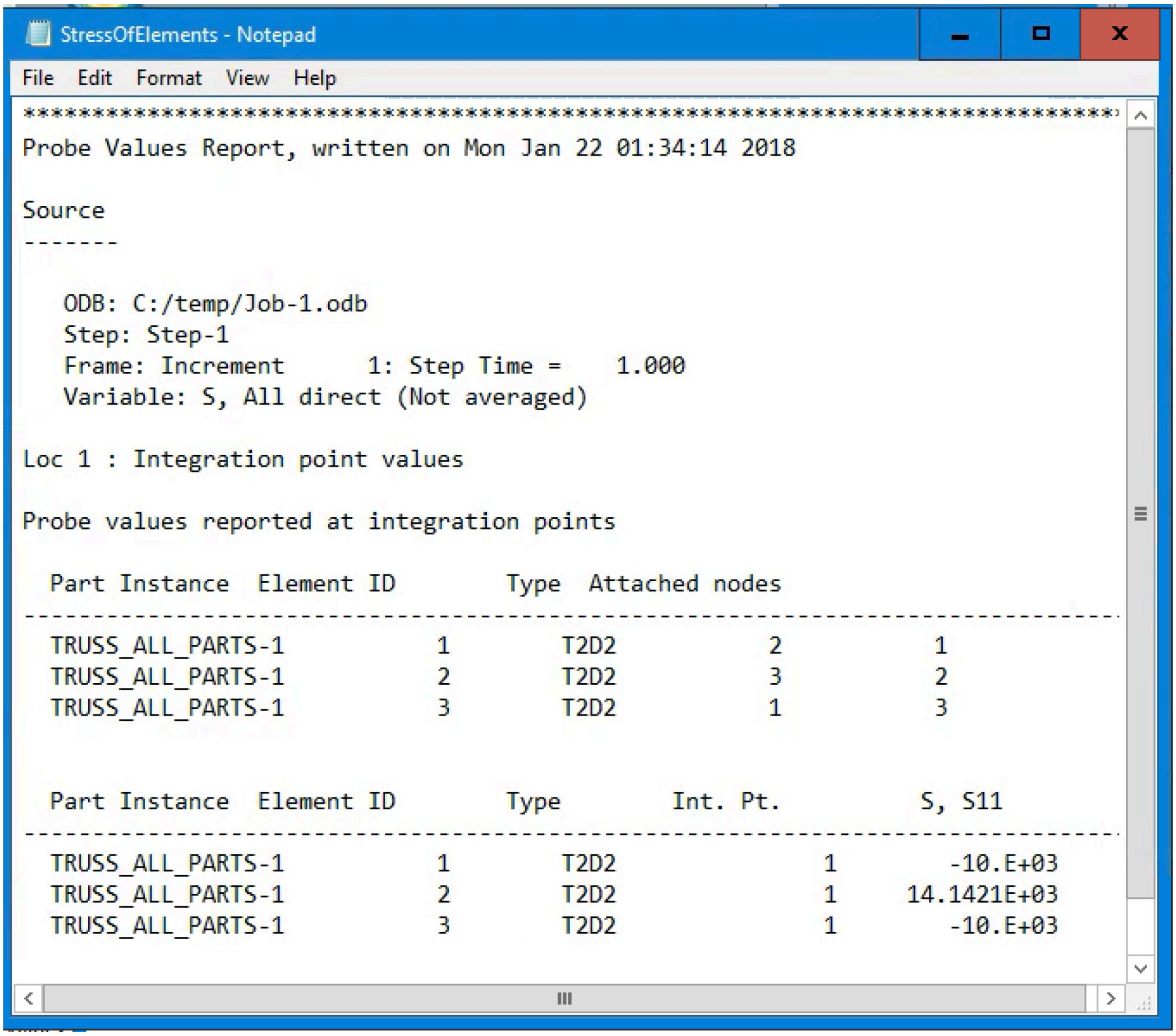


Figure 43 Writing probed values in an output file

- For Instance, Figure 44 demonstrates the stress values for all elements, which are stored in a file named StreeOfElements.txt (See Figure 43). You can double-check these values with the values in reference book [2].



StressOfElements - Notepad

File Edit Format View Help

```
********************************************************************************
Probe Values Report, written on Mon Jan 22 01:34:14 2018

Source
-------

   ODB: C:/temp/Job-1.odb
   Step: Step-1
   Frame: Increment      1: Step Time =    1.000
   Variable: S, All direct (Not averaged)

Loc 1 : Integration point values

Probe values reported at integration points

  Part Instance  Element ID       Type  Attached nodes
--------------------------------------------------------------------------------
  TRUSS_ALL_PARTS-1          1        T2D2          2            1
  TRUSS_ALL_PARTS-1          2        T2D2          3            2
  TRUSS_ALL_PARTS-1          3        T2D2          1            3


  Part Instance  Element ID       Type        Int. Pt.          S, S11
--------------------------------------------------------------------------------
  TRUSS_ALL_PARTS-1          1        T2D2                1        -10.E+03
  TRUSS_ALL_PARTS-1          2        T2D2                1      14.1421E+03
  TRUSS_ALL_PARTS-1          3        T2D2                1        -10.E+03
```

Figure 44 Stored file containing probed values

- Some of the result outputs are stored in text files and presented in the Appendix.

- It is worth mentioning that you can control what information is placed in the output database by modifying the output requests in the **Step** module. You may click on the **Field Output Manager** button to **Edit** current field output request or **Create** a new one. Figure 45 and 46 demonstrates the Field Output Requests Manager and Edit Field Output Request dialog boxes. (For more information see the following links:
http://abaqus.software.polimi.it/v6.14/books/usi/pt03ch14s04s01.html)

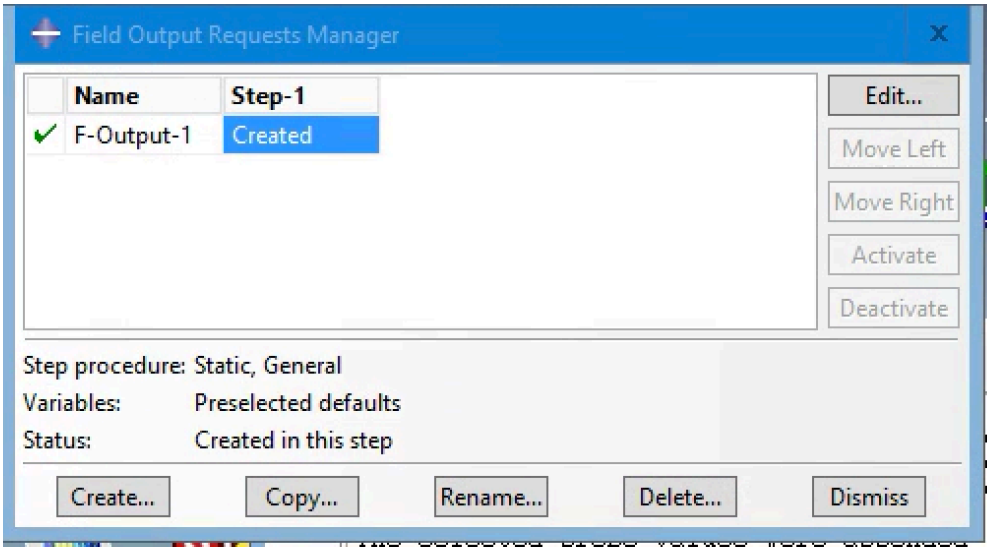


Figure 45 Editing field outputs

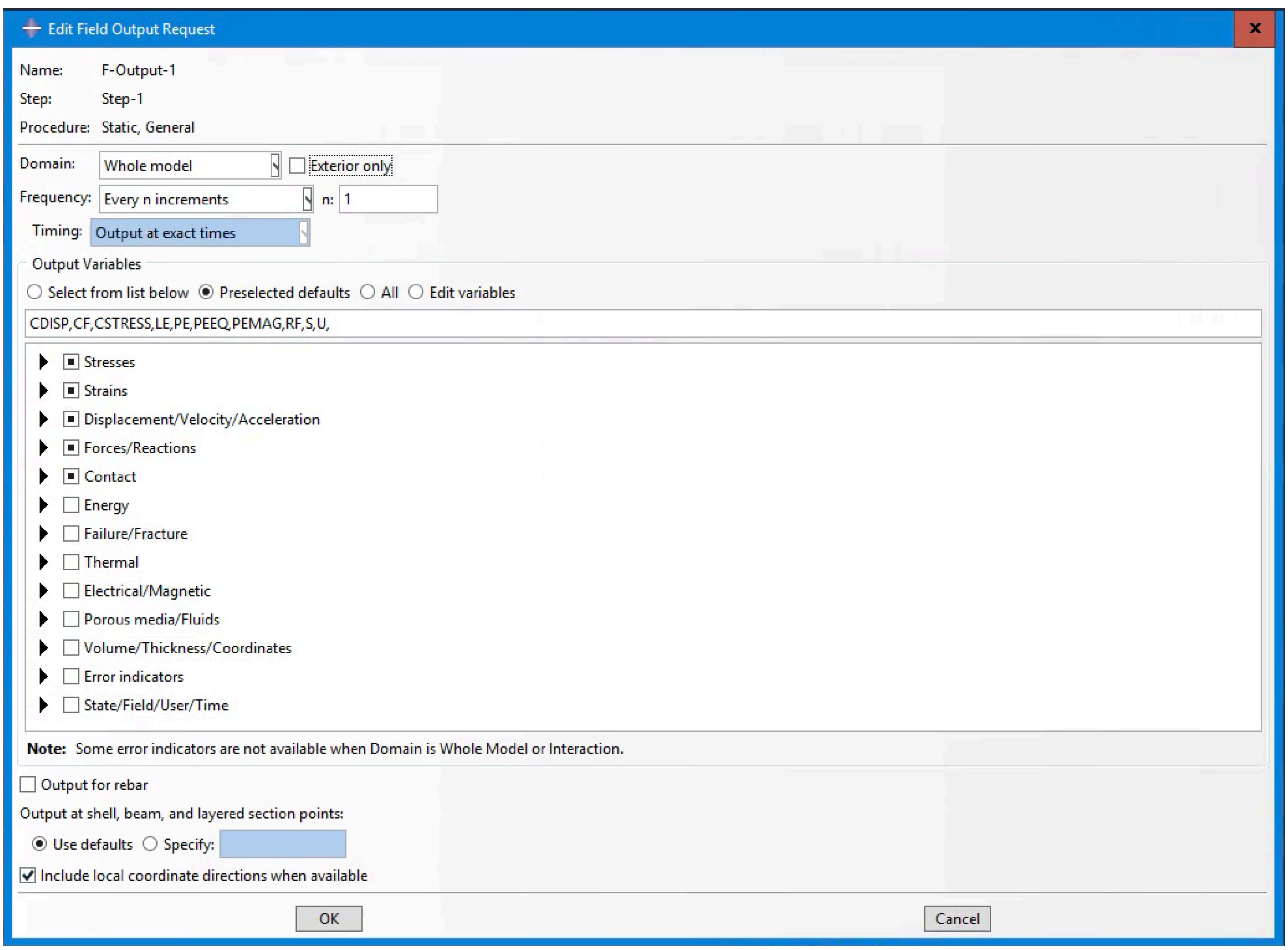


Figure 46 Editing field outputs

Finite Element Analysis of a Frame

Analysis of a frame is pretty much like a truss. The only important differences are assigning sections and also the elements type that you should choose in the Mesh module. The following pages briefly review the procedure for creating a model and running an analysis in Abaqus/CAE, and demonstrate the important considerations for frame analysis.

Problem: Analyze the frame shown in the following figure.

