Abaqus Tutorials

This is the first lesson of eight-course free tutorials available on the CAEassistant.com website (Comprehensive Abacus Software Service and Training Center).

To take the following lessons for free, you can visit our website (CAEassistant.com) and check out the "Free Education" section.

Hello guys

In this course, we start an ABAQUS software free training course from the initial steps.

Here we try to give basic information and starting points for those who want to work on a preliminary level or, later, on an advanced level.

Not to brag, but we have engineers preparing the courses for you who have extensive experience teaching Abaqus software.

So, let us get to the point.

In each tutorial session, we will provide theoretical information and solve some sample problems for you. Let me tell you, we will assign you tasks and expect you to follow us step-by-step!

first session

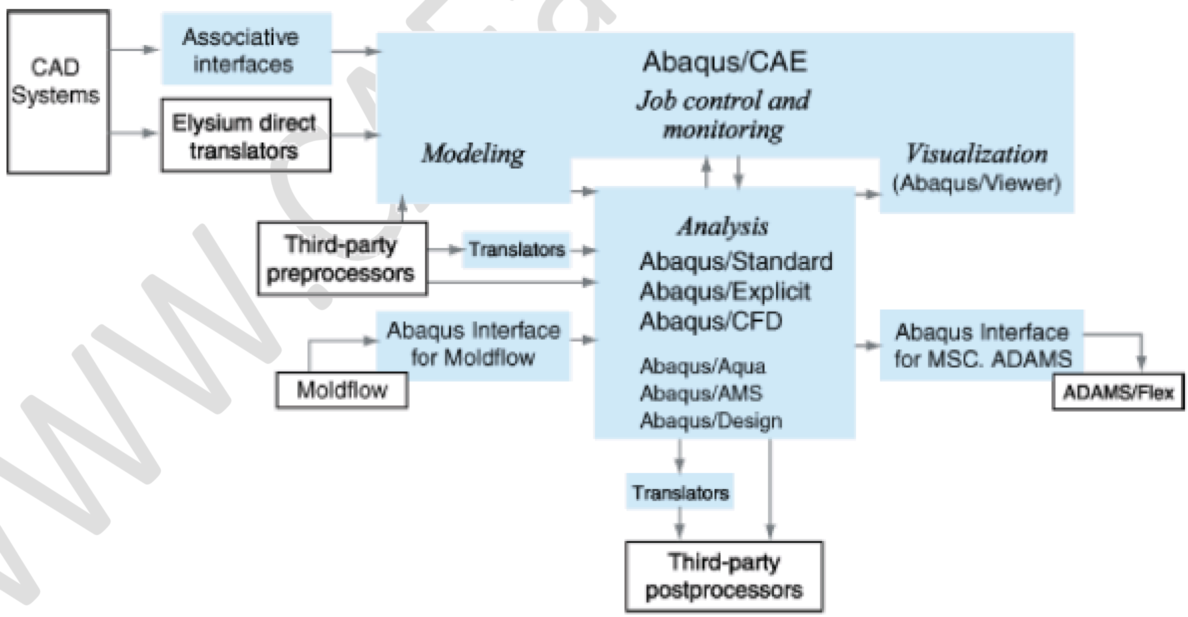
Overview of Abaqus

Dassault System Company has owned the Abaqus software for many years. It is an analytics software based on finite elements. Dassault Systèmes is based in France. One of the world's largest software companies, specializing in 3D product design, simulation, analysis, and manufacturing, such as CATIA and Solid Works.

This software was initially designed for solid analysis, but it has expanded into other phases over time, including fluids, and is constantly improving. Let me tell you that this software is one of the most widely used globally and due to its ease of use, it is trendy!!

Abacus is a CAE (Computer-Aided Engineering) software, similar to Ansys and LSDYNA!

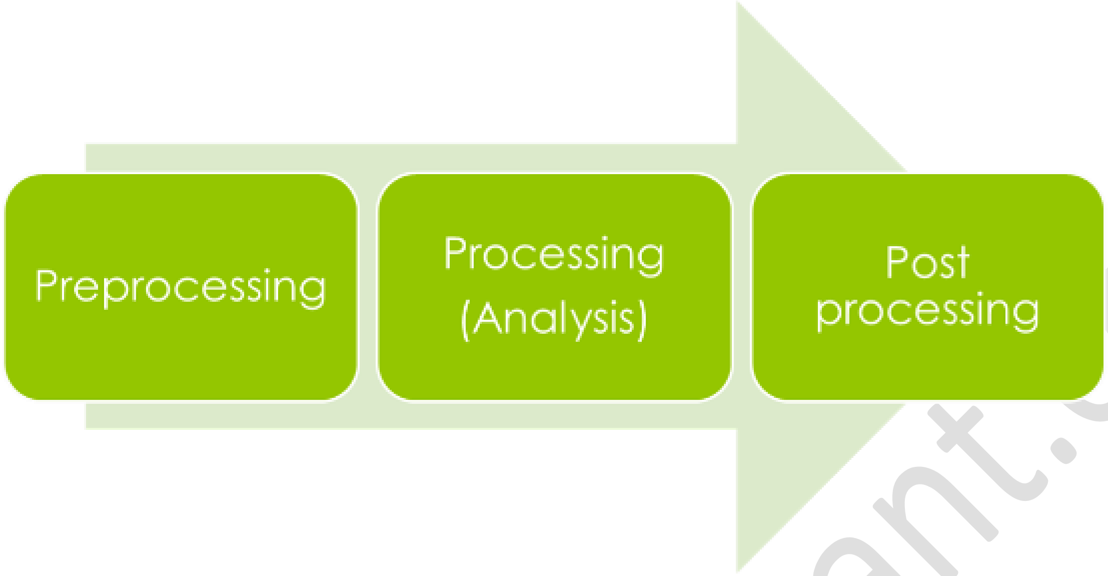
This hierarchy demonstrates how Abaqus works within the specified range. Remember that this shape was created by the Dassault system company .



OK. We will continue...

