

# MAE 656 - Advanced Computer Aided Design

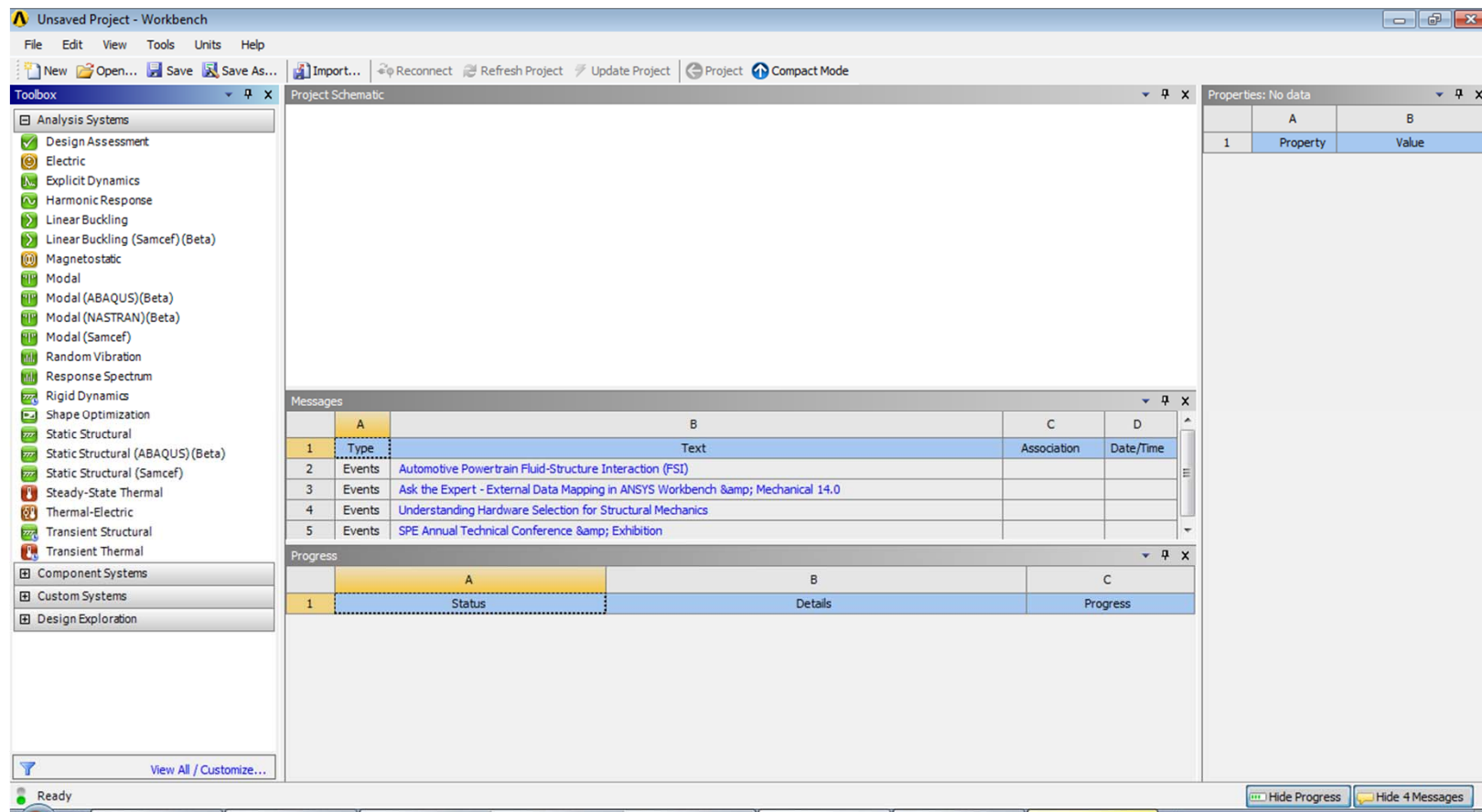
02. Ansys Workbench – Doc 01

Introduction to  
Ansys Workbench

# Main Screen

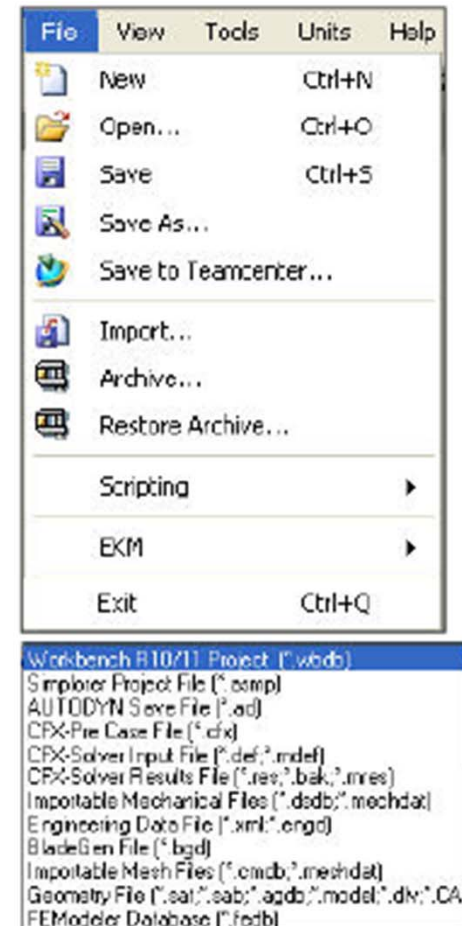
## Components:

Top menu – Toolbox – Messages – Progress – Project Properties



# Top Menu – File

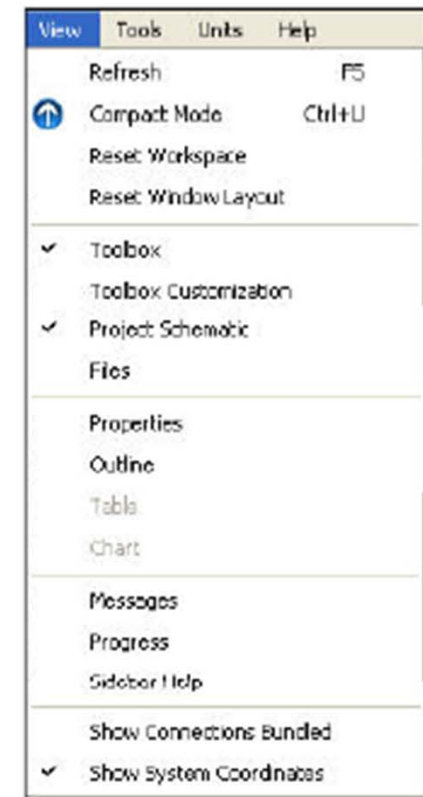
- Create/Open/Save a Workbench project file
- Saves data from all the components
- Import: Import files (for acceptable file types, see Import options)
- Archive: Quickly generate a single compressed file (.wbpz or .zip file) containing all pertinent files for easy transfer. Zipped file can be saved at any required location
- Restore Archive: Unzip and open zipped project files
- Scripting: Record and Run journal files
- EKM: Launch and connect to EKM i.e. Engineering Knowledge Manager



