ABAQUS (50 h)

UPenAUTO : 2017-1-ES01-KA203-03858

Erasmus+

UPenAUTO: UPSKILLED ENGINEERING PROFILES FOR THE AUTOMOTIVE SECTOR AND UPTAKE OF ALTERNATE SCHEME IN EUROPE´S HIGHER EDUCATION.

Gestamp

GTI
GESTAMP TECHNOLOGY INSTITUTE

FABRIKAZIOAREN ZENTRO AURRERATUA
CENTRO AVANZADO DE FABRICACIÓN

somorrostro
centro de formación / irakas-gunea

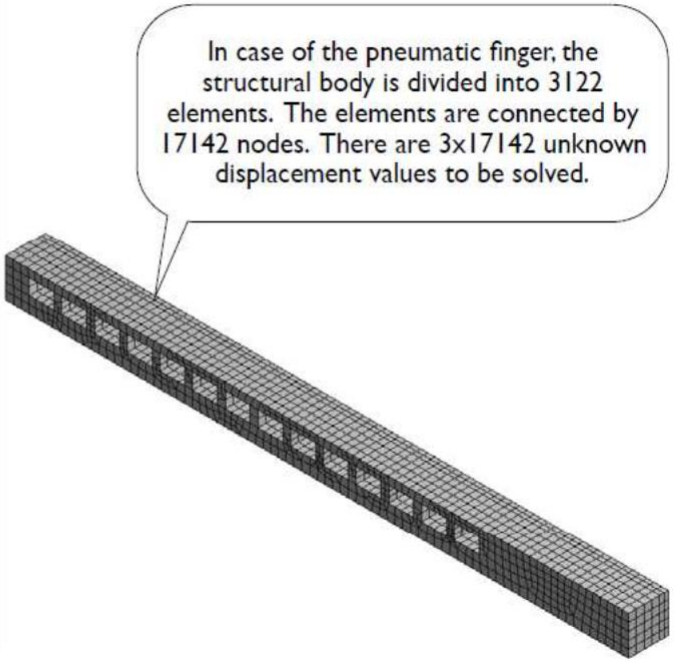
UNIVERSITATEA POLITEHNICA

Finite Element Methods: A Conceptual Introduction

Basic Ideas

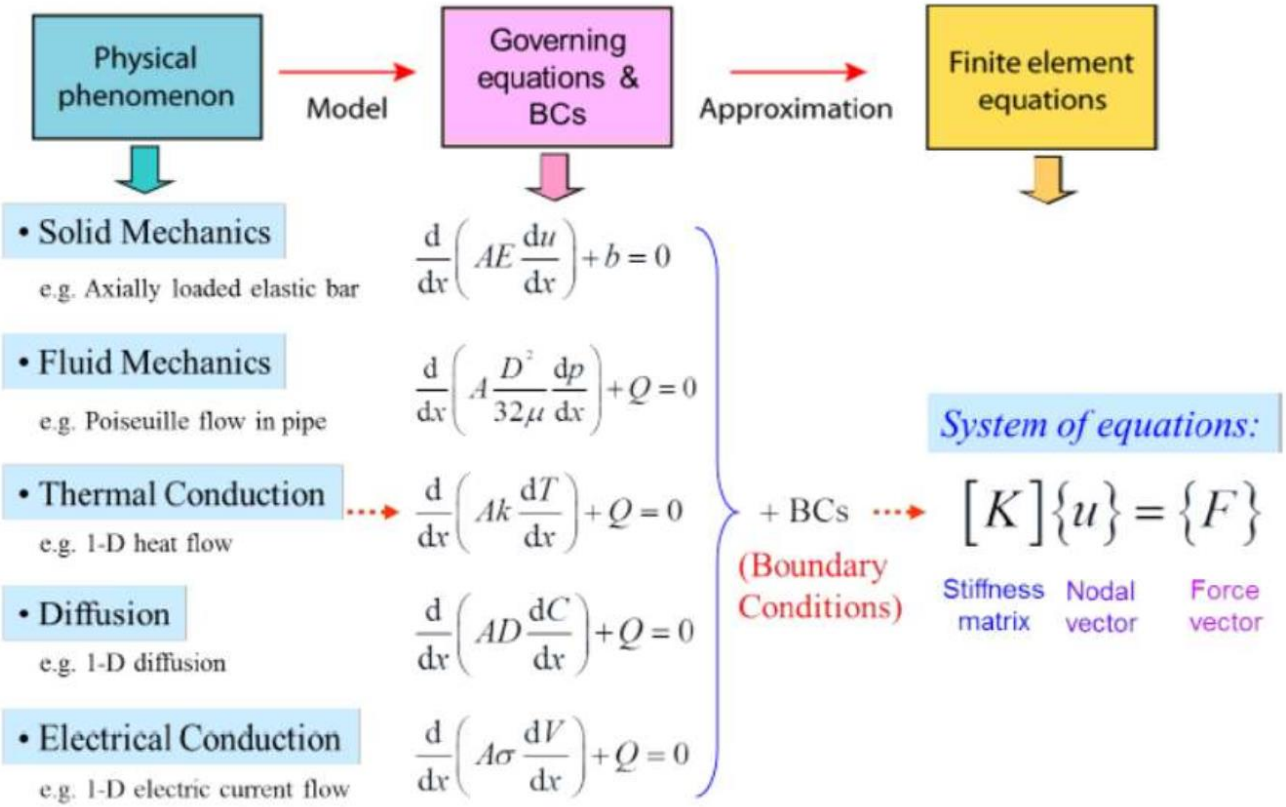
- A basic idea of finite element methods is to divide the structural body into small and geometrically simple bodies, called *elements*, so that equilibrium equations of each element can be written, and all the equilibrium equations are solved simultaneously
- The elements are assumed to be connected by *nodes* located on the elements' edges and vertices.

- Another idea is to solve unknown discrete values (displacements at the nodes) rather than to solve unknown functions (displacement fields).
- Since the displacement on each node is a vector and has three components (in 3D cases), the number of total unknown quantities to be solved is three times the number of nodes.
- The nodal displacement components are called the *degrees of freedom* (DOF's) of the structure.



Introduction

The finite element method (FEM) is a numerical technique for solving a wide range of complex physical phenomena, particularly those exhibiting geometrical and material non-linearities (such as those that are often encountered in the physical and engineering sciences). These problems can be structural in nature, thermal (or thermo-mechanical), electrical, magnetic, acoustic etc. plus any combination of. It is used most frequently to tackle problems that aren't readily amenable to analytical treatments.



- In static cases, the system of equilibrium equations has following form:

$$[K]\{D\}=\{F\}$$

- The displacement vector $\{D\}$ contains displacements of all degrees of freedom.
- The force vector $\{F\}$ contains forces acting on all degrees of freedom.
- The matrix $[K]$ is called the *stiffness matrix* of the structure. In a special case when the structure is a spring, $\{F\}$ as external force, and $\{D\}$ as the deformation of the spring, then $[K]$ is the spring constant.

Basic Procedure of Finite Element Method

1. Given the bodies' geometries, material properties, support conditions, and loading conditions.
2. Divide the bodies into elements.
3. Establish the equilibrium equation: $[K]\{D\} = \{F\}$

   3.1 Construct the $[K]$ matrix, according to the elements' geometries and the material properties.

   3.2 Most of components in $\{F\}$ can be calculated, according to the loading conditions.

   3.3 Most of components in $\{D\}$ are unknown. Some component, however, are known, according to the support conditions.

   3.4 The total number of unknowns in $\{D\}$ and $\{F\}$ should be equal to the total number of degrees of freedom of the structure.

