

Abaqus Tutorial (V2.0) FE Analysis of Trusses and Frames

# • Contact:

<u>shahi@gatech.edu</u> <u>www.sshahi.com</u> 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: <u>http://abaqus.software.polimi.it/v6.14/books/usi/pt01ch02s02.html</u>)

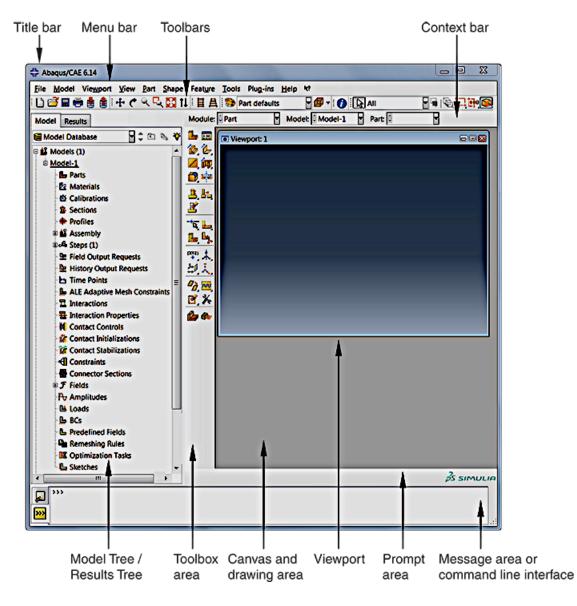


Figure 1 Main Components

#### 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.

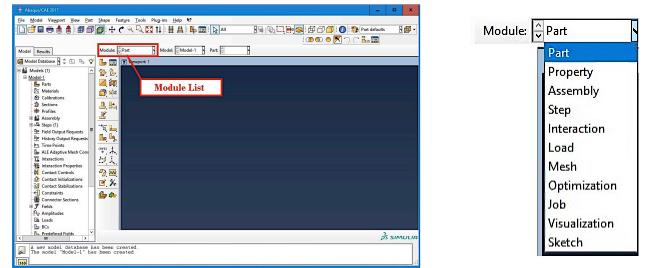


Figure 2 Context bar and the Module list

#### 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: <u>http://abaqus.software.polimi.it/v6.14/books/cmd/pt02.html</u>)

## **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.

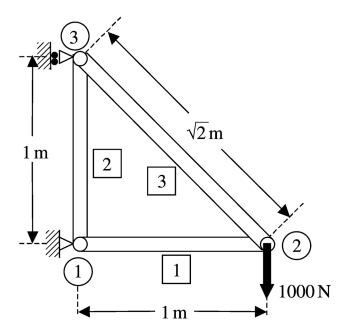


Figure 3 A three member truss structure [2]

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

Table 1 Dimensions and properties of truss members

#### **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).

💠 Save Mo	odel Database As
Directory:	🖹 AbaqusModels 🛛 🔁 🖆 🥕 👘 💼 🗂
<b>—</b>	
i	
<u>F</u> ile Name:	TrussModel QK
File F <u>i</u> lter:	Model Database (*.cae*)

Figure 4 Saving the model database file

- 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

Create Part Part Manager Manager Manager Manager Manager Manager Manager

Figure 5 Module Part toolbox

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: <u>http://abaqus.software.polimi.it/v6.14/books/usi/pt03ch11s19.html</u>). Click on the **Continue** to go to the Sketcher view.

🔶 Create Part	x	
Name: truss_all_parts Modeling Space O 3D • 2D Planar O Axisymmetric		
Туре	Options	
<ul> <li>Deformable</li> <li>Discrete rigid</li> <li>Analytical rigid</li> <li>Eulerian</li> </ul>	None available	
Base Feature O Shell Wire O Point		
Approximate size: 10		
Continue	Cancel	

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.

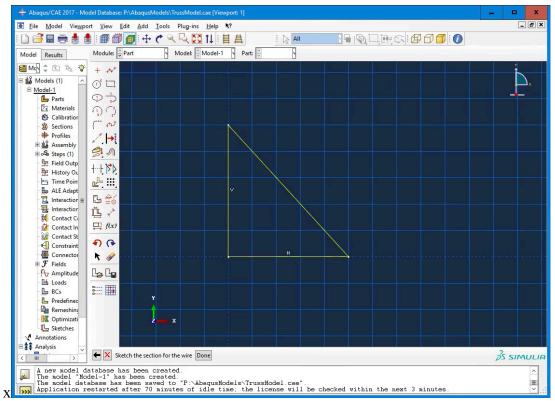
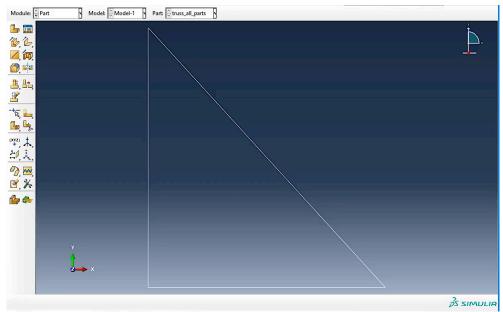


Figure 7 Draw the geometry in Sketcher



- 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.

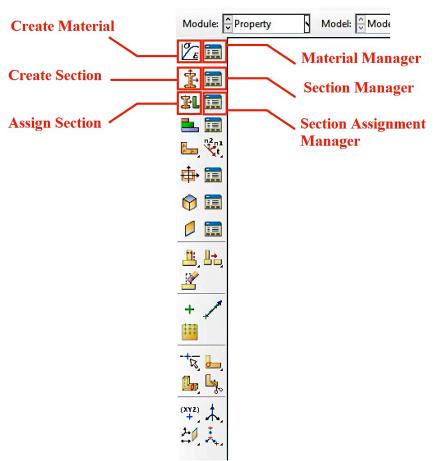


Figure 9 The Functional tools in Property module

- 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).

