# Module 07 Student Reference Guide:

# Analysis Settings, Loads, and Supports

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 07: Learning Objectives

Upon successful completion of this module, the student should be able to:

- Explain the behavior and definition of a Pressure load

- Understand the general process of defining and editing geometry selections

- Explain the behavior and definition of a Frictionless Support

- Appreciate some basic considerations surrounding the relationships between Ansys supports and physical reality

- Understand the basic reasons for the need to constrain all rigid-body motions

# Module 07: Learning Objectives

Upon successful completion of this module, the student should be able to:

- Demonstrate familiarity with the basic features of the environment branch graphics display

- Understand the similarities and differences between contact regions and supports

- Understand the general purpose of the Analysis Settings branch

# Module 07: Graphics

**Goals:**

- **Introduce the environment branch**

- **Define loads and supports**

- **Discuss Analysis Settings**

**/A**nsys

# Module 07: Graphics

**Other Points:**

1. Selecting the environment branch displays all defined loads and supports.

# Module 07: Graphics

Other Points:

2. Contact regions relate modeled bodies to each other. Supports relate modeled bodies to ground.

# Module 07: Graphics

**Other Points:**

3.  **Each body in the model must be fully constrained against rigid-body motion.**

# Module 07: 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.

- Pressure load

https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_Pressure_Loads.html



SETTING UP BOUNDARY CONDITIONS | TYPES OF BOUNDARY CONDITIONS | LOAD TYPE BOUNDARY CONDITIONS |       PRINT PAGE

**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

> **Note:** Eigen response (an Eigenvalue Buckling Analysis or a Modal Analysis) and Harmonic Response (Full) analyses take into account any pressure load stiffness contribution applied in a linked Static Structural analysis.
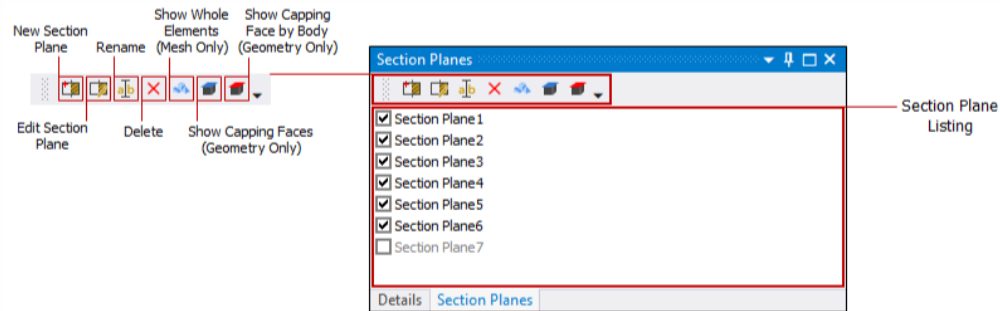
**Common Characteristics**

# Module 07: Reference Material

- Creating Section Planes

https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_New_Sec_Plane.html
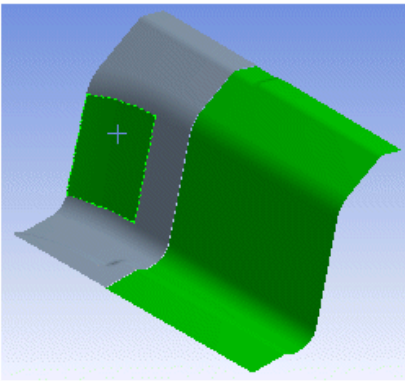
# Module 07: Reference Material

- Selecting Geometry using "Extend To"

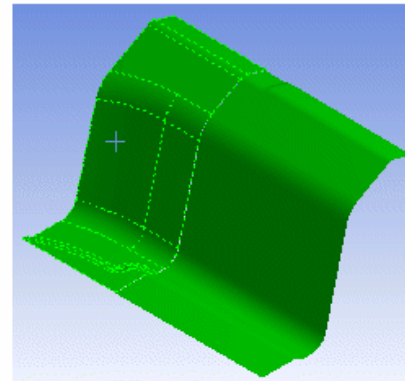[https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_Graphical_Selection.html](https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_Graphical_Selection.html)

## Limits

- For *faces*, the **Limits** option searches for faces that are tangent to the current selection as well as all faces that are tangent to each of the additional selections within the part. The selections must meet an angular tolerance along their shared edges.