4. Solve the equilibrium equation. Now, the nodal displacements {d} of each element are known.
5. For each element:

   5.1 Calculate displacement fields $\{u\}$, using an interpolating method, $\{u\} = [N]\{d\}$. The interpolating functions in $[N]$ are called the *shape functions*.

   5.2 Calculate strain fields according to the strain-displacement relations.

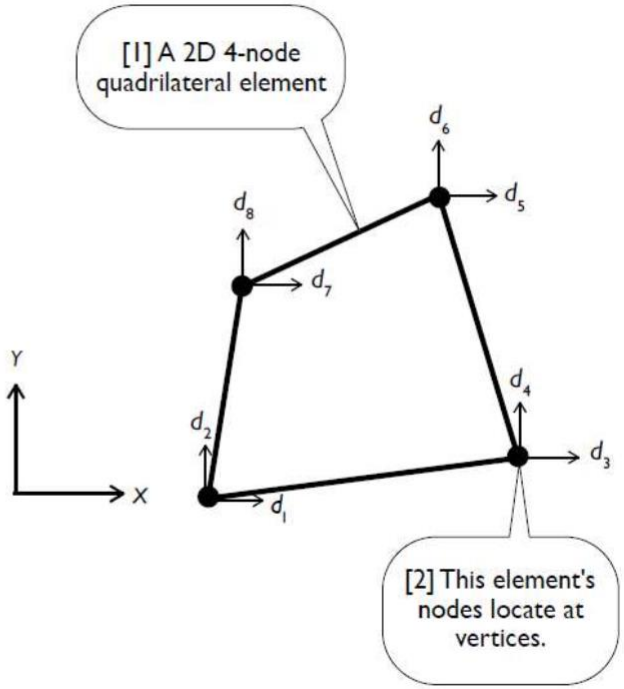
   5.3 Calculate stress fields according to the stress-strain relations (Hooke's law).

Shape Functions

- Shape functions serve as interpolating functions, allowing the calculation of displacement fields (functions of *X*, *Y*, *Z*) from nodal displacements (discrete values).

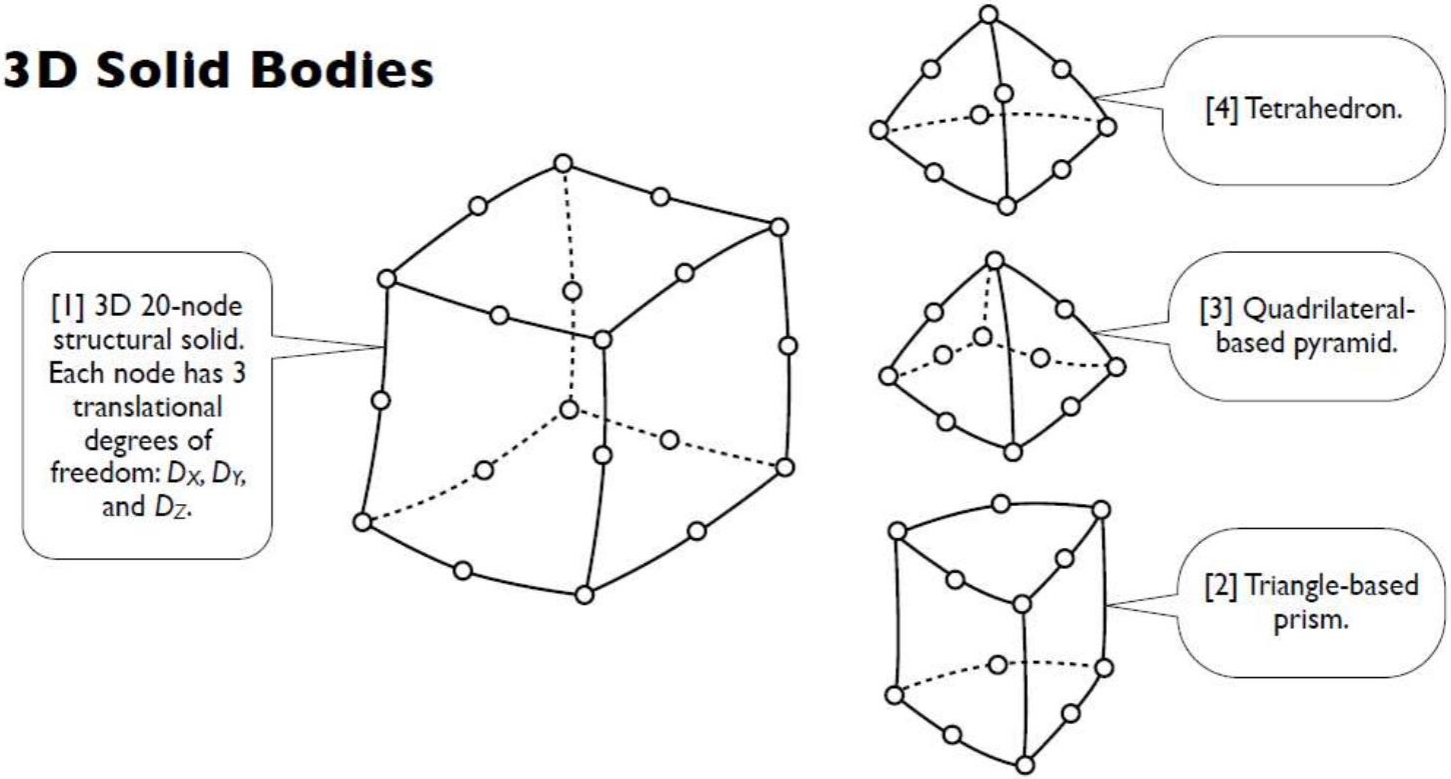
$$\{u\} = [N]\{d\}$$

- For elements with nodes at vertices, the interpolation must be linear and thus the shape functions are linear (of *X*, *Y*, *Z*).

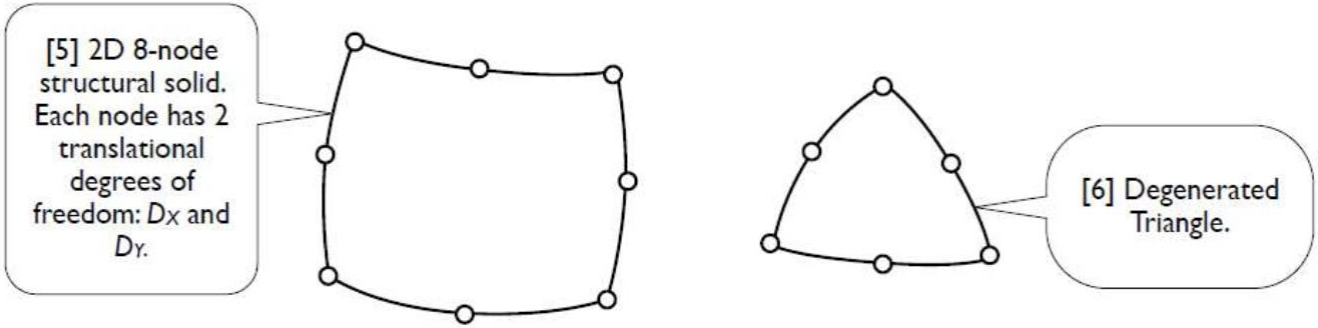


- For elements with nodes at vertices as well as at middles of edges, the interpolation must be quadratic and thus the shape functions are quadratic (of X, Y, Z).
- Elements with linear shape functions are called *linear elements*, *first-order elements*, or *lower-order elements*.
- Elements with quadratic shape functions are called *quadratic elements*, *second-order elements*, or *higher-order elements*.