Name: truss_mat Description: Material Behaviors  General Mechanical Ihermal Electrical/Magnetic Other  Elasticity  Elasticity  Elasticity  Elastic  Plasticity  Damage for Ductile Metals  Damage for Flastomers Damage for Flastomers Damage for Elasticity  Deformation Plasticity Damping Expansion Brittle Cracking Egs Viscosity Super Elasticity  OK Cancel  OK Cancel	🔶 Edit Ma	terial	×
Material Behaviors         General       Mechanical       Ihermal       Electrical/Magnetic       Other         Elasticity       >       Elastic       Hyperelastic         Damage for Ductile Metals       >       Hyperfoam         Damage for Fiber-Reinforced Composites       >       Hypeglastic         Damage for Fiber-Reinforced Composites       >       Porous Elastic         Damage for Elastjomers       >       Porous Elastic         Damping       Egs       Yiscoelastic         Super Elasticity       Super Elasticity       Super Elasticity	Name: trus	s_mat	
General       Mechanical       Ihermal       Electrical/Magnetic       Other       Y         Elasticity       Image       Elastic       Hyperelastic       Hyperfoam         Damage for Ductile Metals       Image       Image       Hyperfoam       Low Density Foam         Damage for Fiber-Reinforced Composites       Image       Image       Porous Elastic       Porous Elastic         Damage for Elastomers       Image       Image       Image       Image       Image       Image         Deformation Plasticity       Image       Image	Description:		
General       Mechanical       Ihermal       Electrical/Magnetic       Other       Y         Elasticity       •       Elastic       Hyperelastic       Hyperfoam         Damage for Ductile Metals       •       Hyperfoam       Low Density Foam         Damage for Fiber-Reinforced Composites       •       Damage for Elastomers       •         Damage for Elastomers       •       Dorous Elastic       Yiscoelastic         Damping       Egs       Yiscosity       Super Elasticity			-
Elasticity       Elastic         Plasticity       Hyperelastic         Damage for Ductile Metals       Hyperfoam         Damage for Traction Separation Laws       Low Density Foam         Damage for Fiber-Reinforced Composites       Hypegelastic         Damage for Elastomers       Porous Elastic         Deformation Plasticity       Viscoelastic         Damping       Expansion         Brittle Cracking       Eos         Viscosity       Super Elasticity	Material B	ehaviors	
Elasticity       Elastic         Plasticity       Hyperelastic         Damage for Ductile Metals       Hyperfoam         Damage for Traction Separation Laws       Low Density Foam         Damage for Fiber-Reinforced Composites       Hypeglastic         Damage for Elastomers       Porous Elastic         Deformation Plasticity       Viscoelastic         Pamping       Expansion         Brittle Cracking       Egs         Viscosity       Super Elasticity			
Elasticity       Elastic         Plasticity       Hyperelastic         Damage for Ductile Metals       Hyperfoam         Damage for Traction Separation Laws       Low Density Foam         Damage for Fiber-Reinforced Composites       Hyperelastic         Damage for Elastomers       Porous Elastic         Deformation Plasticity       Viscoelastic         Damping       Expansion         Brittle Cracking       Egs         Viscosity       Super Elasticity			
Elasticity       Elastic         Plasticity       Hyperelastic         Damage for Ductile Metals       Hyperfoam         Damage for Traction Separation Laws       Low Density Foam         Damage for Fiber-Reinforced Composites       Hypeglastic         Damage for Elastomers       Porous Elastic         Deformation Plasticity       Viscoelastic         Damping       Expansion         Brittle Cracking       Eos         Viscosity       Super Elasticity			
Elasticity       Elastic         Plasticity       Hyperelastic         Damage for Ductile Metals       Hyperfoam         Damage for Traction Separation Laws       Low Density Foam         Damage for Fiber-Reinforced Composites       Hypeglastic         Damage for Elastomers       Porous Elastic         Deformation Plasticity       Viscoelastic         Damping       Expansion         Brittle Cracking       Eos         Viscosity       Super Elasticity			
Elasticity       Elastic         Plasticity       Hyperelastic         Damage for Ductile Metals       Hyperfoam         Damage for Traction Separation Laws       Low Density Foam         Damage for Fiber-Reinforced Composites       Hypelastic         Damage for Elastomers       Porous Elastic         Deformation Plasticity       Viscoelastic         Damping       Egs         Viscosity       Super Elasticity			
Plasticity       Hyperelastic         Damage for Ductile Metals       Hyperfoam         Damage for Traction Separation Laws       Low Density Foam         Damage for Fiber-Reinforced Composites       Porous Elastic         Damage for Elastomers       Porous Elastic         Deformation Plasticity       Viscoelastic         Damping       Expansion         Brittle Cracking       Egs         Viscosity       Super Elasticity	<u>G</u> eneral		
Damage for Ductile Metals Hyperfoam Damage for Traction Separation Laws Low Density Foam Damage for Fiber-Reinforced Composites Hypelastic Damage for Elastomers Deformation Plasticity Damping Expansion Brittle Cracking Eos Viscosity Super Elasticity			
Damage for Traction Separation Laws       Low Density Foam         Damage for Fiber-Reinforced Composites       Hypoelastic         Damage for Fiber-Reinforced Composites       Porous Elastic         Damage for Fiber-Reinforced Composites       Viscoelastic         Deformation Plasticity       Viscoelastic         Damping       Expansion         Brittle Cracking       Eos         Viscosity       Super Elasticity			
Damage for Fiber-Reinforced Composites       Hypoelastic         Damage for Elastomers       Porous Elastic         Deformation Plasticity       Viscoelastic         Damping       Expansion         Brittle Cracking       Eos         Viscosity       Super Elasticity			
Damage for Elastomers     Porous Elastic       Deformation Plasticity     Viscoelastic       Damping     Expansion       Brittle Cracking     Eos       Viscosity     Super Elasticity			
Deformation Plasticity <u>Viscoelastic</u> Damping Expansion Brittle Cracking Eos Viscosity Super Elasticity			
Damping Expansion Brittle Cracking Eos Viscosity Super Elasticity		-	
Expansion Brittle Cracking Eos Viscosity Super Elasticity		<u>-</u> iscoclastic	
Brittle Cracking Eos Viscosity Super Elasticity			
Eos Viscosity Super Elasticity			
<u>V</u> iscosity Super Elasticity		Second	
Super Elasticity			
OK	3	<u>Super classicity</u>	
OK Cancel			
		OK	

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.)

🕂 Edit Material	X
Name: truss_mat	
Description:	/
	×
Material Behaviors	
<u>G</u> eneral <u>M</u> echanical <u>T</u> hermal <u>E</u> lectrical/Magne	tic <u>O</u> ther
Elastic	
Type: Isotropic	▼ Suboptions
Use temperature-dependent data	
Number of field variables: 0	
Moduli time scale (for viscoelasticity): Long-term	Β
No compression	
No tension	
Data	
Young's Poisson's Modulus Ratio	
1 70e9 0.3	
OK	Cancel

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)

+ Create Section X			
Name: truss_sect			
Category Type			
○ Solid	Beam		
O Shell	Truss		
Beam			
○ Other			
Continue Cancel			
Figure 12 Create new section			

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<sup>2</sup> and click on **OK**.

+ Edit Section	x
Name: truss_sect Type: Truss	
Material: truss_mat	
Temperature variation: Constant through thick	<b>cne</b> ss
OK Cancel	

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.

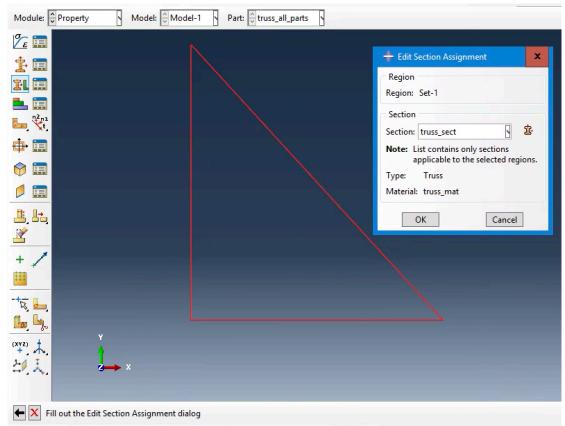


Figure 14 Assigning sections to elements

- 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://abagus.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.

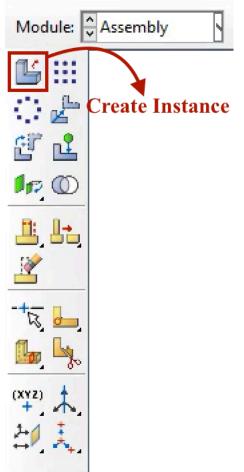


Figure 15 Assembly toolbox

- 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**.

🐥 Create Instance	x
Create instances from: Parts O Models Parts	
truss_all_parts	
Instance Type	
Oppendent (mesh on part)	
O Independent (mesh on instance)	
Note: To change a Dependent instanc mesh, you must edit its part's m	
Auto-offset from other instances	
OK Apply Canc	el

Figure 16 Create an instance

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

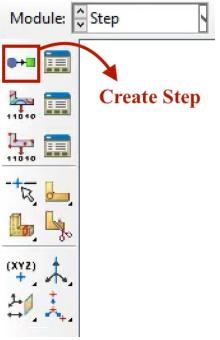


Figure 17 Step toolbox

- 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** (Figure 18).

