

Introduction to ANSYS Mechanical

Realize Your Product Promise®

ANSYS Chapter Overview

In this chapter we introduce the basic features in Mechanical :

- A. Basic Analysis Procedure
- **B.** The Mechanical Interface
- C. Menus
- **D.** Toolbars
- E. Outline Tree and Details
- F. Graphics Window
- **G.** Scoping Loads and Supports
- H. Demonstration
- I. The Engineering Data application
- J. Assigning Material Properties
- K. Demonstration
- L. Workshop 2-1
- M. Appendix
- 2 © 2015 ANSYS, Inc. February 27, 2015

ANSYS A. Basic Analysis Procedure

A finite element analysis is used to determine the *response* of a system based on some type of *loading*.

It is important to remember that a finite element solution is an approximation:

- CAD geometry is an idealization of the physical model.
- The *mesh* is a combination of discrete "elements" representing the geometry.
- The accuracy of answers is determined by various factors, one of which is the mesh density.



ANSYS ... Basic Analysis Procedure

An analysis can be described in the context of four main steps:

- Preliminary Decisions
 - What type of analysis: Static, modal, etc.?
 - What to model: Part or Assembly?
 - Which elements: Surface or Solid Bodies?
 - Should I start with a simpler model?
 - Symmetry?
- Preprocessing
 - Attach the model geometry
 - Define and assign material properties to parts
 - Mesh the geometry
 - Apply loads and supports
 - Request results
- Solve the Model
- Postprocessing
 - Review results
 - Check the validity of the solution



ANSYS B. The Mechanical Interface

The components of the user interface are shown below:



^{5 © 2015} ANSYS, Inc. February 27, 2015

ANSYS ... The Mechanical Interface

A range of menus and toolbars provide much of the general functionality in Mechanical.

• The title bar lists analysis type, product and active ANSYS license.



Some menu and toolbar items are self explanatory. We'll cover the basic controls on the next few slides.

• Additional controls and features will be introduced throughout the course as they are encountered.

Note: the windows in Workbench and the Mechanical application can be customized. If you wish to return to a default layout use in either:

Mechanical: "View > Windows > Layout > Reset Window Layout".

Workbench: "View > Reset Window Layout".

ANSYS D. Toolbars

The "Standard" toolbar is shown below (details will be covered later):



- A. Activate the Mechanical Wizard.
- B. Object Generator.
- C. Solve.
- D. Populate the message windows with the appropriate error message for any tree objects that are not properly defined.
- E. Create slice planes, annotations, charts and tables.
- F. Add comments and figures to the tree.
- G. Activate optional Worksheet view.
- H. Activate selection information window.
- 7 © 2015 ANSYS, Inc. February 27, 2015



The Context toolbar updates depending on current tree selection.



- Context sensitive features are exposed as different branches are highlighted.
- In almost all cases an alternate path to these features is available by right clicking (RMB), when the branch is highlighted.

ANSYS ... Toolbars

The Graphics toolbar graphical selection section:

- Select Geometry (vertex, face, etc.):
 - Use single or box select modes.
- Select Mesh (nodes):

© 2015 ANSYS, Inc.

- Use single or box select mode (including box, box volume and lasso volume).
- Select mesh is only available when the mesh is displayed.

February 27, 2015





9



Selection planes allow for users to easily select entities that are overlaying one another.

- An initial selection point acts as the starting point for a path through the model.
- Each entity encountered is displayed by a selection plane.
- The path through the model is in the normal Z (viewing) direction.



• Example using surfaces: initial selection location.

ANSYS ... Toolbars

The Graphics toolbar graphical manipulation section (left mouse button):

- A. Common features (rotate, pan, zoom, box zoom) are to the left of the toolbar.
- **B.** A "fit" button and magnifier window toggle are available.
- **C.** When zooming, a "stack" is stored and can be retraced using previous or next buttons.
- D. Isometric view.
- E. "Look At" selected entities, reorients view normal to current selection.
- F. Manage Views.
- **G.** Resize annotations on loads and supports after zooming.
- H. Tags.

ANSYS E. Outline Tree and Details

The Outline Tree branches represent various operations. Each branch has an associated status symbol.

 Becoming familiar with the status symbols will allow you to debug Mechanical problems quickly.

Solution Branch Icons



Symbol

2 Pressure

1 Thermal Condition

Mapped Face Meshing

, Fixed Support

👘 Result Object

😭 Rec_Bar

🗙 🍘 Solid

🗙 🗊 Beam

4 Result Object

Status Symbol Name

Needs to be Updated

Mapped Face or Match Control Failure

Underdefined

Error

Ok

Hidden

Meshed

Suppress

Solve

ANSYS ... Outline Tree and Details

The Details View contains input and output fields (the contents will change depending on the branch selected):

- White field: input data that can be edited.
- Yellow field: incomplete input data.
- Gray field: information only, cannot be modified.
- Red field: Result no longer up to date (must re-solve).