+1 (740) 966-1814
+43 68181750465
Support@CAEassistant.com
Enviroflex GmbH, Sterngasse 3/2/6 1010, Vienna, Austria (Sales Representative)

Abaqus performs a step-by-step analysis. Take note of the figure below!!



Yes, that's correct...

Pre-processing, processing, and post-processing. The first step is to prepare the model in the software by you, and then Abaqus delivers it to a solver to process it using the governing equations and finite element method. Finally, in the post-processing section, you can see the results in various forms!!

For a better understanding, consider the following explanations and figures:

Pre-processing:

Pre-processing consists of the following steps:

1. Create geometry in Abaqus or import geometry from another software.

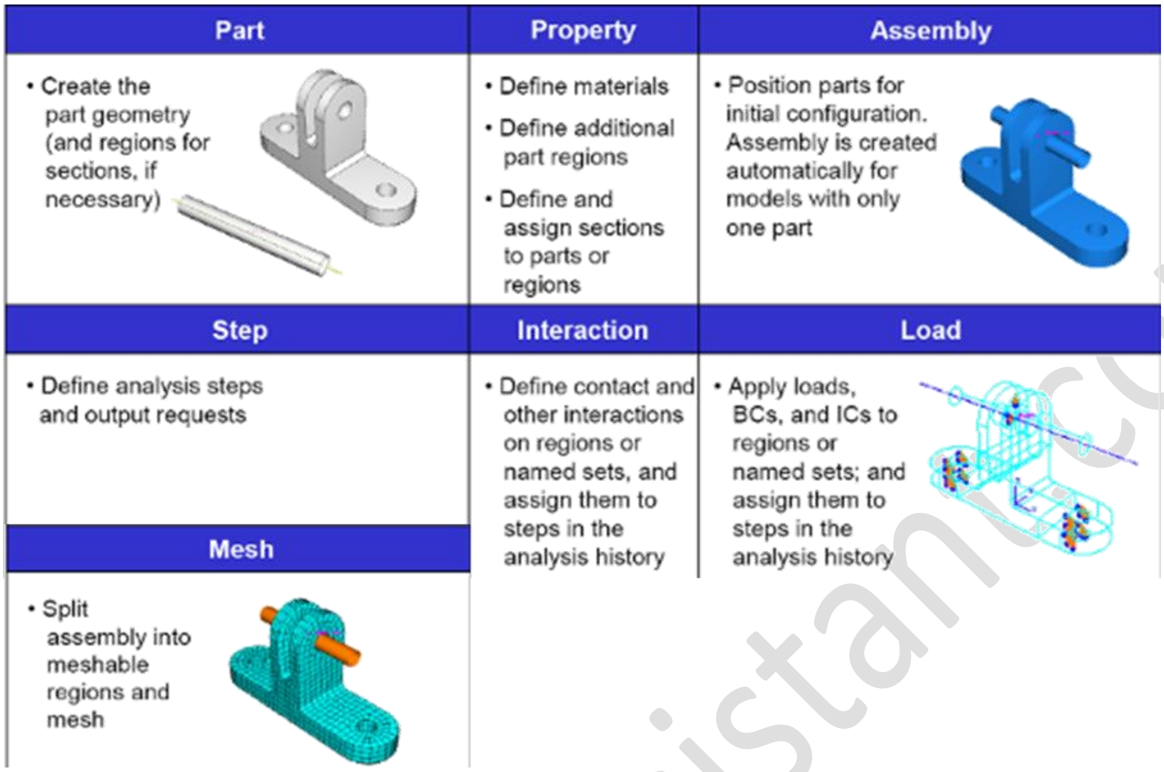
2. Create the material and assign it to your model.

3. Assemble your parts.

4. Select the solver type and desired output.

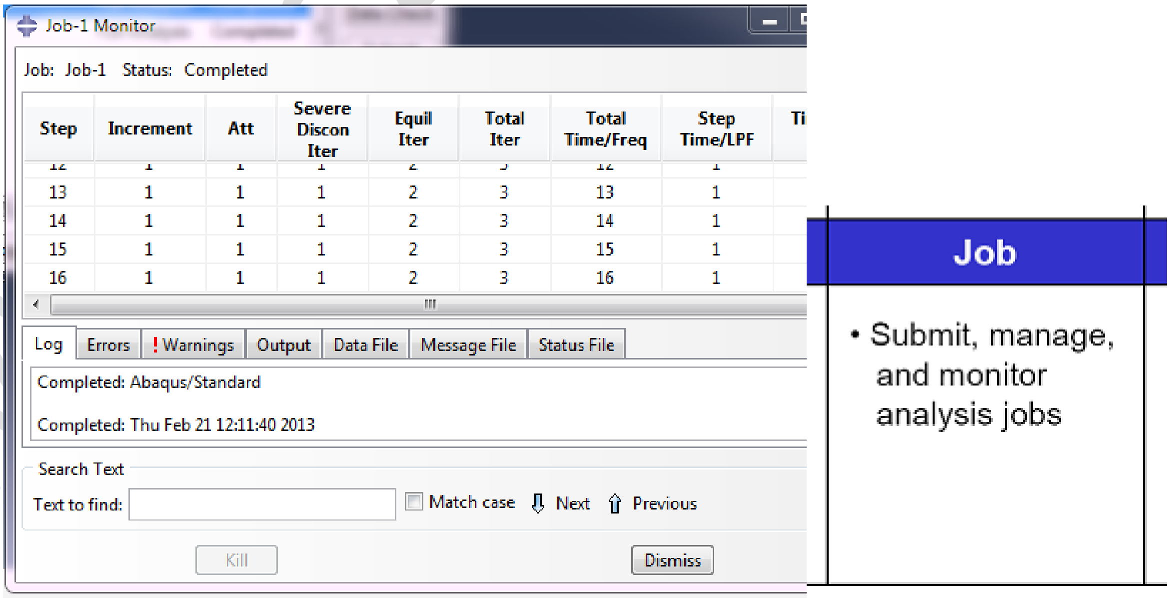
5. Apply the boundary conditions and loading.

Finally, select the appropriate element and mesh the model.