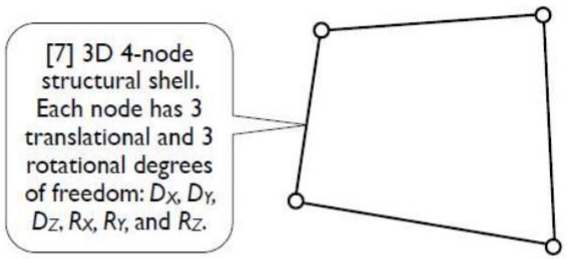
3D Solid Bodies

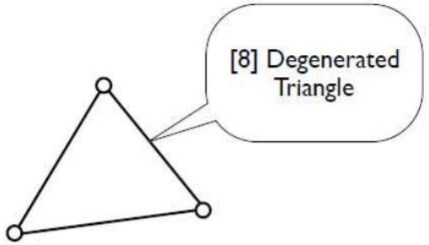


2D Solid Bodies

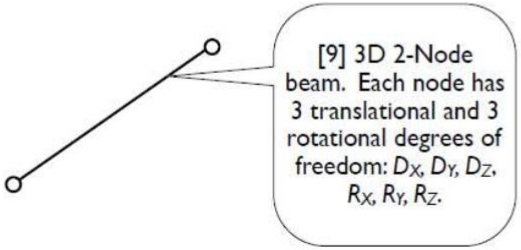


3D Surface Bodies





3D Line Bodies

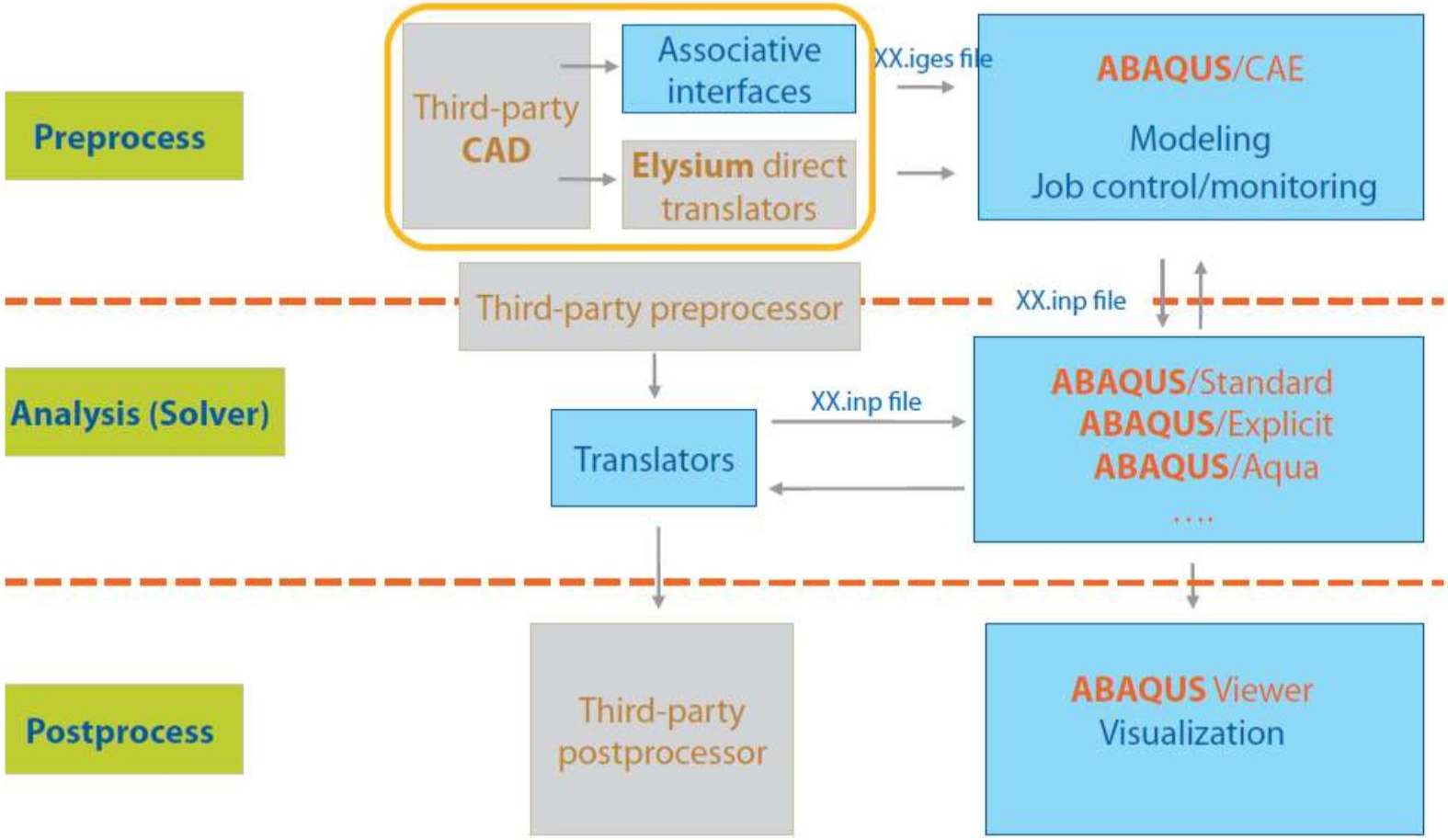


Applications of Finite Element Method

Structural Problem	Non-structural Problem
Stress Analysis - truss & frame analysis - stress concentrated problem Buckling problem Vibration Analysis Impact Problem	Heat Transfer Fluid Mechanics Electric or Magnetic Potential

Introduction to ABAQUS

ABAQUS Ecosystem



ABAQUS Workflow

Preprocessing

ABAQUS/CAE
Modules:
Part, Property, Assemble, Step, Interaction, Load, Mesh

inp file

Simulation

ABAQUS/Standard
ABAQUS/Explicit
Module:
Job

odb, fil, dat, res files

Postprocessing

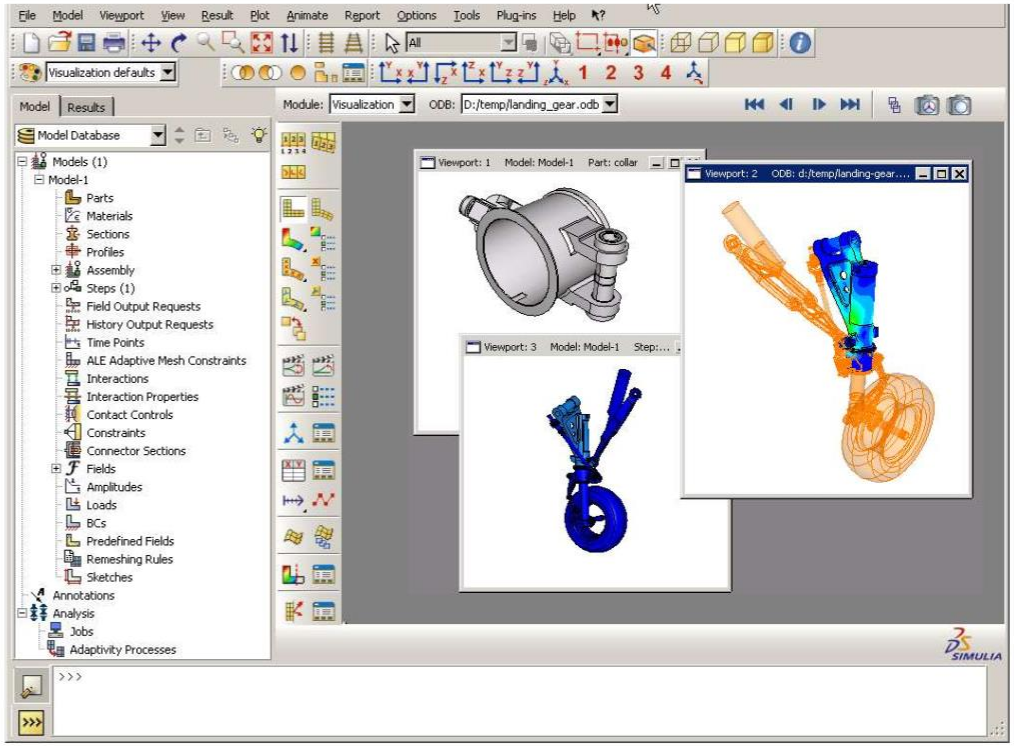
ABAQUS/CAE
Module:
Visualization

What is Abaqus FEA?

