

2. ANSYS Basics

3. ANSYS Basics

- In this chapter, we will discuss the basics of how to enter and exit ANSYS, how to use the GUI and on-line help, and the database and files created by ANSYS.
- Topics Covered:
 - A. Starting ANSYS
 - B. The GUI
 - C. Graphics & Picking
 - D. On-Line Help
 - E. The Database and Files
 - F. Exiting ANSYS

ANSYS Basics

A. Starting ANSYS

- There are two ways to start ANSYS:

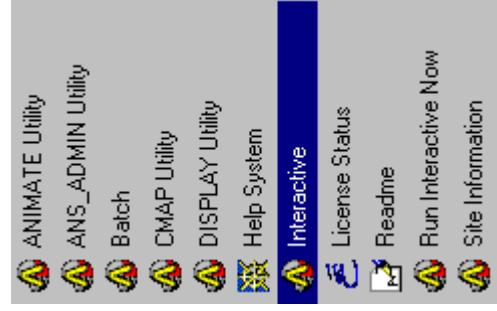
- By Launcher
- By Command Line

Launcher



Unix launcher

- Allows you to start ANSYS and other ANSYS utilities by pressing buttons on a menu.



Windows launcher

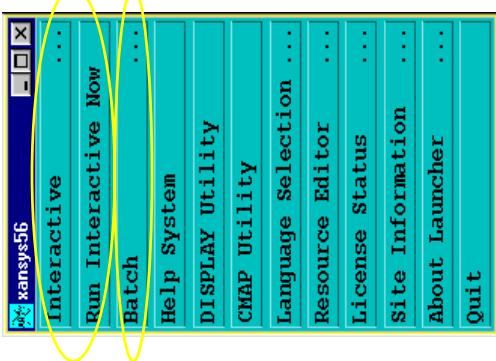
- On Unix systems, issue xansys56 & to bring up the launcher.
- On Windows systems, press Start > Programs > ANSYS 5.6.



...Starting ANSYS

Launcher (cont'd)

- Automatically brings up the GUI (Graphical User Interface) when ANSYS is started in interactive mode.



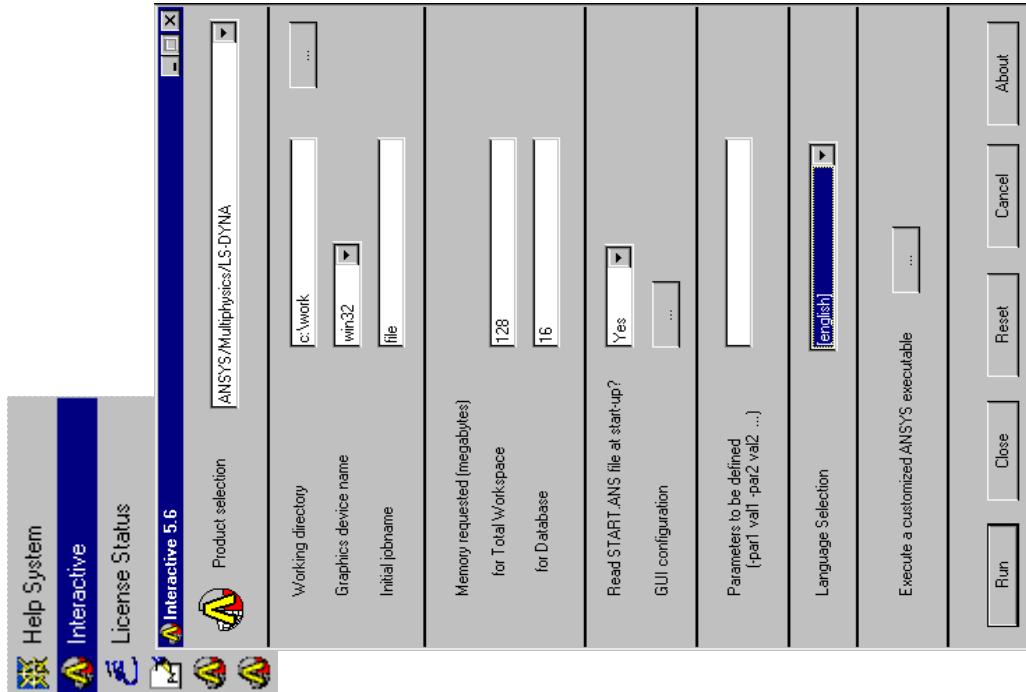
- A note on Interactive vs. Batch mode:
 - Interactive mode allows you to interact “live” with ANSYS, reviewing each operation as you go.
 - Batch mode works on an input file of commands and allows you to run ANSYS in the background.
 - No live interaction, therefore any errors in input will cause the batch run to stop.
 - Best suited for operations that don’t need user interaction, such as a solve.
 - We will mainly cover interactive mode in this course.

