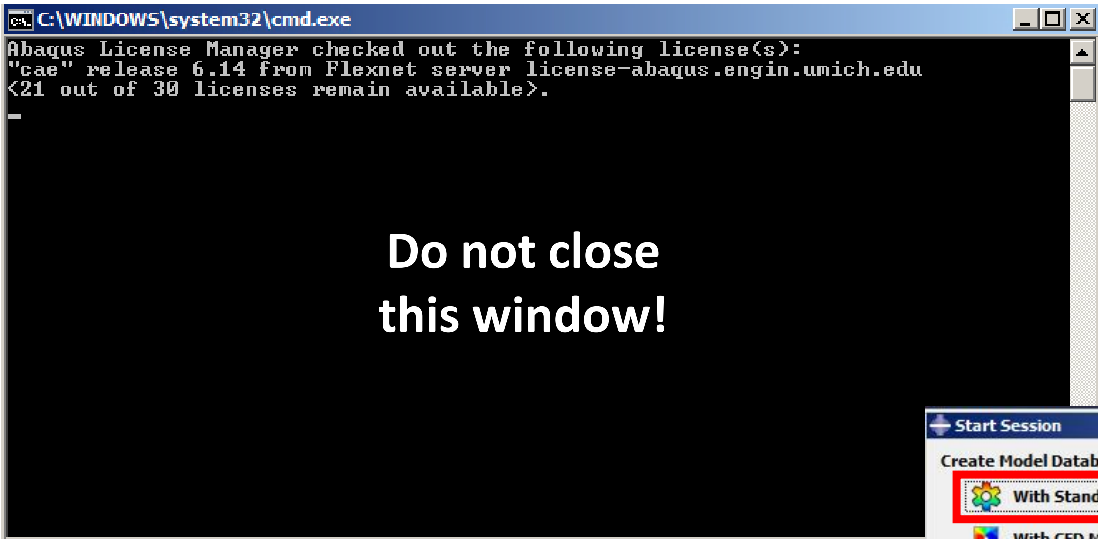
ASEE Introduction to Abaqus Workshop

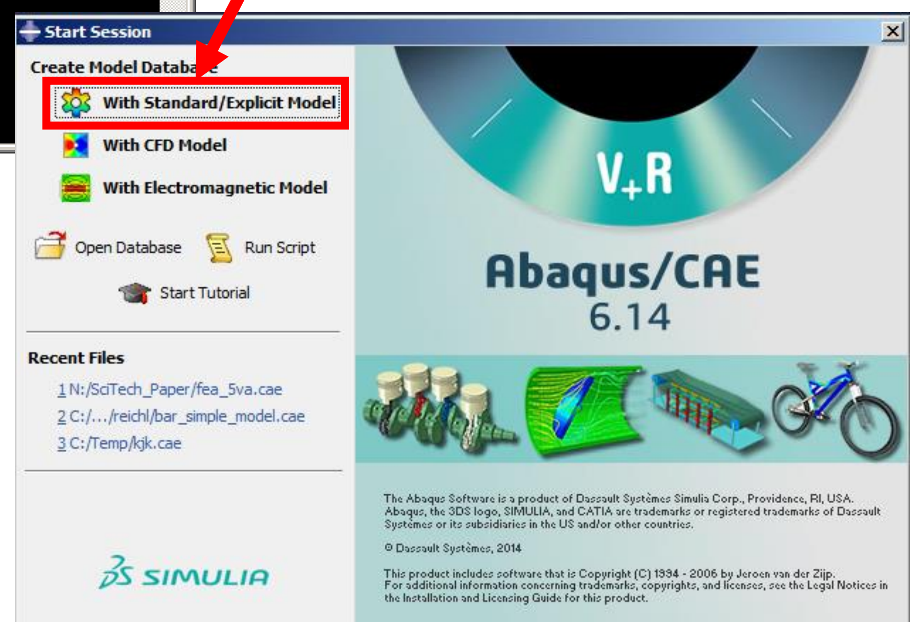


What to do to get started

Open Abaqus 6.14

You can close the 3D mouse window

Choose this option

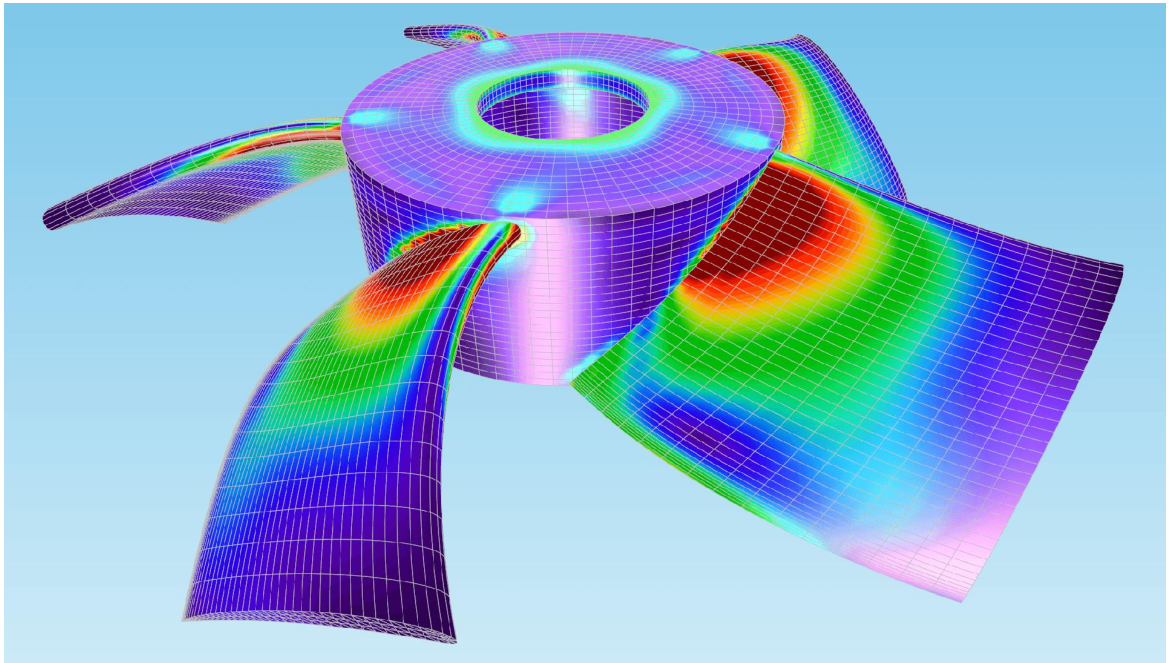


What we will cover today

- What is Finite Element Analysis
- What can we use Finite Element Analysis for
- Overview of how to navigate Abaqus
- Overview of the basic procedure for performing Finite Element Analysis
- Specific instructions for performing static structural analysis

What is the Finite Element Method?

The finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. FEM subdivides a large problem into smaller, simpler, parts, called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. FEM then uses variational methods from the calculus of variations to approximate a solution by minimizing an associated error function.

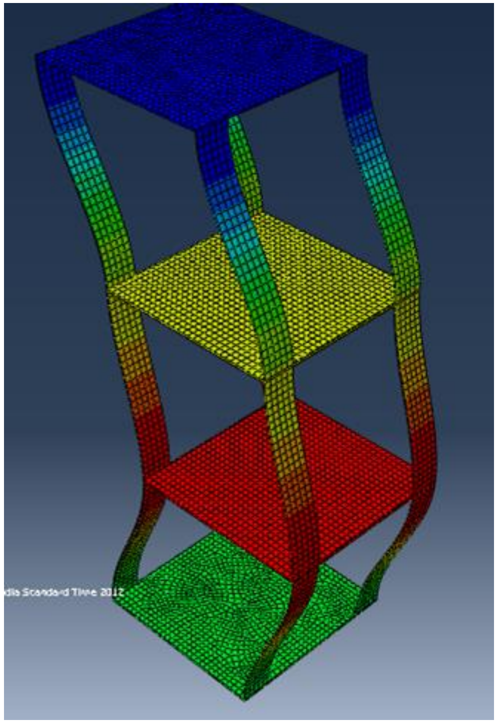


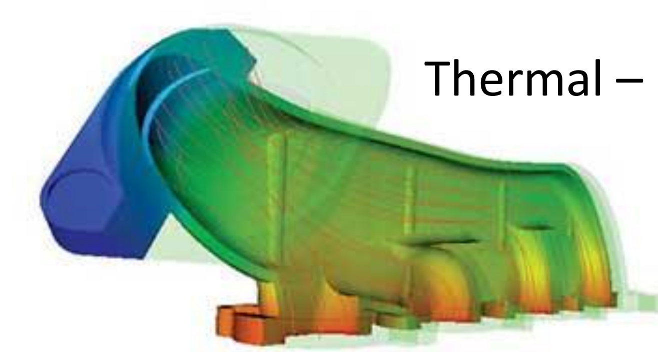
Definitions

Elements: smaller simpler parts in with equations are solved over
Nodes: corners of the elements
Mesh: pattern in which you subdivide your problem into

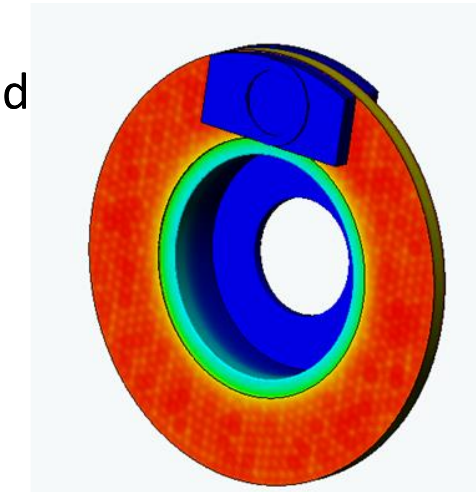
What can you do with FEA?