Single face selected in part on the left.



Additional tangent faces selected after **Extend to Limits** option is chosen.

- For *edges*, the **Limits** option searches for edges that are tangent to the current selection as well as all edges that are tangent to each of the additional selections within the part. The selections must meet an angular tolerance along their shared vertices.

# Module 07: Reference Material

- Selecting Geometry using "Mini Selection Toolbar"

https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_Graphical_Selection.html

## Mini Selection Toolbar

When you are making geometric selections on your model, such as scoping contact conditions, boundary conditions, and/or results, a mini toolbar automatically displays in the **Geometry** window. This toolbar enables you to make selection changes "on the fly." Toolbar options include:

- **Apply Selection**: Replace scoping with the current geometry selection.

- **Add to:** Add the current geometry selection to the existing scoping.

- **Remove from**: Remove the current geometry selection from the existing scoping.

In addition, when you are using the Smart Select option option, an option to select the parent body of your current selection is also available on the toolbar.
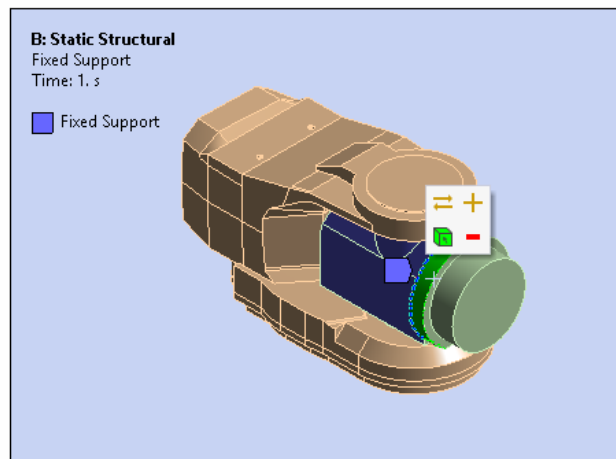
# Module 07: Reference Material

- Frictionless Support

https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_Frictionless_Surface.html

## Frictionless Support

You use this boundary condition to prevent one or more flat or curved faces from moving or deforming in the normal direction. The normal direction is relative to the selected geometry face. No portion of the surface body can move, rotate, or deform normal to the face.

For tangential directions, the surface body is free to move, rotate, and deform tangential to the face.

For a flat surface body, the frictionless support is equivalent to a symmetry condition.

> **Important:** Due to an internal processing requirement, if you specify a user-defined (local) Coordinate System when defining this boundary condition, the nodal coordinate system axes may differ from the local Coordinate System axes. As needed, you can verify the actual nodal orientation in the Mechanical APDL application.
>
> Also, see the Converting Boundary Conditions to Nodal DOF Constraints (Mechanical APDL Solver) section for more information about how the application manages nodal DOF constraints.

## Analysis Types

A **Frictionless Support** is available for the following analysis types:

- Coupled Field Analyses
- Harmonic Acoustics
- Harmonic Response
- Modal
- Modal Acoustics
- Static Structural

# Module 07: Reference Material

- Analysis Settings for Most Analysis Types
  https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_Analysis_Settings_Types.html

CONFIGURING ANALYSIS SETTINGS  |                                                                                    PRINT PAGE

## Analysis Settings for Most Analysis Types

When you define an analysis type, an Analysis Settings object is automatically inserted in the Mechanical application tree. With this object selected, you can define various solution options in the Details view that are customized to the specific analysis type, such as enabling large deflection for a stress analysis.

The available control groups as well as the control settings within each group vary depending on the analysis type you have chosen. The sections that follow outline the availability of the control settings for each of these groups and describe the controls available in each group.