ANSYS Basics

...Starting ANSYS

Launcher (cont'd)

- Pressing the Interactive button on the launcher brings up a dialog box containing start-up options, e.g:
 - ANSYS product
 - Working directory - the directory in which all files will be stored.
 - Graphics device - set to 3-D if you have a 3-D graphics device card. Otherwise, set to X11 on Unix, win32 on Windows.
 - Jobname - file name prefix, up to 32 characters, assigned to all files produced by this session. Defaults to “file” or last specified name.
 - Amount of memory - default values should suffice in most cases.



...Starting ANSYS

Launcher (cont'd)

- After choosing the desired start-up options, press the Run button to start ANSYS.

Command Line

- ANSYS is started by typing a command at the system level.

For example:

- ansys56
- ansys56 -g
- ansys56 -g -j plate
- ansys56 -g -p ANE3FL -d 3d -j proj1 -m 128

Command Line (cont'd)

- **Typical start-up options, commonly known as *command line options*, are:**

`-g` (to automatically bring up the GUI upon start-up)

`-p product_code`

`-d graphics_device`

`-j jobname`

`-m memory`

- **The working directory is the directory in which the command is issued.**

- **Refer to your *ANSYS Installation and Configuration Guide* for details on the command line options.**

ANSYS Basics

B. The GUI

- Entering ANSYS brings up the following GUI windows:

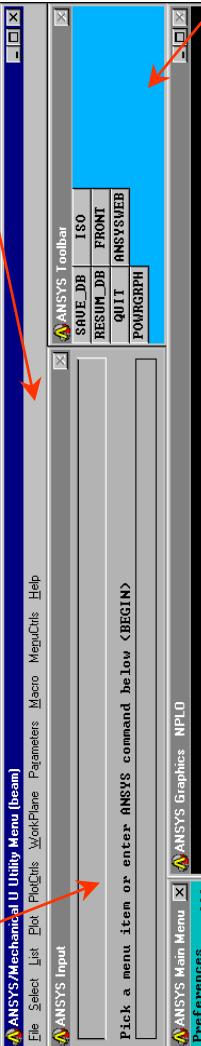
Input

Displays program prompt messages and a text field for typing commands. All previously typed commands appear for easy reference and access.



Utility Menu

Contains functions which are available throughout the ANSYS session, such as file controls, selecting, graphics controls, parameters, controls, and exiting.



Main Menu

Contains the primary ANSYS functions, organized by processors (preprocessor, solution, general postprocessor, etc.).



Toolbar

Contains push buttons for executing commonly used ANSYS commands and functions. Customized buttons can be created.

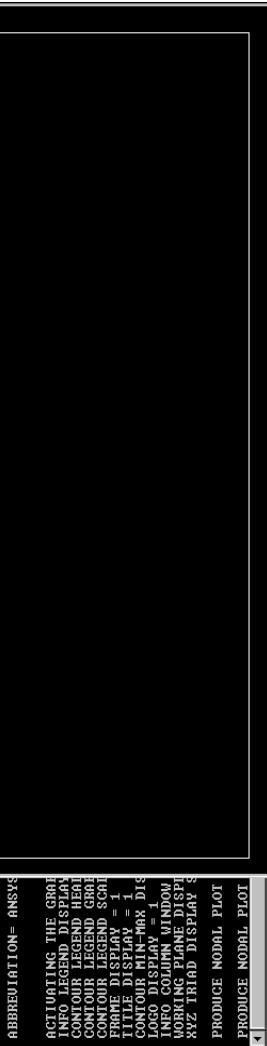


Output

Displays text output from the program. It is usually positioned behind the other windows and can be raised to the front when necessary.