De	etails of "Equivaler	nt Stress"	μ			
Ξ	Scope					
	Scoping Method	Geometry Selection				
	Geometry	All Bodies				
Ŧ	Definition					
Ŧ	Integration Point Results					
Ξ	Results					
Minimum		3.1675e-003 MPa				
	Maximum 7.1737 MPa					
Ξ	Minimum Value Over Time					
	Minimum 3.1675e-003 MPa					
	Maximum 3.1675e-003 MPa					
	Maximum Value Over Time					
	Minimum	7.1737 MPa				
	Maximum 7.1737 MPa					
Ŧ	Information					



ANSYS ... Outline Tree and Details

File Edit View Units Tools Help	📁 Solve 👻 ?./ Show Errors 🟥 📷 🕪 📣 💽 🖝 💕 Worksheet	ik
😨 🚧 🧚 🗽 🕟 🗕 🗞 🖛 🚺 🙀) + 'G 💠 <mark>9/4</mark> (Q Q Q 🐺 /2 🗐 🖴 🗞 🗖 +	•
Fri Show Vertices 🙀 Wireframe 🖓 Show Mesh	🤸 📕 Pandom Colors 🖉 Annotation Preferences 📗 📕 Edge Coloring 👻	$/_{1} \leftarrow /_{1} \leftarrow /_{2} \leftarrow /_{3} \leftarrow /_{X} \leftarrow \neq \parallel \rightarrow \parallel \qquad \qquad$
Environment 🍕 Inertial 🔻 🍕 Loads 👻 🍕 Support	s - 🕵 Conditions マ 🖏 Direct FE マ 👔	
Outline		
Filter: Name 🔻	A: Static Structural Fixed Support	
Project	Time: 1. s	
Em Model (A4)	11/25/2013 11:27 AM	
	Fixed Support	
Static Structural (A5)		
Analysis Settings		
Fixed Support		
Solution Information		
Details of "Fixed Support"		
E Scope		
Scoping Method Geometry Selection		
Geometry 1 Face		
Type Fixed Support		
Suppressed No		0.00
		0.00
		500.00
	Geometry Print Preview Report Preview	
	Messages	
	Test	Association
	Error You need at least one structural load to proceed with the solution.	Project>Model>Static Structural

As said before, becoming familiar with the status symbols will allow you to debug Mechanical problems quickly. To help you in this task, you can use the "Show Errors" button.

14 © 2015 ANSYS, Inc. February 27, 2015

ANSYS F. Graphics Window

The Graphics Window shows the geometry and results. Tabs allow access to Print and Report Previews as well.



ANSYS ... Graphics Window

Worksheet views are available for many objects in the tree (i.e. geometry, connections, etc.).

Provides a list view of the data in the tree.



^{16 © 2015} ANSYS, Inc. February 27, 2015

ANSYS[®]

... Graphics Window



Keyboard Shortcuts are supported in Mechanical for common actions such as: Select All Objects (Ctrl+ A), Body Filter Selection (Ctrl+B), Zoom to Fit (F7).

ANSYS G. Scoping Loads & Supports

Loads and supports can be applied as:

🗣 Inertial 🔹 🗣 Loads 🔹 🗣 Supports 🔹 🗣 Conditions 🔹 🖤 Direct FE 🔹

- **1.** Scope, then action (pre-select):
 - Select geometry entity in Graphics Window, then select load or support (context toolbar or RMB > insert).
 - Define magnitude and direction (if required).

OR

- 2. Action, then scope:
 - Select load or support from the context menu or RMB > Insert.
 - Select the scope, then "Apply" in the details.
 - Define magnitude and direction (if required).

Notes:

- Preselecting (1) is more efficient since it avoids having to use the Apply/Cancel function (preselection is automatically "applied").
- If you wish to change a boundary condition's location simply click in the geometry field to bring up the Apply/Cancel selections and make a new selection.



ANSYS ... Scoping Loads & Supports

Some loads require a direction. There are 2 methods of direction control:

Component Method:

- In the details view set "Define By" to "Components".
- Select the desired coordinate system (local or global).
- Enter X, Y, and Z magnitudes.



ANSYS ... Scoping Loads & Supports

Vector Method:

- In the details view set "Define By: to "Vector".
- Enter the load magnitude.
- Click the "Direction" field and choose the control geometry (vertex pairs, edges or surfaces).
- "Apply" to confirm.
- To modify directions using the vector method:
 - Click in the "Direction" field and select new control geometry.
 - Use the arrows in the Graphics window to reverse directions.
 - "Apply" when finished.

Default method can be set in: "Tools > Options ... > Mechanical: Miscellaneous> Load Orientation Type".