+ Create Step X	🜩 Edit Step	x
Name: Step-1 Insert new step after Initial	Name: Step-1         Type: Static, General         Basic       Incrementation         Obscription:         Time period:         1         NIgeom:       Off         (This setting controls the inclusion of nonlinear effects         On       of large displacements and affects subsequent steps.)         Automatic stabilization:       None	
Procedure type: General Dynamic, Temp-disp, Explicit Geostatic Heat transfer Mass diffusion Soils Static, General Static, Riks Continue Continue	Include adiabatic heating effects	
Continue Cancel	OK	

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.

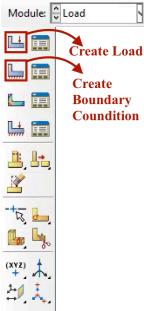


Figure 17 Load toolbox

- 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.

💠 Create Load	×
Name: Load-1	
Step: Step-1	]
Procedure: Static, General	k.
Category	Types for Selected Step
Mechanical	Concentrated force
O Thermal	Moment
O Acoustic	Pressure
O Fluid	Shell edge load 🗧
O Electrical/Magnetic	Surface traction Pipe pressure
O Mass diffusion	Body force
○ Other	Line load Gravity Body force
	Bolt load 🗸
Continue	Cancel

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**.

		<b>b</b> ,
(XY2)		
ka ya ka daga ya ka		
X X		
	• Setc	35 5 10 10 10
★ X Select points for the load ( ✓ Create set: Set-1) Done	Sets	<u>⊰</u> S SIMULIA

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).

<table-cell-rows> Edit Load</table-cell-rows>	i i	×
Name: Loa	d-1	
Type: Cor	centrated force	
Step: Step	o-1 (Static, General)	
Region: Set-	1	
CSYS: (Glo	obal) 🔉 🙏	
Distribution:	Uniform	f(x)
CF1:	0	
CF2:	-1000	
Amplitude:	(Ramp)	₽÷
Follow nodal rotation		
Note: Force will be applied per node.		
OK Cancel		

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).

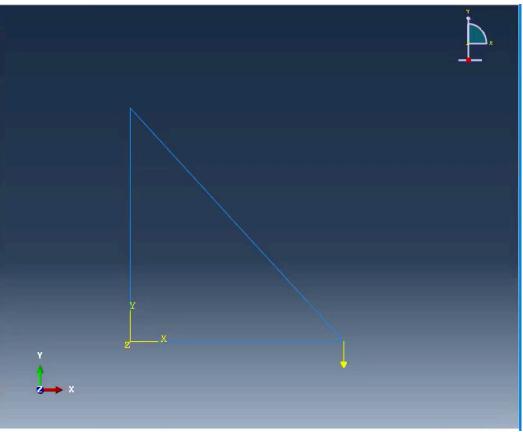


Figure 21 Demonstrating the defined concentrated load on the viewport

- 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).

🔶 Create Bounda	ry Condition	x
Name: BC-1		
Step: Initial Procedure:		
Category	Types for Selected Step	
Mechanical	Symmetry/Antisymmetry/Encastre	
<ul> <li>Electrical/Magr</li> <li>Other</li> </ul>	netic Displacement/Rotation Velocity/Angular velocity Acceleration/Angular acceleration Connector displacement Connector velocity Connector acceleration	
Continue	e Cancel	1

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).

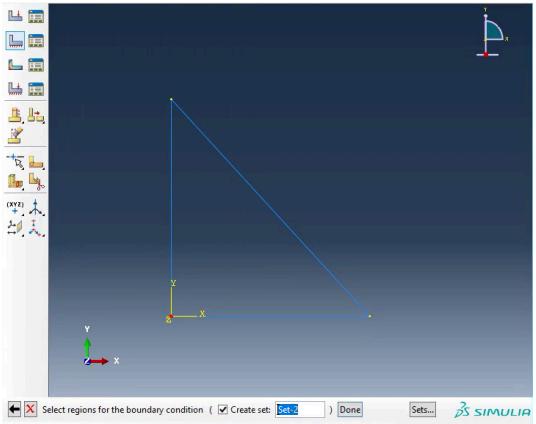


Figure 23 Selecting region imposed to the boundary condition

- 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).

<table-cell-rows> Edit Boundary Condition</table-cell-rows>	×
Name: BC-1	
Type: Displacement/Rotation	
Step: Initial	
Region: Set-2	
CSYS: (Global) 🔈 🙏	
🕑 U1	
✓ U2	
UR3	
Note: The displacement value will be maintained in subsequent steps.	
OK Cancel	

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).

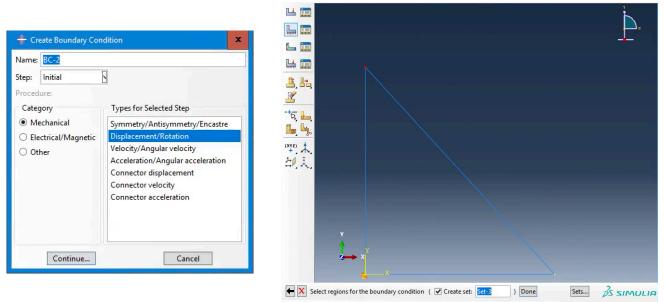


Figure 25 Defining the roller boundary condition

+ Edit Boundary Condition X
Name: BC-2
Type: Displacement/Rotation
Step: Initial
Region: Set-3
CSYS: (Global) 🔉 🙏 🗹 U1
□ U2
UR3
Note: The displacement value will be maintained in subsequent steps.
OK

Figure 26 Defining the roller boundary condition

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

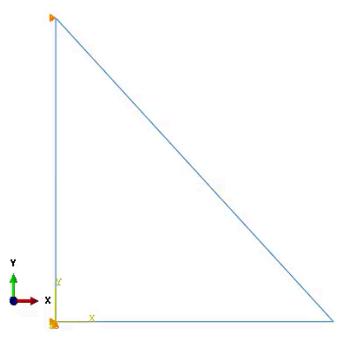


Figure 27 Displaying constrained degrees of freedom in the viewport

- 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).

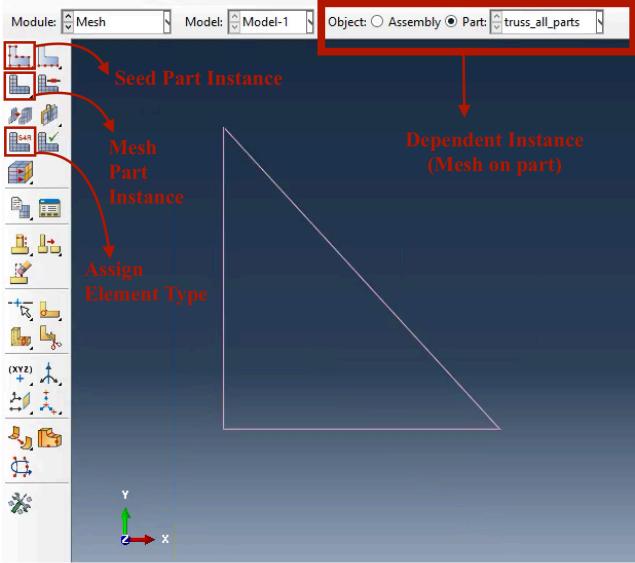


Figure 28 Mesh toolbox

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: <a href="http://abaqus.software.polimi.it/v6.14/books/usb/pt06.html">http://abaqus.software.polimi.it/v6.14/books/usb/pt06.html</a>)

🔶 Element Type	×
Element Library	Family
● Standard ○ Explicit	Piezoelectric
Geometric Order	Pipe Thermal Electric
● Linear ○ Quadratic	Truss
Line	
Hybrid formulation	bulk viscosity: 1
T2D2: A 2-node linear 2-	D truss.
Note: To select an element select "Mesh->Contr	shape for meshing, ols" from the main menu bar.
ОК	Defaults Cancel

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).