Processing:

In this section, run your model and monitor the process!! (Check the correctness of the process while problem-solving)

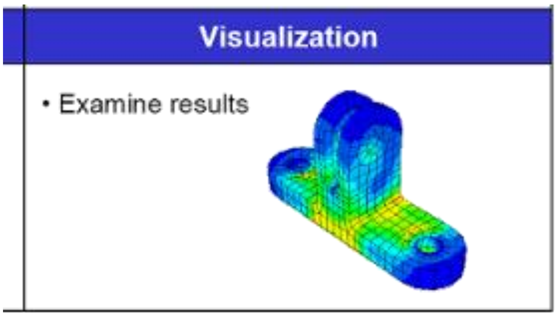


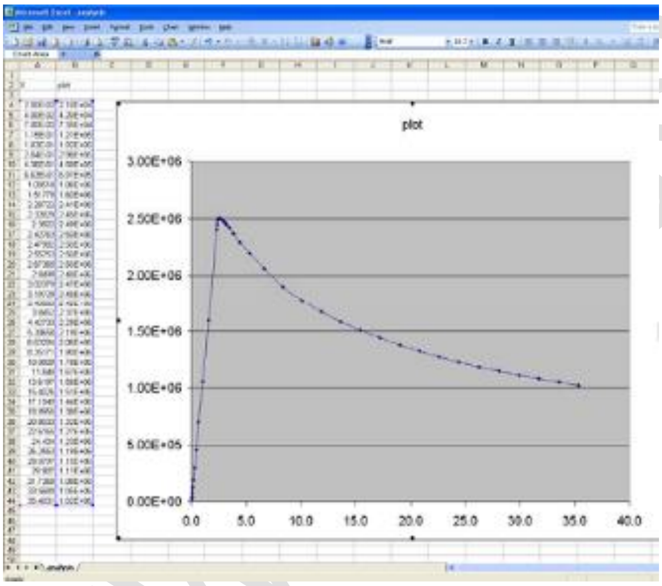
post-processing:

Post-processing:

Oh, this is the best part.

Because we're done and we'll be able to get the results. Of course, this section includes tools for determining whether the answers are correct or not, and, maybe, I will give you intuition about them later. There are various methods to extract results, including contour, number, path definition, symbol, chart, and a thousand of other things. You can also use other software, such as Excel, to generate your output.





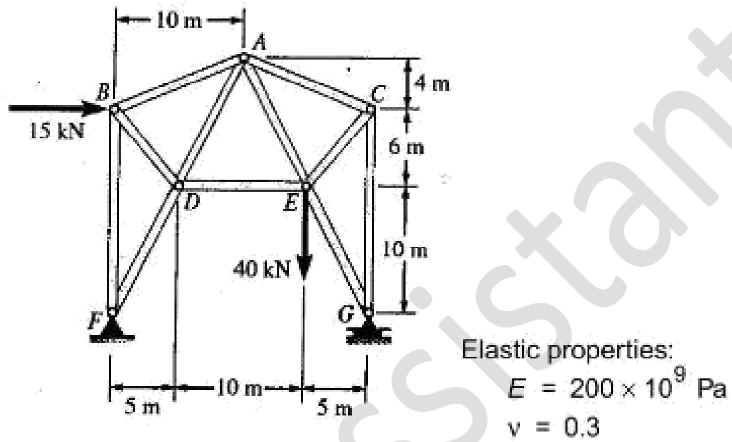
Enough study! We're exhausted!!

Let's look at an example to work with this software.

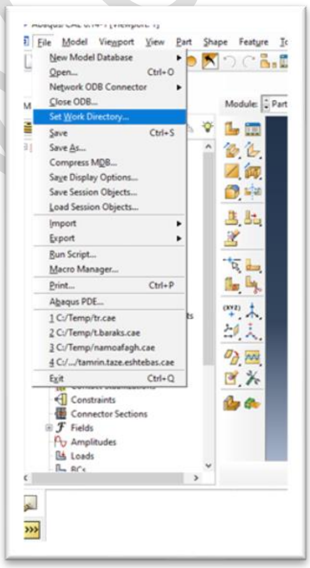
2D truss

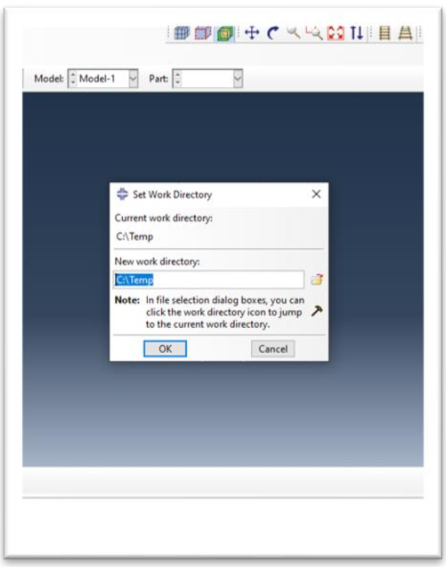
This is the question!

A Statics problem, tell me how much force and deflection each truss member experiences (I mean displacement - this is Mechanics of materials2!! Do you recall?!) Also, which one is the critical member, and what are their reactions? (The cross-section radius is 10 mm.)



To begin, we must specify the working directory for the Abaqus file. When you run the software and choose ‘Set Work Directory’ from the main menu bar, the Work Directories toolbar is updated to reflect the new setting. (As illustrated below)

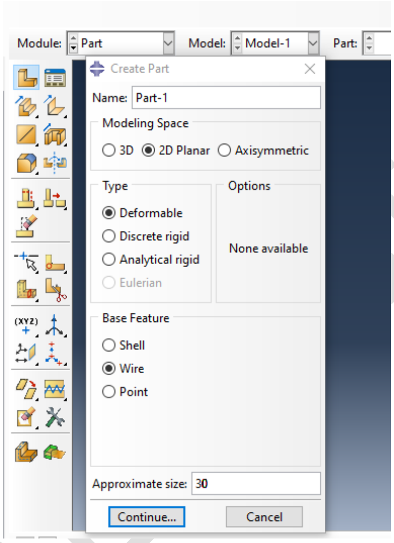




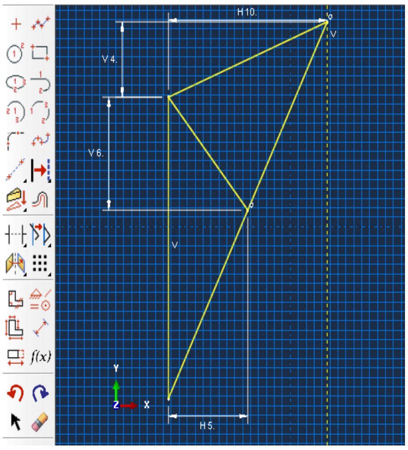
After you've specified your working directory, you can begin geometric modeling.