Graphics

Displays graphics created in ANSYS or imported into ANSYS.

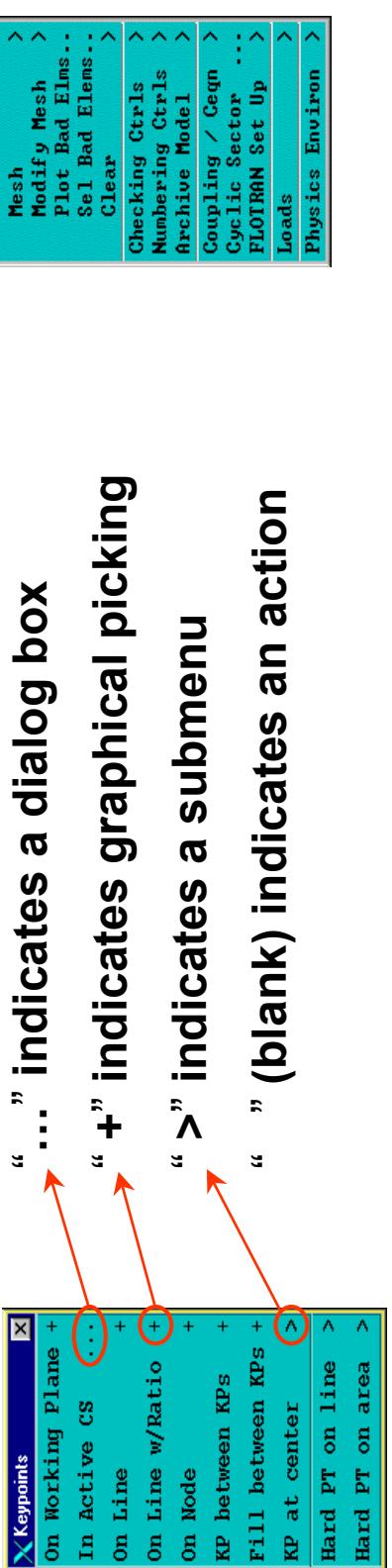


ANSYS Basics ... The GUI

Main Menu

- Contains the main functions required for an analysis.
- Independent, “sticky” windows allow you to complete all necessary steps before moving on to the next function.

Conventions:



ANSYS Basics

... The GUI

Utility Menu

- Contains utilities that are generally available throughout the ANSYS session: graphics, on-line help, select logic, file controls, etc.
- Same conventions as Main Menu:
 - “...” indicates a dialog box
 - “+” indicates graphical picking
 - “>” indicates a submenu
 - “ ” (blank) indicates an action

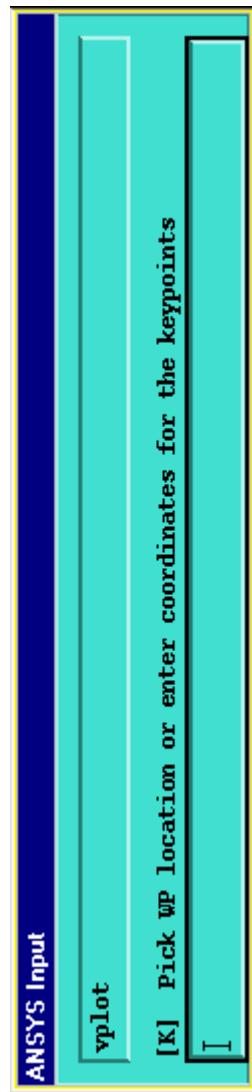
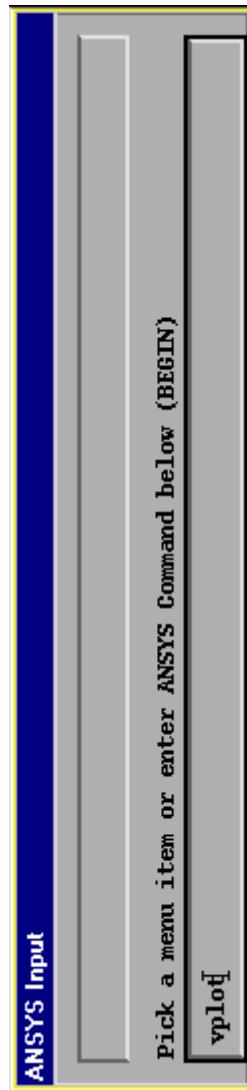