🕂 Global Seeds 🛛 🗙
Sizing Controls
Approximate global size: 2
Curvature control
Maximum deviation factor (0.0 < h/L < 1.0): 0.1
(Approximate number of elements per circle: 8)
Minimum size control
By fraction of global size (0.0 < min < 1.0)     0.1
O By absolute value (0.0 < min < global size) 0.2
OK Apply Defaults Cancel

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).

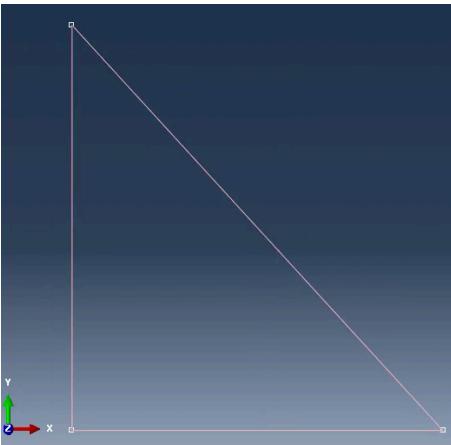


Figure 31 Displaying seeds

- 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)

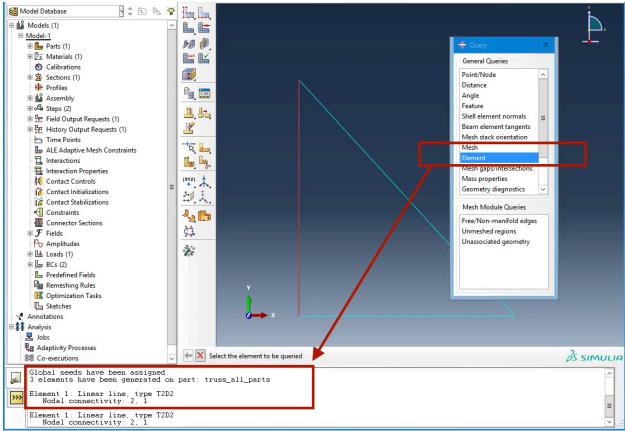


Figure 32 Query elements

- 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).

Module:	Job
1	

🕂 Create Job	x
Name: Job-1	
Source: Model	
Model-1	
Continue Canc	el

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.

💠 Edit Job						x
Name: Job-1						
Model: Mode	I-1					
Analysis produ	ict: Abaqu	s/Standard				
Description:						
Submission	General	Memory	Parallelization	Precision		
Job Type Full analy Recover Restart Run Mode Backgrou	(Explicit)	ue:	Host	name:		
Submit Tim Immedia Wait: At:	tely	in.				
	ОК	]		Can	cel	

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.

Name	Model	Туре	Status	Write Input
Job-1	Model-1	Full Analysis	None	Data Check
				Submit
				Continue
				Monitor
				Results
				Kill

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)

Step     Increment     Att     Discon     Iter     Iter     Time/Freq     Time/LPF     Inc       1     1     1     0     1     1     1     1     1       1     1     0     1     1     1     1     1     1       1     1     0     1     1     1     1     1     1       1     1     0     1     1     1     1     1     1		1 Monitor -1 Status: Co	mpleted		· · · · · · · · · · · · · · · · · · ·				
Log Errors ! Warnings Output Data File Message File Status File   Completed: Abaqus/Standard Completed: Mon Jan 22 01:15:03 2018 Image: Completed: Status File	Step	Increment	Att	Discon		1.			Time/LPF Inc
Completed: Abaqus/Standard Completed: Mon Jan 22 01:15:03 2018	1	1	1	0	1	1	1	1	1
	Comple Comple Search	eted: Abaqus/St eted: Mon Jan 2 Text	andard					vious	

Figure 36 Monitoring the job analysis

la la	Element 1: Linear line, type T2D2	^
>>>> >>>>	The job "Job-1" has been created. The job input file has been written to "Job-1.inp". The job input file "Job-1.inp" has been submitted for analysis. Job Job-1: Ahalysis Input File Processor completed successfully. Job Job-1: Abaques/Standard completed successfully. Job Job-1 completed successfully.	■ ▼ .::