# Top Menu – View

---

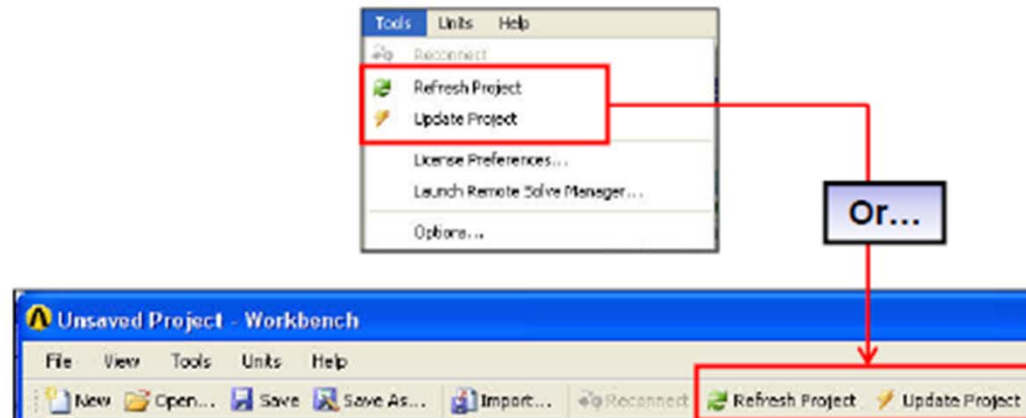
- Refresh and Reset Window layout
- View Toolbox, Toolbox Customization, Project Schematic, Messages, Progress , etc. windows on Project page
- Reset Workspace restores current workspace to default settings
- Reset Window Layout restores original window layout



# Top Menu – Tools

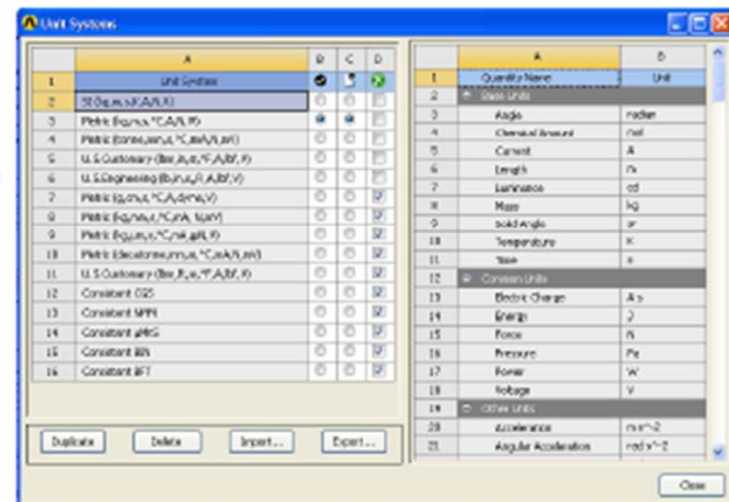
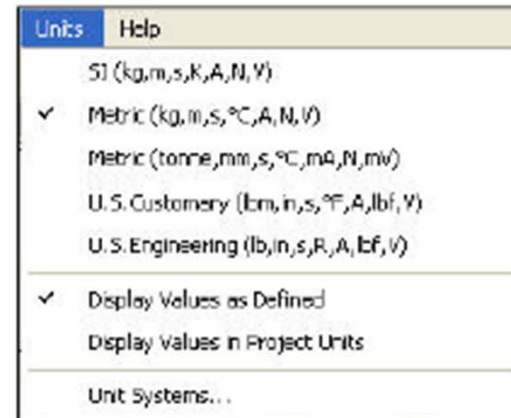
---

- Refresh or Update entire project
- Modify Global Settings through 'Options ...'



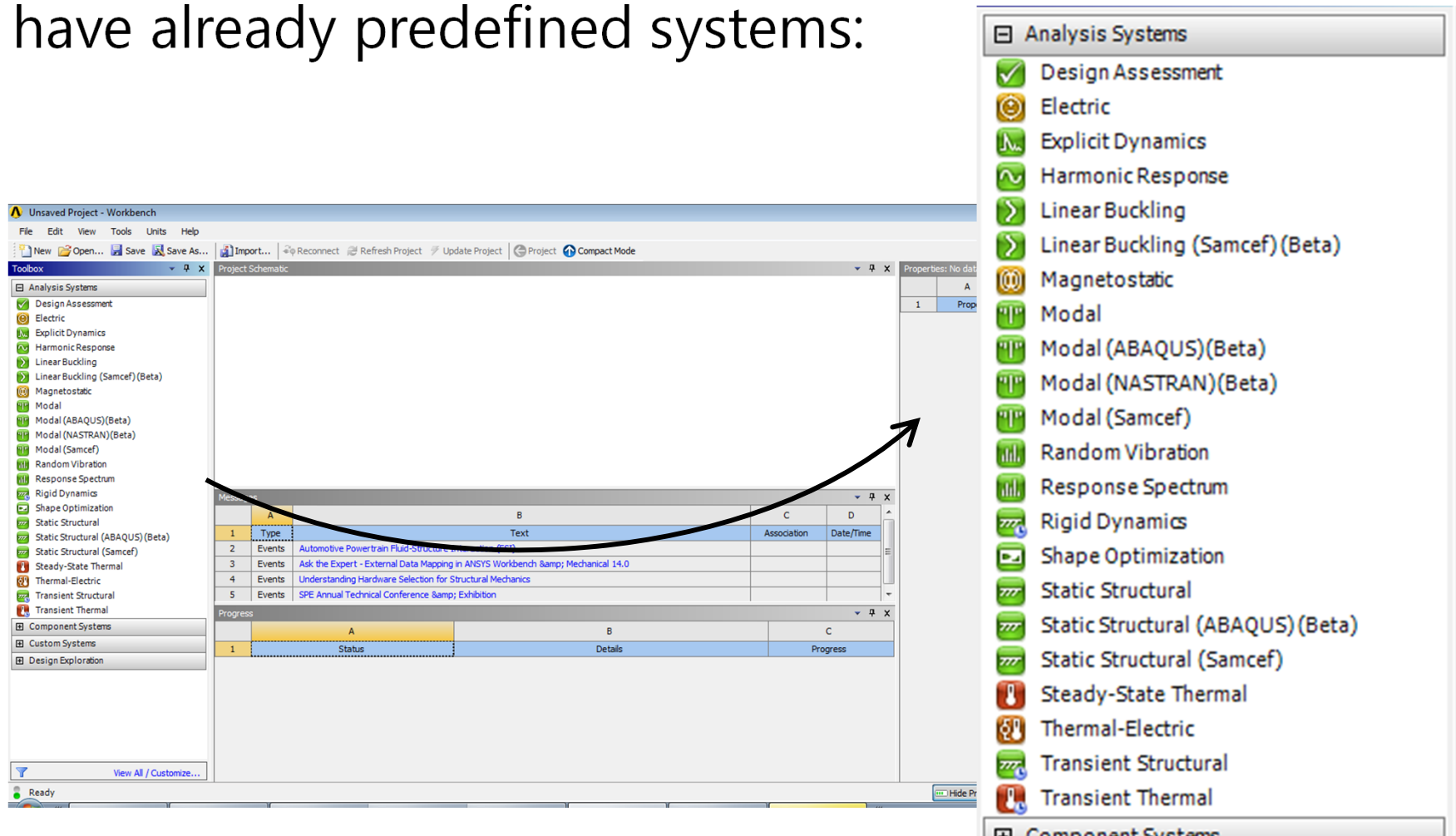
# Top Menu – Units

- Select Unit System to be used for the project
- Sets the default unit system for all systems
- Units can be changed within components like DM, AMP
- Display Values as Defined : displays value and unit as defined in Workbench or original source application
- Display Values in Project Units : displays values converted to selected project unit system
- Modify Unit Systems through 'Unit System ...'