- **The structural analysis "solver" modules, Abaqus/Standard and Abaqus/Explicit, are complementary and integrated analysis tools.***
  - Abaqus/Standard is a general-purpose finite element module. It provides a large number of capabilities for analyzing many different types of problems, including many nonstructural applications.
  - Abaqus/Explicit is an explicit dynamics finite element module.
  - Abaqus/CAE incorporates the analysis modules into a **C**omplete **Abaqus E**nvironment for modeling, managing, and monitoring Abaqus analyses and visualizing results.

Overview of Abaqus/CAE

- Integrates modeling, analysis, job management, and results evaluation seamlessly.
- Provides the most complete interface with the Abaqus solver programs available.
- Uses neutral database files that are machine independent.
- Can be customized to create application-specific systems.

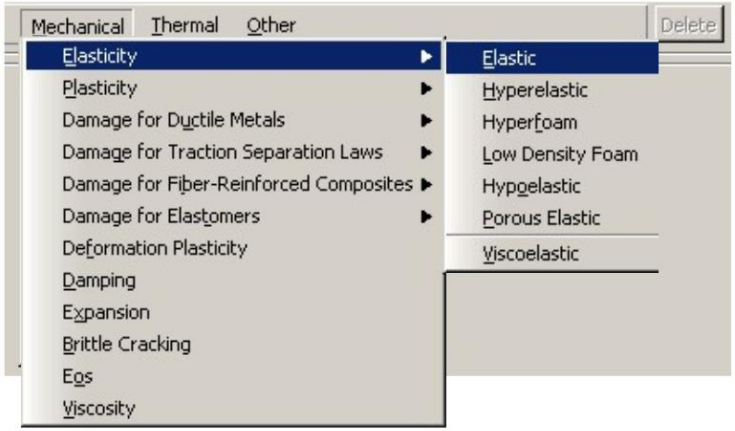


Abaqus/CAE main user interface

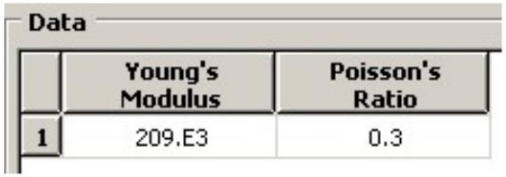
Overview of Abaqus/CAE

- **Modern graphical user interface (GUI) of menus, icons, and dialog boxes**
  - Menus provide access to all capabilities.
  - Icons accelerate access to frequently used features.
  - Dialog boxes allow you to input alphanumeric information and to select various options.

Visualization toolbox icons



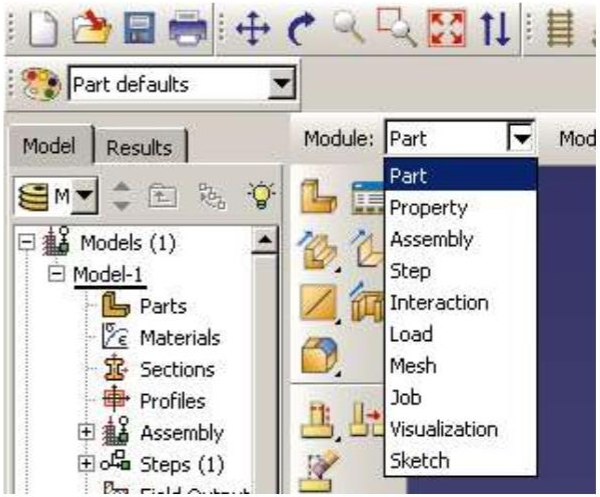
Mechanical property submenu



Elastic material form

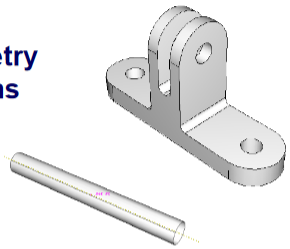
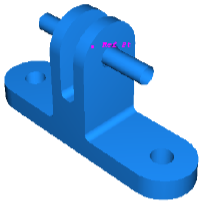
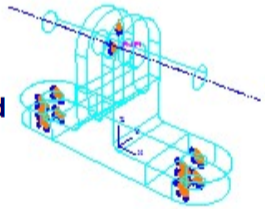
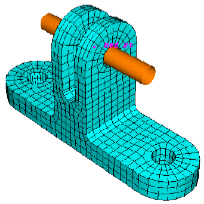
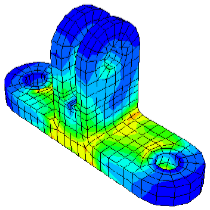
Overview of Abaqus/CAE

- **Consistent environment**
  - Functionality is presented in modules.
  - Each module contains a logical subset of the overall functionality.
  - Once you understand the presentation of one module, you can easily understand the presentation of the other modules.



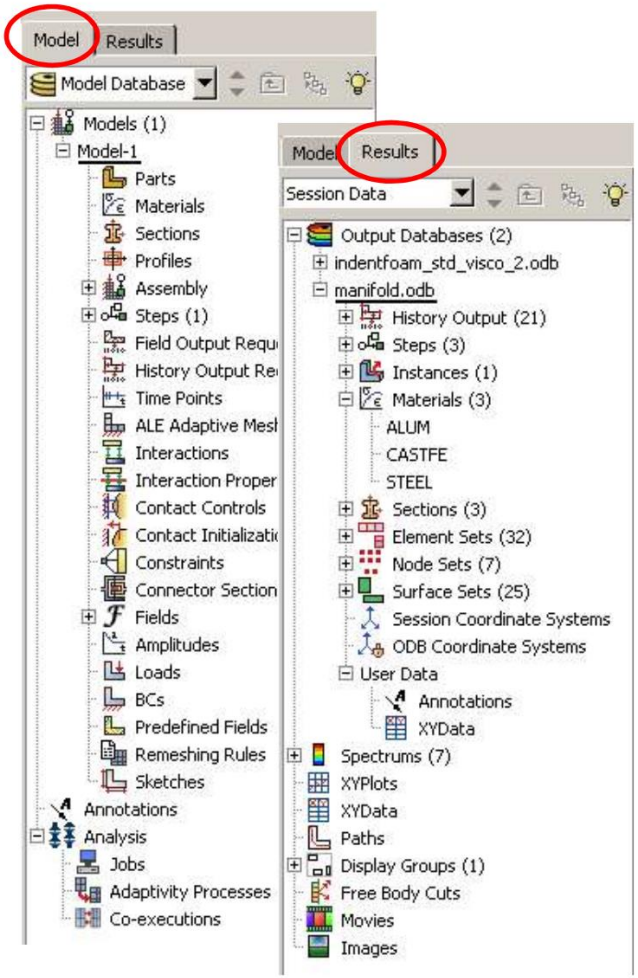
ABAQUS

Overview of Abaqus/CAE

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. 
Step	**Interaction**	**Load**
• Define analysis steps and output requests	• 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 
Mesh	**Job**	**Visualization**
• Split assembly into meshable regions and mesh 	• Submit, manage, and monitor analysis jobs	• Examine results 