- Step Controls for Static and Transient Analyses
- Step Controls for Harmonic Analysis Types
- Additive Manufacturing Controls
- Solver Controls
- Restart Analysis
- Restart Controls
- Adaptivity Remeshing Controls
- Creep Controls
- Fracture Controls
- Cyclic Controls
- Radiosity Controls
- Options for Analyses
- Scattering Controls
- Advanced
- Damping Controls
- Nonlinear Controls
- Output Controls
- Analysis Data Management
- Rotordynamics Controls
- Visibility

# Module 07: Reference Material

- Understanding Solving
  https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_solve_interface.html



**Solve Options**

**Home Tab**                    **Context Tab**
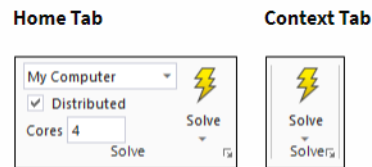
The **Solve** group provides options that enable you to specify some basic solution configurations and to solve your analysis.

- The **Solver Handler** drop-down menu enables you to specify the desired target machine on which to perform your solution. It contains the options **My Computer** (default) and **My Computer, Background**. When you create remote solution options, they are also displayed in the menu.

- The **Solve** button/option also provides a drop-down menu that contains the same options as the **Solver Handler** menu. These options initiate the solution when selected.

- The **Distributed** option, active by default, indicates the use of distributed-memory parallel (DMP) processing. For more information, see the Using DMP Processing section of the Mechanical APDL Parallel Processing Guide.

- The **Cores** field enables you to change the number of CPU cores to use during the solution. The default is **4**. If your computer does not have four cores, the application automatically scales to the lower number.

- The **Resource Prediction** option opens a window that enables you to select an available analysis and produce a prediction the computing resources needed to perform a solution for the environment.

  **Important:** In the lower right-hand corner of the **Solve** group is an option that launches the Solve Process Settings dialog. This dialog enables you to specify solution settings for remote computers as well as specific process settings to be used during the solution process.

# Module 07: Reference Material

- Solution Progress Display
https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_solve_interface.html

## Solution Progress Display

Once you start a solution, its progress is displayed in the Status Bar. The status bar also displays solution Interrupt and Stop options. If you would like the solver to halt immediately and forego writing any outstanding restart points, select the Stop option. Using the Interrupt option, you instruct the solver to complete its current iteration and record outstanding restart points (available for Static Structural, Transient Structural, and Topology Optimization analyses). Neither of these options affects previous restart points.

In addition, you can select the Progress pane to display the **Solution Status** window. Like the status bar display, this window displays the progress for synchronous solutions. The window also includes a **Stop Solution** button and an **Interrupt Solution** button. Using the **UI Options** preference setting **Progress**, you can choose to always display the progress window.

> **Note:**
>
> - Clicking the **Interrupt Solution** button places a file named `file.abt` in the working directory. This may be useful for those who are familiar with Mechanical APDL functionality.
>
> - The Ansys Distributed Compute Services DCS application does not support the Interrupt option.

# Module 07: Reference Material

- Equivalent (von Mises) Stress and Strain
  https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_Equiv_Stress.html

## 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 $\varepsilon_e$ 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:

$\nu'$ = 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.

# Module 07: Reference Material

- Deformation

https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_Deformation.html

USING RESULTS | STRUCTURAL RESULTS |                                          PRINT PAGE

## Deformation

Physical deformations can be calculated on and inside a part or an assembly. Fixed supports prevent deformation: locations without a fixed support usually experience deformation relative to the original location. Deformations are calculated relative to the part or assembly *world coordinate system*.

$$U = \sqrt{U_x^2 + U_y^2 + U_z^2}$$
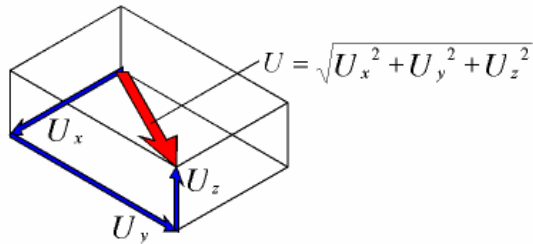
- ■ Component deformations (**Directional Deformation**)

- ■ Deformed shape (**Total Deformation** vector)

The three component deformations $U_x$, $U_y$, and $U_z$, and the deformed shape U are available as individual results.

Scoping is also possible to both geometric entities and to underlying meshing entities (see example below). Numerical data is for deformation in the global X, Y, and Z directions. These results can be viewed with the model under wireframe display, facilitating their visibility at interior nodes.

**End of presentation**