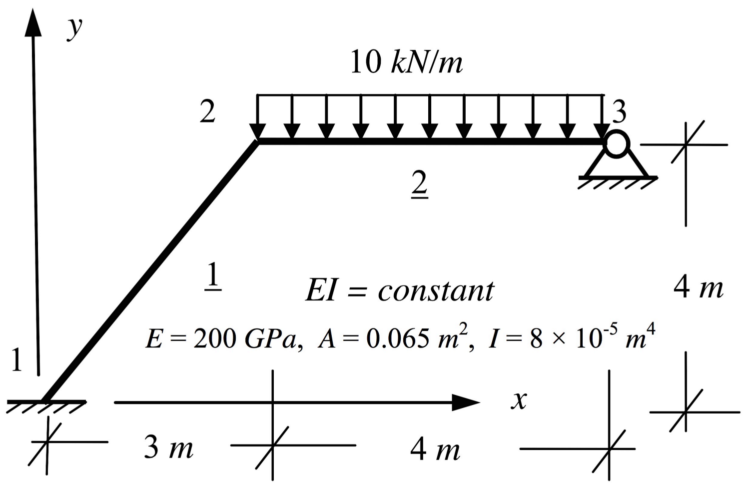


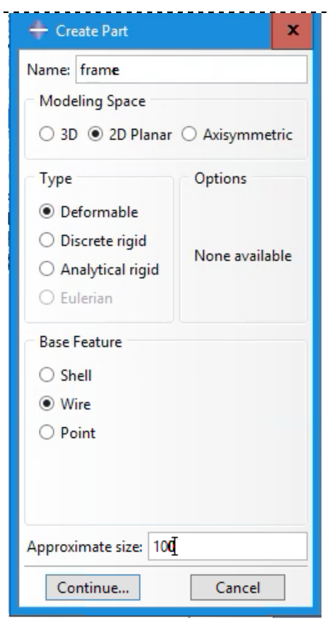
Figure 47 Analyzing a Frame

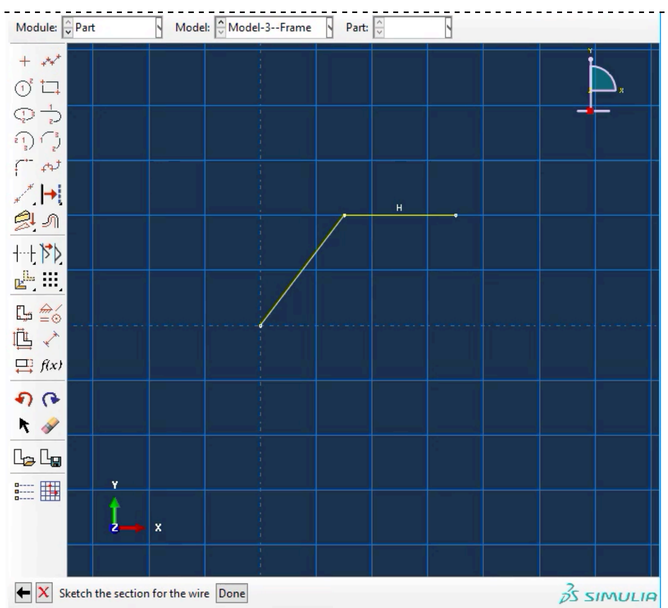
1 - Module: Part

Consideration(s):

- Exactly similar to the truss problem: 2D Planar/Deformable/Wire

Figures:





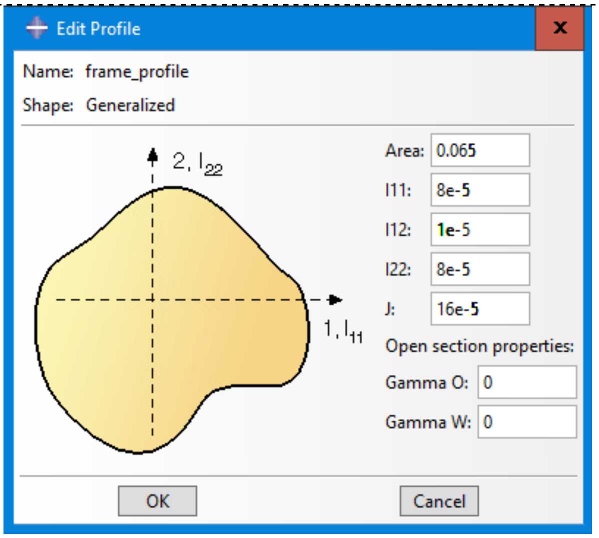
2 - Module: Property

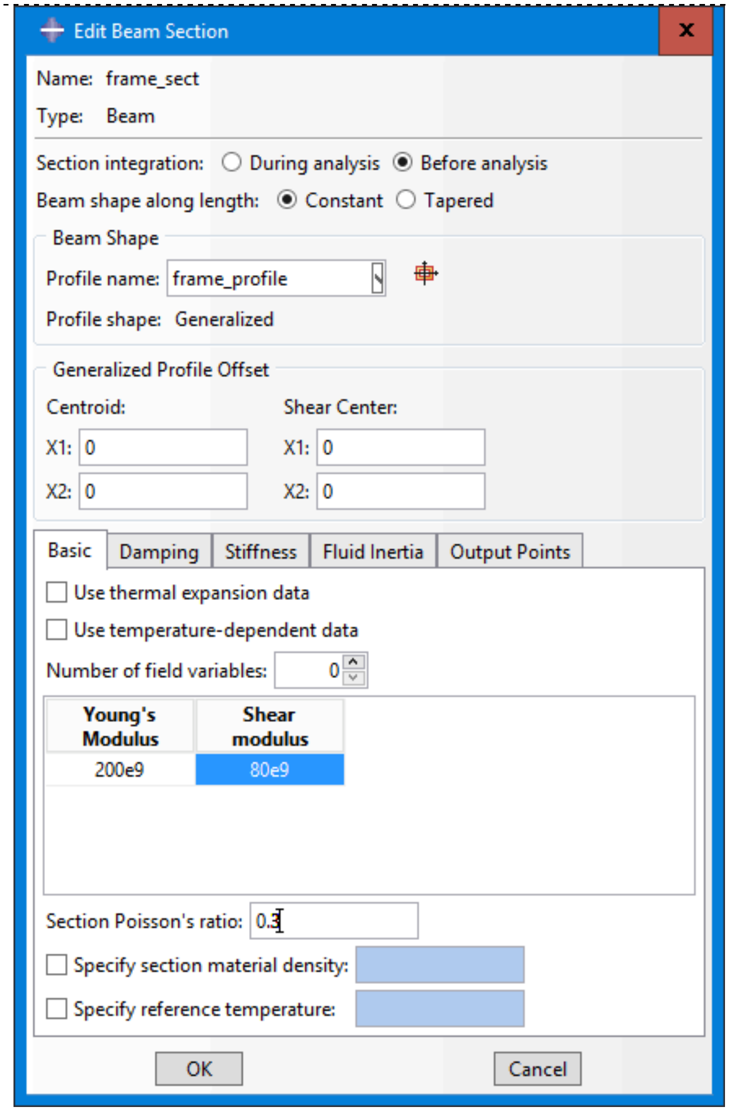
Consideration(s):

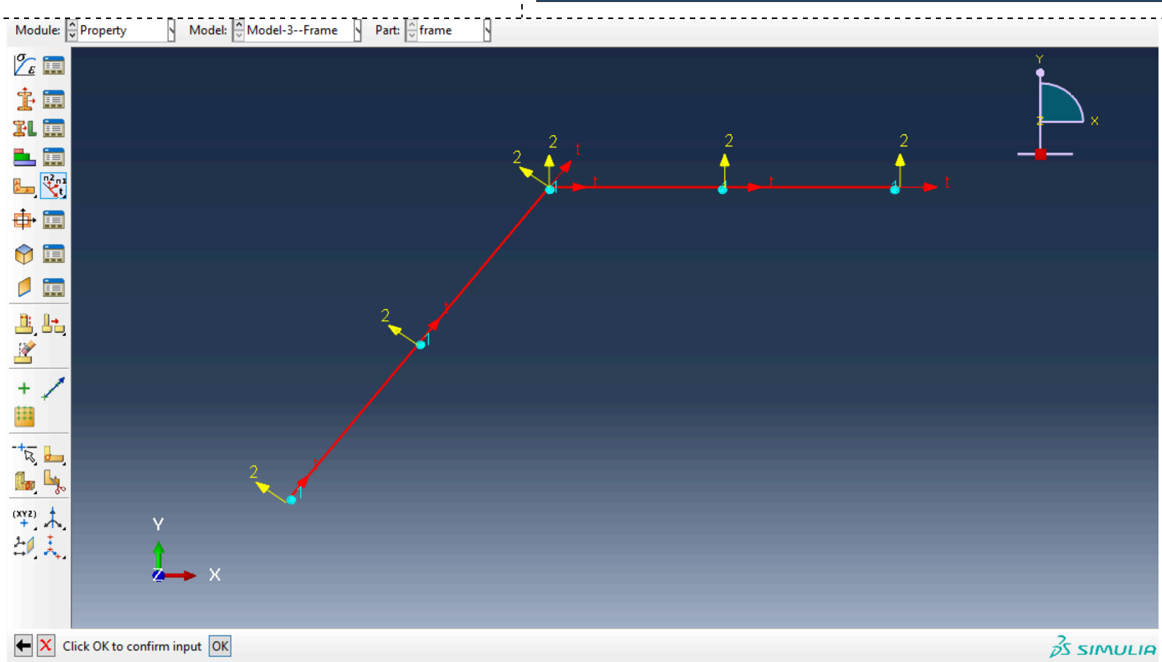
- There is no need to define the material by Create Material tool. Material properties should be defined during defining the **Profile**.
- First, click on the **Create Profile** button  . In the Create Profile dialog box, enter frame_profile as the profile name. Since the problem does not give us any information regarding the section shape, choose **Generalized** in the **Shape** section and click on the **Continue** button.
- In the **Edit Profile** dialog box, enter the section properties. (Note that for this analysis only I_{11} is required. But Abaqus needs the other information. Therefore, enter 8e-4 for both I_{11} and I_{22}. I_{12} is not required but we know $I_{11} \times I_{22} - I_{12}^2$ must be a positive value. J is also equal to $I_{11}+I_{22}$)
- Then, click on the **Create Section** button. Name it frame_sect, and choose **Beam** in the both Category and Type sections. Then Click on the **Continue** button.
- In the **Edit Beam Section** dialog box, toggle on **Before analysis** for Section integration. In the **Beam Shape** section, choose the defined frame_profile in the Profile name list. Then, enter the mechanical properties in the **Basic** tab (Young's Modulus = 200e9, Shear Modulus = 80e9 (steel), and Poisson's ratio = 0.3). Then, click on the **OK** button.
- Next, click on the **Assign Section** button and assign the created section frame_sect to the all part members.
- Moreover, for beam and frame analysis, you should specify the orientation (local axis) of each member. Therefore, click on the **Assign Beam Orientation** button, select all members, and enter 0,0,-1 as an approximate n1 direction (For more information see the following link: http://abaqus.software.polimi.it/v6.14/books/usi/pt03ch12s15hlb03.html)

Figures:

(In the next page)


Edit Profile
Name: frame_profile
Shape: Generalized
2, I22
1, I11
Area: 0.065
I11: 8e-5
I12: 1e-5
I22: 8e-5
J: 16e-5
Open section properties:
Gamma O: 0
Gamma W: 0
OK
Cancel


Edit Beam Section
Name: frame_sect
Type: Beam
Section integration: During analysis Before analysis
Beam shape along length: Constant Tapered
Beam Shape
Profile name: frame_profile
Profile shape: Generalized
Generalized Profile Offset
Centroid:
Shear Center:
X1: 0
X2: 0
X1: 0
X2: 0
Basic
Damping
Stiffness
Fluid Inertia
Output Points
Use thermal expansion data
Use temperature-dependent data
Number of field variables: 0
Young's Modulus
Shear modulus
200e9
80e9
Section Poisson's ratio: 0.3
Specify section material density:
Specify reference temperature:
OK
Cancel