Modal Analysis

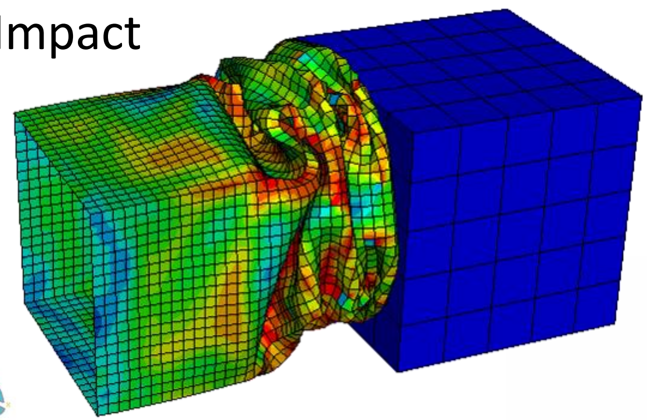




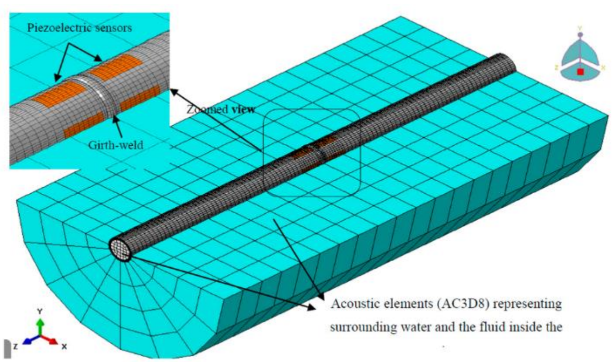
Thermal – Fluid



Impact



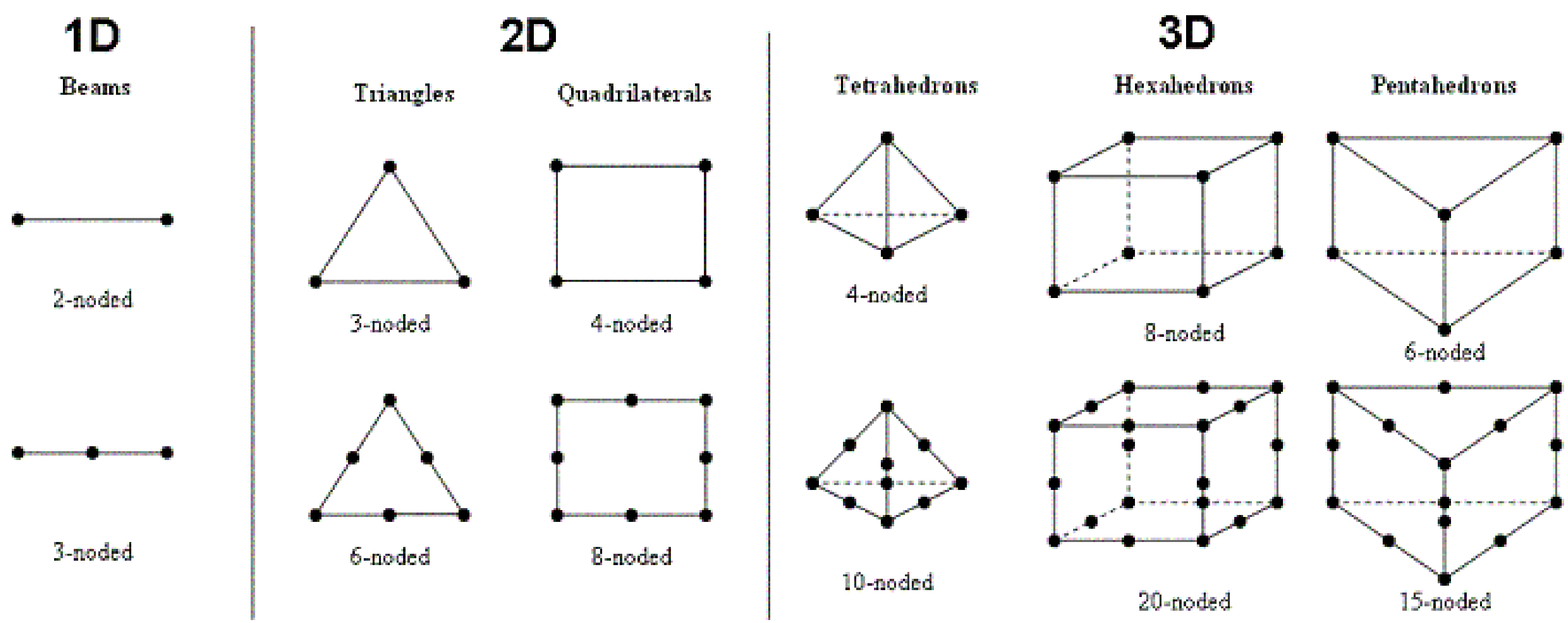
Thermal – Mechanical



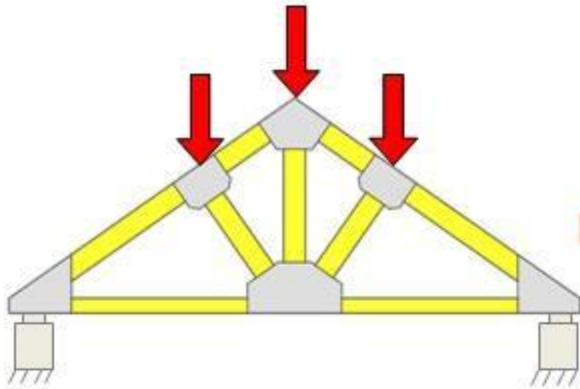
Composites - Dynamic

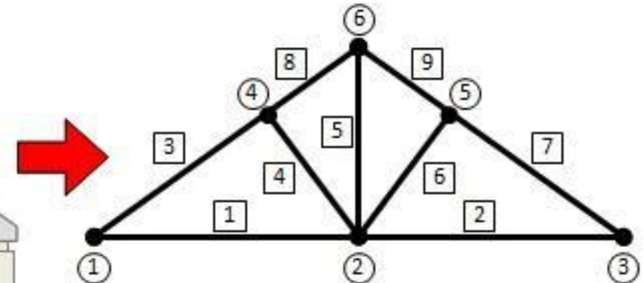
Peizoelectric - Acoustic

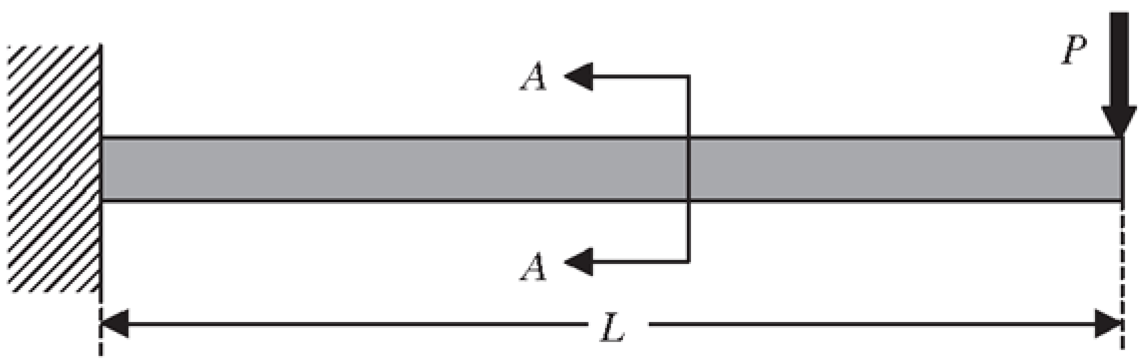