Step 1: Truss geometric modeling

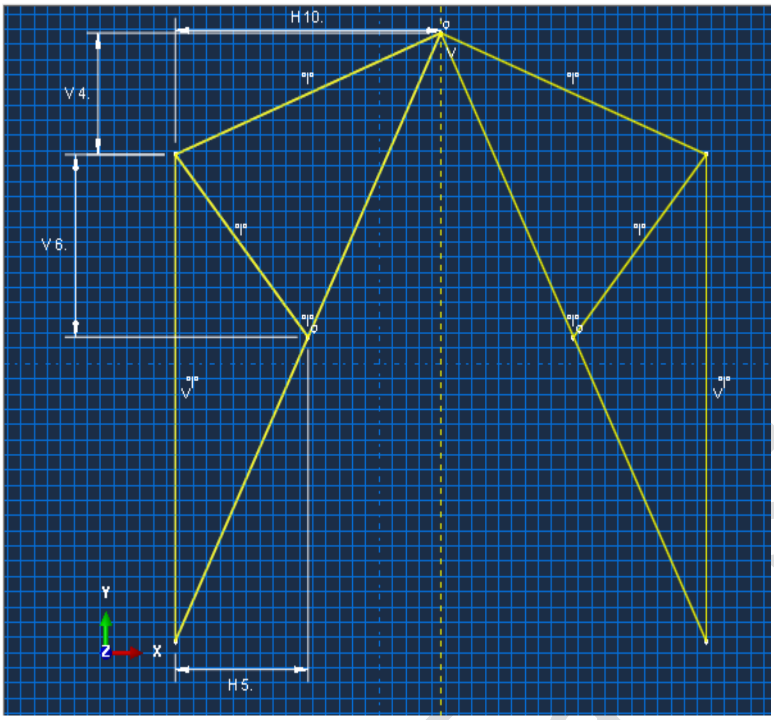
First, in the part module, use tool to create a part. Make the settings shown in the figure. Modeling Space in radio buttons should be two-dimensional (planar), and the Truss type should be deformable. The ‘base feature’ field defines the shape and type of the part's base feature. The wire was used here. Then, using Approximate, set the page dimensions.



Draw half of the geometry with a series of lines in this section (because of symmetry). We accomplish this with the help of tool. And we use the Add Dimension tool to determine the dimensions. Finally, we employ the Create Construction tool. We have the following figure until this step.



Now hold down the mouse button on Translate then select the mirror tool, then select the copy button ('copy' and 'move' you can see at the bottom of the page) and choose the dotted line (same as the symmetrical line) and shape, so we can get the final shape.



Warning! It is not over yet!! We must connect the middle line...

seriously?

Do you need more photos?! :)))

Just do it!

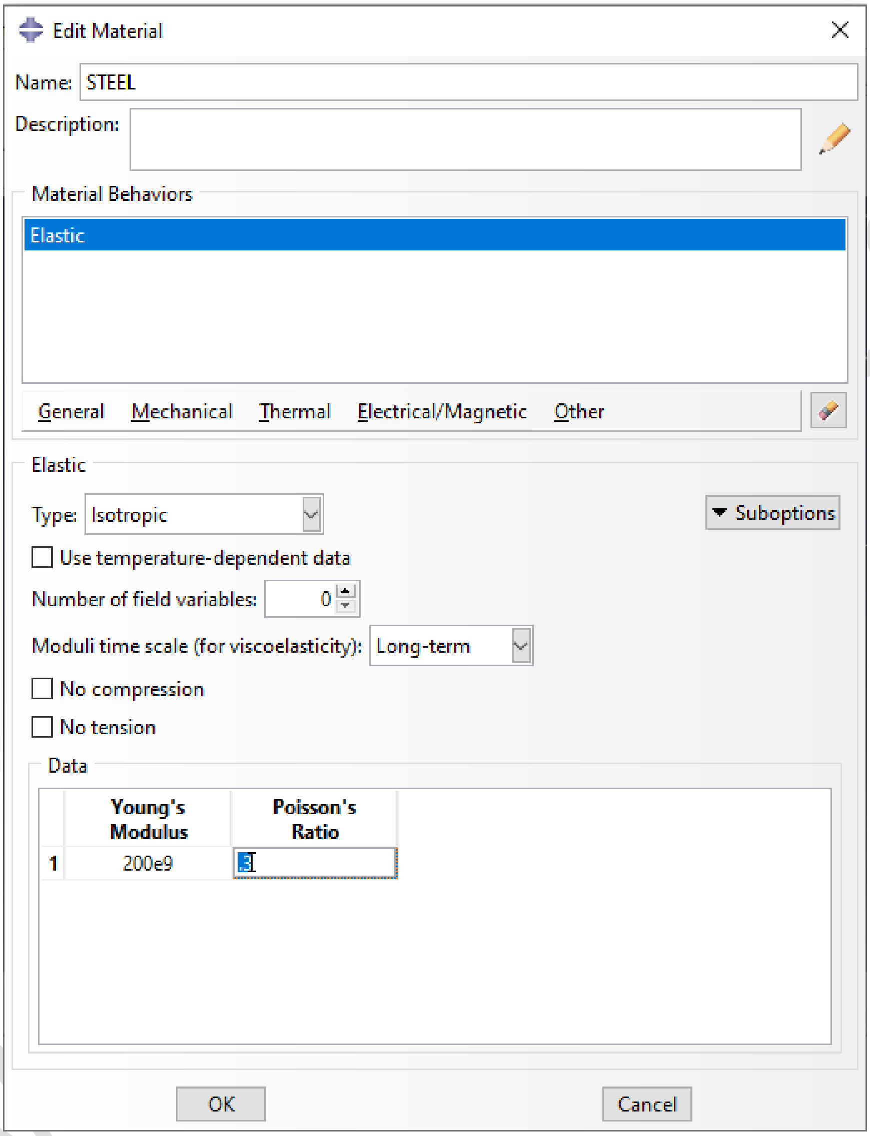
Once the geometry has been created, simply confirm by pressing the middle mouse button. Exit sketcher by pressing the DONE or middle mouse button once more.