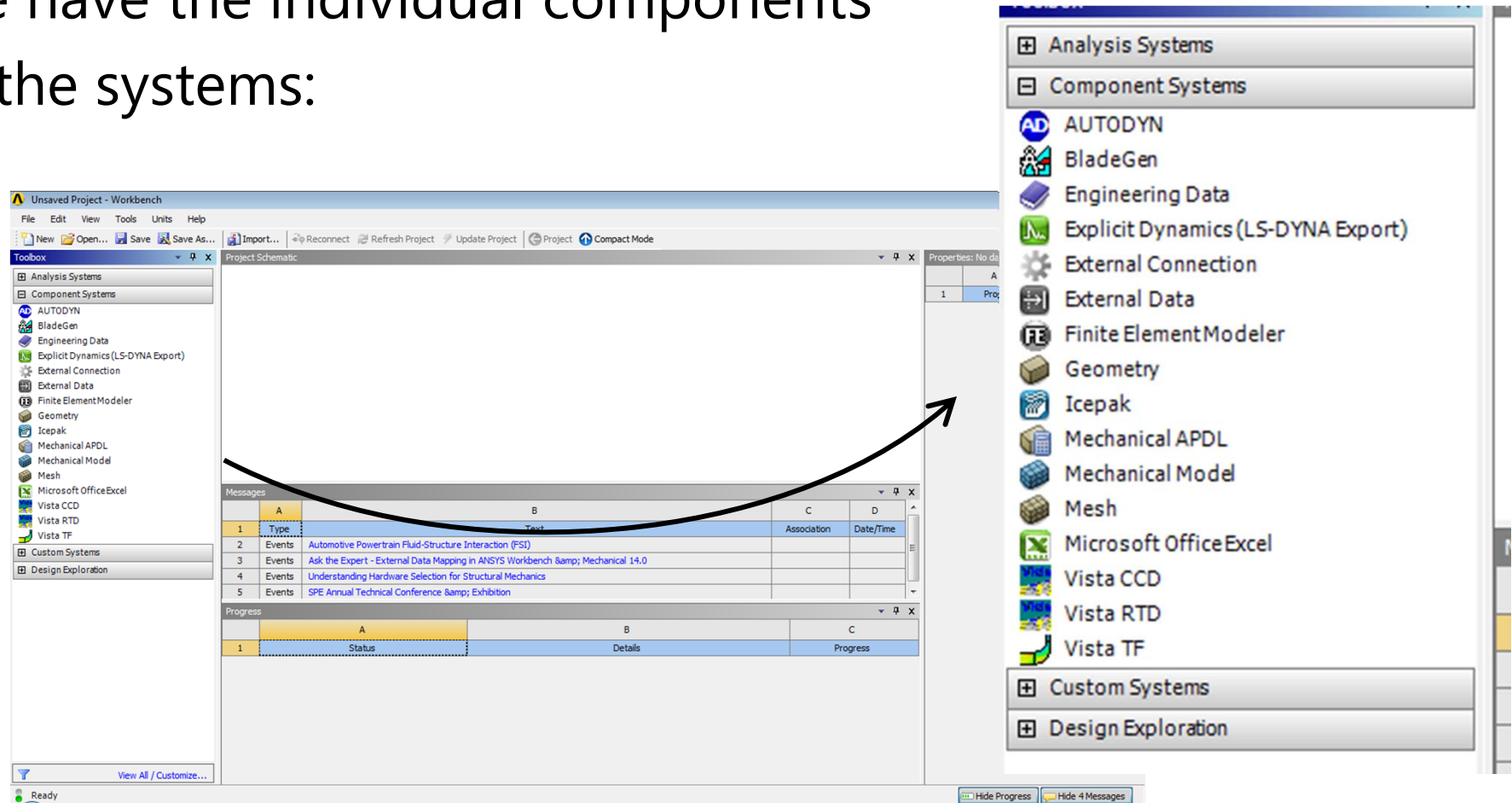
# Toolbox – Analysis Systems

We have already predefined systems:



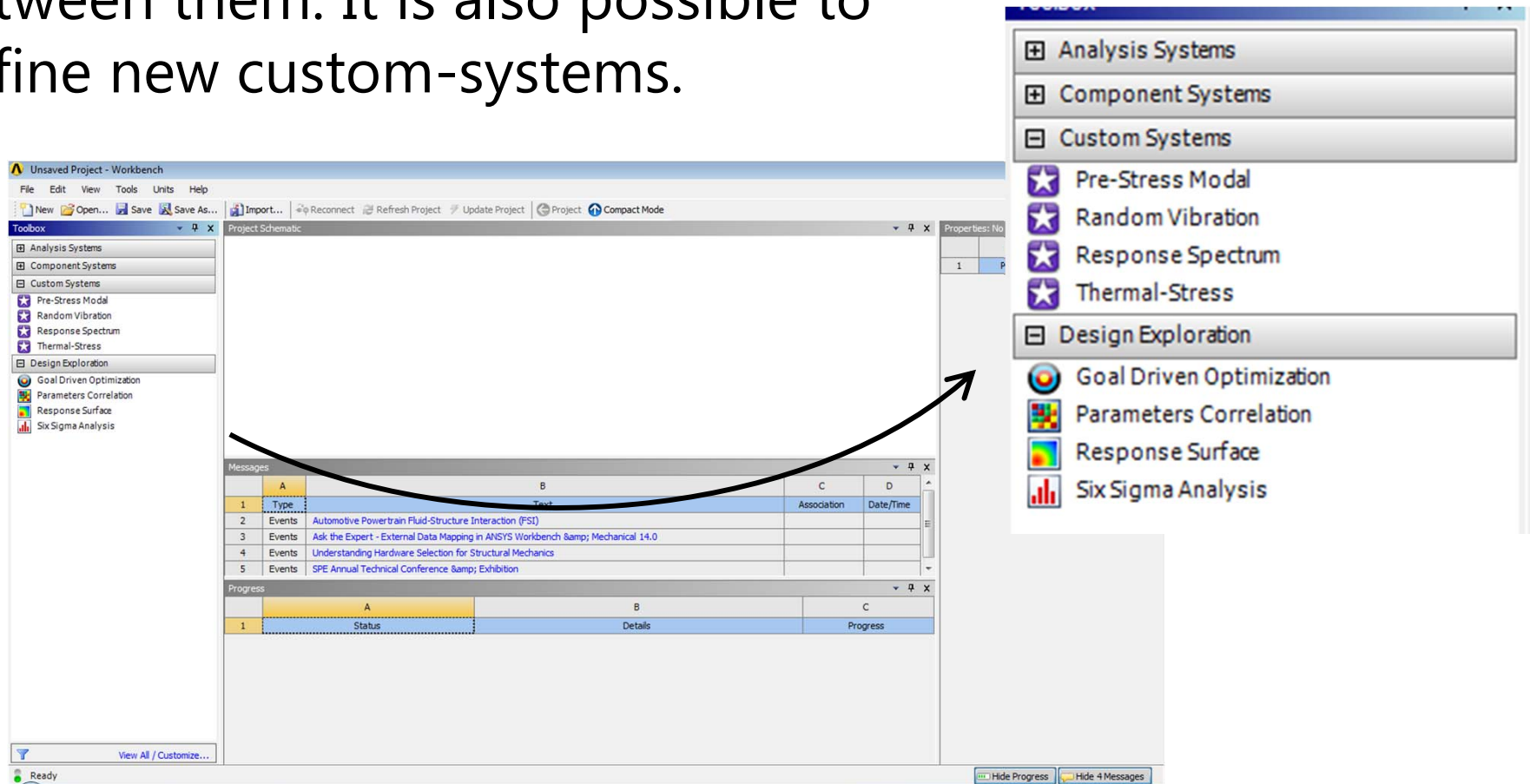
# Toolbox – Component Systems

We have the individual components of the systems:



# Toolbox – Custom Systems

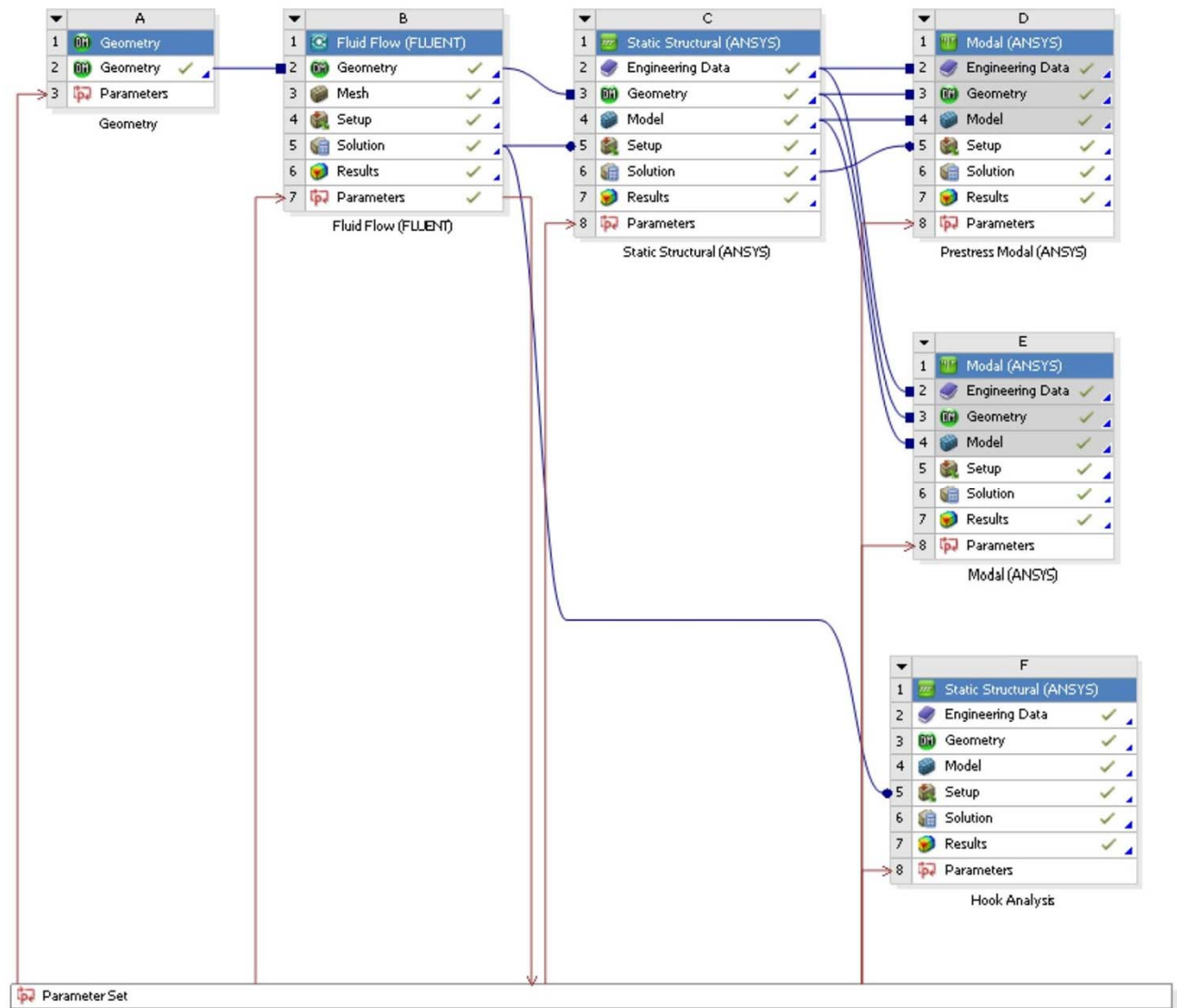
We have predefined some systems with some interactions between them. It is also possible to define new custom-systems.



# Project Schematic

It is possible to solve different problems with shared data.

It is also possible to use results from one problem as input data to another one (i.e. fluid-structure interaction)



# Static Structural System

---

We will work with the Static Structural system.

It is composed by:

Engineering data:

Definition of **material properties**

Geometry:

Def. of the problem **geometry**

Model:

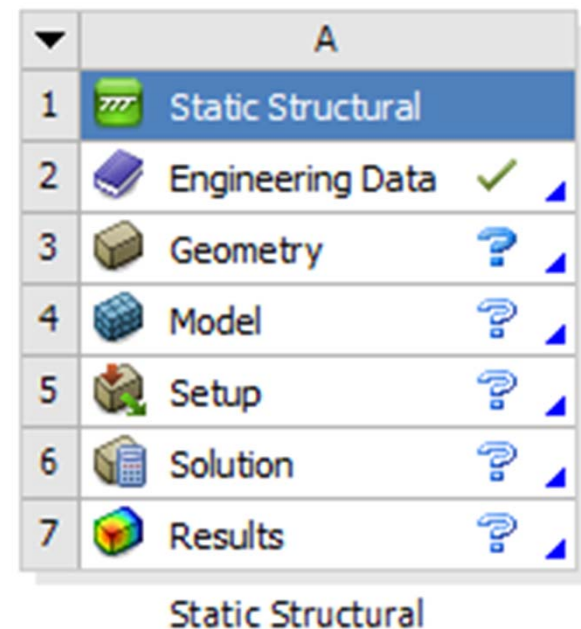
Def. of the **mesh**

Setup:

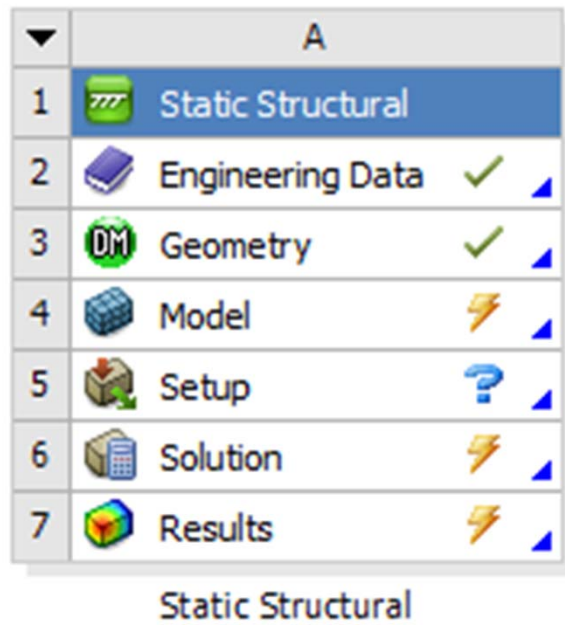
Def. of the **boundary conditions**

Results:

Contains the different results required to the solver.

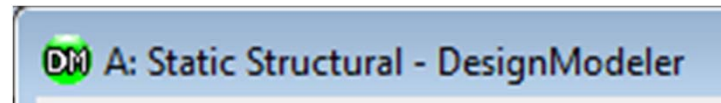


# Static Structural System

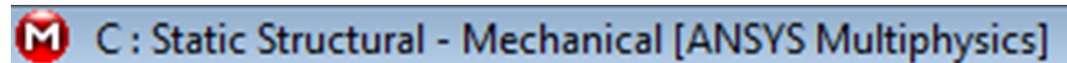


Defined in a new workbench window

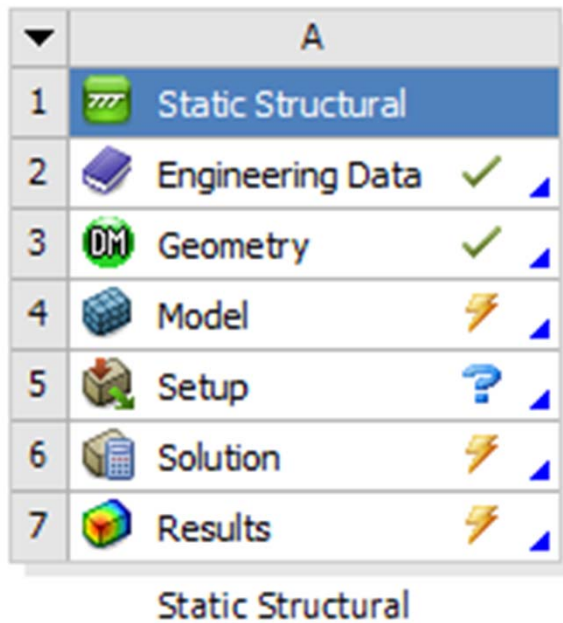
Solved with Design Modeler



Solved with Mechanical module



# Static Structural System



System properly defined and has no errors

System already defined but that has to be updated because there has been modifications in upper levels

The system is yet to be defined

# Engineering Data

The screenshot displays the ANSYS Workbench Engineering Data environment. On the left, the 'Toolbox' contains various material property categories. The 'Orthotropic Elasticity' property is highlighted and is being dragged towards the 'Properties of Outline Row 4: New Material' table. The table has columns for Property, Value, and Unit. The 'Orthotropic Elasticity' property is being added to the table, and its sub-properties (Young's Modulus X, Y, Z directions and Poisson's Ratios) are visible below it. The 'Messages' and 'Progress' panes are also visible at the bottom.