Element Types

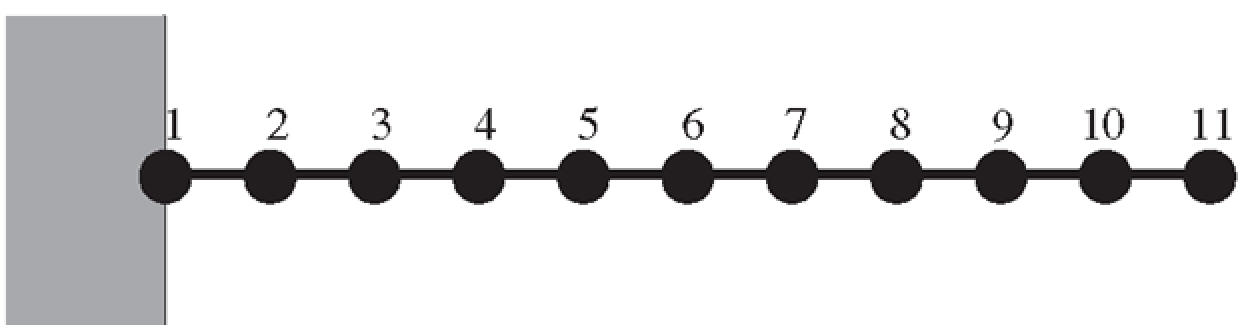


Simplifying Models

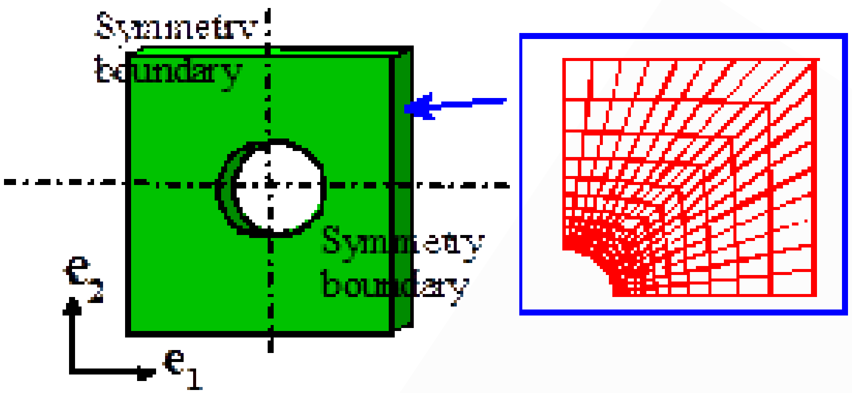


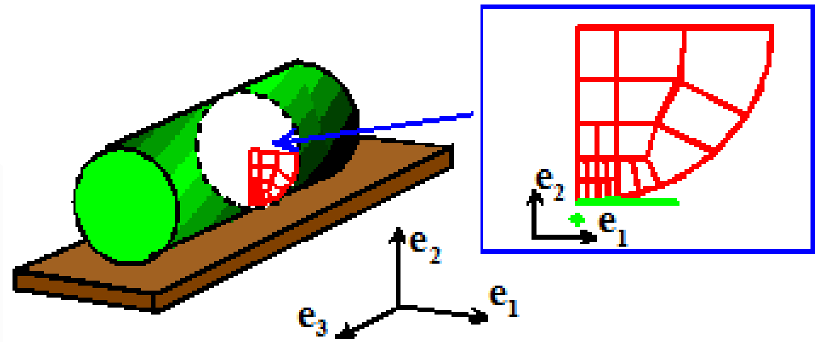




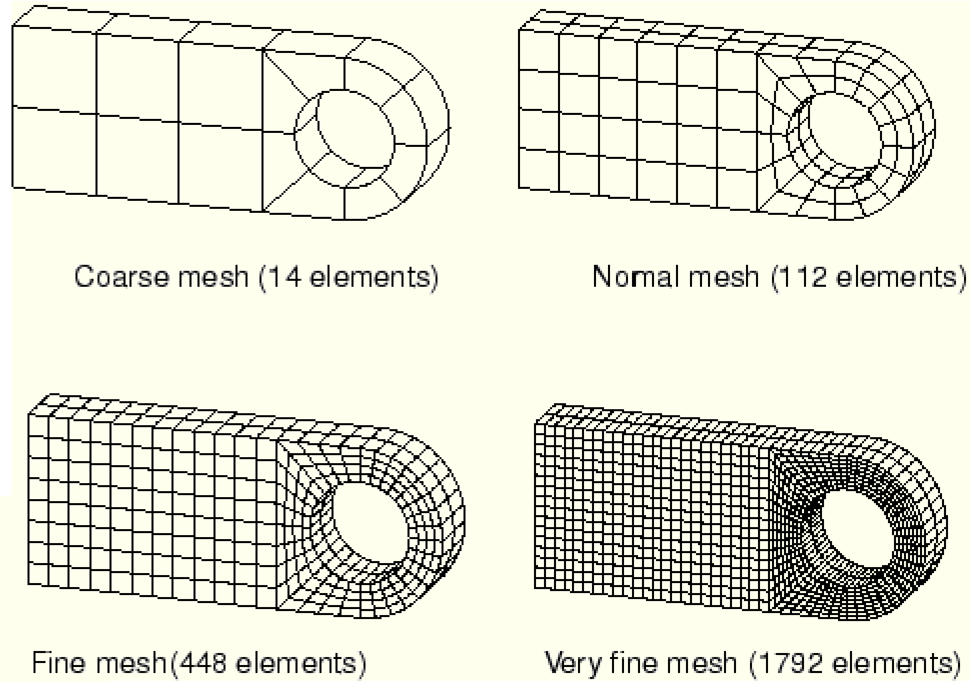


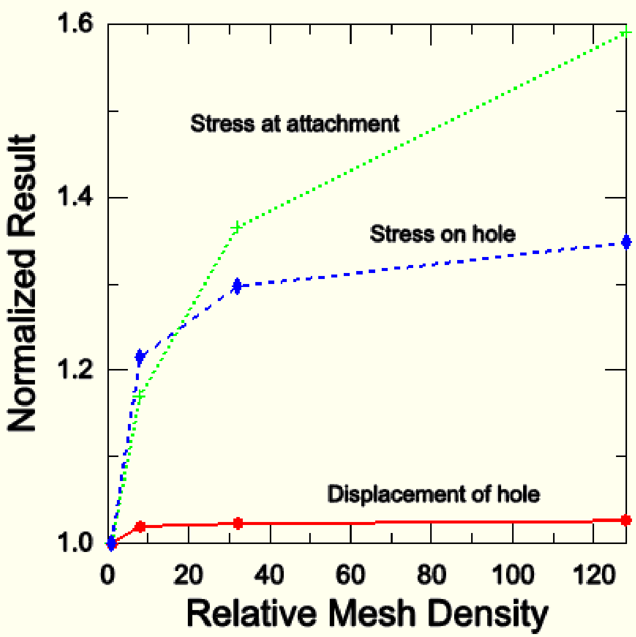
Simplifying Models

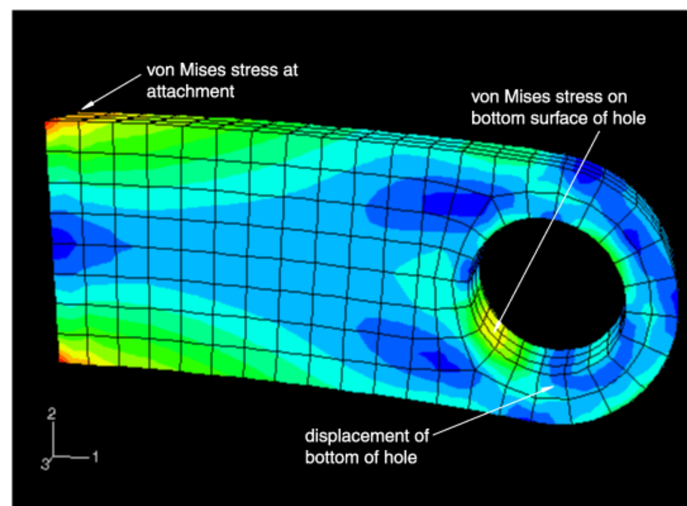




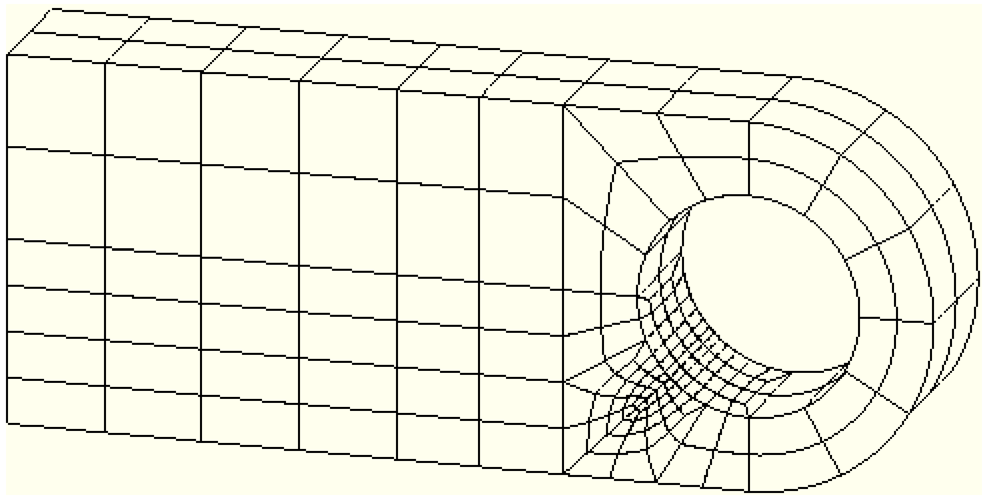