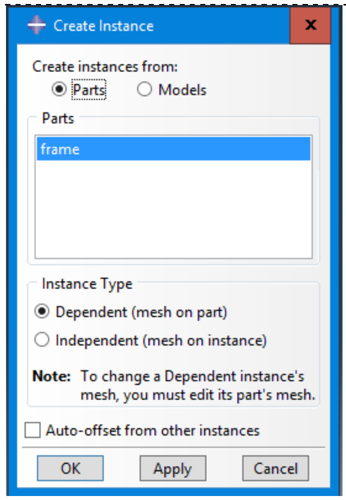

Module: Property
Model: Model-3--Frame
Part: frame
Click OK to confirm input
OK
SIMULIA

3 - Module: **Assembly**

Consideration(s):

- Exactly similar to the truss problem

Figures:

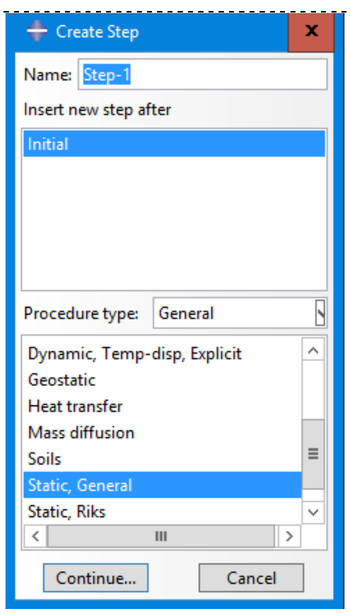


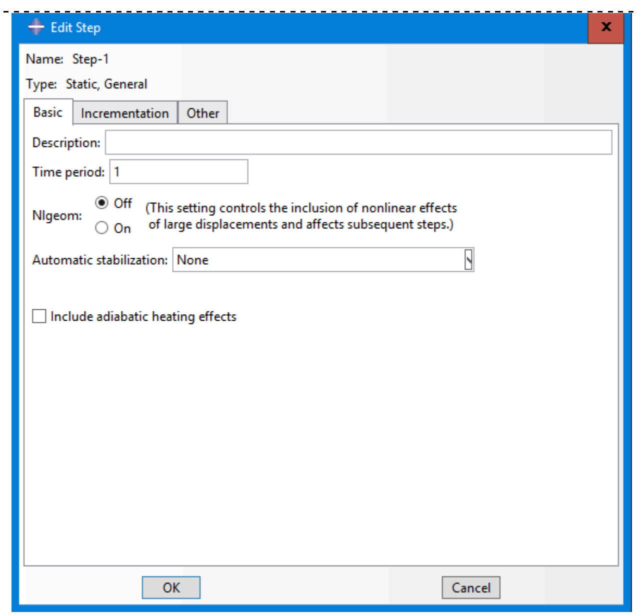
4 - Module: Step

Consideration(s):

- Exactly similar to the truss problem

Figures:



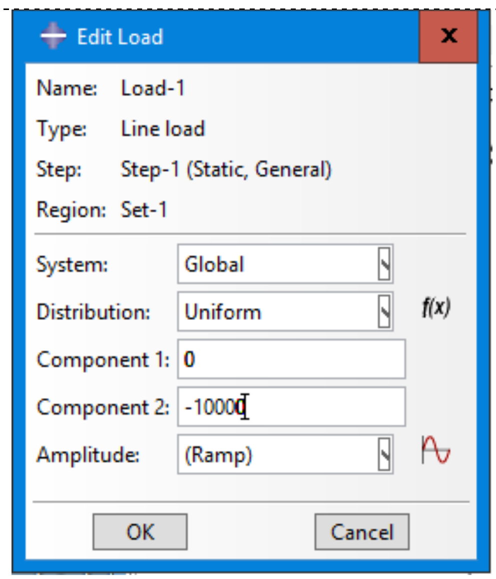


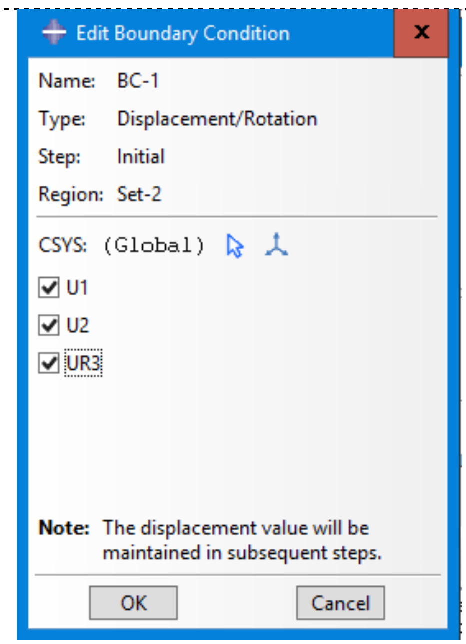
5 - Module: Load

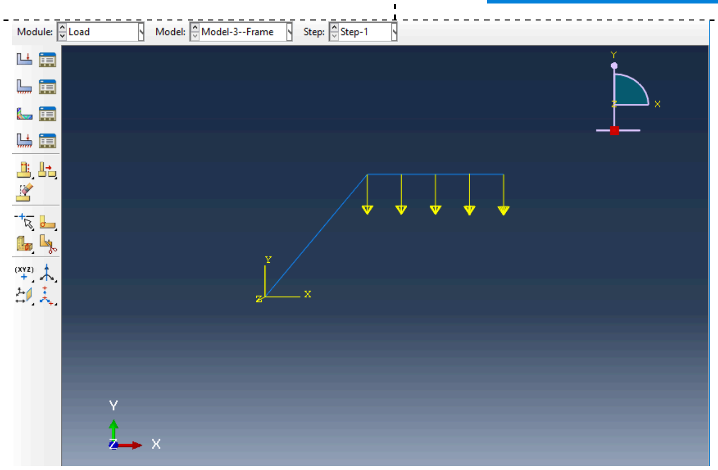
Consideration(s):

- Similar to the truss problem
- In this problem the applied force is distributed. Therefore, in the **Create Load** dialog box, choose **Line load** as the type of mechanical load. Then, select the horizontal member as the **body for the load**, and click on the **Done** button.
- Then, in the **Edit Load dialog box**, enter 0 for **Component 1** (X direction) and -10000 for **Component 2** (Y direction).
- In the **Creating Boundary Conditions**, one of the supports is fixed; therefore, in addition to the both displacements, the rotation should also be checked in the **Edit Boundary Condition** dialog box.

Figures:





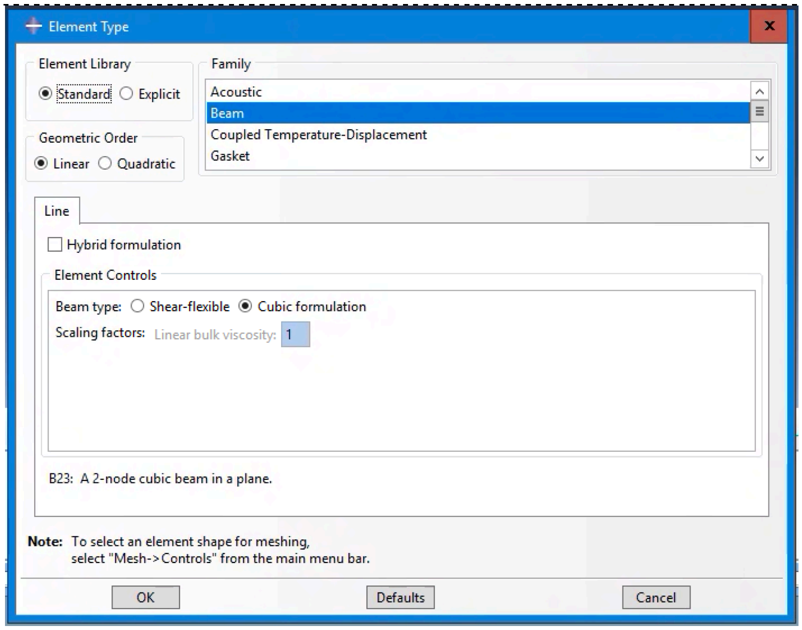


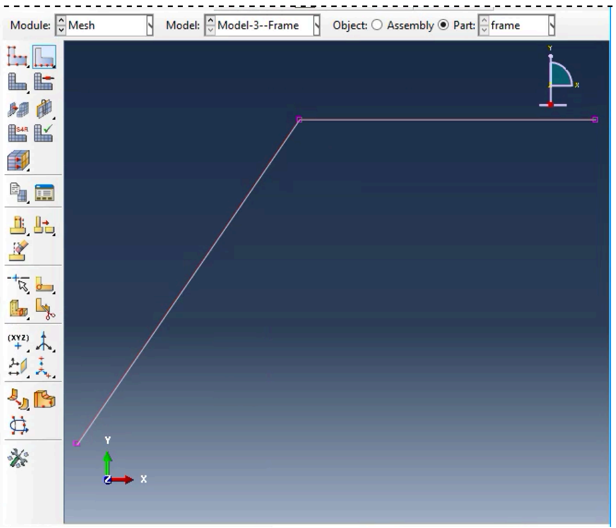
6 - Module: Mesh

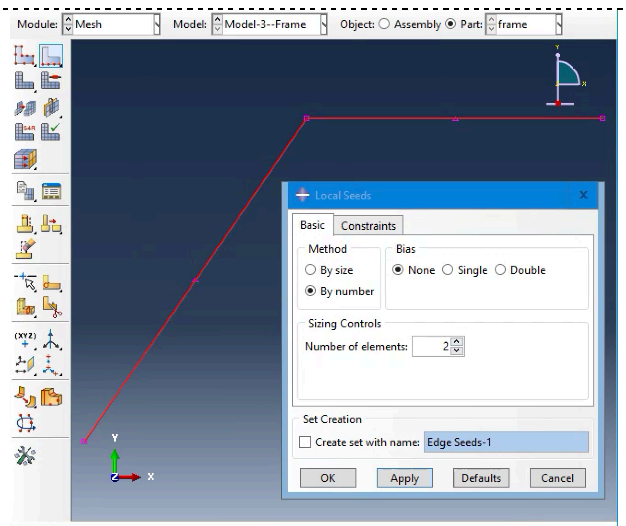
Consideration(s):

- Similar to the truss problem
- The only difference is the **Element Type**. In the Element Type dialog box, choose **Beam** element. This element recognized as **B21** element in Abaqus (2-node linear beam in a plane). Then, in the same dialog box, choose **Cubic-formulation** as Beam type.
- Enter 1 as the **Number of elements** in the **Seed Edges** tool. You can also try to generate more elements to see how the results may be affected.