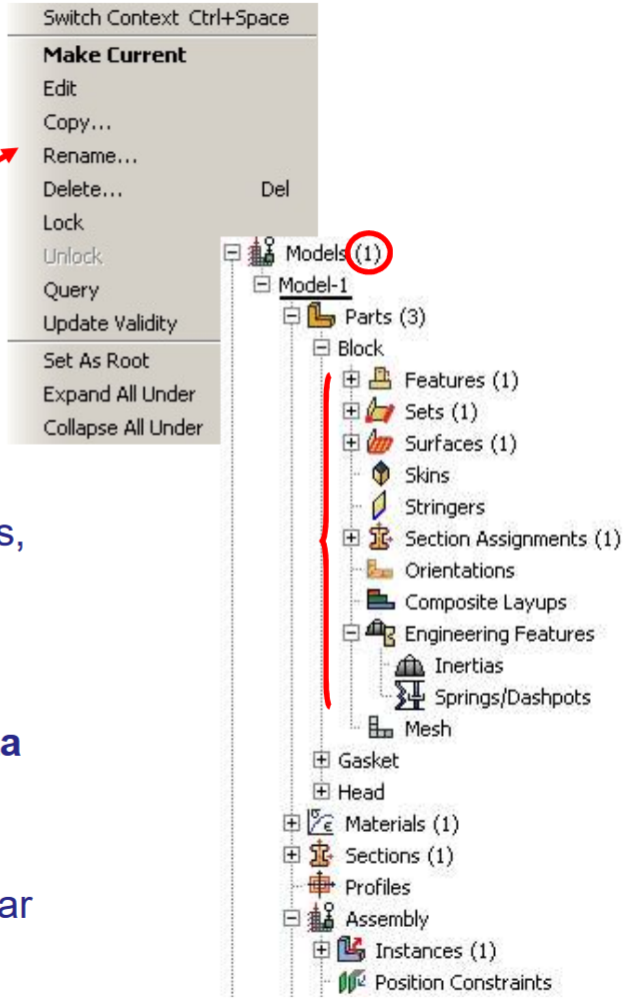
Overview of Abaqus/CAE

- **Model Tree and the Results Tree**
  - The Model Tree provides you with a graphical overview of your model and the objects that it contains.
  - The Results Tree is used to display analysis results from output databases as well as session-specific data such as *X*–*Y* plots.
  - Both trees provide shortcuts to much of the functionality of the main menu bar, the module toolboxes, and the various managers.
  - Some features of the Model and Results trees are discussed next.



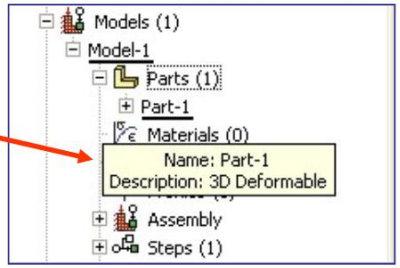
Overview of Abaqus/CAE

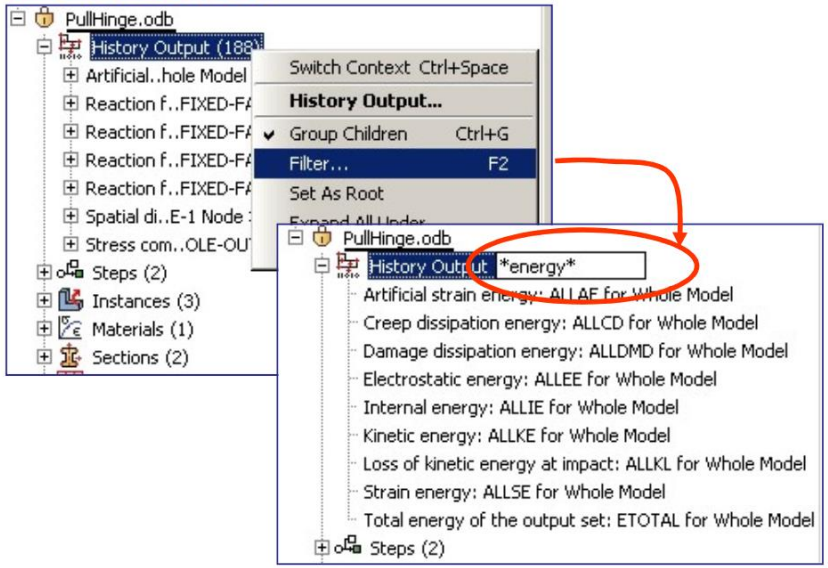
- **Tree features**
  - **Navigation tool**
    - Context sensitive RMB actions → edit/create/suppress/query
    - Can effectively build most of your model from the tree
  - **Model query/auditing**
    - E.g., shows number of sections, materials, constraints currently defined
    - Shows status of certain features/items (invalid part, suppressed feature)
  - **Comprehensive view of Abaqus model data**
    - "Containers" expand to show objects and their hierarchy
    - Step dependent objects (e.g., BCs) appear in the STEP and LOAD containers



Overview of Abaqus/CAE

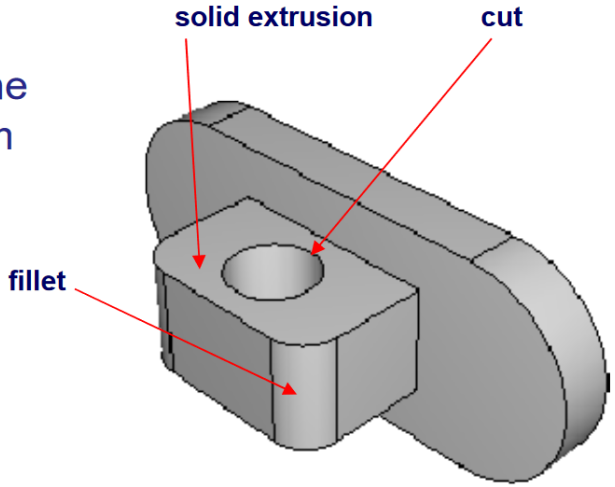
- **Tool tips**
  - **Float the mouse over a container or item…**
- **"Pruning" the tree**
  - **You can set a certain container as the new "root" to reduce clutter**
- **Keyboard shortcuts**
  - **Hide/show**
  - **Expand/collapse**
  - **Search**
  - **Delete items**
  - **Switch context**
- **Filtering**