**Orthotropic Elasticity**

Property	Value	Unit
Orthotropic Elasticity		
Young's Modulus X direction		Pa
Young's Modulus Y direction		Pa
Young's Modulus Z direction		Pa
Poisson's Ratio XY		
Poisson's Ratio YZ		
Poisson's Ratio XZ		

**Messages**

Type	Text	Association	Date/Time
Events	Automotive Powertrain Fluid-Structure Interaction (FSI)		

**Progress**

Status	Details	Progress

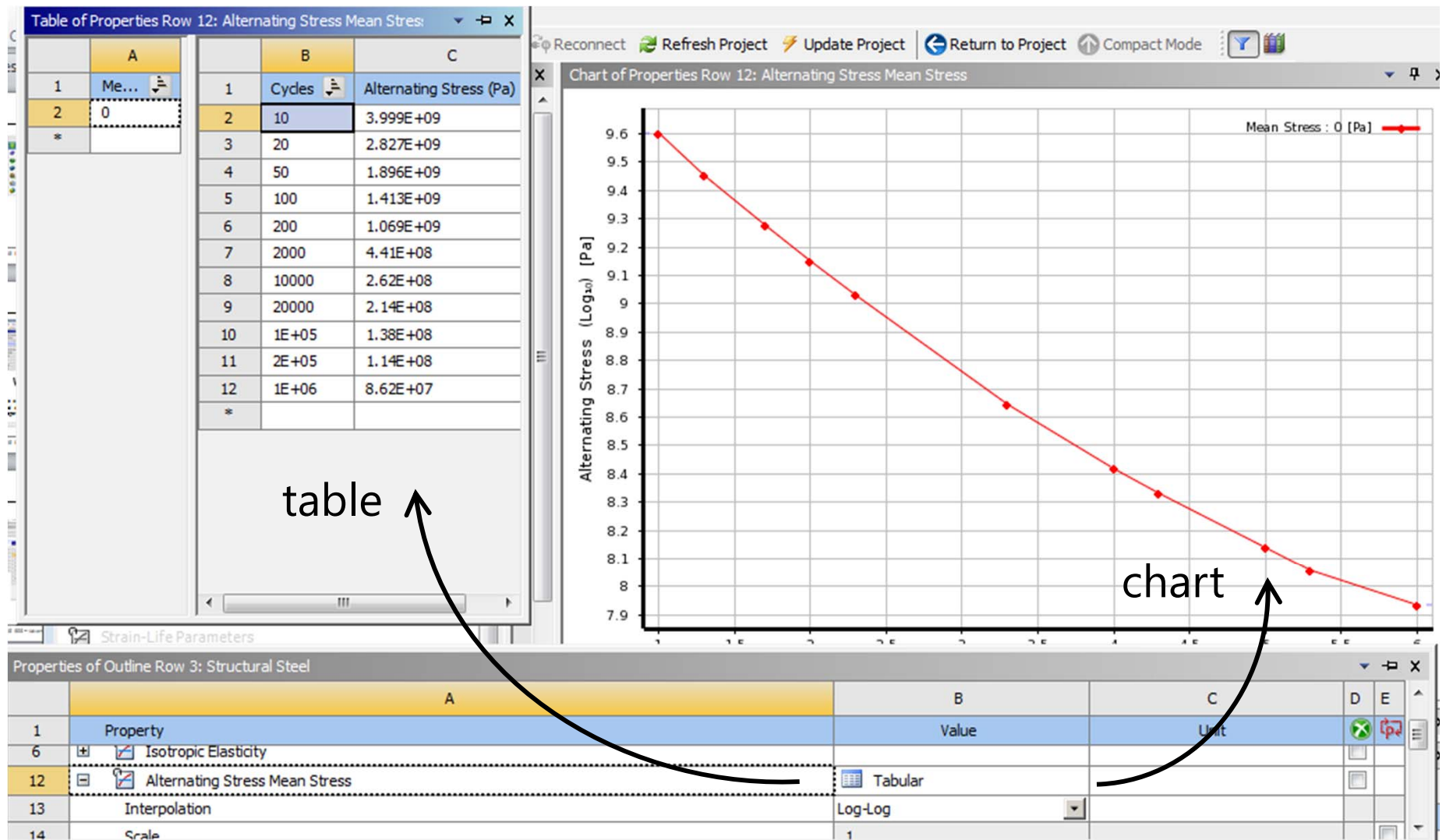
We can drag and drop properties

# Engineering Data Sources

or, right click

The screenshot shows the ANSYS Workbench interface. At the top, there is a toolbar with buttons for 'Save As...', 'Import...', 'Reconnect', 'Refresh Project', 'Update Project', 'Return to Project', and 'Compact Mode'. Below the toolbar is the 'Outline of Favorites' pane, which contains a table with columns A, B, C, D, E, and F. Row 1 has 'Contents of Favorites' in column A, 'Add' in column B, a green checkmark in column C, 'Source' in column E, and 'Description' in column F. A callout bubble with a black border points to the 'Add' button in the 'Outline of Favorites' pane, with the text 'or, right click' next to it. Below the 'Outline of Favorites' pane is the 'Engineering Data Sources' dialog box, which has a table with columns A, B, C, and D. Row 1 has 'Data Source' in column A, 'Location' in column B, and 'Description' in column D. Row 2 has a star icon in column A, 'Favorites' in column B, and 'Quick access list and default items' in column D. Row 3 has a folder icon in column A, 'General Materials' in column B, a checkbox in column C, and 'General use material samples for use in various analyses.' in column D. Row 4 has a folder icon in column A, 'General Non-linear Materials' in column B, a checkbox in column C, and 'General use material samples for use in non-linear analyses.' in column D. Row 5 has a folder icon in column A, 'Explicit Materials' in column B, a checkbox in column C, and 'Material samples for use in an explicit analysis.' in column D. Row 6 has a folder icon in column A, 'Hyperelastic Materials' in column B, a checkbox in column C, and 'Material stress-strain data samples for curve fitting.' in column D. Row 7 has a folder icon in column A, 'Magnetic B-H Curves' in column B, a checkbox in column C, and 'B-H Curve samples specific for use in a magnetic analysis.' in column D. Row 8 has a folder icon in column A, 'Thermal Materials' in column B, a checkbox in column C, and 'Material samples specific for use in a thermal analysis.' in column D. Row 9 has an asterisk in column A, 'Click here to add a new library' in column B, and a three-dot menu icon in column C. At the bottom of the dialog box, there is a 'Messages' pane with columns A, B, and C.

# Engineering Data - Charts & Tables



# Engineering Data - Exercise

---

Create a custom material for Aluminum Alloy type 5052-O.