Ξ	Scope					
	Scoping Method	Geometry Selection				
	Geometry	1 Face				
Ξ	Definition					
	Туре	Bearing Load				
	Define By	Vector				
	🗌 Magnitude	10. N				
	Direction	Click to Change				
	Suppressed	No				



Note: structural simulations allow certain boundary conditions to be applied directly to the finite element nodes. The technique is covered in lecture 5.

ANSYS H.Demonstration

Demonstration : 02_Mechanical_Intro.mp4

- What you can learn in this demonstration :
 - Mechanical Interface presentation
 - Details on graphical window
 - How to manipulate a model
 - Toolbar description
 - Outline Tree branches description
 - Use the Details window
 - Scoping loads and supports



ANSYS I. The Engineering Data Application

The Engineering Data application provides control for material properties.

• Engineering Data can be opened "stand alone" or from the analysis system cell (double click or RMB>Edit).



- Note : A Physics Filter toggles between displaying all materials and properties or only materials relevant to the analysis systems in the project Filter Engineering Data
- A key concept in Engineering Data is that materials must be checked out of a material library and into a project before they can be used in your analysis



ANSYS ... The Engineering Data Application

Project View



IMPORTANT!: Only materials shown in the Project view will be available in the

			anan	VSIS.
20	Strain-Life Parameters			
28	🔁 Tensile Yield Strength	2.5E+08	Pa 📃 🗖	1.1 -
29	Compressive Yield Strength	2.5E+08	Pa 🗾 🗖 🗖	1 I
30	🔀 Tensile Ultimate Strength	4.6E+08	Pa 🗾 🗖	-1
31	Compressive Ultimate Strength	0	Pa 🗾 🗖	



ANSYS ... The Engineering Data Application

Users can define their own material or use one from the « Engineering data source »

To use « Engineering data source », click on

Highlight the desired library

Click the "+" next to the desired material

Note : users can check material properties before import in the project view



Engineering Data Sources

ANSYS . . . The Engineering Data Application

To create a new material:

- Toggle to the project materials display.
- Enter a name for the new material.

Engineering Data Sources = OFF

- From the Toolbox double click or drag and drop the desired properties.
- Enter values for the properties.
- Note: properties can be added to existing materials using the same technique.



Outline of Schematic A2: Engineering Data

0

Material

1

3

A

Structural Steel

Click here to add a new material

Contents of Engineering Data

В

8

9

Ge

-

ANSYS ... The Engineering Data Application

To place a new material in a library you must export it as an xml file first.

- Highlight the material name
- From the "File" menu choose "Export Engineering Data".
- Browse to the desired location to store the file.



Continued . . .

26 © 2015 ANSYS, Inc. February 27, 2015



ANSYS . . . The Engineering Data Application

Toggle to the "Data Sources" display.

🗰 Engineering Data Sources 🛛 = ON



• The material library must be unlocked (edit mode) before new materials can be added.

- From the "File" menu choose to "Import Engineering Data".
- Browse to the xml file for the new material.





ANSYS . . . The Engineering Data Application

• To create a material library toggle to the Data Sources display.

Engineering Data Sources = ON

• Enter a name and select a location (a browser will open automatically).

🔥 Unsaved Project - Workbench		services that Manhalitation				
File View Tools Units Extensions	Help					
🖺 🛃 🔜 Project 🥏 A2:Engi	neering Dat	ta 🗙				
🍸 Filter Engineering Data 🏢 Engineering Data	aSources					
Toolbox 👻 🕂 🗙	Enginee	ring Data Sources			~ д	×
Physical Properties		A	в	С	D	-
	1	Data Source		Location	Description	
Hyperelastic Experimental Data	7	Magnetic B-H Curves			B-H Curve samples specific for use in a magnetic analysis.	1
Hyperelastic ■			$\left \right $		Material camples specific for use in a thermal	1
	8	Thermal Materials			analysis.	-
		Eluid Materials			Material samples specific for use in a fluid analysis.	
	*	Click here to add a new library				١Ļ.
🖽 Life		1			1	1

Engineering Data files (libraries and materials) are stored in .xml format.

ANSYS J. Assigning Material Properties

- Material properties are assigned to parts in the "Material Assignment" field in the part's details (Mechanical application).
- Notice there are several shortcuts allowing quick access to the Engineering Data application in Workbench.



ANSYS K. Demonstration Video

Demonstration : 03_Engineering_Data.mp4

- What you can learn in this demonstration :
 - Define new material and edit properties
 - Export / Import created materials
 - Use Ansys material library
 - Assign material properties to parts



ANSYS L. Workshop 2-1 – Mechanical Basics

- Workshop 2.1 Mechanical Basics
- Goal:
 - Using the Stress Wizard, set up and solve a structural model for stress, deflection and safety factor.