Meshes

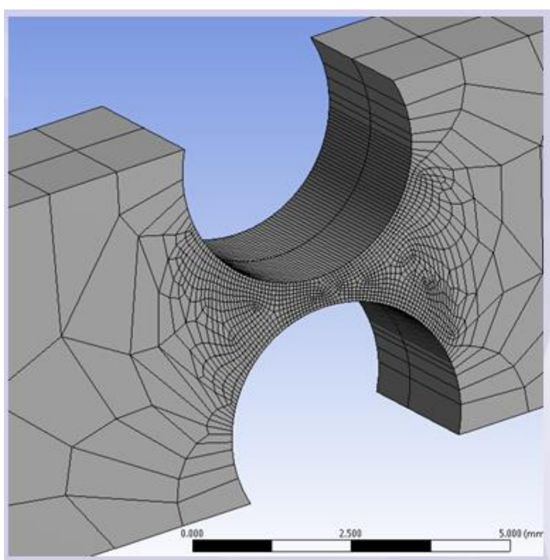

Coarse mesh (14 elements)
Nomal mesh (112 elements)
Fine mesh(448 elements)
Very fine mesh (1792 elements)

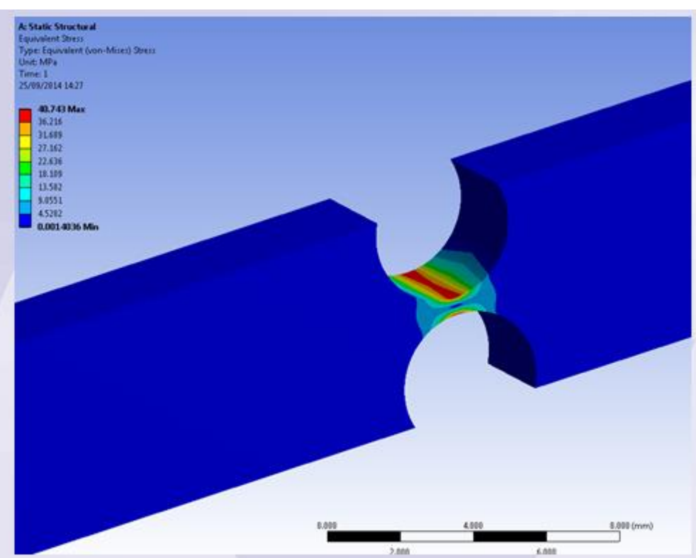

1.6
1.4
1.2
1.0
0
20
40
60
80
100
120
Normalized Result
Relative Mesh Density
Stress at attachment
Stress on hole
Displacement of hole

