

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

CAEassistant.com



Support@CAEassistant.com





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...







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.







Part	Property	Assembly
• Create the part geometry (and regions for sections, if necessary)	 Define materials Define additional part regions Define and assign sections to parts or regions 	Position parts for initial configuration. Assembly is created automatically for models with only one part
Step	Interaction	Load
Define analysis steps and output requests Mesh	 Define contact and other interactions on regions or named sets, and assign them to steps in the analysis history 	• Apply loads, BCs, and ICs to regions or named sets; and assign them to steps in the analysis history
Split assembly into meshable regions and mesh		20

Processing:

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

💠 Job-1	Monitor		10				Ŀ	- C		
Job: Job	o-1 Status: Co	mpleted								
Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Ti		
12	1	1	1	4		12	1			
13	1	1	1	2	3	13	1		1	
14	1	1	1	2	3	14	1			
15	1	1	1	2	3	15	1		Job	
16	1	1	1	2	3	16	1	_		
•				111						
Log	Errors ! Warni	ngs Ou	itput Data	File Mess	age File	Status File			 Submit, manage, 	
Compl	eted: Abagus/St	andard							and monitor	
									analysis jobs	
Compl	eted: Thu Feb 21	12:11:40	2013							
Search Text										
Text to find: Match case $I Next r Previous$										
	Γ	Kill				Di	smiss			

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.

Image: Space Image: Space Image: Space Image: Space	
Discrete rigid Analytical rigid Lulerian	
Ovy1 A. Base Feature 20 Shell Shell 30 Wire Point	
Approximate size: 30	

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 it 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: $Property \Rightarrow Material \rightarrow Create \rightarrow Name: STEEL;$

CAEassistant.com



Support@CAEassistant.com





 $\begin{aligned} & \textit{Mechanical} \rightarrow \textit{Elasticity} \rightarrow \textit{Elastic} \rightarrow \textit{Young's Modulus: 200e9; Poisson's ratio: 0.3} \rightarrow \\ & \textit{OK} \end{aligned}$

🚔 Edit Material	×	
Name CTEL		
Description:		
	×	
Material Behaviors		
Elastic		
<u>General Mechanical Thermal Electrical/Magnetic Other</u>	· · · · · · · · · · · · · · · · · · ·	
Elastic		
Type: Isotropic	▼ Suboptions	
Use temperature-dependent data		
Number of field variables: 0		
Moduli time scale (for viscoelasticity): Long-term		
No compression		
No tension		
Data		
Young's Poisson's Modulus Ratio		
1 200e9		
ОК	Cancel	

After creating the material, we must assign it to our part. So, first, create a section with the $\stackrel{1}{\textcircled{}}$ tool and adjust the settings as shown in the figure. Set the area according to the radius of the Truss cross-section (10mm)







Create Section	×			
Name: Truss section				
Category Type				
O Solid Beam				
O Shell Truss				
Beam				
⊖ Fluid				
○ Other				
 Edit Section Name: Truss section Type: Truss 		×	x	
Material: STEEL Cross-sectional area: 3.14e- 4	V 12		S	
Temperature variation: Consta	nt through thic	kness		
OK	Cancel			

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 ^{LL} tool and accept all default options.

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

Module: Assembly \Rightarrow Instance \rightarrow Create \rightarrow Type: Dependent \rightarrow OK







Create Instance		
Instance Type Dependent (mesh on part) Independent (mesh on instance) Note: To change a Dependent instance's mesh, you must edit its part's mesh.	y k_x	
Auto-offset from other instances		

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: $Step \Rightarrow Step \rightarrow Create \rightarrow Name: static; Procedure type: General \rightarrow Static, General \rightarrow Continue$

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 in and History Output in 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 \Longrightarrow Load \rightarrow Create \rightarrow Name: static load1; Step: static; Mechanical \rightarrow Concentrated force \rightarrow Continue \rightarrow select point $B \rightarrow$ Done \rightarrow CF1: 15000 \rightarrow OK









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

Module: Load \Rightarrow Load \rightarrow Create \rightarrow Name: static load2; Step: static; Mechanical \rightarrow Concentrated force \rightarrow Continue \rightarrow select point $E \rightarrow$ Done \rightarrow CF2: -40000 \rightarrow 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 \rightarrow Mechanical; Types for Selected Step \rightarrow Displacement/Rotation \rightarrow Continue

CAEassistant.com



Support@CAEassistant.com





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 \rightarrow Mechanical; Types for Selected Step \rightarrow Displacement/Rotation \rightarrow 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.

Element Library	Family	
Standard Explicit	Piezoelectric	
	Pipe	
Geometric Order Linear Quadratic	Thermal Electric Truss	
line		
Hybrid formulation		
Element Controls		
Configuration and a second second	and the second	
Scaling factors: Linear	bulk viscosity: 1	
Scaling factors: Linear	bulk viscosity: 1	
Scaling factors: Linear	bulk viscosity: 1	
Scaling factors: Linear	bulk viscosity: 1	
Scaling factors: Linear	bulk viscosity: 1	
Scaling factors: Linear T2D2: A 2-node linear 2	-D truss.	
Scaling factors: Linear	-D truss.	
Scaling factors: Linear T2D2: A 2-node linear 2 ote: To select an element select "Meth-2 Cont	-D truss.	

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.







Basic Constraints	↓ +
Method Bias By size None Single Double By number	
Sizing Controls Number of elements: 1	
Set Creation Create set with name: Edge Seeds-1	
OK Apply Defaults Cancel	λ

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 preprocessing step is finished, and the post-processing step begins!!

ſ	Create Job	💠 Job Manage	r	mari R	The Local	×
	Name: Truss	Name	Model	Туре	Status	Write Input
	Source: Model	Truss	Model-1	Full Analysis	None	Data Check
	Model-1					Submit
						Continue
						Monitor
						Results
						Kill
	Continue Cancel	Create	Edit Copy	Rename	Delete	Dismiss





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.









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

Module: Visualization \Rightarrow *Result* \rightarrow *Filed Output* \rightarrow *Primary variable: RF*

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



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!!!