The parameters that have to be defined are those of:

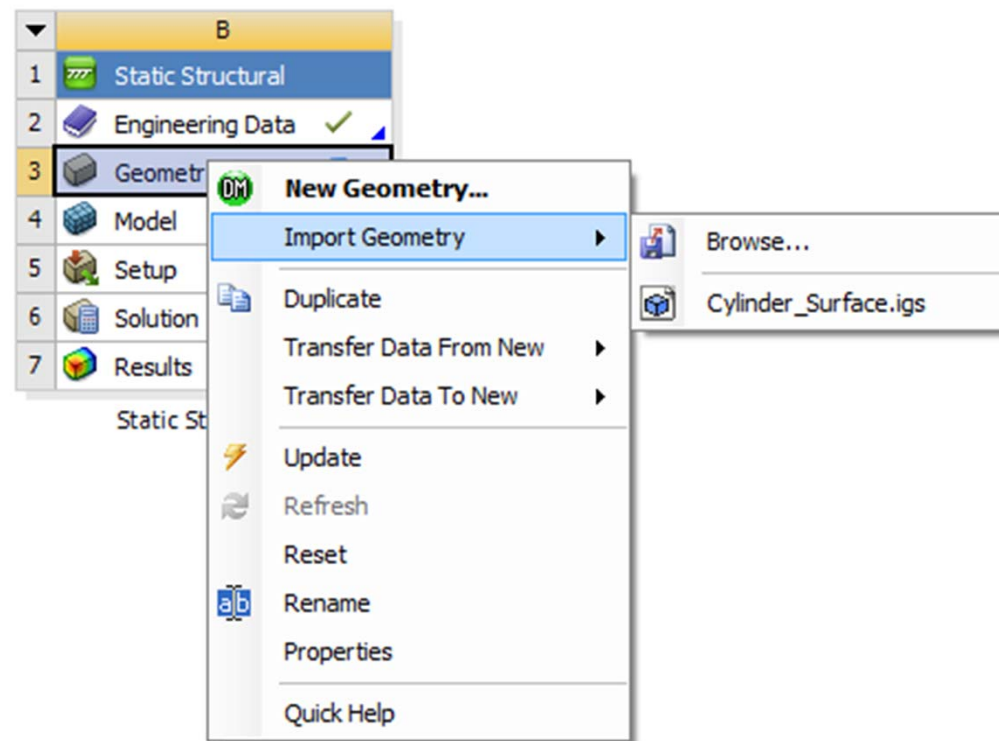
- Density
- Thermal expansion
- Isotropic elasticity
- Yield and ultimate strength (defined as multilinear isotropic hardening)
- Yield and ultimate strength as failure variables to be used in an elastic material.

The numerical value of each parameter can be obtained from <http://www.makeitfrom.com/>

# Design Modeler – Import Geometry

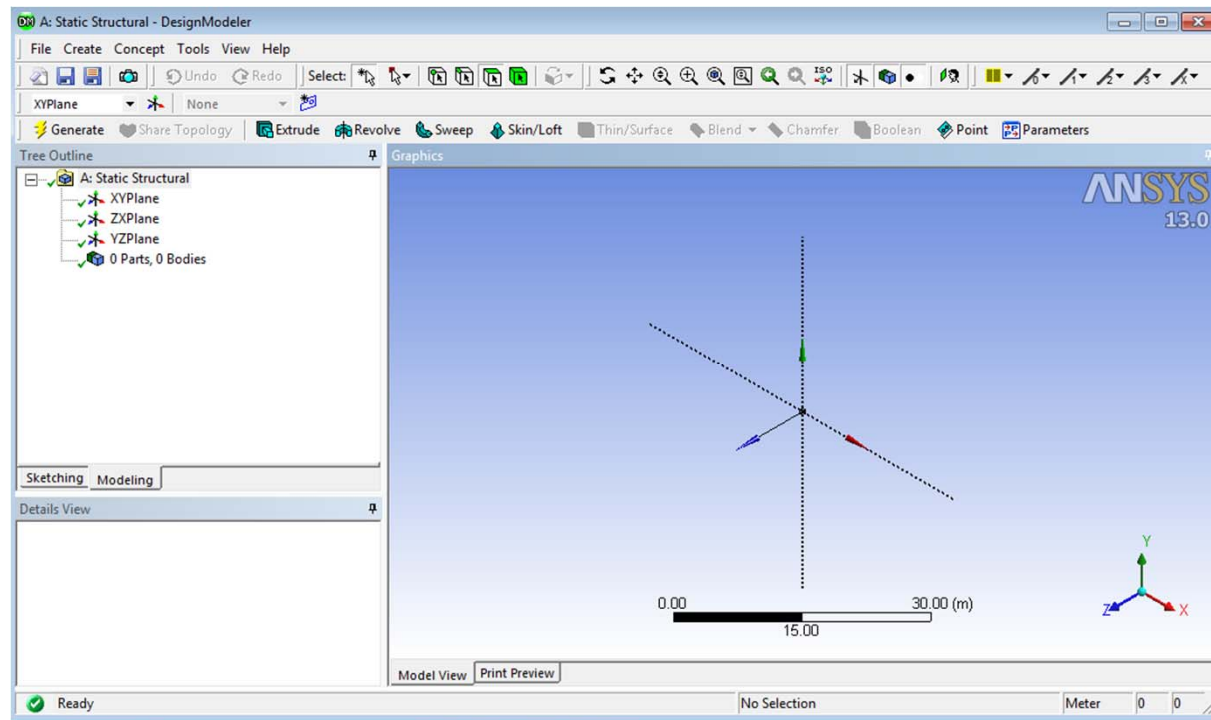
---

It is possible to import the geometry from other codes such as Catia, Rhinoceros, Solid Works, etc.



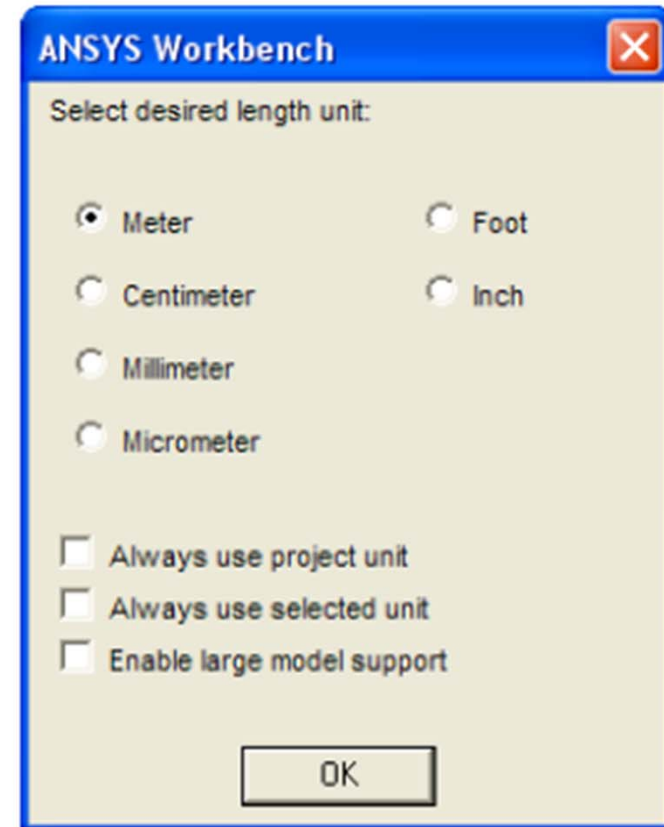
# Design Modeler – New Geometry

If instead of importing the geometry from another code or from another problem defined in the same workbench we decide to create a new geometry, clicking on NEW GEOMETRY will launch the desing modeler.



# Design Modeler – Units

- Unit selection menu pops up immediately after DM launches
- **Always use project unit** : Project units set in Workbench will be used
- **Always use selected unit** : Unit selected from the panel will be used
- **Enable large model support** : enable this to create large models within a bounding box of 1000 cubic km.