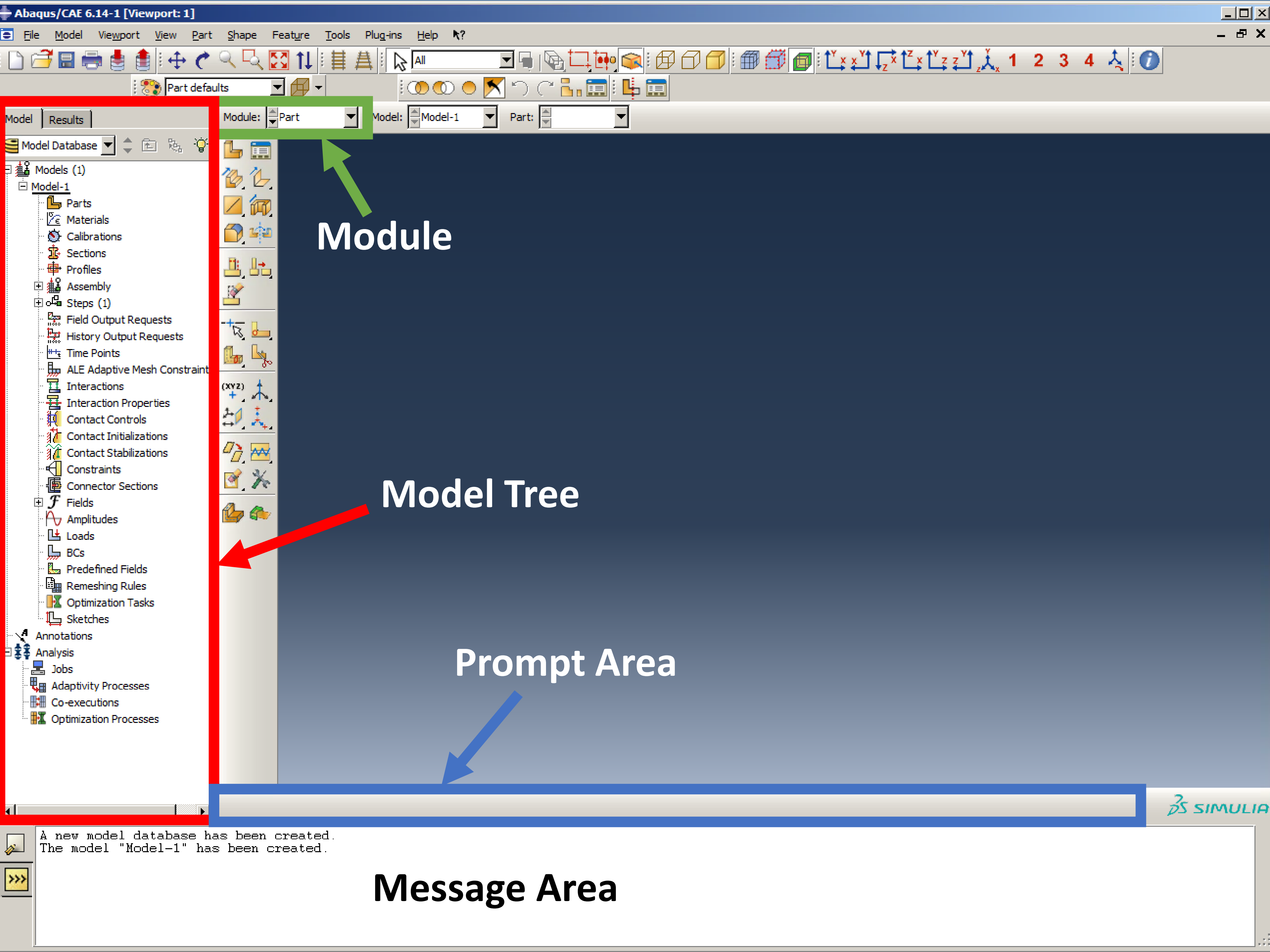

von Mises stress at attachment
von Mises stress on bottom surface of hole
displacement of bottom of hole

Meshes



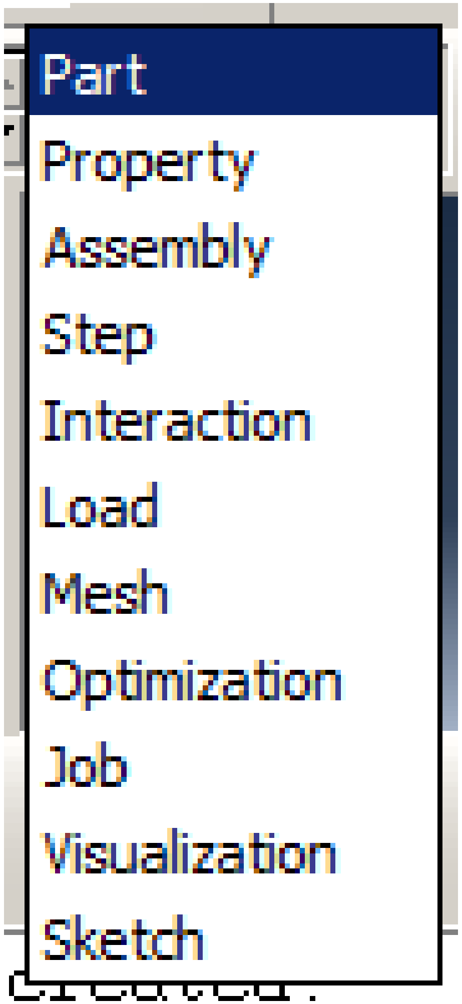

0.000
2.500
5.000 (mm


A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1
25/09/2014 14:27
40.743 Max
36.216
31.689
27.162
22.636
18.109
13.582
9.0551
4.5282
0.0014036 Min
0.000
4.000
8.000 (mm)
2.000
6.000