Figures:





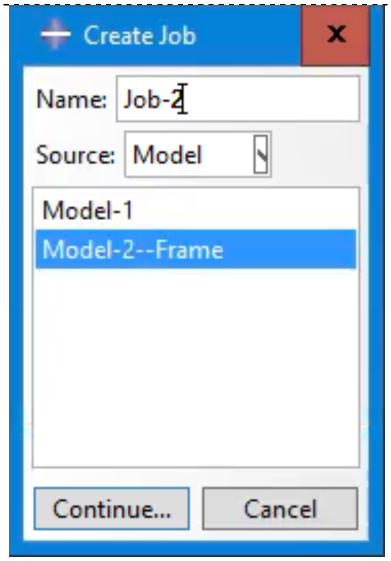


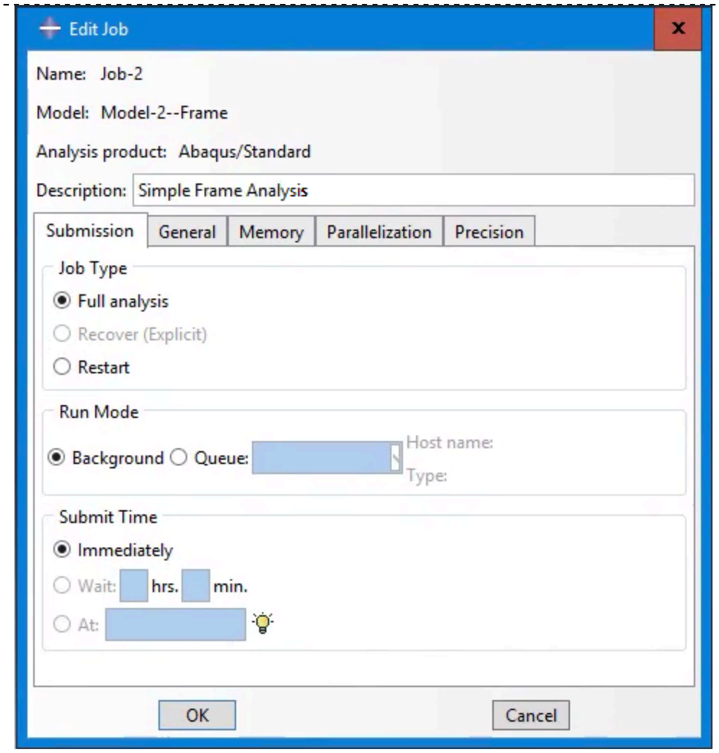
7 - Module: Job

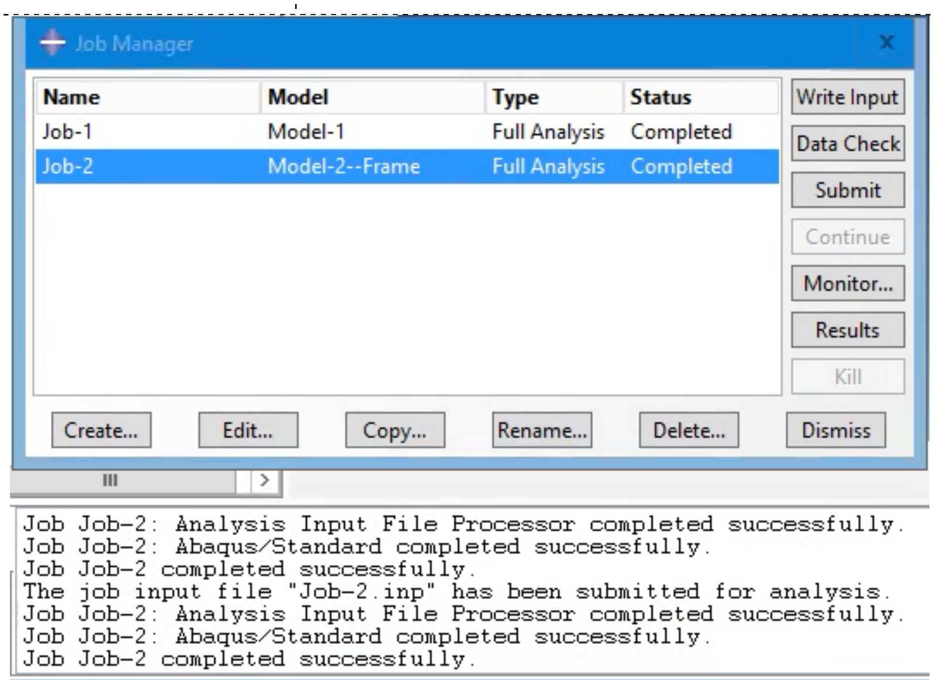
Consideration(s):

- Exactly similar to the truss problem

Figures:





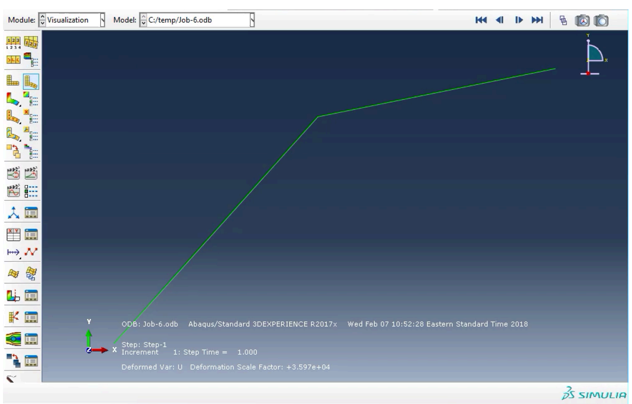


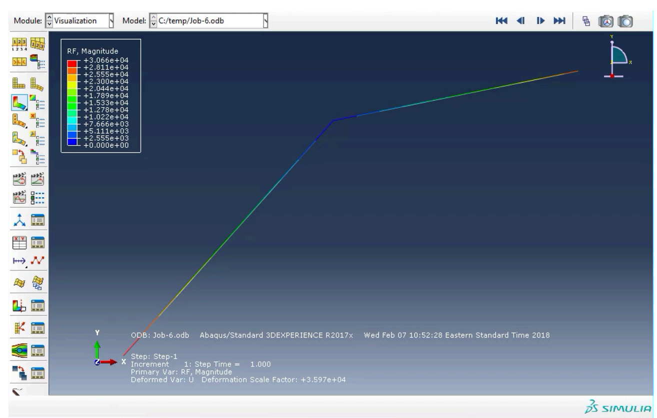
8 - Module: Visualization

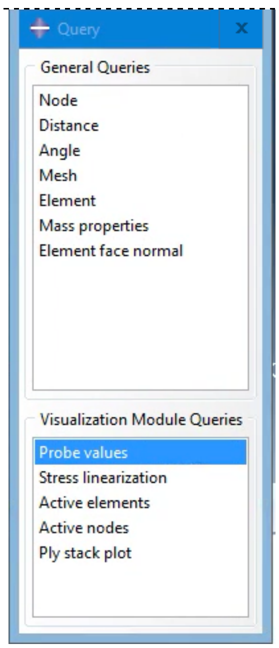
Consideration(s):

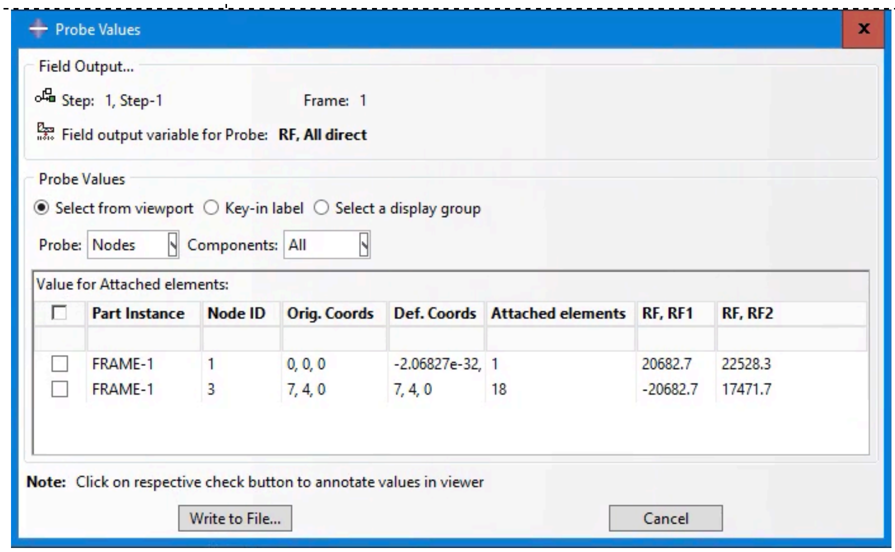
- Exactly similar to the truss problem
- For instance, we can check **Reaction Force (RF)** at the support nodes by using the Query toolset.
- Some results are stored in a text file and presented in the Appendix.

Figures:









References:

[1] Systèmes, D., 2012. Abaqus Documentation. *Providence, RI, United States*.

[2] Liu, G.R. and Quek, S.S., 2013. *The finite element method: a practical course*. Butterworth-Heinemann.

[3] Wikipedia. 2017. *Abaqus*. [ONLINE] Available at: https://en.wikipedia.org/wiki/Abaqus. [Accessed 1 January 2018].

Appendix

Before presenting the results, a list of important output variables and their notations are briefly described in the following. (A complete description can also be found in the following link: http://dsk.ippt.pan.pl/docs/abaqus/v6.13/books/usb/default.htm?startat=pt02ch04s01aus38.html)

No.	Notation	Description
1	**S**	All stress components
2	**Sij**	ij-component of stress
3	**SP**	All principal stresses
4	**MISES**	Mises equivalent stress
5	**TRESC**	Tresca equivalent stress, defined as the maximum difference between principal stresses
6	**E**	All strain components
7	**Eij**	ij-component of strain
8	**EP**	All principal strains
9	**SF**	All section force and moment components
10	**SFn**	Section force component n (n=1..5 for conventional shells; n=1..6 for continuum shells; n=1..3 for beams)
11	**SE**	All section strain, curvature change, and twist components
12	**SEn**	Section strain component n (n=1..6 for shells; n=1..3 for beams)
13	**COORD**	Coordinates of the section point. These are the current coordinates if the large-displacement formulation is being used
14	**COORn**	Coordinate n (n=1..3)
15	**U**	All physical displacement components, including rotations at nodes with rotational degrees of freedom
16	**UT**	All translational displacement components
17	**UR**	All rotational displacement components
18	**Un**	n-th displacement component (n=1..3)
19	**RF**	All components of reaction forces, including components of reaction moments at nodes with rotational degrees of freedom (conjugate to prescribed displacements and rotations)
20	**RT**	All reaction force components
21	**RM**	All reaction moment components
22	**RFn**	Reaction force component n (n=1..3)
23	**RMn**	Reaction moment component n (n=1..3)
24	**CF**	All components of point loads and concentrated moments
25	**CFn**	Point load component n (n=1..3).
26	**LOADS**	Current values of distributed loads