Overview of Abaqus/CAE

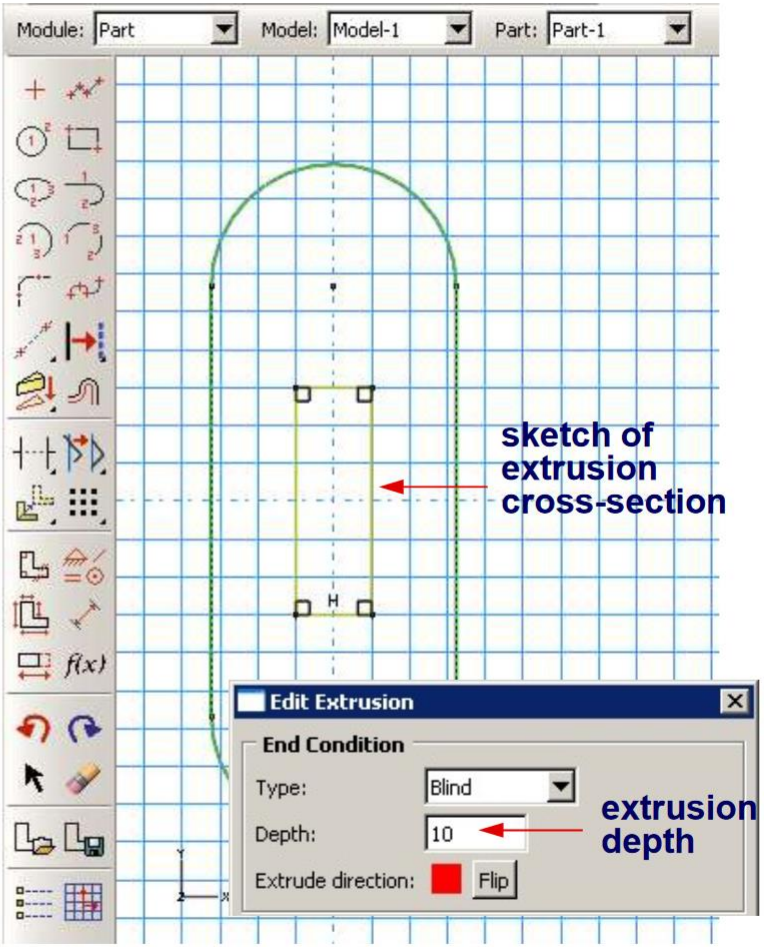
- **Models are feature-based and parametric**
  - A feature is a meaningful piece of the design. Models are constructed from numerous features; for example:
  - **Geometric features**
    - Solid extrusion, wire, cut, fillet, etc.
  - **Assembly features**
    - Wheel must be concentric with the axle, the blank lies exactly in contact with the rigid die, etc.
  - **Mesh features**
    - Partition the mesh into different regions for different meshing techniques, seed different edges with different mesh densities, etc.



Part with several annotated features

Overview of Abaqus/CAE

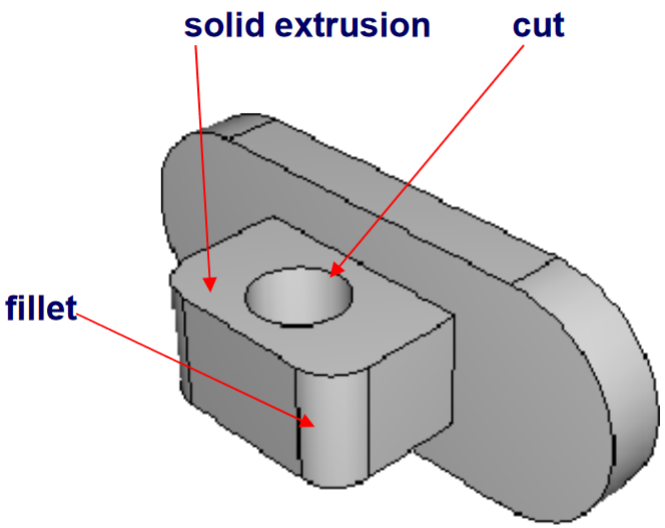
- A parameter is a modifiable quantity that provides additional information for a feature; for example:
- **Solid extrusion parameters**
  - Sketch of extrusion cross-section, depth of extrusion.
- **Cut**
  - Sketch of cut cross-section, depth of cut.
- **Fillet parameter**
  - Fillet radius.



Solid extrusion parameters

Overview of Abaqus/CAE

- Features often have parent-child relationships, such that the existence of the child depends on the existence of the parent; for example:
  - Delete the solid extrusion, and the hole and fillet cannot exist.
  - Delete the part, and the mesh cannot exist.
- Abaqus/CAE always asks to make sure that you want to delete the parent and its child features.

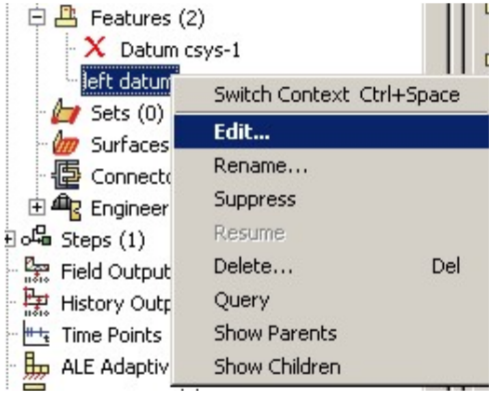


- **parent: solid extrusion**
- **child: cut**
- **child: fillet**

Example of Parent-Child Relationships among Features

Overview of Abaqus/CAE

- Features can be modified by editing their parameters.
  - Aspects of the model can be regenerated.
  - Parametric studies are easy and natural.
- Features can also be deleted or temporarily suppressed with the option to resume them later.
  - If a parent feature is deleted or suppressed, all its child features are also deleted or suppressed.

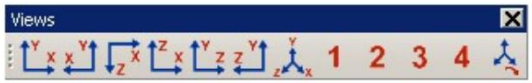


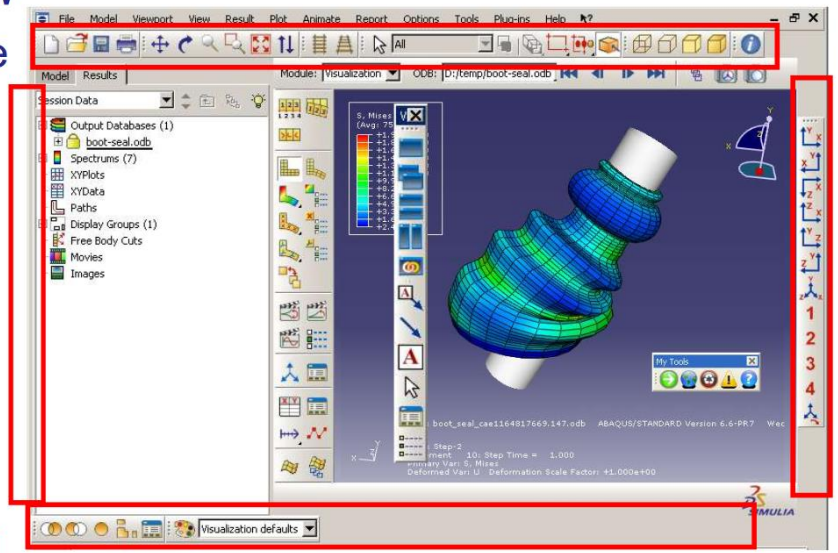
Overview of Abaqus/CAE

- **Interoperability**
  - Abaqus/CAE is based on the concepts of parts and assemblies, which are common to many CAD systems.
  - Parts can be created within Abaqus/CAE.
  - Geometry can be imported from other packages and exported to other packages.
  - Existing Abaqus meshes can be imported for further processing.
  - Individual models can be copied between databases.