Abaqus/CAE 6.14-1 [Viewport: 1]
File Model Viewport View Part Shape Feature Tools Plug-ins Help
Part defaults
Module: Part
Model: Model-1
Part:
Model Results
Model Database
Models (1)
Model-1
Parts
Materials
Calibrations
Sections
Profiles
Assembly
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraint
Interactions
Interaction Properties
Contact Controls
Contact Initializations
Contact Stabilizations
Constraints
Connector Sections
Fields
Amplitudes
Loads
BCs
Predefined Fields
Remeshing Rules
Optimization Tasks
Sketches
Annotations
Analysis
Jobs
Adaptivity Processes
Co-executions
Optimization Processes
Module
Model Tree
Prompt Area
SIMULIA
A new model database has been created.
The model "Model-1" has been created.
Message Area

Detailed Procedure

- Part: Create the Part
- Property
  - Create the Materials
  - Create the Section
  - Assign the Section to the Part
- Assembly: Create the Assembly
- Step: Define the Analysis Type
- Interaction: Define how different parts interact within an assembly (if there is only one part this will be skipped)
- Load
  - Define Loads
  - Define Boundary Conditions
- Mesh: Create the Mesh
- Job: Run the analysis
- Visualization: View the results



Units in Abaqus

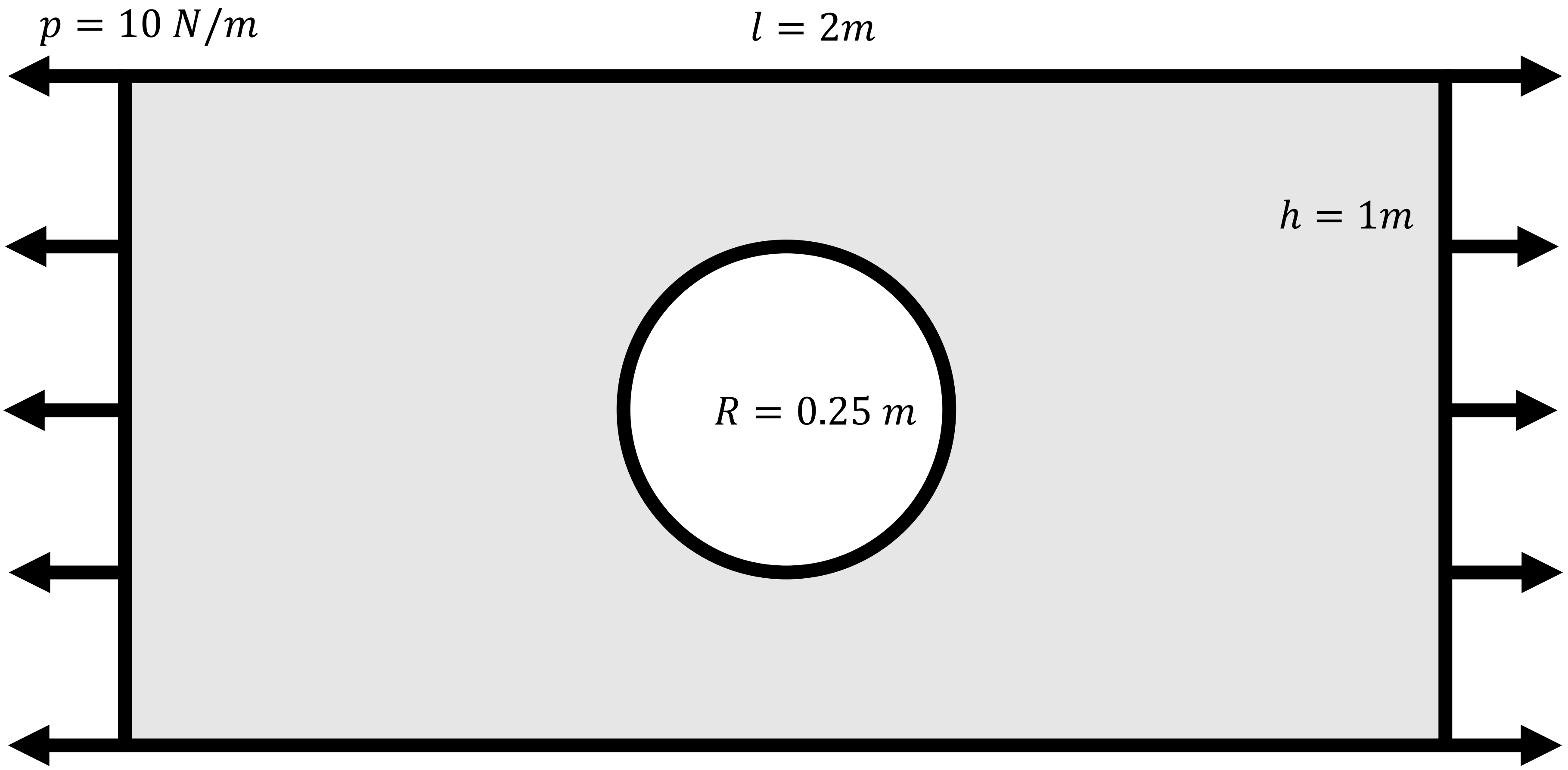
Abaqus does not have units, therefore you must use consistent units

Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne (10^3 kg)	slug	lbf s^2/in
Time	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3} J)	ft lbf	in lbf
Density	kg/m^3	tonne/mm^3	slug/ft^3	lbf s^2/in^4