Truss Analysis Outputs

```
********************************************************************************
Probe Values Report, written on Tue Jan 30 18:56:46 2018

Source
-------

   ODB: C:/temp/Job-1.odb
   Step: Step-1
   Frame: Increment      1: Step Time =    1.000
   Variable: S, Mises (Not averaged)

Loc 1 : Integration point values

Probe values reported at integration points

  Part Instance  Element ID        Type  Attached nodes
--------------------------------------------------------------------------------
  TRUSS_ALL_PARTS-1           1        T2D2           2           1
  TRUSS_ALL_PARTS-1           2        T2D2           3           2
  TRUSS_ALL_PARTS-1           3        T2D2           1           3


  Part Instance  Element ID        Type   Int. Pt.   S, Mises
--------------------------------------------------------------------------------
  TRUSS_ALL_PARTS-1           1        T2D2          1    10.E+03
  TRUSS_ALL_PARTS-1           2        T2D2          1    14.E+03
  TRUSS_ALL_PARTS-1           3        T2D2          1    10.E+03


  Minimum                                          10.E+03
  at Element                                             3
    Int. Pt.                                             1
  in Part Instance                                    TRUSS_ALL_PARTS-1

  Maximum                                          14.E+03
  at Element                                             2
    Int. Pt.                                             1
  in Part Instance                                    TRUSS_ALL_PARTS-1

       Total                                       34.E+03
```

```
********************************************************************************
Probe Values Report, written on Tue Jan 30 18:29:52 2018

Source
-------

   ODB: C:/temp/Job-1.odb
   Step: Step-1
   Frame: Increment      1: Step Time =    1.000
   Variable: S, All principals (Not averaged)

Loc 1 : Integration point values

Probe values reported at integration points

  Part Instance  Element ID        Type  Attached nodes
--------------------------------------------------------------------------------
  TRUSS_ALL_PARTS-1           1        T2D2           2           1
  TRUSS_ALL_PARTS-1           2        T2D2           3           2
  TRUSS_ALL_PARTS-1           3        T2D2           1           3


  Part Instance  Element ID        Type   Int. Pt.  S, Max. In-Plane Principal  S, Max. In-Plane Principal (Abs)  S, Min. In-Plane Principal  S, Max. Principal  S, Max. Principal (Abs)  S, Min. Principal
--------------------------------------------------------------------------------
  TRUSS_ALL_PARTS-1           1        T2D2          1        0.0   -10.E+03   -10.E+03        0.0   -10.E+03   -10.E+03
  TRUSS_ALL_PARTS-1           2        T2D2          1    14.E+03    14.E+03        0.0    14.E+03    14.E+03        0.0
  TRUSS_ALL_PARTS-1           3        T2D2          1        0.0   -10.E+03   -10.E+03        0.0   -10.E+03   -10.E+03


  Minimum                                              0.0   -10.E+03   -10.E+03        0.0   -10.E+03   -10.E+03
  at Element                                             3          3          3          3          3          3
    Int. Pt.                                             1          1          1          1          1          1
  in Part Instance                                     TRUSS_ALL_PARTS-1  TRUSS_ALL_PARTS-1  TRUSS_ALL_PARTS-1  TRUSS_ALL_PARTS-1  TRUSS_ALL_PARTS-1  TRUSS_ALL_PARTS-1

  Maximum                                          14.E+03    14.E+03        0.0    14.E+03    14.E+03        0.0
  at Element                                             2          2          2          2          2          2
    Int. Pt.                                             1          1          1          1          1          1
  in Part Instance                                     TRUSS_ALL_PARTS-1  TRUSS_ALL_PARTS-1  TRUSS_ALL_PARTS-1  TRUSS_ALL_PARTS-1  TRUSS_ALL_PARTS-1  TRUSS_ALL_PARTS-1

       Total                                       14.E+03    -6.E+03   -20.E+03    14.E+03    -6.E+03   -20.E+03
```

```
********************************************************************************
Field Output Report, written Tue Jan 30 18:24:01 2018

Source 1
---------

   ODB: C:/temp/Job-1.odb
   Step: Step-1
   Frame: Increment      1: Step Time =    1.000

Loc 1 : Nodal values from source 1

Output sorted by column "Node Label".

Field Output reported at nodes for part: TRUSS_ALL_PARTS-1
   Computation algorithm: EXTRAPOLATE_COMPUTE_AVERAGE
   Averaged at nodes
   Averaging regions: ODB_REGIONS

      Node Label    CF.Magnitude
                          @Loc 1
---------------------------------
               1              0.
               2              0.
               3         1.E+03


  Minimum                     0.
         At Node               2

  Maximum                1.E+03
         At Node               3

           Total         1.E+03


      Node Label          CF.CF1
                          @Loc 1
---------------------------------
               1              0.
               2              0.
               3              0.


  Minimum                     0.
         At Node               3

  Maximum                     0.
         At Node               3

           Total              0.


      Node Label          CF.CF2
                          @Loc 1
---------------------------------
               1              0.
               2              0.
               3        -1.E+03


  Minimum               -1.E+03
         At Node               3

  Maximum                     0.
         At Node               2

           Total        -1.E+03

      Node Label  COORD.Magnitude
                          @Loc 1
----------------------------------
               1              0.
               2              1.
               3              1.


  Minimum                     0.
         At Node               1

  Maximum                     1.
         At Node               3

           Total              2.

      Node Label     COORD.COOR1
                          @Loc 1
---------------------------------
               1              0.
               2              0.
               3              1.


  Minimum                     0.
         At Node               2

  Maximum                     1.
         At Node               3

           Total              1.


      Node Label     COORD.COOR2
                          @Loc 1
---------------------------------
               1              0.
               2              1.
               3              0.


  Minimum                     0.
         At Node               3

  Maximum                     1.
         At Node               2

           Total              1.


      Node Label    RF.Magnitude
                          @Loc 1
---------------------------------
               1     1.41421E+03
               2          1.E+03
               3              0.


  Minimum                     0.
         At Node               3

  Maximum            1.41421E+03
         At Node               1

           Total     2.41421E+03
```

```
     Node Label          RF.RF1
                         @Loc 1
---------------------------------
              1          1.E+03
              2         -1.E+03
              3              0.


 Minimum                -1.E+03
        At Node               2

 Maximum                 1.E+03
        At Node               1

          Total              0.

     Node Label          RF.RF2
                         @Loc 1
---------------------------------
              1          1.E+03
              2              0.
              3              0.


 Minimum                     0.
        At Node               3

 Maximum                 1.E+03
        At Node               1

          Total          1.E+03

     Node Label    RT.Magnitude
                         @Loc 1
---------------------------------
              1     1.41421E+03
              2          1.E+03
              3              0.


 Minimum                     0.
        At Node               3

 Maximum            1.41421E+03
        At Node               1

          Total     2.41421E+03

     Node Label          RT.RT1
                         @Loc 1
---------------------------------
              1          1.E+03
              2         -1.E+03
              3              0.


 Minimum                -1.E+03
        At Node               2

 Maximum                 1.E+03
        At Node               1

          Total              0.
```

```
     Node Label          RT.RT2
                         @Loc 1
---------------------------------
              1          1.E+03
              2              0.
              3              0.


 Minimum                     0.
        At Node               3

 Maximum                 1.E+03
        At Node               1

          Total          1.E+03

     Node Label    TF.Magnitude
                         @Loc 1
---------------------------------
              1     1.41421E+03
              2          1.E+03
              3          1.E+03


 Minimum                 1.E+03
        At Node               3

 Maximum            1.41421E+03
        At Node               1

          Total     3.41421E+03

     Node Label          TF.TF1
                         @Loc 1
---------------------------------
              1          1.E+03
              2         -1.E+03
              3              0.


 Minimum                -1.E+03
        At Node               2

 Maximum                 1.E+03
        At Node               1

          Total              0.

     Node Label          TF.TF2
                         @Loc 1
---------------------------------
              1          1.E+03
              2              0.
              3         -1.E+03


 Minimum                -1.E+03
        At Node               3

 Maximum                 1.E+03
        At Node               1

          Total              0.
```

Node Label	U.Magnitude @Loc 1
1	0.
2	142.857E-09
3	704.413E-09
Minimum	0.
At Node	1
Maximum	704.413E-09
At Node	3
Total	847.270E-09

Node Label	U.U1 @Loc 1
1	-1.00000E-33
2	1.00000E-33
3	-142.857E-09
Minimum	-142.857E-09
At Node	3
Maximum	1.00000E-33
At Node	2
Total	-142.857E-09

Node Label	U.U2 @Loc 1
1	-1.00000E-33
2	-142.857E-09
3	-689.775E-09
Minimum	-689.775E-09
At Node	3
Maximum	-1.00000E-33
At Node	1
Total	-832.632E-09

Node Label	E.Max. In-P @Loc 1
1	0.
2	101.015E-09
3	101.015E-09
Minimum	0.
At Node	1
Maximum	101.015E-09
At Node	3
Total	202.031E-09

Node Label	E.Max. In-P(a) @Loc 1
1	-142.857E-09
2	29.5867E-09
3	29.5867E-09
Minimum	-142.857E-09
At Node	1
Maximum	29.5867E-09
At Node	3
Total	-83.6838E-09

Node Label	E.Min. In-P @Loc 1
1	-142.857E-09
2	-71.4286E-09
3	-71.4286E-09
Minimum	-142.857E-09
At Node	1
Maximum	-71.4286E-09
At Node	3
Total	-285.714E-09

Node Label	E.Max. Prin @Loc 1
1	0.
2	101.015E-09
3	101.015E-09
Minimum	0.
At Node	1
Maximum	101.015E-09
At Node	3
Total	202.031E-09

Node Label	E.Max. Prin(a) @Loc 1
1	-142.857E-09
2	29.5867E-09
3	29.5867E-09
Minimum	-142.857E-09
At Node	1
Maximum	29.5867E-09
At Node	3
Total	-83.6838E-09

Node Label	E.Min. Prin @Loc 1
1	-142.857E-09
2	-71.4286E-09
3	-71.4286E-09
Minimum	-142.857E-09
At Node	1
Maximum	-71.4286E-09
At Node	3
Total	-285.714E-09

Node Label	E.E11 @Loc 1
1	-142.857E-09
2	29.5867E-09
3	29.5867E-09
Minimum	-142.857E-09
At Node	1
Maximum	29.5867E-09
At Node	3
Total	-83.6838E-09

Node Label	S.Mises @Loc 1
1	10.E+03
2	12.0711E+03
3	12.0711E+03
Minimum	10.E+03
At Node	1
Maximum	12.0711E+03
At Node	3
Total	34.1421E+03

Node Label	S.Max. In-P @Loc 1
1	0.
2	7.07107E+03
3	7.07107E+03
Minimum	0.
At Node	1
Maximum	7.07107E+03
At Node	3
Total	14.1421E+03

Node Label	S.Max. In-P(a) @Loc 1
1	-10.E+03
2	2.07107E+03
3	2.07107E+03
Minimum	-10.E+03
At Node	1
Maximum	2.07107E+03
At Node	3
Total	-5.85786E+03

Node Label	S.Min. In-P @Loc 1
1	-10.E+03
2	-5.E+03
3	-5.E+03
Minimum	-10.E+03
At Node	1
Maximum	-5.E+03
At Node	3
Total	-20.E+03

Node Label	S.Max. Prin @Loc 1
1	0.
2	7.07107E+03
3	7.07107E+03
Minimum	0.
At Node	1
Maximum	7.07107E+03
At Node	3
Total	14.1421E+03

Node Label	S.Max. Prin(a) @Loc 1
1	-10.E+03
2	2.07107E+03
3	2.07107E+03
Minimum	-10.E+03
At Node	1
Maximum	2.07107E+03
At Node	3
Total	-5.85786E+03

Node Label	S.Min. Prin @Loc 1
1	-10.E+03
2	-5.E+03
3	-5.E+03
Minimum	-10.E+03
At Node	1
Maximum	-5.E+03
At Node	3
Total	-20.E+03

Node Label	S.Tresca @Loc 1
1	10.E+03
2	12.0711E+03
3	12.0711E+03
Minimum	10.E+03
At Node	1
Maximum	12.0711E+03
At Node	3
Total	34.1421E+03

Node Label	S.Pressure @Loc 1
1	3.33333E+03
2	-690.356
3	-690.356
Minimum	-690.356
At Node	3
Maximum	3.33333E+03
At Node	1
Total	1.95262E+03

Node Label	S.Third Inv @Loc 1
1	-10.E+03
2	2.07107E+03
3	2.07107E+03
Minimum	-10.E+03
At Node	1
Maximum	2.07107E+03
At Node	3
Total	-5.85786E+03

Node Label	S.S11 @Loc 1
1	-10.E+03
2	2.07107E+03
3	2.07107E+03
Minimum	-10.E+03
At Node	1
Maximum	2.07107E+03
At Node	3
Total	-5.85786E+03

