Ansys Mechanical Getting Started

Module 01 Student Reference Guide: Introduction

Release 2023 R1

Please note:
- These training materials were developed and tested in Ansys Release 2023 R1. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.

Ansys

Module 01: Learning Objectives

- Start Ansys from the Windows desktop

- Identify the geometry file to be used

- Open the Engineering Data application
  - Describe the purposes of the Engineering Data, Outline, and Properties tables
  - Open the Engineering Data Sources view
  - Describe the purposes of the Engineering Data Sources and material properties tables
  - Add a material from an Engineering Data Source to Engineering Data
  - Change the active system of units for the project
  - Close the Engineering data Sources view

- Open the Ansys Mechanical application

- Assign a material to a part

- Use mouse button shortcuts to rotate, pan, and zoom the graphics display

Module 01: Learning Objectives

- Add a fixed support to the model and select (scope) the associated surface geometry

- Change the active system of units in Mechanical

- Add a pressure load to the model, select (scope) the associated surface geometry, and specify the pressure magnitude

- Add a force load to the model, select (scope) the associated surface geometry, specify the force magnitude, and specify the force direction

- Add a results branch to the model

- Run a solution

- Display contour results
  - Identify the predicted magnitudes of the maximum and minimum results
  - Identify the predicted locations of the maximum and minimum results

Module 01: Graphics

Goals:

▪ Show the end-to-end solution process

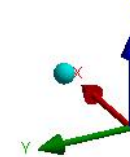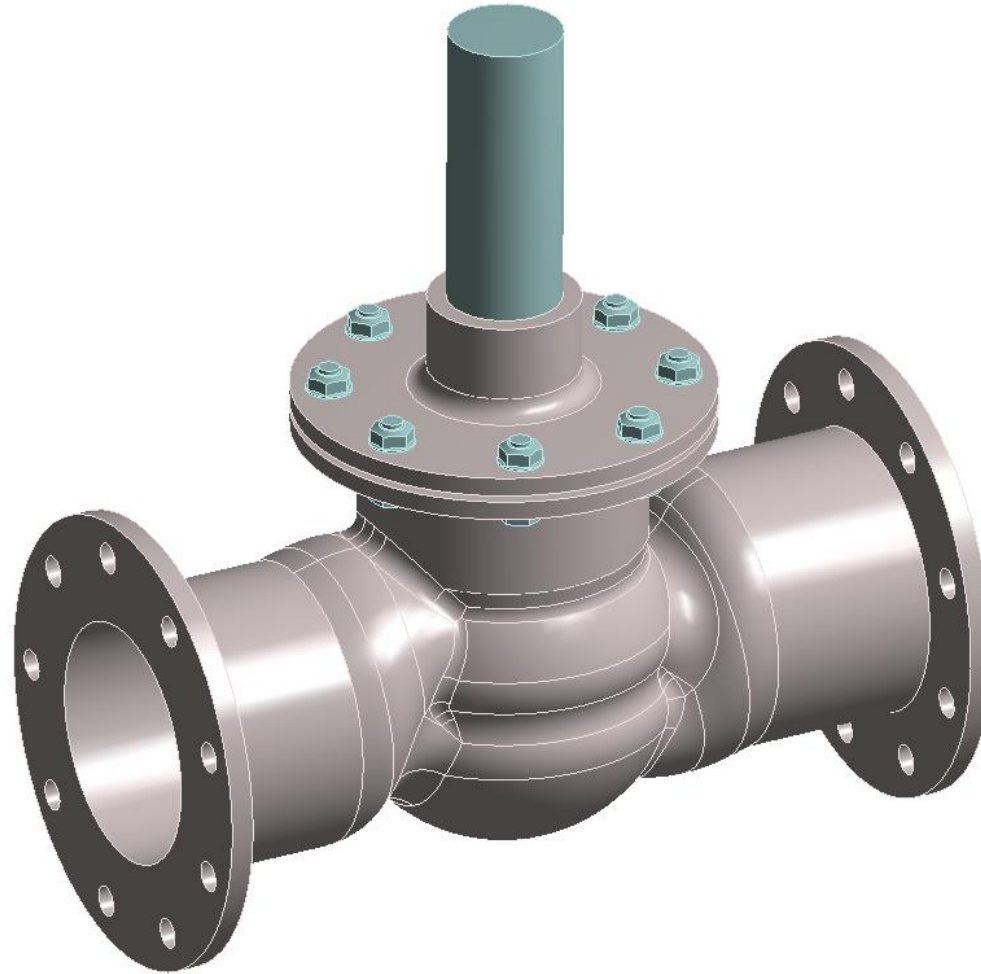▪ Establish a foundation for the rest of the course