Step 2: define the mechanical and geometric properties

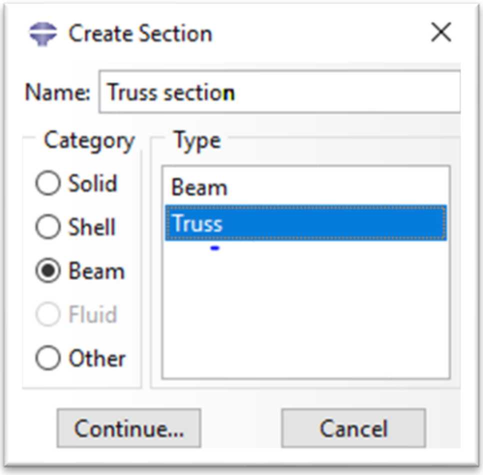
First, create a linear elastic material, such as the one shown in the figure, called Steel:

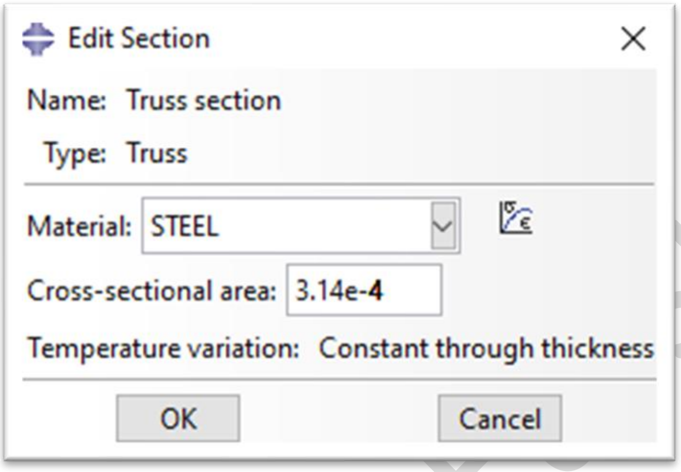
Module: Property⇛ Material→ Create →Name: STEEL;

Mechanical→ Elasticity→ Elastic→ Young's Modulus: 200e9; Poisson's ratio: 0.3→ OK



After creating the material, we must assign it to our part. So, first, create a section with the  tool and adjust the settings as shown in the figure. Set the area according to the radius of the Truss cross-section (10mm)





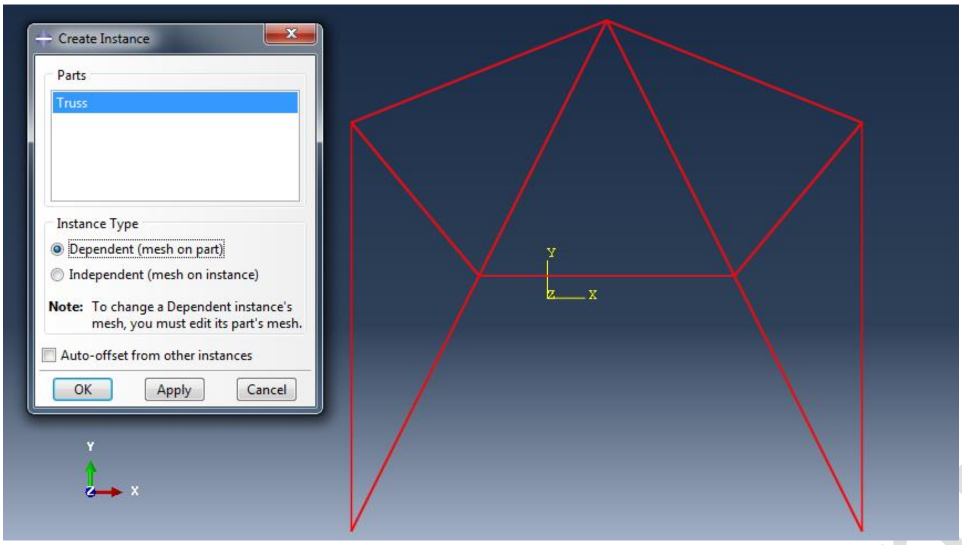
In this step, use tool to assign the cross-section area to the model, take note: we must select the entire section! Finally, in the prompt area, click Done.

Step 3: Assembling the model

We only need to create an instance of each part in this module. Create an instance with the tool and accept all default options.

(I will tell you the difference between these options later.)

Module: Assembly⇛ Instance→ Create→ Type: Dependent→ OK



Step 4: define solver type

The type of analysis will be specified in the Step module. As previously stated, the output results expected from the software analysis will be determined in this module.

Our analysis is static. As a result, we make the following changes:

Module: Step⇛ Step → Create→ Name: static; Procedure type: General→ Static, General→ Continue

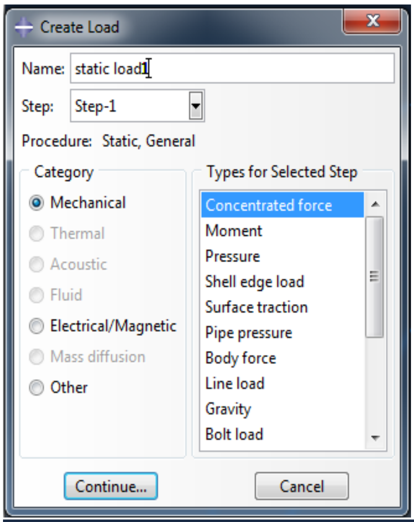
Also, activate the Nlgeom option. This option is for models with large deformations or when we don't know whether the deformation is large or small!