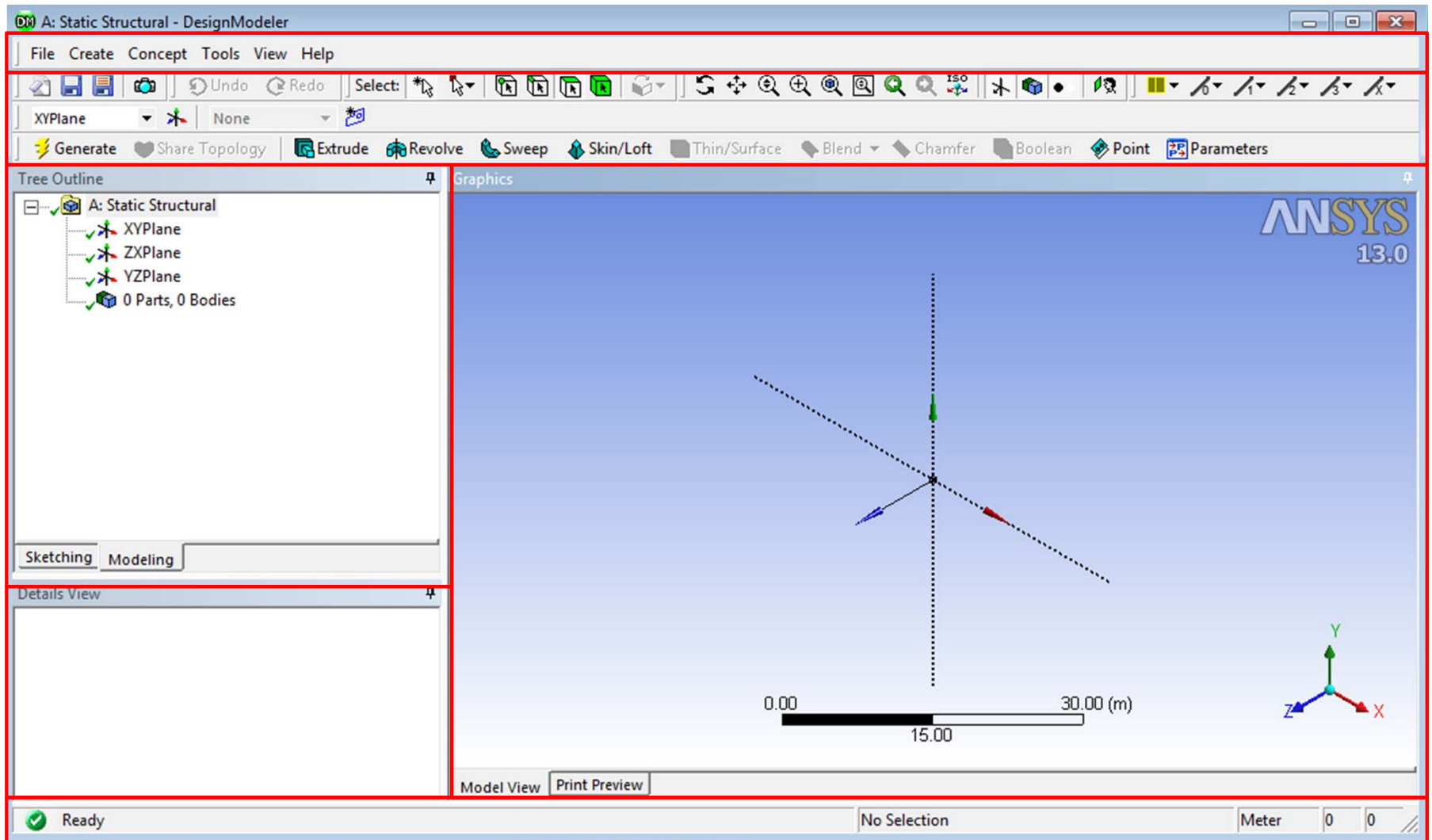


*DM will no longer show the Units pop up panel in subsequent DM sessions if 'Always use project unit' or 'Always use selected unit' is chosen.*

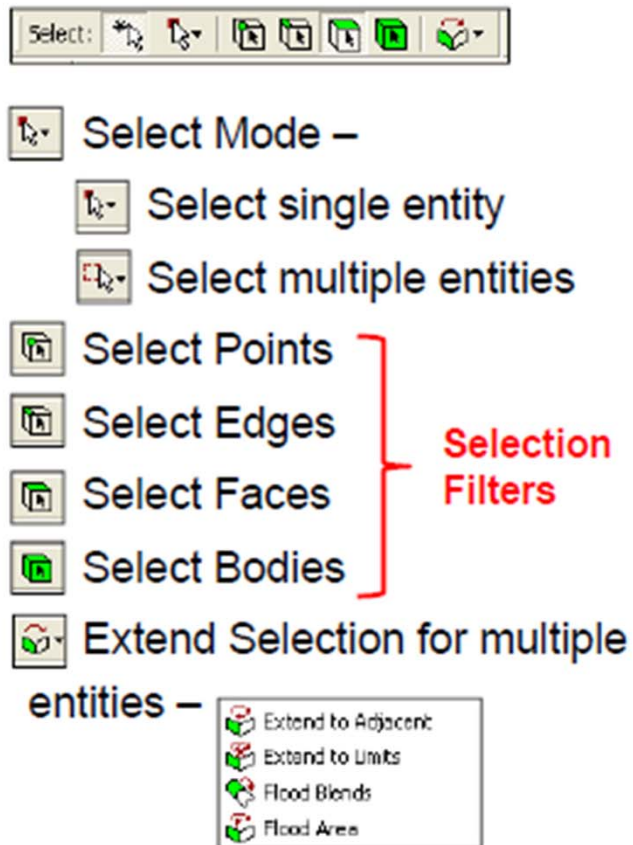


*Units cannot be changed mid-session*

# Design Modeler – Main Window

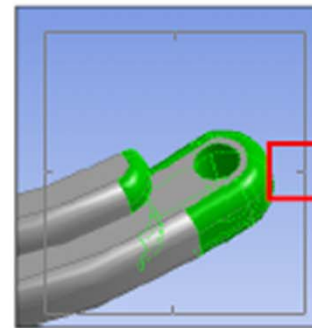


# Design Modeler – Selection

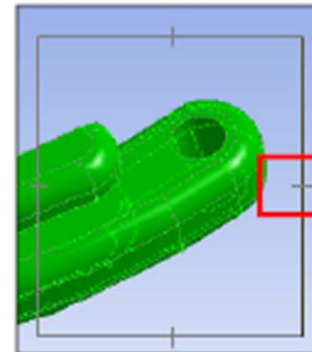


## Tricks on 'Selection'

- To select multiple entities, drag the mouse to create a box enclosing the entities



Drag from **left to right** to select the entities, completely enclosed in the box



Drag from **right to left** to select the entities, completely and partially enclosed in the box



*Hash lines on the edges of box help to determine the type of selection*




# Design Modeler – View Toolbar

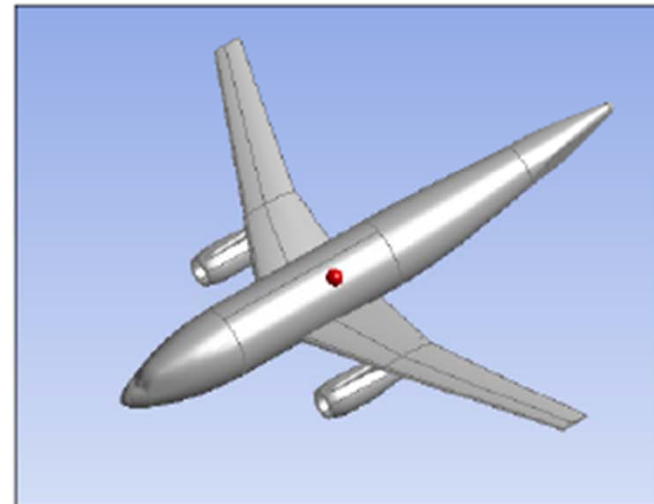
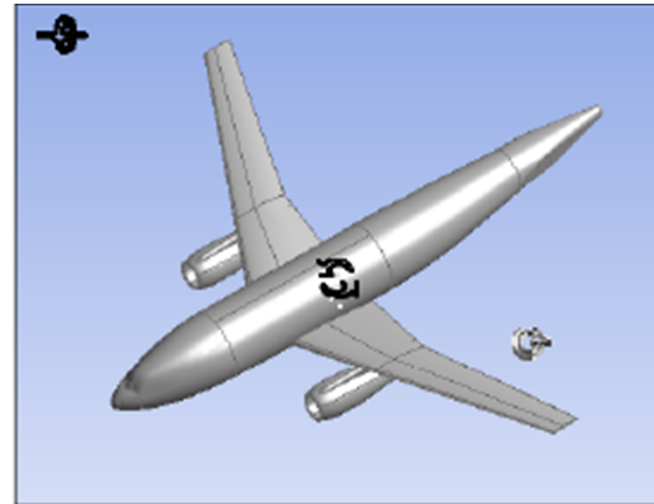
---