Module 01: Graphics

Five Questions:

1. What does my part or assembly look like?

2. What is my part or assembly made of?

3. How is my part or assembly supported?

4. How is my part or assembly loaded?

5. What do I want to learn about my part or assembly?

Module 01: Graphics

Model

Module 01: Reference Material

This section contains links to supporting reference and background information for topics from this module. Unless noted otherwise, their use will require authenticated access to the Ansys Help System or to the Ansys Customer Portal.

- [Analysis System Cell States (Status Icons)](#)

Typical Cell States

State	Icon	Description
Unfulfilled		Required upstream data does not exist. Some applications may not allow you to open them with the cell in this state. For example, if you have not yet assigned a geometry to a system, all downstream cells appear as unfulfilled, because they cannot progress until you assign a geometry.
Refresh Required		Upstream data has changed since the last refresh or update. You may or may not need to update output data. When a cell is in this state, you can edit the cell, refresh the data, update upstream components, or update the cell. The advantage to refreshing rather than updating a cell is that you are alerted to potential effects on downstream cells and make any necessary adjustments before you update it. This option is especially useful if you have a complex system in which an update could take significant time and computer resources.
Attention Required		All of the cell's inputs are current. However, you must take a corrective action to proceed. To complete the corrective action, you may need to interact with this cell or with an upstream cell that provides data to this cell. Cells in this state cannot be updated until the corrective action is taken. This state can also signify that no upstream data is available, but you can still interact with the cell. For instance, some applications support an "empty" mode of operation in which it is possible to enter the application and perform operations regardless of the consumption of upstream data.
Update Required		Local data has changed and the output of the cell must be updated.
Up to Date		An update has been performed on the cell and no failures have occurred. It is possible to edit the cell and for the cell to provide up-to-date generated data to other cells.
Input Changes Pending		The cell is locally up-to-date but may change when next updated as a result of changes made to upstream cells.

Module 01: Reference Material

- [Fixed Supports](#)

Details View Properties

The selections available in the Details view are described below.

| Category	Fields/Options/Description

Scope	**Scoping Method**: Options include:
	
	• **Geometry Selection**: Default setting, indicating that the boundary condition is applied to a geometry or geometries, which are chosen using a graphical selection tools.
	
	    ○ **Geometry**: Visible when the **Scoping Method** is set to **Geometry Selection**. Displays the type of geometry (Body, Face, etc.) and the number of geometric entities (for example: 1 Body, 2 Edges) to which the boundary has been applied using the selection tools.
	
	• **Named Selection**: Indicates that the geometry selection is defined by a Named Selection.
	
	    ○ **Named Selection**: Visible when the **Scoping Method** is set to **Named Selection**. This field provides a drop-down list of available user-defined Named Selections.

Definition	**Type**: Read-only field that describes the object - **Fixed Support**. **Suppressed**: Include (**No** - default) or exclude (**Yes**) the boundary condition.

Ansys

Module 01: Reference Material

- **Rotate/Pan/Zoom and Other Viewing Options**

Rotating the Model

Selecting the **Rotate** option on the Graphics toolbar enables you to turn your model about a default or user-selected center using the left mouse button. This is a common application feature. By default, the rotational center is the center of your model.