Frame Analysis Outputs

****
Field Output Report, written Wed Feb 07 13:25:35 2018

Source 1

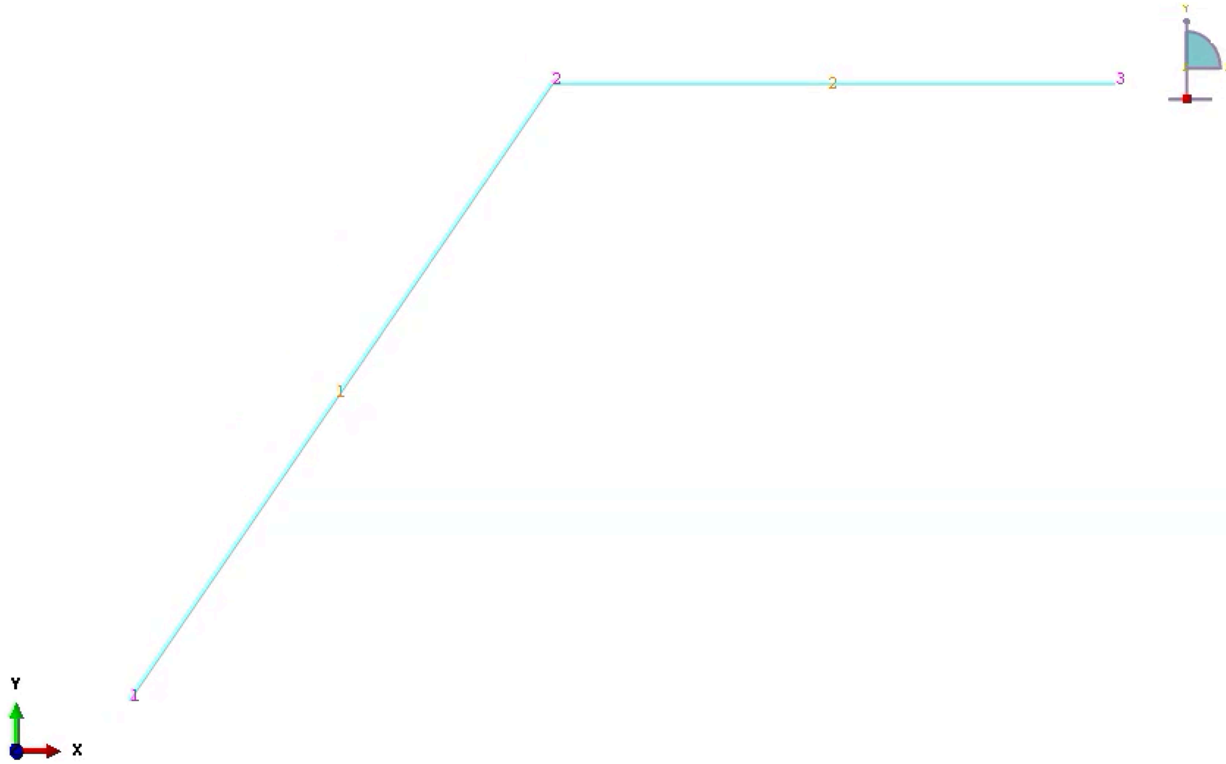
ODB: C:/temp/Job-6.odb
Step: Step-1
Frame: Increment 1: Step Time = 1.000

Loc 1 : Nodal values from source 1

Output sorted by column "Node Label".

Field Output reported at nodes for part: FRAME-1
Computation algorithm: EXTRAPOLATE_COMPUTE_AVERAGE
Averaged at nodes
Averaging regions: ODB_REGIONS

Node Label	COORD.COOR1 @Loc 1	COORD.COOR2 @Loc 1	RF.RF1 @Loc 1	RF.RF2 @Loc 1	RM3 @Loc 1	U.U1 @Loc 1	U.U2 @Loc 1	UR3 @Loc 1
1	0.	0.	20.7639E+03	22.5653E+03	-5.09840E+03	-20.7639E-33	-22.5653E-33	5.09840E-33
2	3.	4.	0.	0.	0.	6.38889E-06	-19.4602E-06	-806.697E-06
3	7.	4.	-20.7639E+03	17.4347E+03	0.	20.7639E-33	2.56531E-33	1.24398E-03
Minimum	0.	0.	-20.7639E+03	0.	-5.09840E+03	-20.7639E-33	-19.4602E-06	-806.697E-06
At Node	1	1	3	2	1	1	2	2
Maximum	7.	4.	20.7639E+03	22.5653E+03	0.	6.38889E-06	2.56531E-33	1.24398E-03
At Node	3	3	1	1	3	2	3	3



1
2
3
1
2
Y
X

****
Field Output Report, written Wed Feb 07 13:25:51 2018

Source 1

ODB: C:/temp/Job-7.odb
Step: Step-1
Frame: Increment   1: Step Time =   1.000

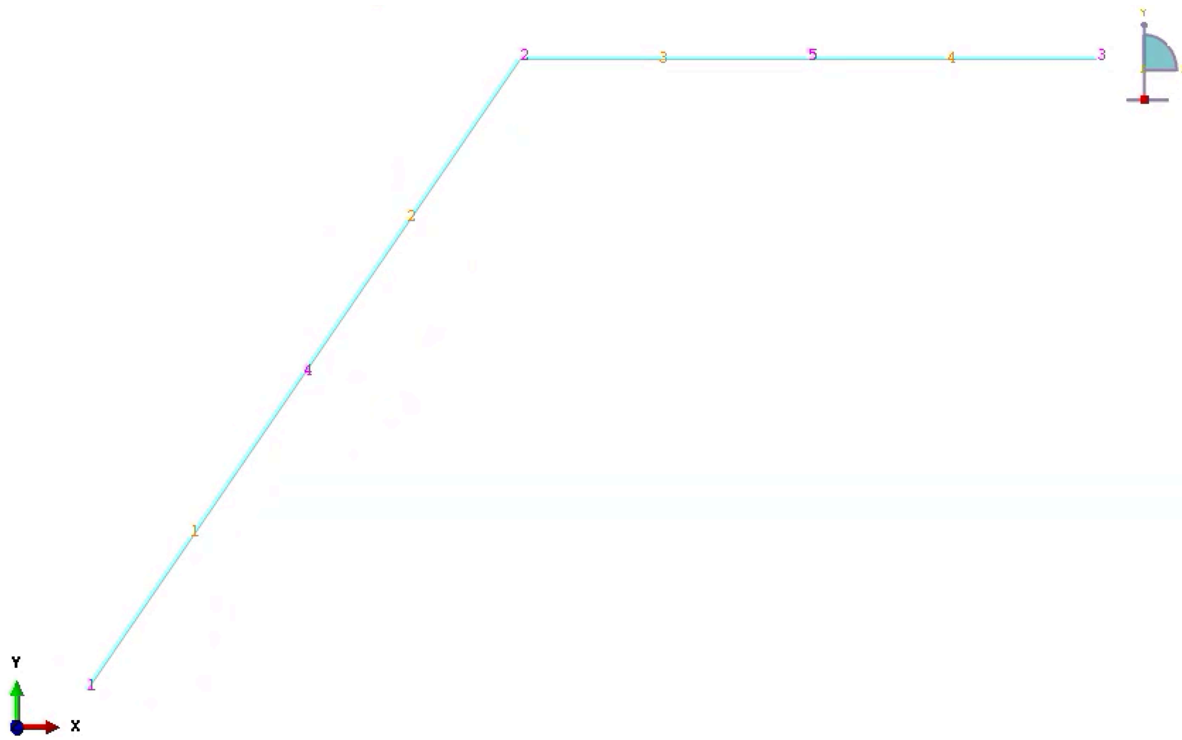
Loc 1 : Nodal values from source 1

Output sorted by column "Node Label".

Field Output reported at nodes for part: FRAME-1
Computation algorithm: EXTRAPOLATE_COMPUTE_AVERAGE
Averaged at nodes
Averaging regions: ODB_REGIONS

Node Label	COORD.COOR1 @Loc 1	COORD.COOR2 @Loc 1	RF.RF1 @Loc 1	RF.RF2 @Loc 1	RM3 @Loc 1	U.U1 @Loc 1	U.U2 @Loc 1	UR3 @Loc 1
1	0.	0.	20.7639E+03	22.5653E+03	-5.09840E+03	-20.7639E-33	-22.5653E-33	5.09840E-33
2	3.	4.	0.	0.	0.	6.38889E-06	-19.4602E-06	-806.697E-06
3	7.	4.	-20.7639E+03	17.4347E+03	0.	20.7639E-33	-7.43469E-33	1.24398E-03
4	1.5	2.	0.	0.	0.	-400.154E-06	292.781E-06	196.638E-06
5	5.	4.	0.	0.	0.	3.19445E-06	-1.45173E-03	-102.023E-06
Minimum	0.	0.	-20.7639E+03	0.	-5.09840E+03	-400.154E-06	-1.45173E-03	-806.697E-06
At Node	1	1	3	5	1	4	5	2
Maximum	7.	4.	20.7639E+03	22.5653E+03	0.	6.38889E-06	292.781E-06	1.24398E-03
At Node	3	5	1	1	5	2	4	3

****
s



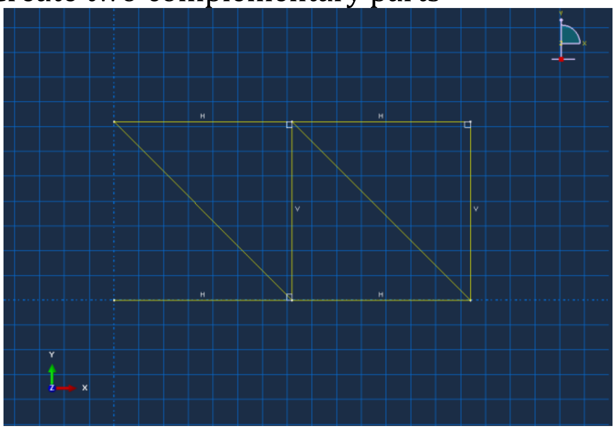
1
1
4
2
2
3
5
4
3
Y
X

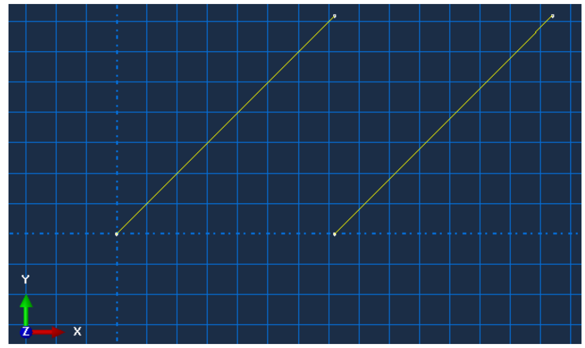
Hints on Homework #5 – Abaqus Problem

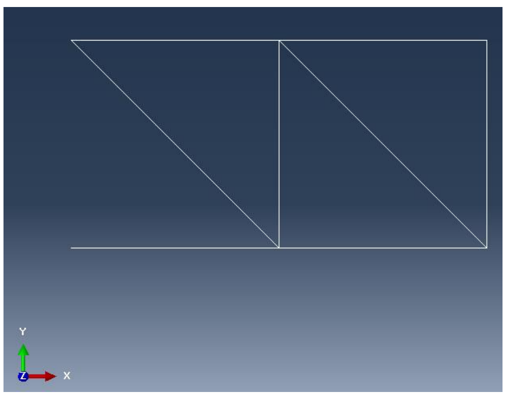
Dealing with Unintended Intersections

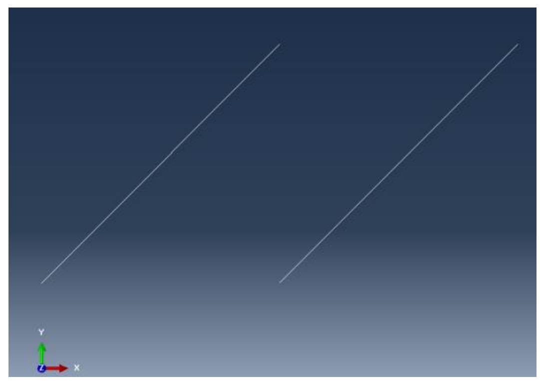
In general, when two lines intersect in the sketcher, a node will be generated at the intersection. There are various ways to handle this problem. One way is the following:

1- Create two complementary parts

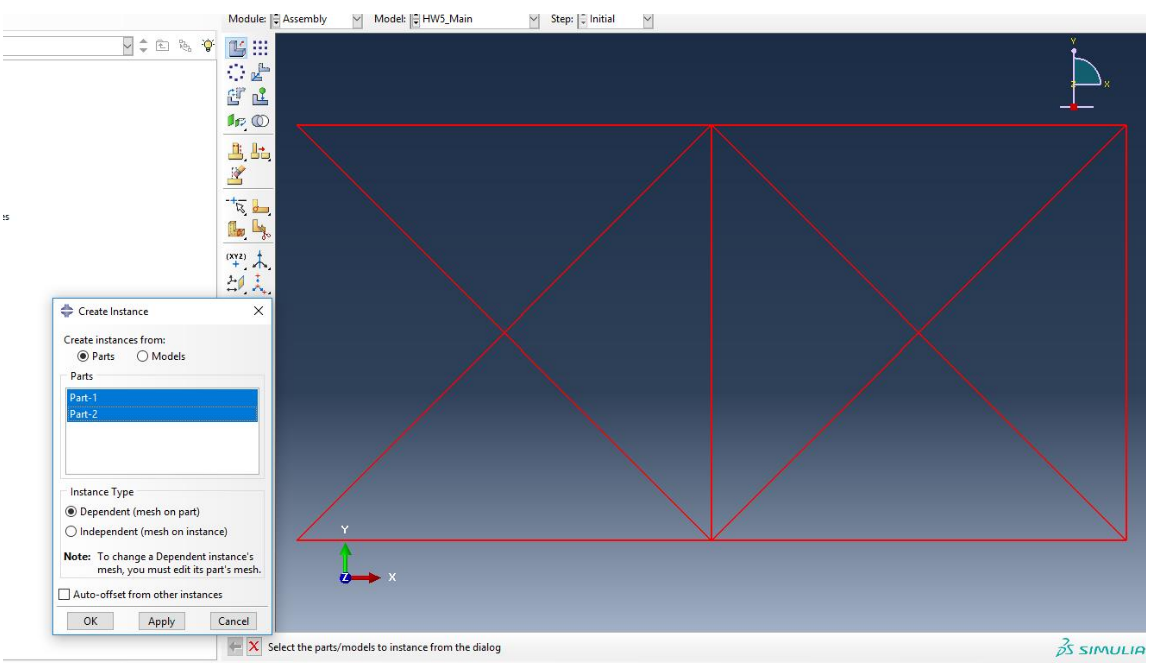




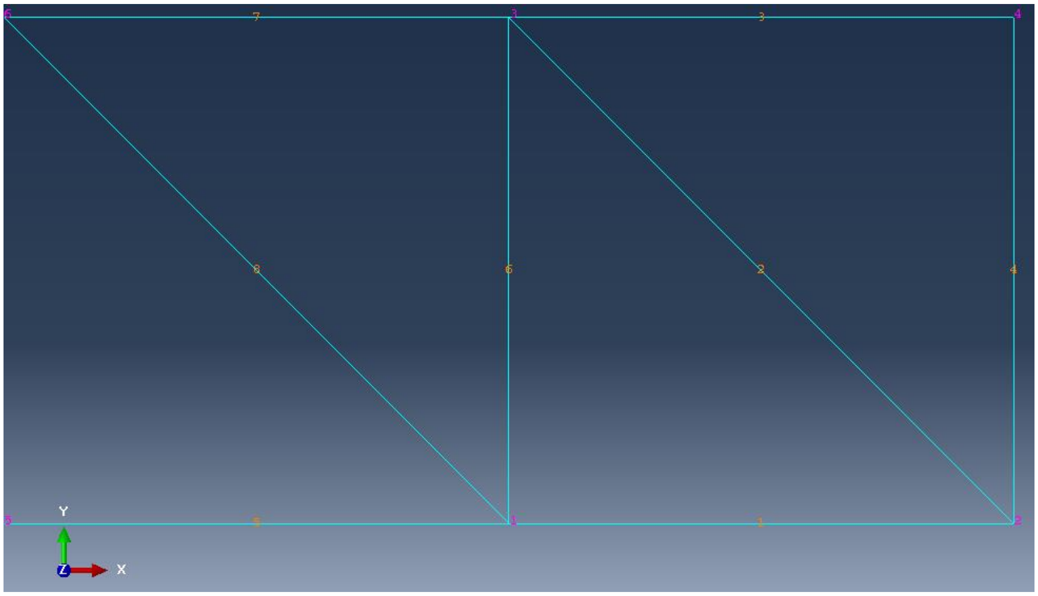


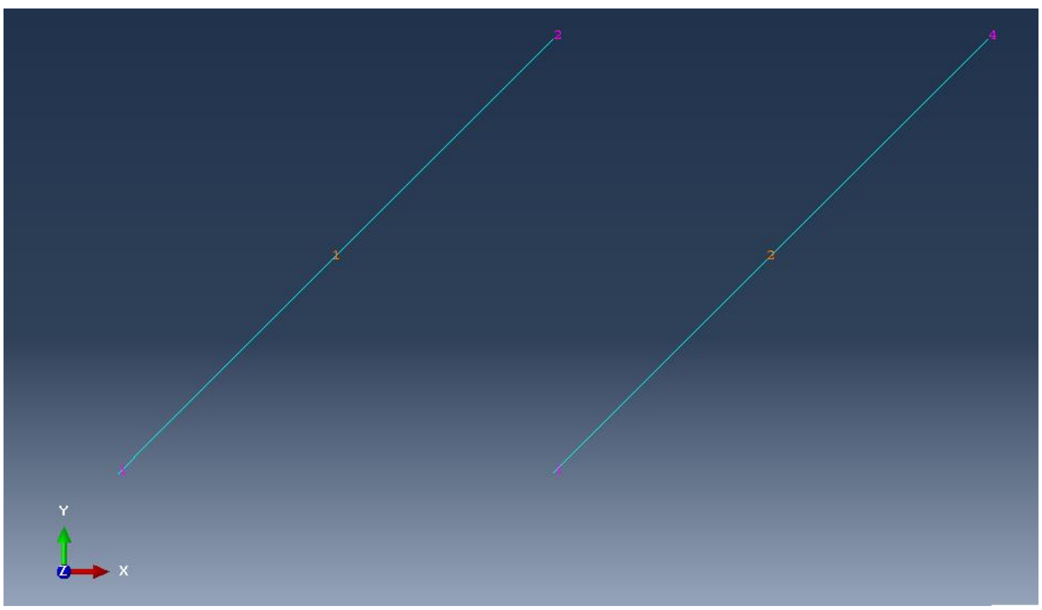


2- After defining material properties and assigning sections to each member, assemble them in the **Assembly** module. Choose **Dependent** Instance Type to generate mesh on parts.

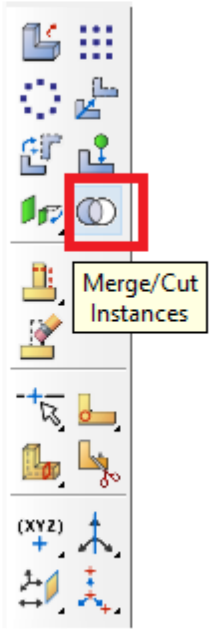


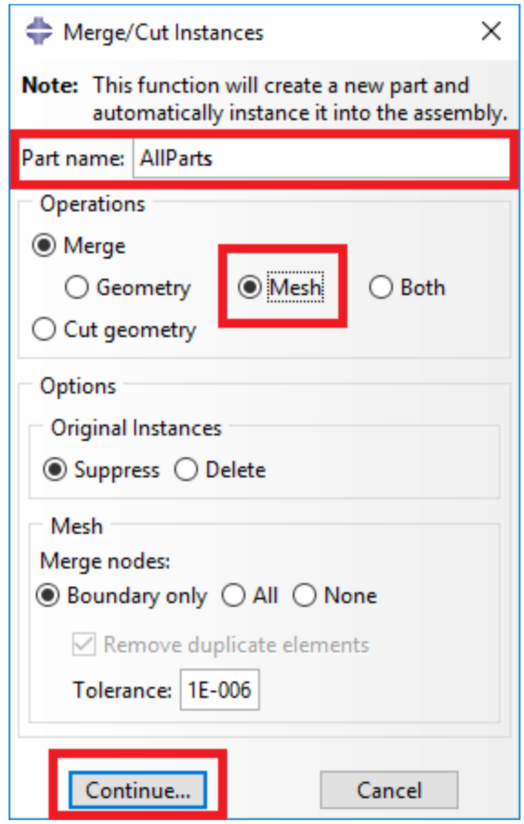
3- <u>Before</u> defining loads and boundary conditions, go to the Mesh module and mesh each part, separately.



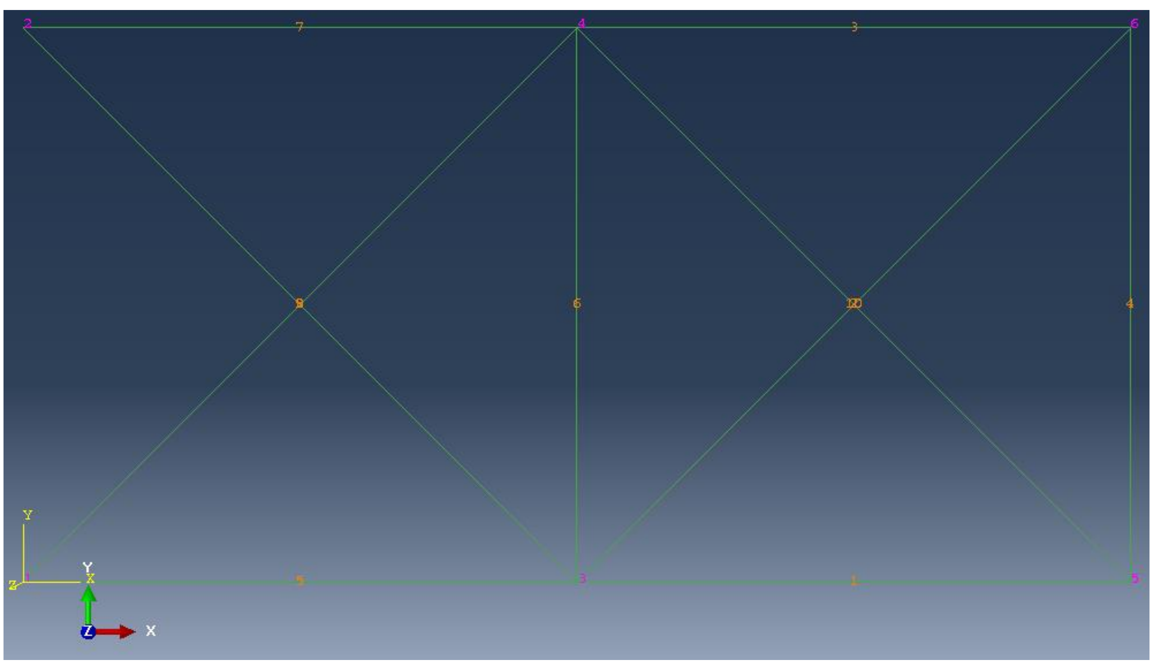


4- Comeback to the Assembly module and click on the **Merge/Cut Instance** button to create a new part with merging two meshed parts.





Picking both instances and click on **Done**.