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

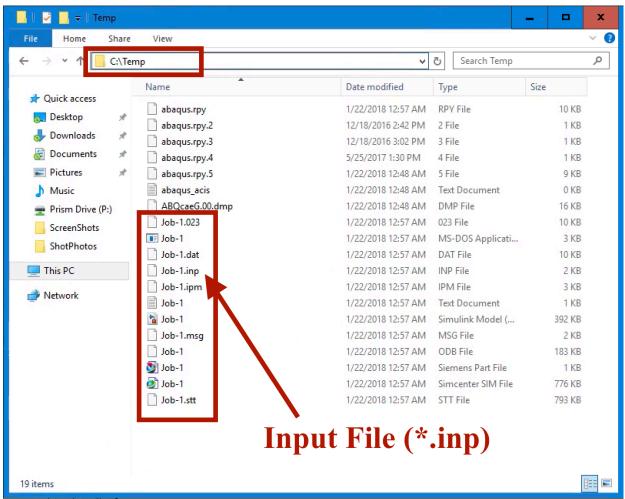


Figure 38 Generated files during the analysis

- 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

```
📕 Job-1 - Notepad
                                                             x
                                                       File Edit Format View Help
*Heading
                                                               ~
** Job name: Job-1 Model name: Model-1
** Generated by: Abagus/CAE 2017
*Preprint, echo=NO, model=NO, history=NO, contact=NO
** PARTS
se se
*Part, name=truss_all_parts
*Node
      1,
                   0.,
                                  0.
                                                               =
      2,
                   0.,
                                  1.
      3,
                                  0.
                   1.,
*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: <a href="http://abaqus.software.polimi.it/v6.14/books/usi/pt05ch40.html">http://abaqus.software.polimi.it/v6.14/books/usi/pt05ch40.html</a>)

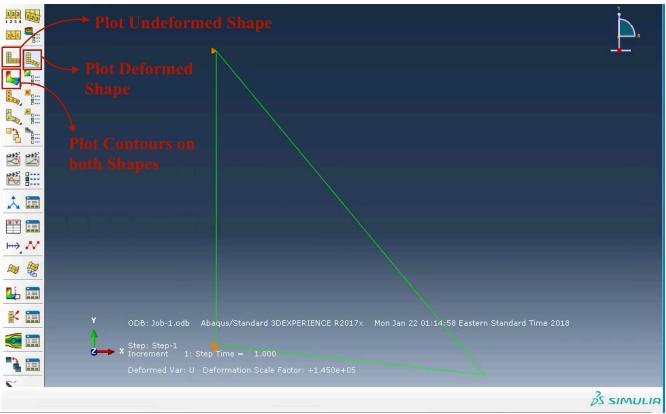


Figure 40 Visualization toolbox and output results (displacement)

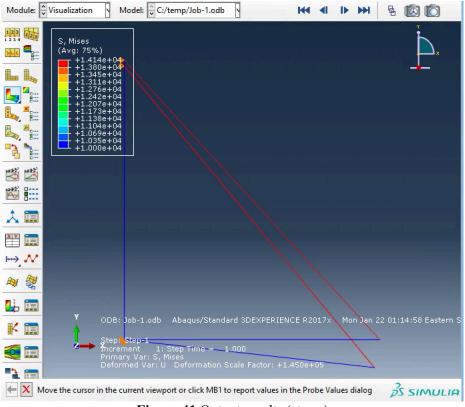
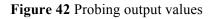


Figure 41 Output results (stress)

- 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).

General Queries	🜩 Prob	oe Values				
Node Distance Angle Mesh Distance Element Mass properties Element face normal	०म्बि Step मुख्य Fiel ─ Probe ● Selee	ct from viewport	○ Key-in lab	oel 🔿 Sel	ect a display group	
		Elements Co or Attached nodes		All Direct	Position: Inte	Egration Pt
		Part Instance	Element ID	Туре	Attached nodes	S, S11
isualization Module Queries		TRUSS_ALL_PAR TRUSS_ALL_PAR		T2D2 T2D2	2, 1 3, 2	-10000 14142.1
robe values tress linearization ctive elements		TRUSS_ALL_PAR		T2D2	1, 3	-10000
ctive nodes ly stack plot	Note: C		check buttor	n to annot	ate values in viewer	Cancel



- 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).

🕂 Report Probe Values	x			
File				
Name: StressOfElements.txt Sele	ct			
Append to file				
Output Format				
Page width (characters):   No limit   Specify: 80				
Number of significant digits: 6				
Data Values				
Write: Column totals Column min/max				
OK				

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						- x
File Edit Format View Help						
********	*******	*********	********	****	********	kakakakaka (
Probe Values Report, writ	tten on M	lon Jan 22 01	:34:14 2018			
Source						
ODB: C:/temp/Job-1.odb	b					
Step: Step-1						
Frame: Increment			000			
Variable: S, All dired	ct (Not a	veraged)				
and a Tabaanation						
Loc 1 : Integration point	c values					
Prohe values reported at	integrat	ion noints				
Probe values reported at	integrat	ion points				
Probe values reported at Part Instance Element			ched nodes			
Part Instance Element	ID 1	Type Atta	2		1	
Part Instance Element	ID 1	Type Atta	2		1 2	
Part Instance Element	ID 1 2	Type Atta	2			
Part Instance Element TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1	ID 1 2	Type Atta T2D2 T2D2	2 3		2	
Part Instance Element TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1	ID 1 2 3	Type Atta T2D2 T2D2 T2D2 T2D2	2 3 1		2 3	
Part Instance Element TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1 Part Instance Element	ID 1 2 3 ID	Type Atta T2D2 T2D2 T2D2 T2D2	2 3 1 Int. Pt.		2 3 S, S11	
Part Instance Element TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1 Part Instance Element TRUSS_ALL_PARTS-1	ID 1 2 3 ID	Type Atta T2D2 T2D2 T2D2 T2D2 Type T2D2	2 3 1 Int. Pt.	1	2 3	
Part Instance Element TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1 Part Instance Element	ID 1 2 3 ID	Type Atta T2D2 T2D2 T2D2 T2D2 Type T2D2	2 3 1 Int. Pt.	1	2 3 5, 511 -10.E+0	)3 )3
Part Instance Element TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1 Part Instance Element TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1	ID 1 2 3 ID 1 2	Type Atta T2D2 T2D2 T2D2 T2D2 Type T2D2 T2D2 T2D2	2 3 1 Int. Pt.	1 1	2 3 5, 511 -10.E+0 14.1421E+0	)3 )3

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)

💠 Field Output	Requests Manager			×
Name	Step-1			Edit
✓ F-Output-1	Created			Move Left
				Move Right
				Activate
				Deactivate
Step procedure:	Static, General			
Variables:	Preselected defaults			
Status:	Created in this step			
Create	Сору	Rename	Delete	Dismiss
	Figure 4	5 Editing field	outputs	<del></del>

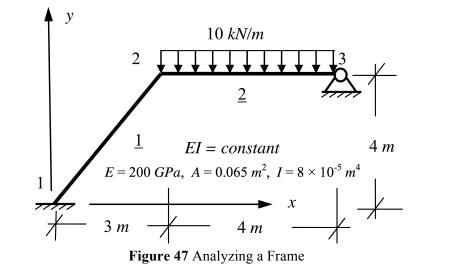
+ Edit Field Output Request				
Name: F-Output-1				
Step: Step-1				
Procedure: Static, General				
Domain: Whole model				
Frequency: Every n increments n: 1				
Timing: Output at exact times				
Output Variables				
○ Select from list below				
CDISP, CF, CSTRESS, LE, PE, PEEQ, PEMAG, RF, S, U,				
Stresses				
Strains				
Displacement/Velocity/Acceleration				
Forces/Reactions				
Contact				
Energy				
Failure/Fracture				
Thermal				
Electrical/Magnetic				
Porous media/Fluids				
Volume/Thickness/Coordinates				
Error indicators				
State/Field/User/Time				