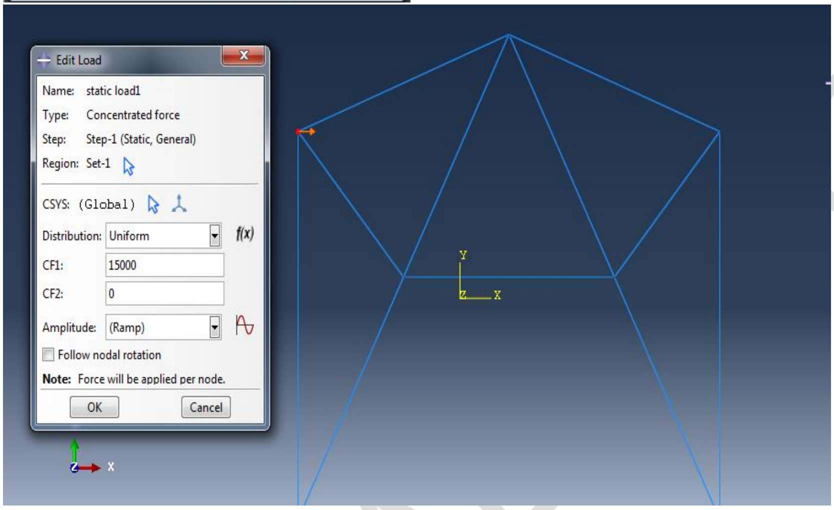
By the way, we don't change the output type and leave it as the default. But outputs can be organized by Field output  and History Output  tool.

Step 5: Specifying the boundary conditions and appling loads

It's true.! In the Load module, the load and boundary conditions should be applied to the truss. Follow me to put it into practice.

Module: Load ⇛ Load→ Create→ Name: static load1; Step: static; Mechanical→ Concentrated force→ Continue→ select point B → Done→
CF1: 15000→ OK





Apply the same force to point E with a magnitude of 40 KN.

Module: Load ⇛ Load→ Create→ Name: static load2; Step: static; Mechanical→ Concentrated force→ Continue→ select point E→ Done→
CF2: -40000→ OK

Finally, we must define the boundary condition in this module. There is pin support on the left, and on the right, there is roller support. To accomplish this, use  tool to create a boundary condition.

The path of boundary conditions is shown here!

Name: Fixed; Step: Initial; Category → Mechanical;

Types for Selected Step→ Displacement/Rotation→ Continue

Choose point F. Select U1 and U2 in the popup window to limit the movement of the selected degrees of freedom (X and Y-axis) to zero.

The measures for point G is as follows:

Name: Fixed2; Step: Initial; Category → Mechanical;

Types for Selected Step→ Displacement/Rotation→ Continue

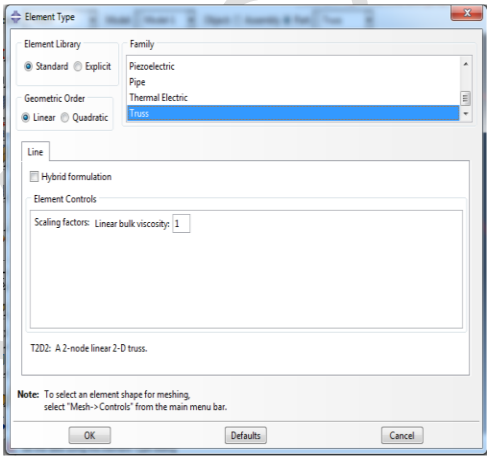
Choose the G point. Select U2 in the popup window. Then click OK to prevent the part from moving vertically. That's the roller support!

Step 6: Meshing

In this step, we mesh the model to be prepared for analysis.

Notes: First, the element type must be specified. The second is the element shape. Third, the number and method of meshing in various locations, and finally, do the meshing.

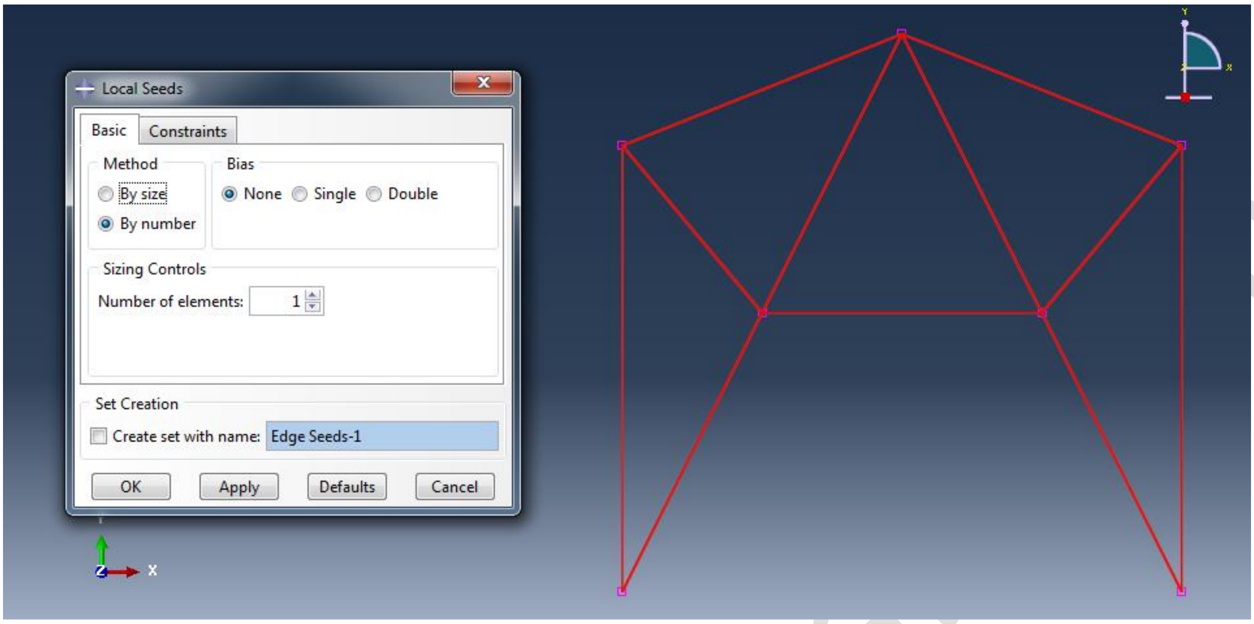
Using the tool, assign a specific element type to the model. Select Truss from the Family section, and the remaining settings are as follows.