Overview of Abaqus/CAE

- **Dockable toolbars**
  - Allow you to modify the layout and appearance of toolbars
  - Individual toolbars can be moved by dragging the toolbar grip
  - Toolbars can be “docked” at any of four docking regions located around the main window
  - Floating toolbars can be located anywhere
    - Orientation of floating toolbars can be controlled

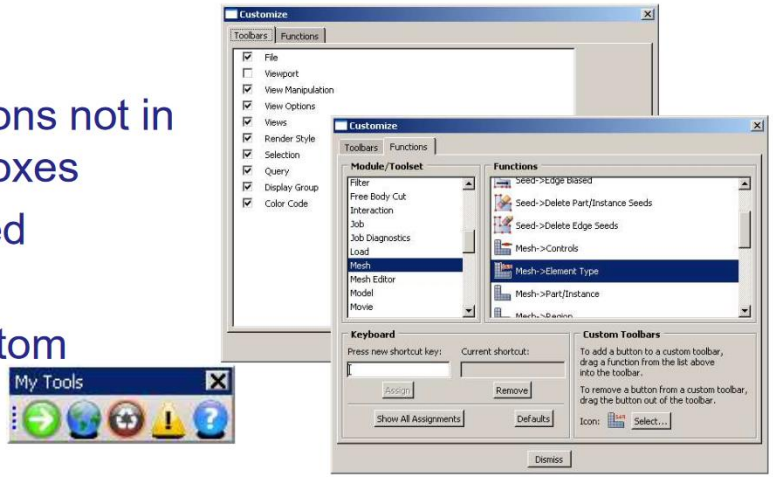


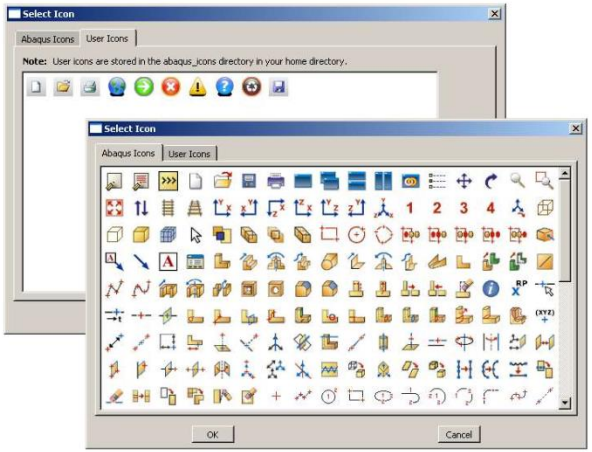


Dock Sites

Overview of Abaqus/CAE

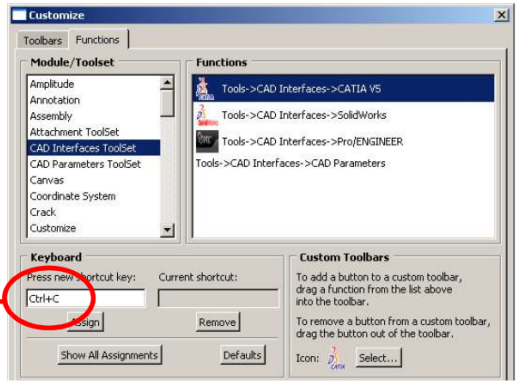
- **Custom toolbars**
  - Include shortcuts to functions not in standard toolbars or toolboxes
  - Can collect commonly used functions
  - To add a function to a custom toolbar:
    - **Tools → Customize**
    - Select the function in the **Customize** dialog box
    - Drag it onto the custom toolbar.
    - Assign an icon to represent the toolbar
  - Can be moved, docked, floated, or hidden in the same way as standard toolbars

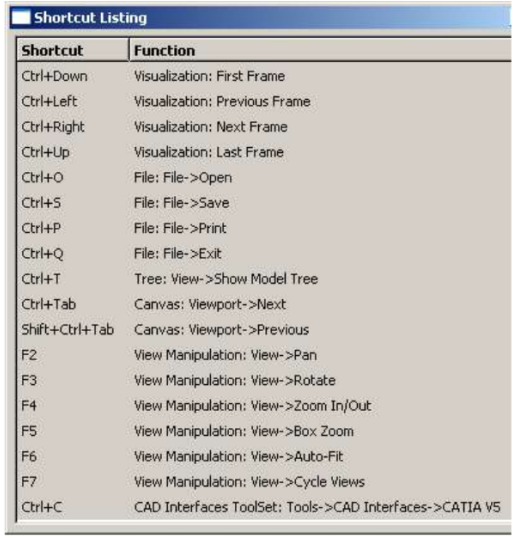




Overview of Abaqus/CAE

- **Custom keyboard shortcuts**
  - Available for most functions.
    - E.g., the key combination **[Shift] + [Ctrl] + P** may be assigned to **Create Part** dialog box
  - Default keyboard shortcuts for common functions (save, print, etc.) can also be reassigned.
  - Keyboard shortcuts must use one of the following keys or key combinations:
    - Any function key except **F1**
    - **[Alt] + [Shift] + key**
    - **[Ctrl] + key**. You can also add **[Alt]** or **[Shift]** to modify any keyboard shortcut that includes the **[Ctrl]** key.

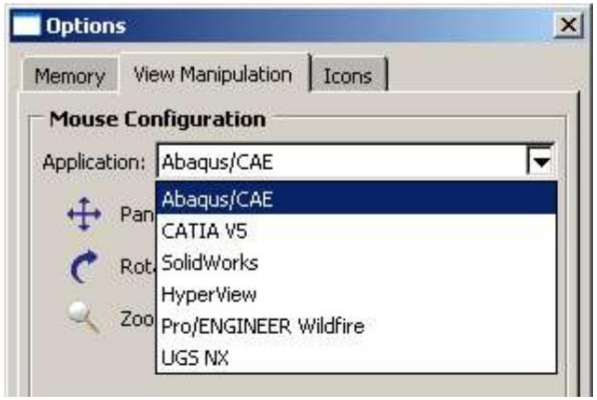




Shortcut	Function
Ctrl+Down	Visualization: First Frame
Ctrl+Left	Visualization: Previous Frame
Ctrl+Right	Visualization: Next Frame
Ctrl+Up	Visualization: Last Frame
Ctrl+O	File: File->Open
Ctrl+S	File: File->Save
Ctrl+P	File: File->Print
Ctrl+Q	File: File->Exit
Ctrl+T	Tree: View->Show Model Tree
Ctrl+Tab	Canvas: Viewport->Next
Shift+Ctrl+Tab	Canvas: Viewport->Previous
F2	View Manipulation: View->Pan
F3	View Manipulation: View->Rotate
F4	View Manipulation: View->Zoom In/Out
F5	View Manipulation: View->Box Zoom
F6	View Manipulation: View->Auto-Fit
F7	View Manipulation: View->Cycle Views
Ctrl+C	CAD Interfaces ToolSet: Tools->CAD Interfaces->CATIA V5

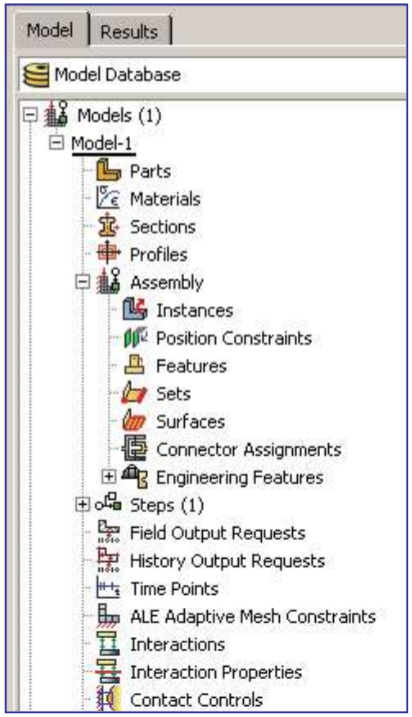
Overview of Abaqus/CAE

- **View manipulation**
  - Toolbar to control view (pan, zoom, rotate, etc.).
  - Alternatively, can use a combination of keyboard and mouse actions:
    - Rotate: [Ctrl]+[Alt]+MB1.
    - Pan: [Ctrl]+[Alt]+MB2.
    - Zoom: [Ctrl]+[Alt]+MB3.
  - You can reconfigure these combinations to mimic the view manipulation interfaces used by other common CAD applications



Working with the Model Tree

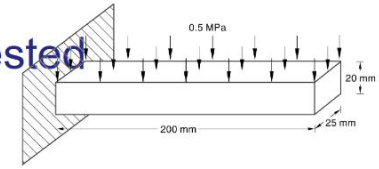
- **The Model Tree is a convenient tool for navigating and managing your models and analysis results.**
  - The Model Tree provides a visual description of the hierarchy of items in a model.
  - The arrangement of the containers and items in the Model Tree reflects the order in which you are likely to create your model.
    - A similar logic governs the order of modules in the module menu—you create parts before you create the assembly, and you create steps before you create loads.