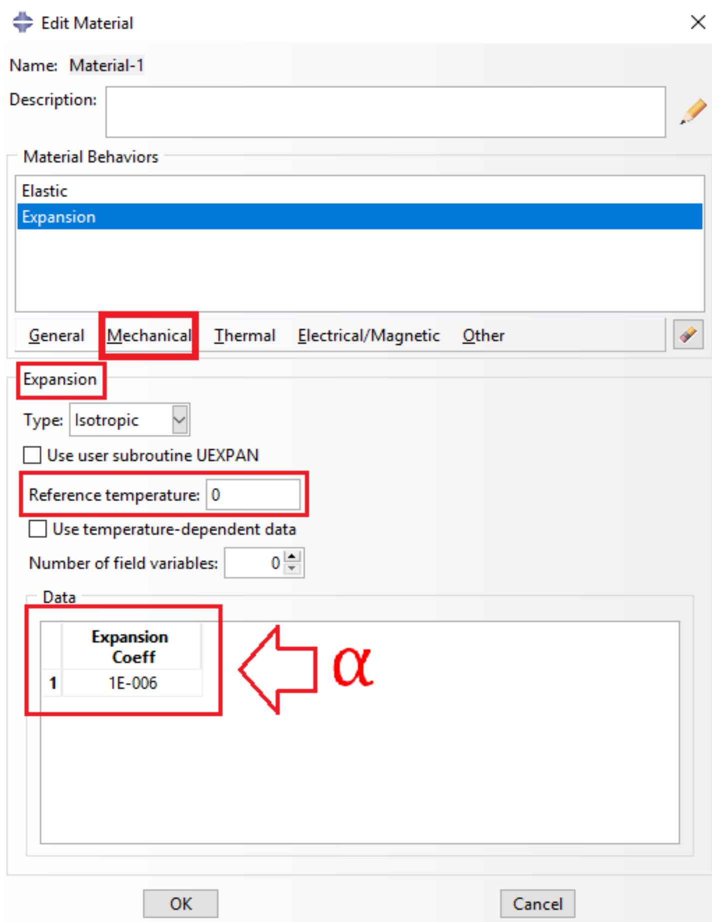
Now, you can continue the modeling procedure with this new assembled meshed part.

5- Create a new step in the **Step** module (As it is explained in the tutorial).
6- Define applied loads and boundary conditions in the **Load** module (As it is explained in the tutorial).
7- Create a new job in the **Job** module and run the analysis by clicking on the Submit button in Job Manager dialog box (As it is explained in the tutorial).

Applying Temperature Change

There are various ways to model temperature change in Abaqus. Regardless what approach will be used, at first, you should add the coefficient of thermal expansion (α) in Material Behaviors:

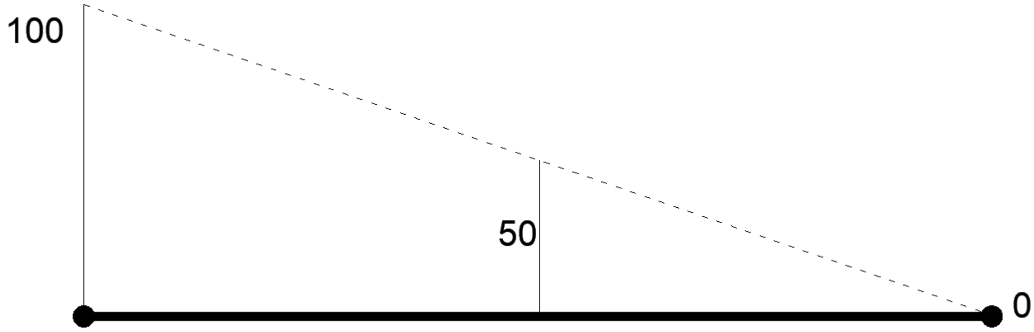
Module: **Property**
Button: **Create Material** or **Material Manager**
Edit Material dialog box
Tab: **Mechanical / Expansion**



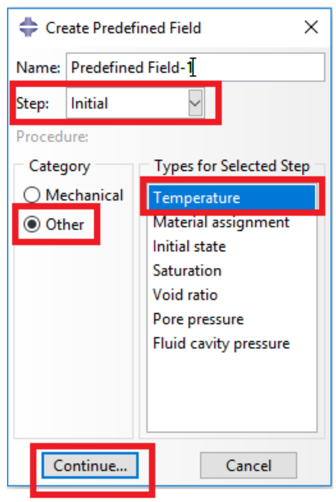
Then, you can follow one of the following ways in the **Load** module:

(1) Defining the thermal increasing as **Predefined Temperature** on nodes.

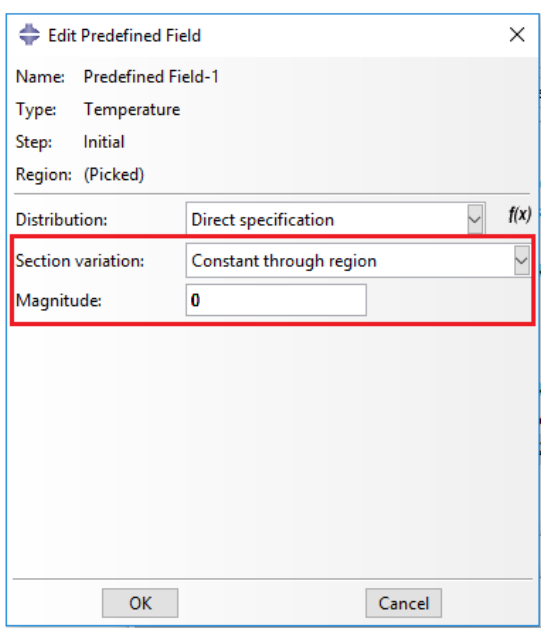
In this model, linear truss elements are employed. To increase of temperature of 50°F in elements 1, 3, 7, and 8, it is enough to apply 100°F in their starting nodes and leave the other end 0°F. Due to the linear interpolation (see the following figure), the value of temperature in the element integration points will be 50°F, as required.



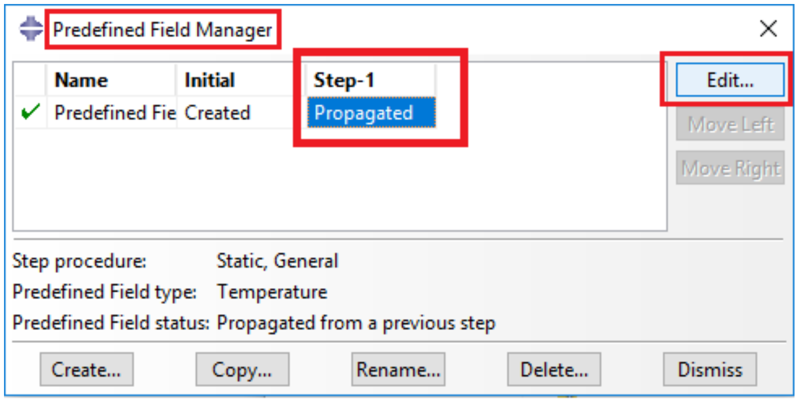
Therefore, click on the **Create Predefined Field** button. In the appeared dialog box, choose the **initial** step, **Other** category, and **Temperature**:

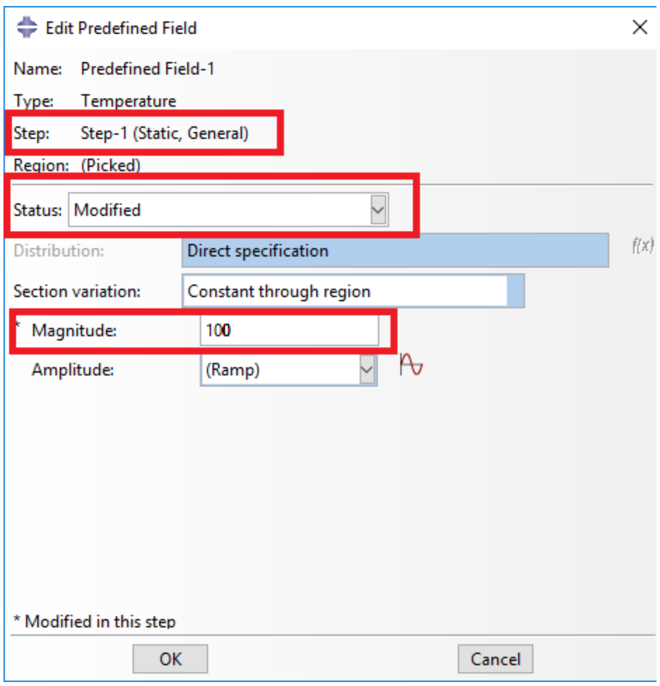


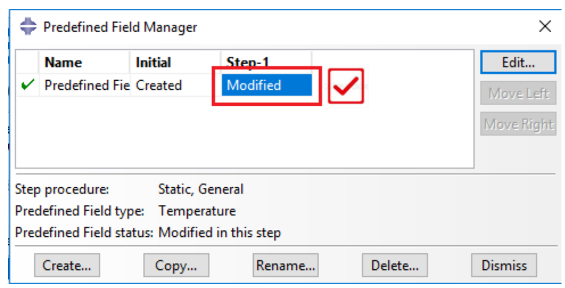
Pick node 5 and 6 to apply predefined temperatures. In the initial step, the magnitude is zero:



Then, modify the Magnitude for the Step-1 (Magnitude = 100):





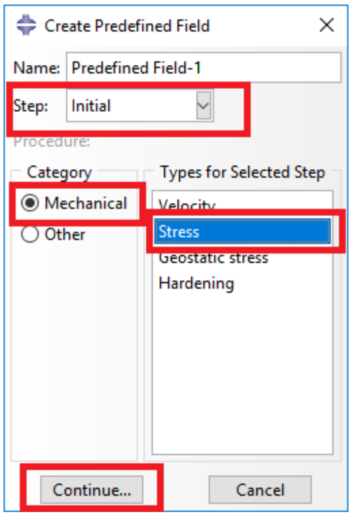


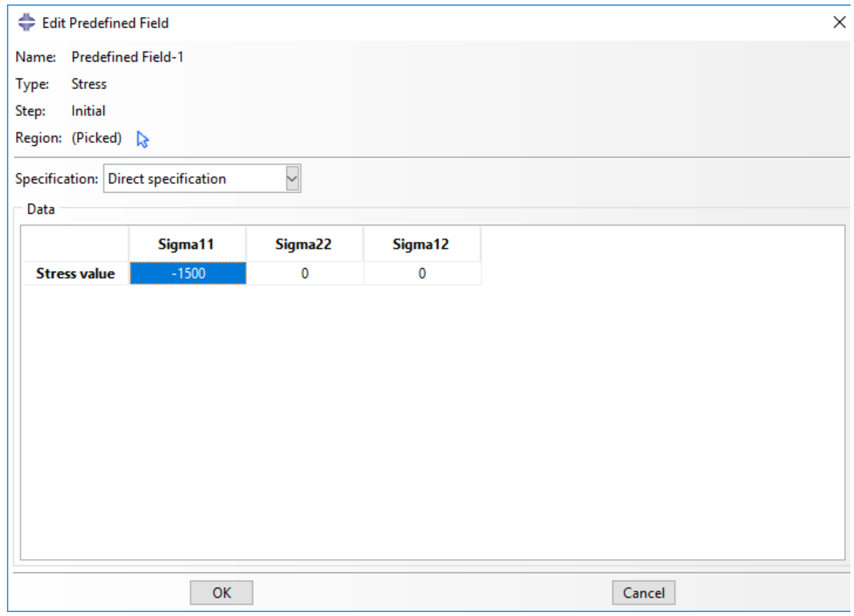
(2) Defining the thermal increasing as **Predefined Stress** on elements

Another way to apply thermal change in elements is calculating the equivalent thermal stresses and applying that as a predefined stress:

$$\sigma = E\varepsilon_0 = E \times \alpha \Delta T$$

The result should be applied as σ_{11} in the predefined stress dialog box.





(3) Defining the thermal increasing as **Equivalent Nodal Forces** on nodes

The third way to model thermal change in elements is calculating the equivalent nodal forces and apply them as concentrated force. The equivalent forces can be obtained by:

$$F = EA\varepsilon_0 \begin{bmatrix} -m \\ -l \\ m \\ l \end{bmatrix}$$

where $\varepsilon_0 = \alpha \Delta T$, and m and l are $\cos(\theta)$ and $\sin(\theta)$, respectively. The equivalent forces should be calculated for elements 1, 3, 7, and 8. Then the corresponding concentrated loads should be applied on nodes 3, 4, 5, and 6.