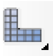


To select the shape of an element, we should select 'Assign Mesh Control' , but there is no need to use this tool because our element is a truss. (Don't overthink it; of course, the truss element has no choice but to be linear!!!)

Following that, the method for meshing should be specified. It should be noted that each truss member must be defined as one element.

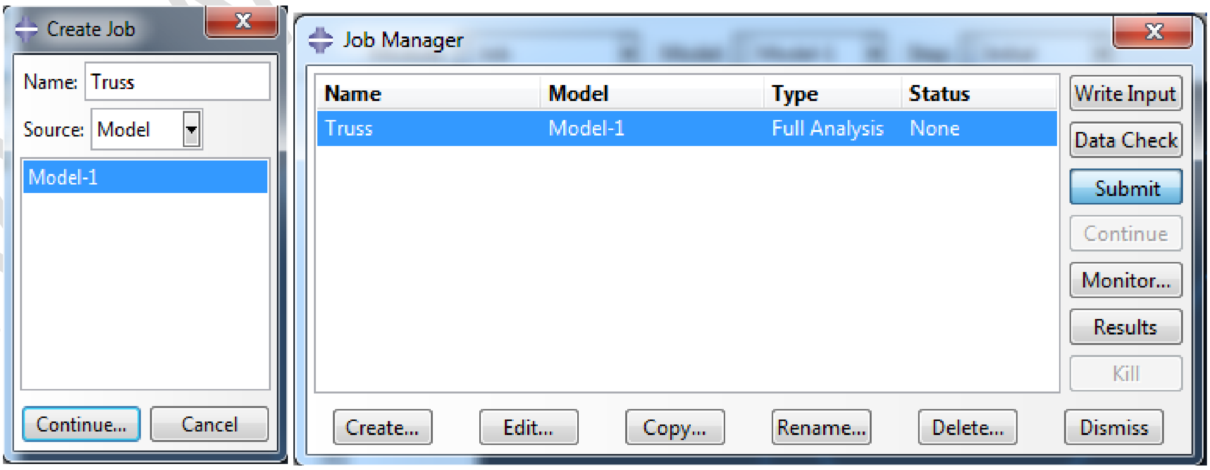
Set the Number of Element to 1 when using the Seed Edge tool.



Finally, the Mesh Part;  tool is used to mesh the data. Yes, the model's color has changed to blue!!

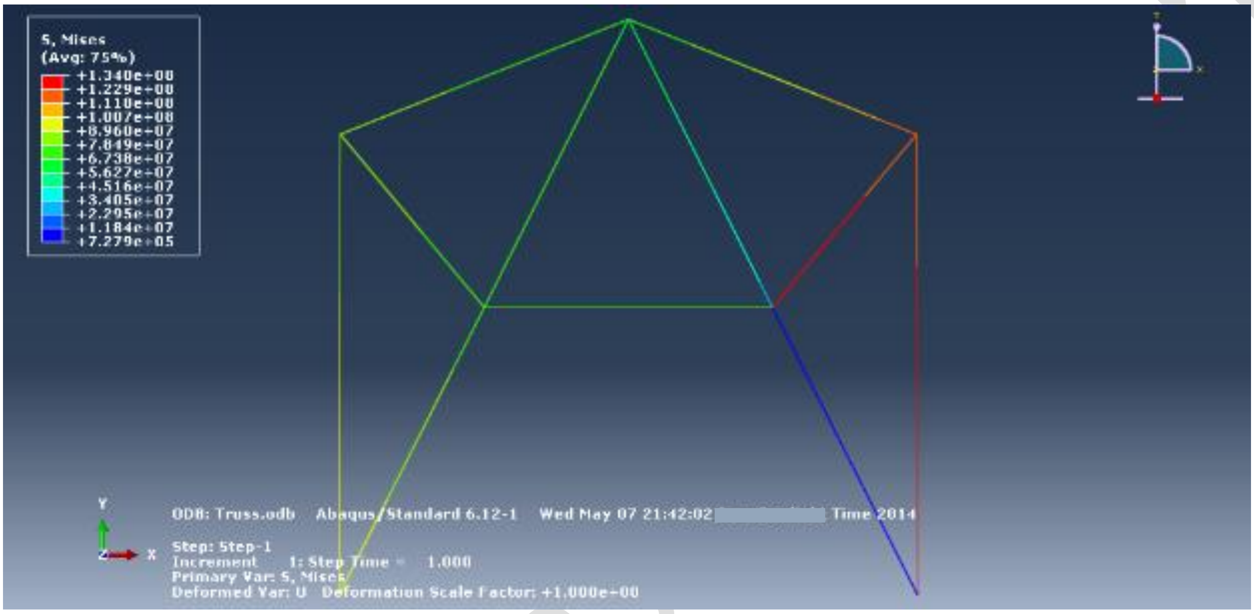
Step 7: Solving the problem

Create a Job with  tool in this module, name it Truss, and submit it. So the pre-processing step is finished, and the post-processing step begins!!