Note: Some error indicators are not available when Domain is Whole Model or Interaction.				
Output for rebar				
Output at shell, beam, and layered section points:				
● Use defaults ○ Specify:				
Include local coordinate directions when available				
OK				

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.



#### 1 - Module: Part

Consideration(s):

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

💠 Create Part	x	Module: Part Model: Model-3Frame Part:
Name: frame		+ **
Modeling Space		
⊖ 3D	O Avis martin	
0 SD @ 2D Planar		
Туре	Options	
Deformable		
O Discrete rigid		
<ul> <li>Analytical rigid</li> </ul>	None available	+: <u>+</u> ;
O Eulerian		
Base Feature		
O Shell		
<ul> <li>Wire</li> </ul>		
O Point		
Approximate size: 100	[	

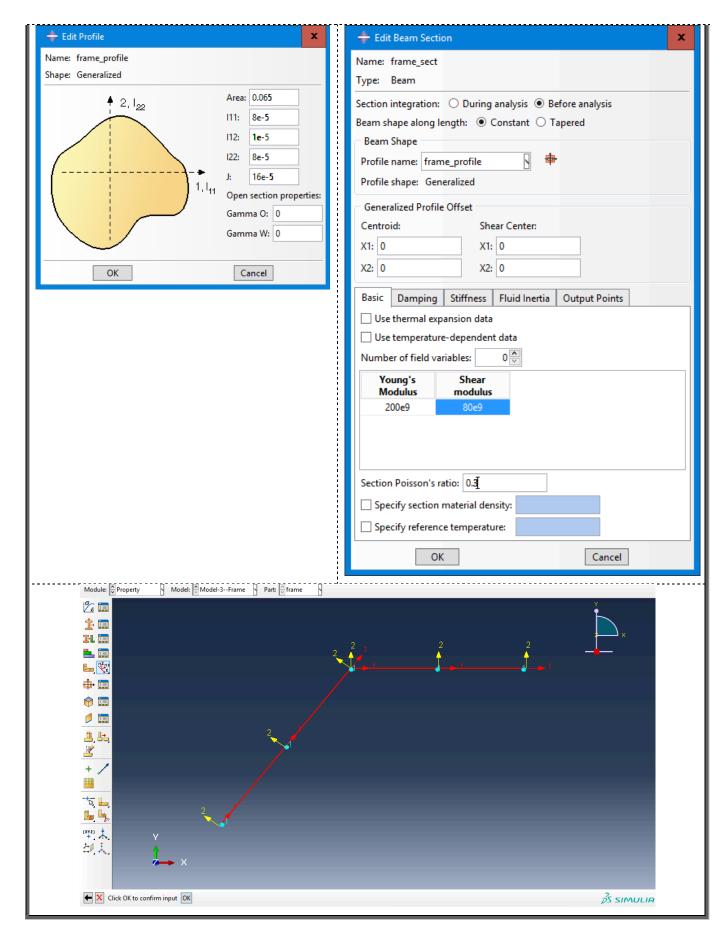
#### 2 - Module: Property

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 Profile. 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: <a href="http://abaqus.software.polimi.it/v6.14/books/usi/pt03ch12s15hlb03.html">http://abaqus.software.polimi.it/v6.14/books/usi/pt03ch12s15hlb03.html</a>)

#### **Figures:**

(In the next page)



## 3 - Module: Assembly

Consideration(s):

- Exactly similar to the truss problem

## Figures:

8	
	+ Create Instance
	Create instances from:
	Instance Type
	<ul> <li>Dependent (mesh on part)</li> </ul>
	O Independent (mesh on instance)
	Note: To change a Dependent instance's mesh, you must edit its part's mesh.
	Auto-offset from other instances
	OK Apply Cancel

4 - Module: Step					
Consideration(s):					
- Exactly similar to the truss problem					
Figures:					
+ Create Step	+ Edit Step				
Name: Step-1 Insert new step after	Name: Step-1 Type: Static, General Basic Incrementation Other Description:				
	Time period:       1         NIgeom:       Off On       (This setting controls the inclusion of nonlinear effects of large displacements and affects subsequent steps.)         Automatic stabilization:       None				
Procedure type:     General       Dynamic, Temp-disp, Explicit     ^       Geostatic     ^       Heat transfer	☐ Include adiabatic heating effects				
Continue Cancel	OK				

#### 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.

💠 Edit Load		x	+ Edit Boundary Condition 🗙
Name: Load-1			Name: BC-1
Type: Line load			Type: Displacement/Rotation Step: Initial
	Static, General)		Region: Set-2
Region: Set-1			CSYS: (Global) 🔓 🙏
System: G	lobal		<b>☑</b> U1
Distribution: U	niform	f(x)	✓ U2
Component 1: 0			UR3
Component 2: -	1000 <b>d</b>		
Amplitude: (F	Ramp)	Ъ	
ОК	Cancel		Note: The displacement value will be maintained in subsequent steps.
	Module: 🗘 Load 🕥 Model: 🗍	Aodel-3Frame	
			ľ –
	<u></u>		
	<u>≝</u> -t≅, <b>⊾</b> ,		
	<u>⊫</u> <u>₩.</u> <sup>(¥<sup>2</sup>)</sup> <del>↓</del>		
	(111) 木. 전 王.	z	
	Y		

### 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:		
🔶 Element Type		x
C Element Library	Family	
● Standard ○ Expli		
Geometric Order	Beam Coupled Temperature-Displacement	
● Linear ○ Quadrati	Gasket	<b>v</b>
Line		
Hybrid formulation	1	
Element Controls	r-flexible   Cubic formulation	
	ar bulk viscosity: 1	
B23: A 2-node cubic	peam in a plane.	
Note: To select an elem select "Mesh->C	ent shape for meshing, ntrols" from the main menu bar.	
ОК	Defaults	Cancel
		e: ()Mesh Nodel: )Model-3Frame Object: O Assembly @ Part: )frame
Module: Mesh Model: Model-3Frame Object: Ass	nbly 🔍 Part: 🗁 frame	
		4
		🗉 🕂 Local Seeds 🛛 🗙
<u>* h</u>	La La	
		Method Bias O By size  None O Single O Double
	- <del>1</del> 2 -	By number
	(XY2) ‡	Sizing Controls
<sup>(112)</sup> 未, 권,美,	と見え	
		Set Creation
		Create set with name: Edge Seeds-1
* 📩 ×	X.	X OK Apply Defaults Cancel
L		

# 7 - Module: Job

Consideration(s):

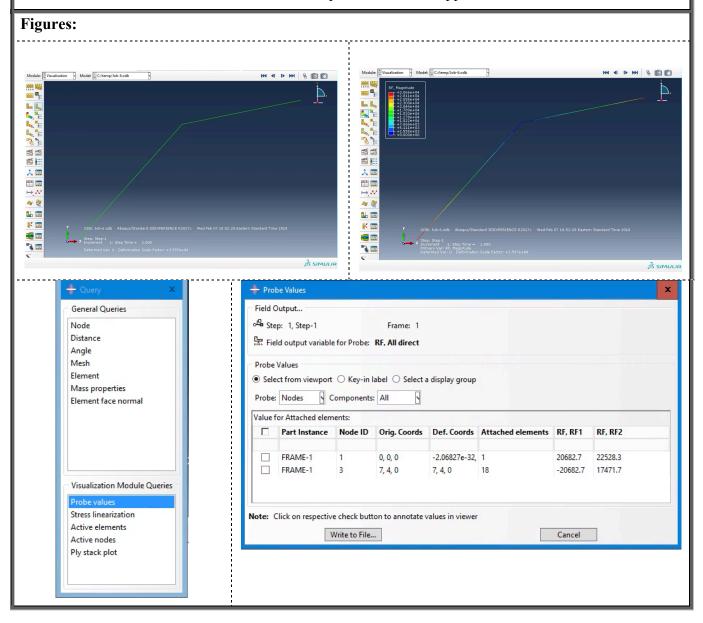
- Exactly similar to the truss problem

Figures:						
🕂 Create J	ob X		💠 Edit Job			x
Name: Job-	λ.		Name: Job-2			
			Model: Model-2Frame			
Source: Mo	del N		Analysis product: Abaqus/St Description: Simple Frame A			
Model-1				emory Parallelizati	on Precision	
Model-2F	rame		Job Type			
			<ul> <li>Full analysis</li> <li>Recover (Explicit)</li> </ul>			
			O Restart			
			Run Mode			
			Background O Queue:	N	Host name: Type:	
Continue	Cancel		Submit Time			
			Immediately			
			O Wait: hrs. min.	<del>ç</del> .		
			ОК		Cancel	
	💠 Job Manager				×	
	Name	Model	Туре	Status	Write Input	
	Job-1 Job-2	Model-1 Model-2Fi	Full Analysis rame Full Analysis	Completed Completed	Data Check	
	505-2	Model-211	Torre Torr Analysis	completed	Submit	
					Continue	
					Monitor	
					Results	
					Kill	
	Create	Edit	Rename	Delete	Dismiss	
	III	2				
	Job Job-2: Ar Job Job-2: Ab	nalysis Input Dagus/Standard	File Processor co d completed succes	mpleted suc sfully.	cessfully.	
ĺ	Job Job-2 con The job input	pleted succes file "Job-2.	ssfully. .inp" has been sub	mitted for	analysis.	
	Job Job-2: Ar Job Job-2: Ab	nalysis Input Daqus/Standard	File Processor co d completed succes	mpleted suc	cessfully.	
	Job Job-2 com	pleted succes	ssfully.			