To rotate about a specific point on the model, select a new point of rotation on your model with the left mouse button. This action recenters your model in the **Geometry** window and displays a red sphere that indicates the newly selected center of rotation. From this position, you can rotate your model freely about the new rotation point. To restore the default rotation point, simply click off of the model.
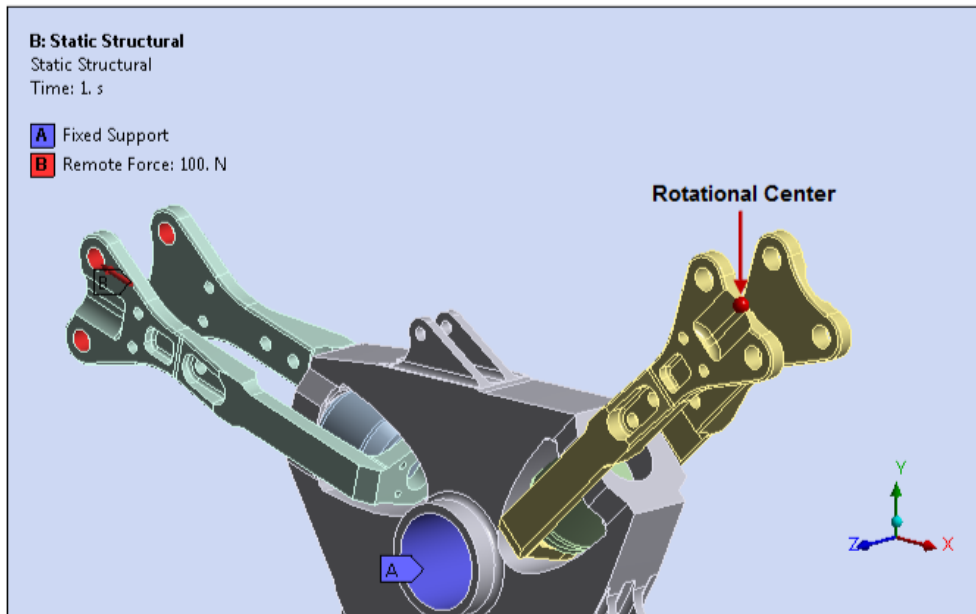
Module 01: Reference Material

- **Pressure Loads**

Pressure

A pressure load applies a constant pressure or a varying pressure in a single direction (x, y, or z) to one or more flat or curved faces. A positive value for pressure acts into the face, compressing the solid body.

Analysis Types

Pressure is available for the following analysis types:

- Coupled Field Harmonic

- Coupled Field Static

- Coupled Field Transient

- Explicit Dynamics

- Harmonic Acoustics

- Harmonic Response

- Static Structural

- Transient Structural

Module 01: Reference Material

- [Force Loads](#)

Force

Force is specified based on the following topologies:

- **Vertex**: Applies a force vector to one or more vertices.

- **Edge**: Distributes a force vector along one or more straight or curved edges, resulting in uniform line load along the edge.

- **Face**: Distributes a force vector across one or more flat or curved faces, resulting in uniform traction across the face.

- **Node**: Applies a force to an individual node or a set of nodes. This scoping is the same as using an Nodal Force except that you scope the nodes directly (no Named Selection is required). As such, the force is applied using the Mechanical APDL **F** command.

> **Note:** Node-based scoping is not supported for Harmonic Response or Explicit Dynamics analyses.

- **Element Face**: Distributes a force across one or more element faces.

Module 01: Reference Material

- **Equivalent Stress Results**

Equivalent (von Mises)

Equivalent stress is related to the principal stresses by the equation:

$$\sigma_e = \left[\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2} \right]^{1/2}$$

Equivalent stress (also called *von Mises stress*) is often used in design work because it allows any arbitrary three-dimensional stress state to be represented as a single positive stress value. Equivalent stress is part of the maximum equivalent stress failure theory used to predict yielding in a ductile material.

The von Mises or equivalent strain ε_e is computed as:

$$\varepsilon_e = \frac{1}{1+\nu} \left(\frac{1}{2} \left[(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_3 - \varepsilon_1)^2 \right] \right)^{\frac{1}{2}}$$

where:

ν' = effective Poisson's ratio, which is defined as follows:

- Material Poisson's ratio for elastic and thermal strains computed at the reference temperature of the body.

- 0.5 for plastic strains.

> **Note:** Currently, for Linked MSUP analyses with the **Expand Results From** detail under **Output Controls** set to **Modal Solution**, the Mechanical APDL solver does not calculate equivalent strains. If you choose to display equivalent strain results, you will see zero contours.

/Ansys

Module 01: Reference Material

- Outline Tree Object States (Status Icons)

Understanding the Tree Outline

The tree Outline uses the following conventions:

- Icons appear to the left of objects in the tree. Their intent is to provide a quick visual reference to the identity of the object. For example, icons for part and body objects (within the **Geometry** object folder) can help distinguish solid, surface and line bodies.

- A ⊞ symbol to the left of an item's icon indicates that it contains associated subitems. Click to expand the item and display its contents.

- To collapse all expanded items at once, double-click the Project name at the top of the tree.

- Drag-and-drop function to move and copy objects.

- To delete a tree object from the Outline, right-click the object and select **Delete**. A confirmation dialog asks if you want to delete the object.

- Filter tree contents and expand the tree by setting a filter and then clicking the **Expand on Refresh** button.

Ansys

End of presentation