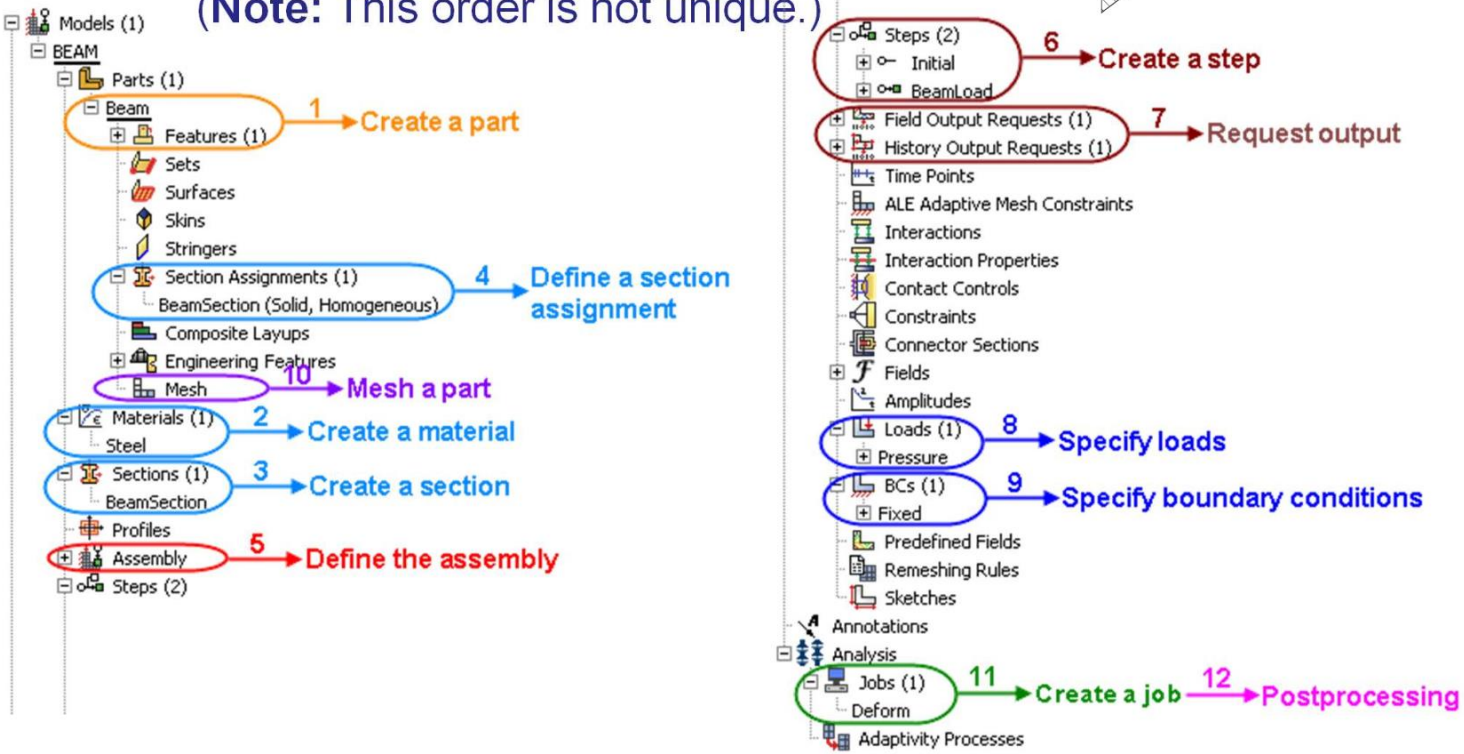


ABAQUS

Working with the Model Tree

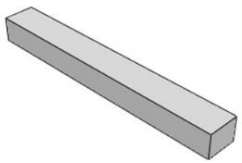
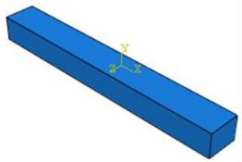
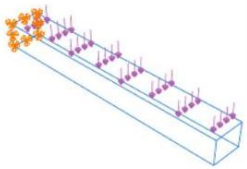
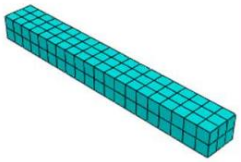
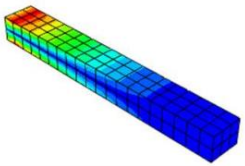
- Example: The following figure shows a suggested order to create the cantilever beam model . (**Note:** This order is not unique.)





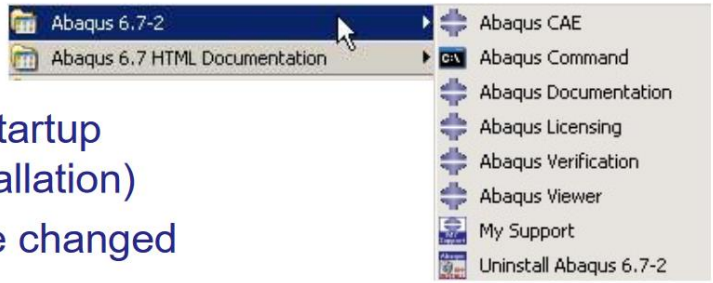
Working with the Model Tree

- Alternatively, follow the order of modules in the module menu to create the model.

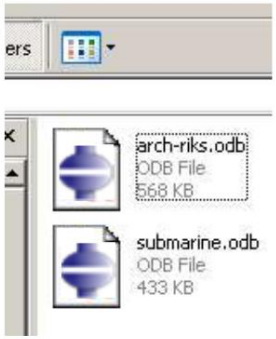
Part	Property	Assembly
• Create the part geometry 	• Define materials • Define and assign sections to parts or regions	• Position the part for initial configuration. 
Step	**Interaction**	**Load**
• Define analysis steps and output requests	• Not applicable for this example	• Apply loads and BCs to regions or named sets; and assign them to steps in the analysis history 
Mesh	**Job**	**Visualization**
• Mesh the part 	• Submit, manage, and monitor analysis jobs	• Examine results 

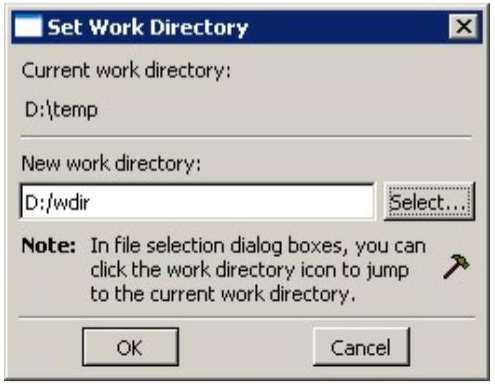
Starting Abaqus/CAE

- **Three options available:**
  - Command line
    - **`abaqus cae = filename.cae`**
    - Opens Abaqus/CAE in current directory
  - Start menu (Windows)
    - Opens Abaqus/CAE in startup directory (set during installation)
    - Working directory can be changed (see next slide)



  - Double-click `.cae` or `.odb` file in Windows folder
    - Opens Abaqus/CAE in current directory



Starting Abaqus/CAE

- **Selecting a working directory**
  - Can select a working directory
    - Default is startup directory
  - Subsequent file operations will use this directory for reading/writing
  - Job files will be written to the new working directory
  - This makes it easier to manage job files
    - E.g., keep all job files in a per-job directory