### 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.



# **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: <u>https://en.wikipedia.org/wiki/Abaqus</u>. [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=15 for conventional shells; n=16 for continuum shells; n=13 for beams)
11	SE	All section strain, curvature change, and twist components
12	SEn	Section strain component n (n=16 for shells; n=13 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=13)
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=13)
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=13)
23	RMn	Reaction moment component n (n=13)
24	CF	All components of point loads and concentrated moments
25	CFn	Point load component n (n=13).
26	LOADS	Current values of distributed loads

**Truss Analysis Outputs** 

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 n	odes
TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1	1 2 3		2 1 3 2 1 3
Part Instance Element ID		Type Int. Pt.	S, Mises
TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1 TRUSS_ALL_PARTS-1	1 2 3	T2D2 T2D2 T2D2 T2D2	1 10.E+03 1 14.E+03 1 10.E+03
Minimum at Element Int. Pt. in Part Instance			10.E+03 3 1 TRUSS_ALL_PARTS-1
Maximum at Element Int. Pt. in Part Instance			14.E+03 2 1 TRUSS_ALL_PARTS-1
Total			34.E+03

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 Attache	d 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.P	t. S, Max.	In-Plane P	incipal	S, Max.	In-Plane Pr	incipal (Abs	) S, Min.	In-Plane	e Principal	S, Max.	Principal	S, Max.	Principal (	Abs) S, I	1in. Princ	ipal
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	LE+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+0	3	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_PART	S-1 TRUSS_	ALL_PARTS-1	TRUSS_ALI	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_PART	S-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+0	3 14.E	+03 -6.E	+03 -20.E+	03									

Field Output Report, written Tue Jan 30 18:24:01 2018 Source 1	ude
	1
0DB: C:/temp/Job-1.odb         1         0.           Step: Step-1         2         1.           Frame: Increment         1: Step Time = 1.000         3         1.	1.
Loc 1 : Nodal values from source 1 Minimum 0.	
Dutput sorted by column "Node Label". At Node 1	
Field Output reported at nodes for part: TRUSS_ALL_PARTS-1 At Node 3 Computation algorithm: EXTRAPOLATE_COMPUTE_AVERAGE Averaged at nodes Averaged regions: ODB_REGIONS	3
Node Label CF.Magnitude Node Label COORD.COOR1 @Loc 1 @Loc 1	
1       0.         2       0.         3       1.E+03	0.
Minimum         0.         Minimum         0.           At Node         2         At Node         2	
Maximum         1.E+03         Maximum         1.           At Node         3         At Node         3	
Total 1.E+03 Total 1.	1.
Node Label CF.CF1 Node Label COORD.COOR2 @Loc 1 @Loc 1	1
1       0.         2       0.         3       0.	0. 1.
Minimum 0. At Node 3 Minimum 0. At Node 3	
Maximum         0.         Maximum         1.           At Node         3         At Node         2	1. 2
Total 0. Total 1.	1.
Node Label         CF.CF2         Node Label         RF.Magnitude           @Loc 1         @Loc 1         @Loc 1	
1       0.       1       1.41421E+03         2       0.       2       1.E+03         3       -1.E+03       3       0.	03
Minimum -1.E+03 Minimum 0. At Node 3 At Node 3	
Maximum         0.         Maximum         1.41421E+03           At Node         2         At Node         1	
Total -1.E+03 Total 2.41421E+03	03

Loc 1
Minimum A
Maximum A
Node
Watan
Minimum A
Maximum A
Node
Minimum A:
A: Maximum
A: Maximum A:
A: Maximum A
A: Maximum A:
A Maximum A Node  Minimum

Node Label	U.Magnitude @Loc 1	Node Label
1	0.	1
2	142.857E-09	2
3	704.413E-09	3
Minimum	0.	imum
At Node	1	At Node
Maximum	704.413E-09	imum
At Node	3	At Node
Total	847.270E-09	Total
Node Label	U.U1 @Loc 1	Node Label
1	-1.00000E-33	1
2	1.00000E-33	2
3	-142.857E-09	3
Minimum	-142.857E-09	imum
At Node	3	At Node
Maximum At Node	1.00000E-33	imum At Node
Total	-142.857E-09	Total
Node Label	U.U2 @Loc 1	Node Label
1	-1.00000E-33	1
2	-142.857E-09	2
3	-689.775E-09	3
Minimum	-689.775E-09	imum
At Node	3	At Node
Maximum	-1.00000E-33	imum
At Node	1	At Node
Total	-832.632E-09	Total
Node Label	E.Max. In-P @Loc 1	Node Label
1	0.	1
2	101.015E-09	2
3	101.015E-09	3
Minimum	0.	imum
At Node	1	At Node
Maximum	101.015E-09	imum
At Node	3	At Node

Node Label	E.Min. Prin @Loc 1		S.Max. In-P(a) @Loc 1
	-142.857E-09 -71.4286E-09 -71.4286E-09	1 2 3	2.07107E+03
Minimum At Node	-142.857E-09 1	Minimum At Node	-10.E+03 1
Maximum At Node	-71.4286E-09 3	Maximum At Node	2.07107E+03 3
Total	-285.714E-09	Total	-5.85786E+03
Node Label	E.E11 @Loc 1	Node Label	S.Min. In-P @Loc 1
1 2 3		1 2 3	-10.E+03 -5.E+03
Minimum At Node		Minimum At Node	
Maximum At Node	29.5867E-09 3	Maximum At Node	
Total	-83.6838E-09	Total	-20.E+03
Node Label	S.Mises @Loc 1	Node Label	S.Max. Prin @Loc 1
1 2 3	10.E+03 12.0711E+03 12.0711E+03	1 2 3	
Minimum At Node		Minimum At Node	0. 1
Maximum At Node	12.0711E+03 3	Maximum At Node	7.07107E+03 3
Total	34.1421E+03	Total	14.1421E+03
Node Label	S.Max. In-P @Loc 1		S.Max. Prin(a) @Loc 1