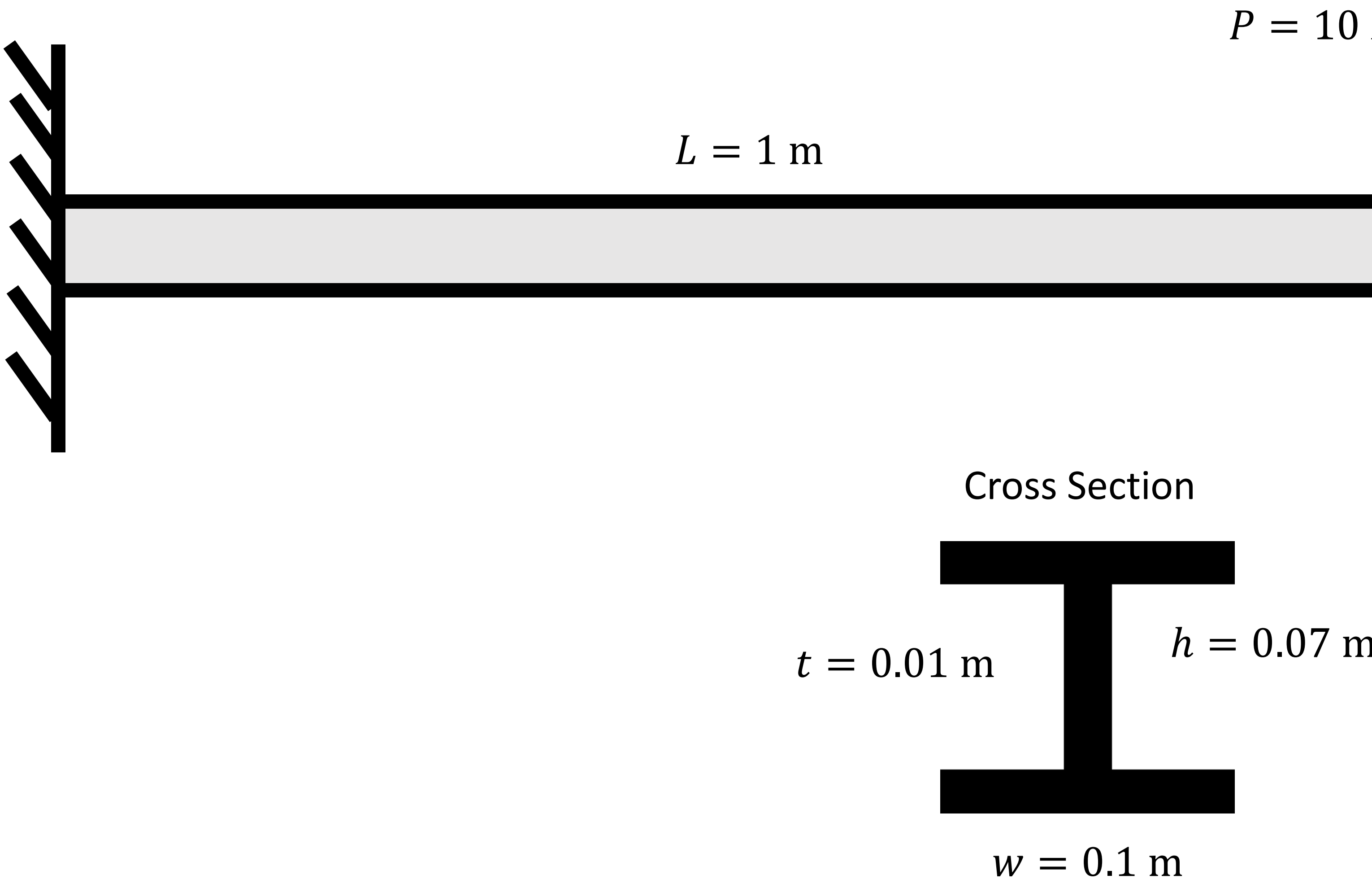
Other Abaqus Tips

- Files
  - .CAE is a file that contains everything needed to run the analysis
  - .ODB is the file created while the analysis is being run and contains the results of the simulation
- View the part
  - Ctrl + Alt + Left Button: Rotate Part
  - Ctrl + Alt + Center/Scroll Button: Translate Part

Example #1



Example #2



References

Slide 3

https://en.wikipedia.org/wiki/Finite_element_method

https://www.comsol.com/multiphysics/fea-software

Slide 4

http://www.creocommunity.com/print_article.php?cpfeatureid=8763

http://www.cerom.lsu.edu/serhan.htm

http://www.mdpi.com/1424-8220/14/9/17174/htm

https://www.simuleon.com/simulia-abaqus/abaqus-explicit/

http://www.iitrpr.ac.in/sites/default/files/navin_web/gallery.html

Slide 5

http://illustrations.marin.ntnu.no/structures/analysis/FEM/theory/index.html

Slide 6

http://article.sapub.org/10.5923.j.mechanics.20130305.02.html

http://what-when-how.com/the-finite-element-method/using-abaqus-finite-element-method-part-1/

Slide 7

http://solidmechanics.org/Text/Chapter7_1/Chapter7_1.php

http://plmsource.industrysoftware.automation.siemens.com/blog/2013/03/plysim-automates-composite-fe-modeling-reduces-design-time/

Slide 8

http://www.egr.msu.edu/software/abaqus/Documentation/docs/v6.7/books/gsa/default.htm?startat=ch04s04.html

Slide 9

http://www.egr.msu.edu/software/abaqus/Documentation/docs/v6.7/books/gsa/default.htm?startat=ch04s04.html

http://confluence.diamond.ac.uk/display/EDG/Design+of+flexural+hinges

Slide 12

https://www.researchgate.net/post/can_someone_help_me_with_on_Abaqus_CAE_data_and_measurement_units