ANSYS Basics

... The GUI

Input Window

- Allows you to enter commands. (Most GUI functions actually “send” commands to ANSYS. If you know these commands, you can type them in the Input Window.)
- Also used for prompts during graphical picking.

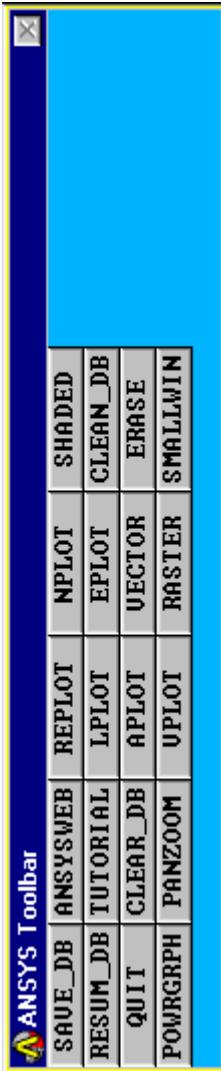


ANSYS Basics

... The GUI

Toolbar

- Contains *abbreviations* -- short-cuts to commonly used commands and functions.
- A few predefined abbreviations are available, but you can add your own. Requires knowledge of ANSYS commands.
- A powerful feature which you can use to create your own “button menu” system!

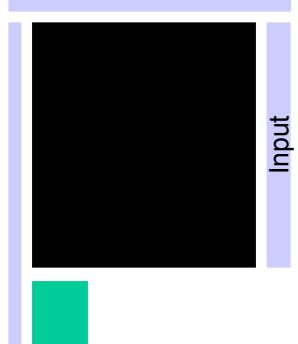
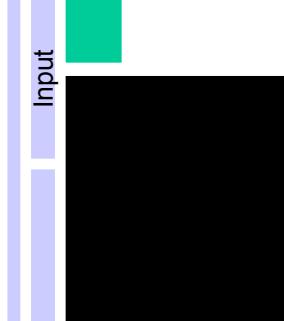
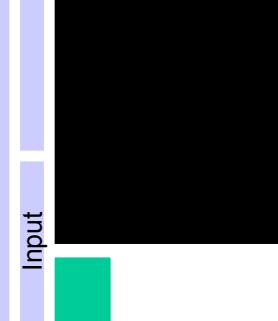


ANSYS Basics

... The GUI

Layout

- **Three predefined menu layouts are available** (Launcher > Interactive > GUI Configuration):
 - Input window on left (default)
 - Input window on right
 - Input window on bottom
- **You can also create your own layout and then save it using Utility Menu > MenuCtrls > Save Menu Layout.**
- Unix systems store the layout in an **ASCII** resource file called **ANSYS56**, located in **\$HOME**.
- Windows systems store the layout in the **system registry**.