1 2 3	0. 7.07107E+03 7.07107E+03	1 2 3	
Minimum At Node	0. 1	Minimum At Node	
Maximum At Node	7.07107E+03 3	Maximum At Node	
Total	14.1421E+03	Total	-5.85786E+03

Node Label	S.Min. Prin @Loc 1
1	-10.E+03
2 3	-5.E+03 -5.E+03
5	512.05
Minimum	-10.E+03
At Node	1
Maximum At Node	-5.E+03 3
Total	-20.E+03
Node Label	S.Tresca
	@Loc 1
1 2	10.E+03 12.0711E+03
3	12.0711E+03
Minimum	10.E+03
At Node	1012105
Maximum	12.0711E+03
At Node	3
Total	34.1421E+03
Node Label	S.Pressure
	@Loc 1
1 2	3.33333E+03
2 3	-690.356 -690.356
Minimum At Node	-690.356 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 2.07107E+03
5	210/20/2005
Minimum	-10.E+03
At Node	1
Maximum At Node	2.07107E+03 3
Total	-5.85786E+03
10000	51057002.05

Frame Analysis Outputs

#### \*\*\*\*\*\*\*

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

Source 1

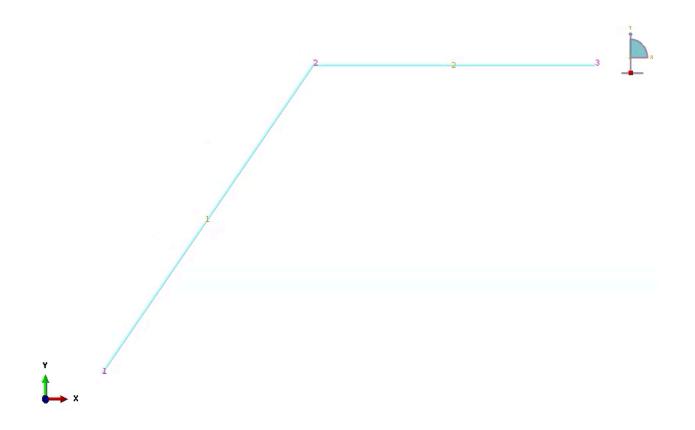
ODB: C:/temp/Job-6.odb Step: Step-1 Frame: Increment 1: Step Time = 1.000

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	COORD.COOR2	RF.RF1	RF.RF2	RM3	U.U1	U.U2	UR3
	@Loc 1	@Loc 1	@Loc 1	@Loc 1	@Loc 1	@Loc 1	@Loc 1	@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



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

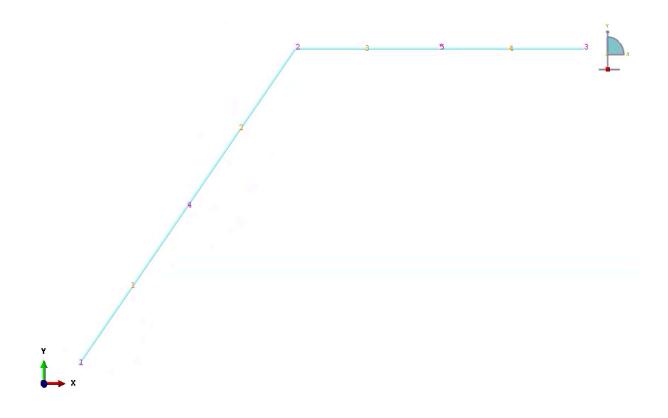
Loc 1 : Nodal values from source 1

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	COORD.COOR2	RF.RF1	RF.RF2	RM3	U.U1	U.U2	UR3
	@Loc 1	@Loc 1	@Loc 1	@Loc 1	@Loc 1	@Loc 1	@Loc 1	@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

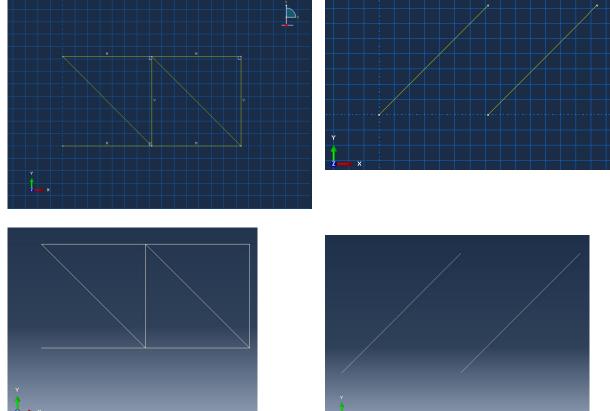


# Hints on Homework #5 - Abaqus Problem

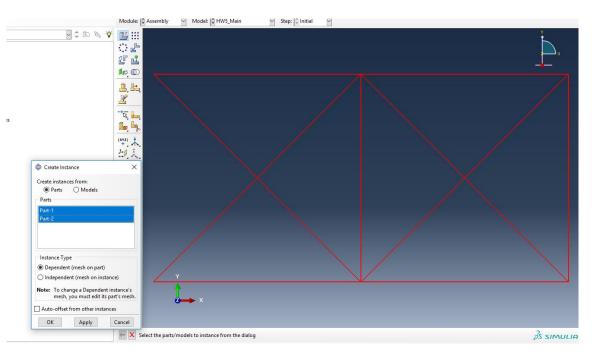
# **Dealing with Unintended Intersections**

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:

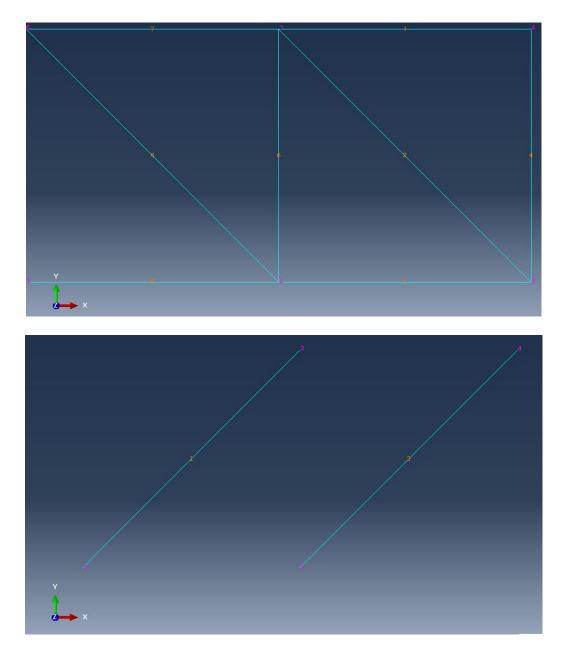
1- Create two complementary parts



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.



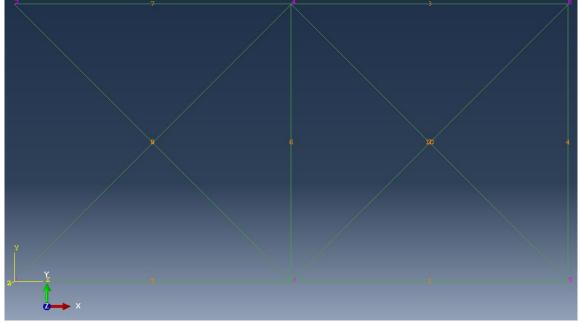
3- <u>Before</u> defining loads and boundary conditions, go to the Mesh module and mesh each part, separately.



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

14 :::	+ Merge/Cut Instances X
	Note: This function will create a new part and automatically instance it into the assembly.
<u>er e</u>	Part name: AllParts
IF <mark></mark>	Operations Merge
Merge/Cut Instances	<ul> <li>Geometry</li> <li>Geometry</li> <li>Cut geometry</li> </ul>
	Options Original Instances
🚂 🖳	● Suppress ○ Delete
(XYZ)	Mesh Merge nodes:
<u>さ</u> (美)	Boundary only      All      None
	Remove duplicate elements
	Tolerance: 1E-006
	Continue Cancel
king both instances and click on <b>Done</b> .	

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 ( $\alpha$ ) in Material Behaviors:

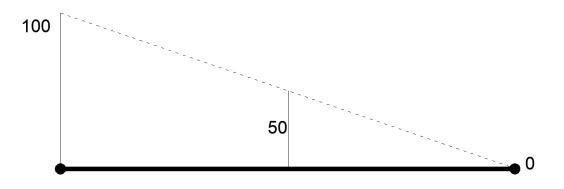
Module: **Property** Button: **Create Material** or **Material Manager Edit Material** dialog box Tab: **Mechanical / Expansion** 

💠 Edit Material X
Name: Material-1
Description:
Material Behaviors
Elastic
Expansion
General Mechanical Thermal Electrical/Magnetic Other
Expansion
Type: Isotropic
Use user subroutine UEXPAN
Reference temperature: 0
Use temperature-dependent data
Number of field variables:
Data
Expansion
1 1E-006
OK Cancel
Canter

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**:

💠 Create Predefined Field 🛛 🗙					
Name:	Predefined Field-1				
Step:	Initial	$\sim$			
Proced	ure:				
Categ	jory	Types for Selected Step			
⊖ Me	chanical	Temperature			
Ot	her	Material assignment			
		Initial state			
		Saturation			
		Void ratio			
		Pore pressure			
		Fluid cavity pressure			
С	ontinue	Cancel			

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

🜩 Edit Predefined Field					
Name: Predefined Fi	ld-1				
Type: Temperature					
Step: Initial					
Region: (Picked)					
Distribution:	Direct specification 🗸	f(x)			
Section variation:	Constant through region	~			
Magnitude:	0				
ОК	Cancel				

Then,	modify the	Magnitude	for the Step-1	(Magnitude =	100):
- ,	,		<b>r</b>	( . 0	

	Predefined Field Man	nager	×	]
	Name Initia ✓ Predefined Fie Creat		Edit Move Left Move Right	
	Predefined Field type: To Predefined Field status: Pr	tatic, General emperature ropagated from a previous opy Rename	step Delete Dismiss	-
	_	(x)		
* Magnitude:	100 (Ramp)		<ul> <li>Predefined Field Manager</li> <li>Name Initial Step-1</li> <li>Predefined Fie Created Modified</li> <li>Step procedure: Static, General</li> <li>Predefined Field type: Temperature</li> <li>Predefined Field status: Modified in this step</li> <li>Create Copy Rename</li> </ul>	Edit Move Left Move Right Delete
ОК		Cancel		

(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  $\sigma_{11}$  in the predefined stress dialog box.

💠 Create Predefined Field 🛛 🗙	💠 Edit Predefine	d Field			×
Name: Predefined Field-1 Step: Initial Procedure: Category Types for Selected Step	Name: Predefine Type: Stress Step: Initial Region: (Picked)	R			
Mechanical     Velocity	Specification: Dir	ect specification	~		
Other Stress Geostatic stress Hardening	Stress value	Sigma11 -1500	<b>Sigma22</b> 0	Sigma12 0	
Continue Cancel					
		ОК			Cancel

#### (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.