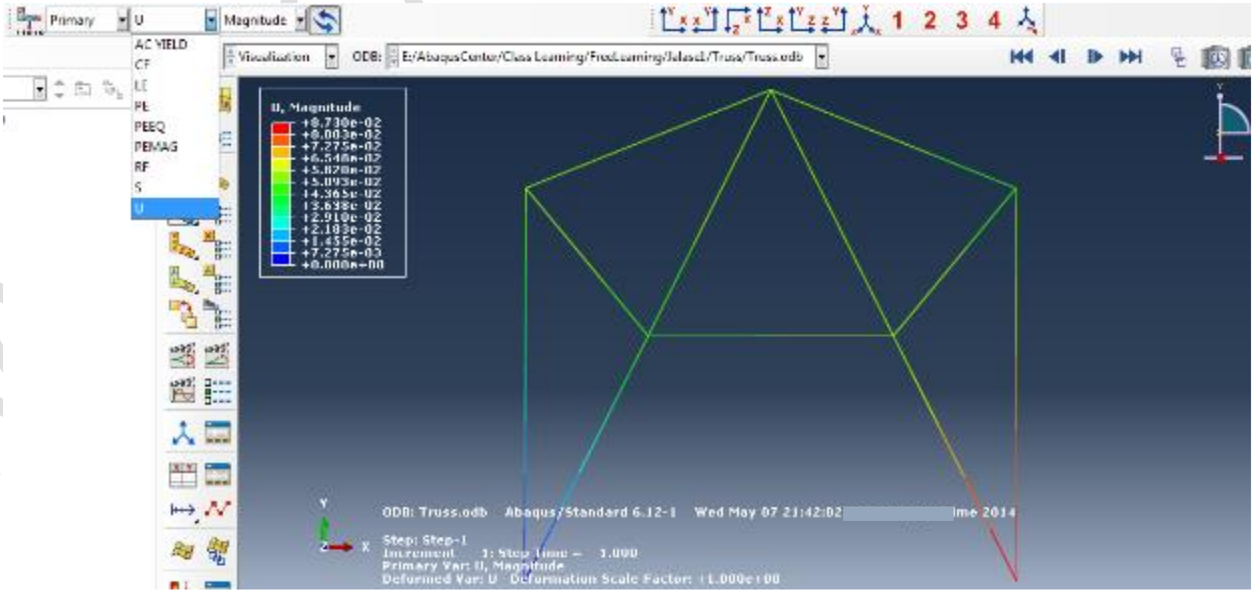


Step 8: Results

The results are available in the Visualization module. The stress contour is depicted below. As you can see, the maximum stress is located on the right side of the truss and has a value of 134 MPa.



We can also see the displacement by changing the toolbar to the U options.



+1 (740) 966-1814
+43 68181750465
Support@CAEassistant.com
Enviroflex GmbH, Sterngasse 3/2/6 1010, Vienna, Austria (Sales Representative)

Another output request for this problem was reaction forces, as shown by the path below.

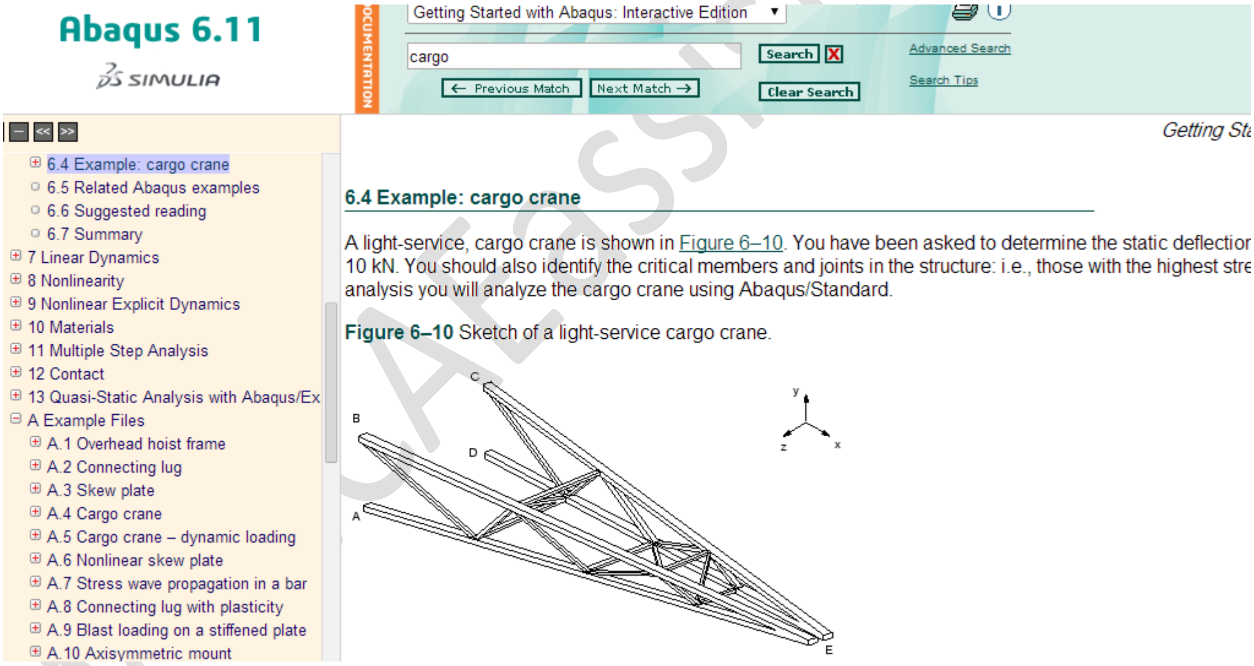
Module: Visualization ⇛ Result→ Filed Output→ Primary variable: RF

So, that's the end of the example. It's now up to you to do something nice for me!!!

Problem:

You can find a 3D Truss example on Abaqus documentation! Find that example and simply model it to familiarize yourself with the Abaqus documentation! It is not required to solve or mesh. (Do only the Creating Parts step; doing the rest is up to you; however, don't be upset if you do not know something and don't understand it! it's OK)

Tip for finding it: Pay attention to the image!!!



Thank you for being with us…

I'm here for you with other examples in the next session.

Have fun!!!

+1 (740) 966-1814
+43 68181750465
Support@CAEassistant.com
Enviroflex GmbH, Sterngasse 3/2/6 1010, Vienna, Austria (Sales Representative)