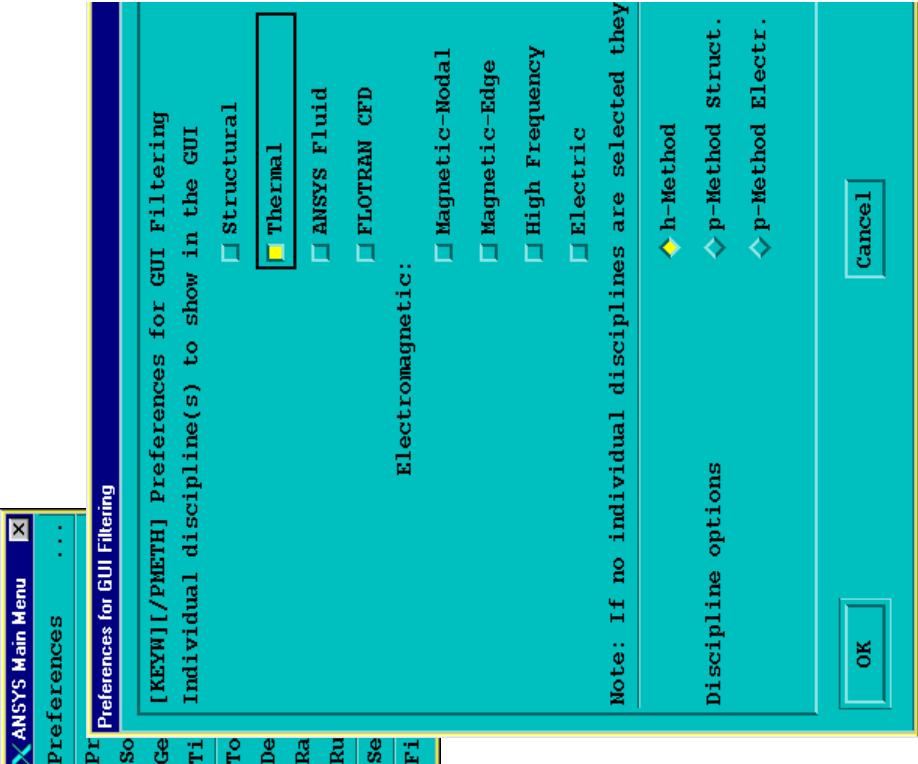


ANSYS Basics

... The GUI

Preferences

- The Preferences dialog (Main Menu > Preferences) allows you to filter out menu choices that are not applicable to the current analysis.
- For example, if you are doing a thermal analysis, you can choose to filter out other disciplines, thereby reducing the number of menu items available in the GUI:
 - Only thermal element types will be shown in the element type selection dialog.
 - Only thermal loads will be shown.
 - Etc.



ANSYS Basics

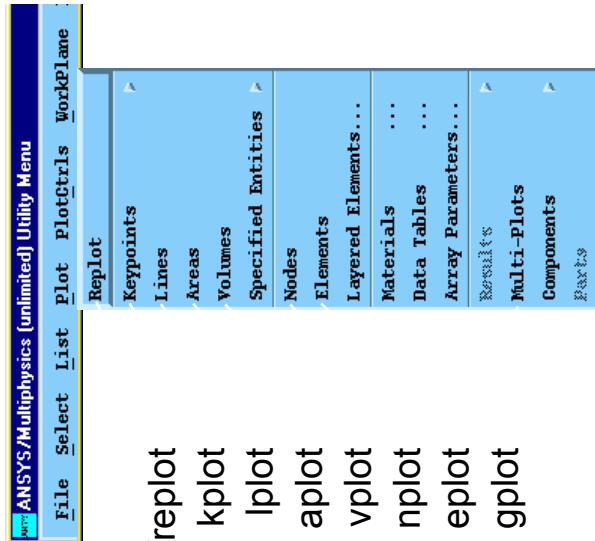
... The GUI

Other GUI Notes

- Some dialog boxes have both Apply and OK buttons.
 - Apply applies the dialog settings, but retains (does not close) the dialog box for repeated use.
 - OK applies the dialog settings and closes the dialog box.
- The Output Window is independent of the ANSYS menus.
Caution: Closing the output window closes the entire ANSYS session!
- Remember that you are not restricted to using the menus. If you know the command, feel free to enter it in the Input Window!

C. Graphics & Picking

- The most heavily used interactive capabilities are graphics and graphical picking.
 - Graphics is used to visualize the model, loading, results, and other input and output data.
 - Picking is used for model creation, meshing, loading, etc.
- Use Plot in the Utility menu to produce plots, or issue the commands shown.