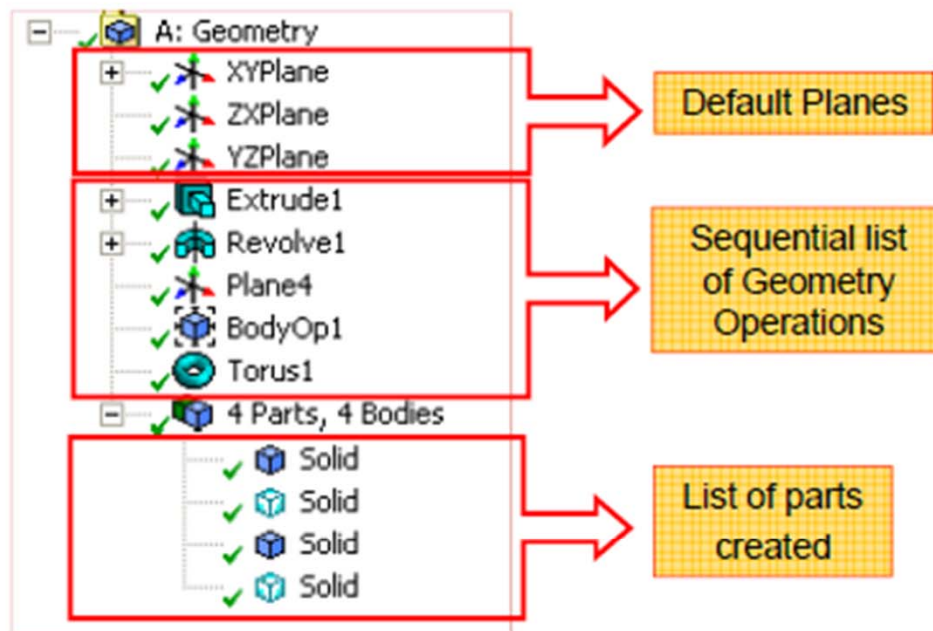
1. **Rotate** : For dynamic rotation of the object
2. **Pan**: To scroll the object around
3. **Zoom**: Zoom in or Zoom out the view by dragging LMB up or down respectively
4. **Box Zoom**: Zooms to the view enclosed by the box created by dragging the LMB
5. **Fit Model to Screen**: Zooms in to fit the full model in the screen
6. **Magnifier Window**: Zoomed in model will be displayed in a new window
7. **Previous View**: Works as Undo for the Displays
8. **Next View**: Works as Redo for the displays
9. **Set Iso View**: Set the model in Isometric view
10. **Display Plane**: Toggle to display the plane on or off
11. **Display Model**: Toggle to display the model on or off
12. **Display Points**: Toggle to display the points on or off
13. **Look at**: Orients the view normal to the selected Face or active Plane, or active Sketch

# Design Modeler – View Toolbar

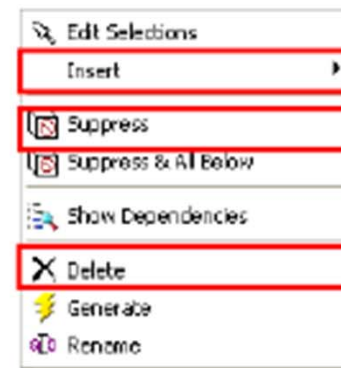
- Rotate Behavior (LMB):
  - Cursor outside center :  Rotation about Z view
  - Cursor near center of graphics screen :  Free rotations
  - Cursor near edge of graphics screen :  Rotations about X (top/bottom) or Y (left/right) axes
- While in Rotate, Pan, or Zoom mode:
  - Left click on model: resets center of view and rotation at cursor location (identified by **red dot**)
  - Left click in open area re-centers model and rotation center to Centroid



# Design Modeler – Tree Outline



Contextual menu upon RMB click



**Insert** can be used to add operations anywhere in the tree

**Suppress** can be used to deactivate a selected operation

**Delete** can be used to delete a selected operation

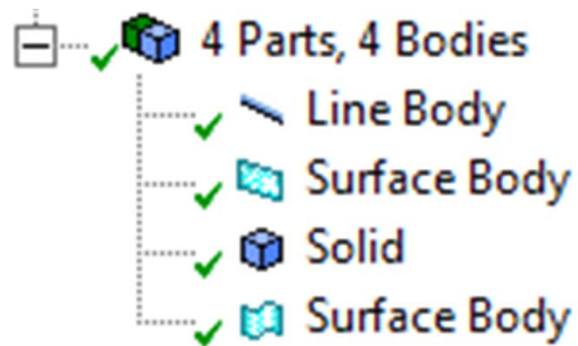


*It is recommended to suppress an operation instead of deleting it. It helps the user with the option of unsuppressing it if needed in the future.*

# Design Modeler – Parts

---

There are four different types of parts. Each one can be used in a type of simulation: BARS, 2D, 3D and SHELLS.



# Design Modeler – More info.

---

More information about how to work with design modeler can be found in the Ansys Training Lectures:

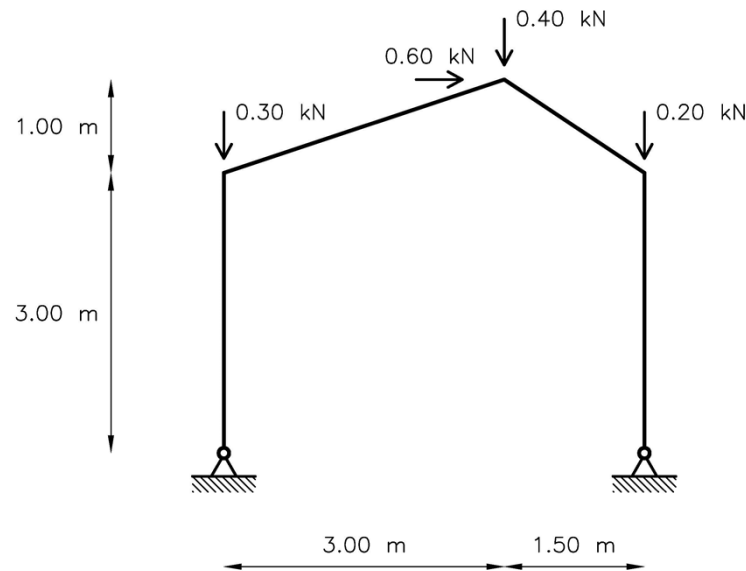
DM-13.0 – 01 – Planes and Sketches.pdf

DM-13.0 – 02 – 3D Operations.pdf

# Design Modeler – Exercises

---

1. Import geometry Cylinder\_Surface.igs to design modeler
2. Create structural frame defined in Assignment 01.
  1. Defining points manually
  2. Using point data "Frame Nodes.txt"  
In this second case, the roof beams have a cross section: 5 x 10 x 0.1 cm



# Design Modeler – Exercises

---

3. Create the triangular plate defined in the following figure to be solved as a 2D structure.
4. Create the triangular plate to be used as a 3D structure