M. APPENDIX

- Menus
- Graphics Control and Selection
- Tagging and tree filtering
- Grouping tree object
- The Mechanical application wizard



The View menu:

• Control basic graphics (shaded, wireframe, etc.).

Control graphical expansion of shells and beams.

Control display utilities (legend, triad, ruler, etc.).

• Set preferences for annotation display.

• Select the desired toolbars and windows to be displayed.



33 © 2015 ANSYS, Inc. February 27, 2015



The Units menu:

- Specify the unit system for Mechanical.
 - Note, Mechanical may use any unit system regardless of the one specified in Workbench. Where needed, automatic conversions will be made.
 - Mechanical will "remember" the units used in the previous session (<u>regardless</u> of the units set in Workbench).

• Specify additional units for angular, rotational and thermal references.





Annotation Preferences:

 Controls the display of annotations for loads/constraints, user labels, remote conditions and meshing displays.

 Make preference selections then "Apply" or "OK" to reflect the changes.

nnotation Preferences						
Basic Annotations						
View Annotations						
View User Defined Graphics Annotations						
✓ View Annotation Labels						
Remote Boundary Conditions						
✓ Point Mass ✓ Beam Connections ✓ Springs ✓ Bearings						
ţ						
Small Default Large Small Default Large						
dditional Display Preferences						
✓ Crack Annotations						
✓ Individual Force Arrows On Surface Reactions						
✓ Body Scoping Annotations						
1esh Display						
✓ Mesh Annotations						
Node Numbers Min 1 Max 100000 Inc 1						
Element Min 1 Max 100000 Inc 1						
Plot Elements Attached to Named Selections						

ANSYS Graphics Control and Selection

Graphics Shortcuts:

- Middle Mouse button (wheel):
 - Free rotation
 - + CTRL = panning
 - + Shift = zoom
- Right Mouse Button:
- Scroll mouse wheel to zoom in/out
- Hold RMB + drag = box zoom

RMB Context Menu from graphics window:

- Access isometric view controls and fit.
- Standard views built in.

Click the axes of the triad to reorient view.

Click the blue "iso" ball for isometric view.





... Graphics Control and Selection

Graphics Options toolbar:

NNSYS

- A. Show Vertices: Accentuates vertex display for easier identification.
- B. Wireframe: Toggles view (solid/wireframe).
- C. Show Mesh: Show mesh instead of geometry.
- D. Show all coordinate systems.
- E. Random Colors: Use random Colors for load and name selection annotations
- F. Annotation Preferences: explain in the next slide



ANSYS ... Graphics Control and Selection

Graphics Options toolbar:

- A. Edge Controls: Edge color control and display options based on edge connectivity (number of faces connected to an edge).
- B. Show edge direction.
- C. Show edges where mesh connections are used.
- D. Thicken line display where lines have boundary conditions scoped.

The Tree Filter:

- To streamline the management of tree objects, especially for large models, a tree filter can be used to find objects or to reduce the length of the tree that is displayed.
- Filter by: Name, Tag (discussed later), Type or State.
- Examples:



Filter by state = underdefined to find only those objects in the tree.





Filter by type = results to only display result items.

The Tree Filter Example:

- Filter Name = "shaft" (user entry)
- Notice here there are a number of branches containing the word "shaft". By using the tree filter each can be quickly found.



Tags are designations that can be assigned to any branch in the tree and used with the previously discussed filtering capability.

Activate the tags window from the "View" menu or the Tags icon.



lags

Creating Tags:

- From the Tags window choose to "Add a Tag".
- Enter a name for the new Tag in the dialog box.

19		
1003	Add a Tag	
ld Ne	ew Tag	
inter	a name for the tag:	
	01/	7



Tags Summary:

- Any object branch in the tree can be tagged.
- Objects can be associated with more than one tag.
- Manage tags from the tags window.
- Highlight Tags and RMB to find items.



Find objects in a single tag



Find objects in multiple tags

ANSYS Grouping Tree Object

For the following object types, Mechanical enables you to organize and group together like-objects :

- Geometry
- Coordinate System
- Connection features: Springs, Beam Connections, End Release, and Bearings
- Named Selections (Named Selections within the Fracture folder cannot be grouped.)
- Boundary conditions
- Results (child objects of the Solution folder)

For boundary conditions and results,

the Group Similar Objects groups together objects of the same type (e.g., Pressure, Displacement, etc.) and renames the group folder according to that type



ANSYS The Mechanical Application Wizard

The Mechanical Wizard is a useful aid in guiding users in locating the features required to complete an analysis:

- Provides a list of required tasks based on the wizard type (e.g. structural, thermal, etc.).
- Icons indicate the status of each task (similar to tree status symbols).

By selecting an item on the "Required Steps" checklist, a callout appears, illustrating where that task is performed.