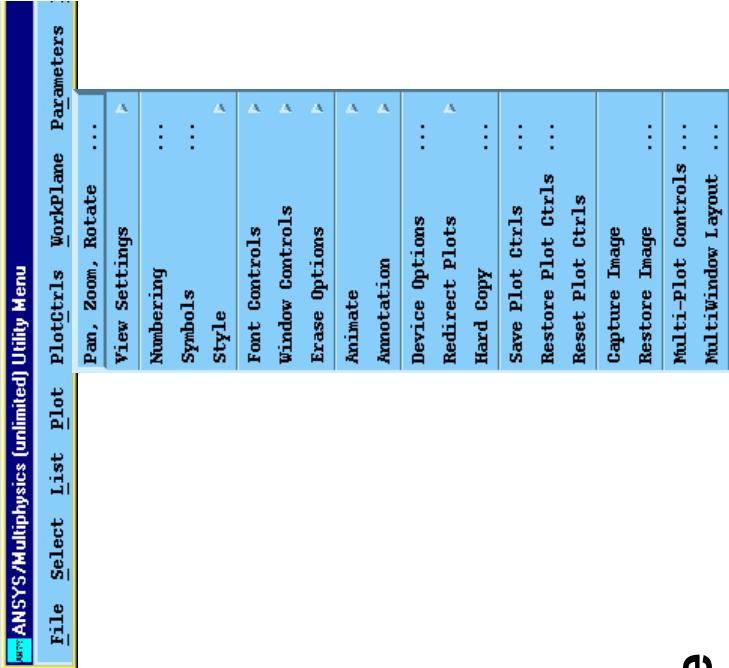
ANSYS Basics

...Graphics & Picking

- The PlotCtrls menu is used to control how the plot is displayed:

- plot orientation
- zoom
- colors
- symbols
- annotation
- animation
- etc.

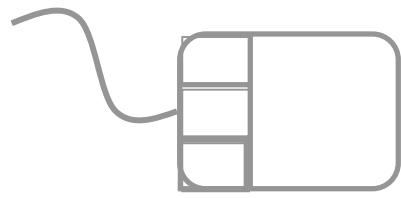
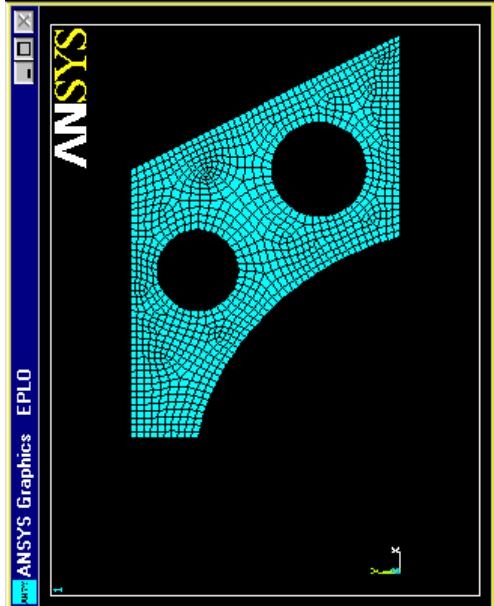
- Among these, changing the plot orientation (view) and zooming are the most commonly used functions.



ANSYS Basics

...Graphics & Picking

- The **default view for a model is the front view**: looking down the +Z axis of the model.
- To change it, use **dynamic mode** — a way to orient the plot dynamically using the **Control key and mouse buttons**.
 - Ctrl + Left mouse button pans the model.
 - Ctrl + Middle mouse button:
 - ↑ zooms the model
 - ↔ spins the model (about **screen Z**)
 - Ctrl + Right mouse button rotates the model:
 - ↑ about screen X
 - ↔ about screen Y

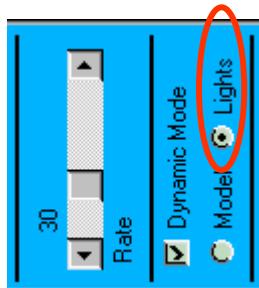
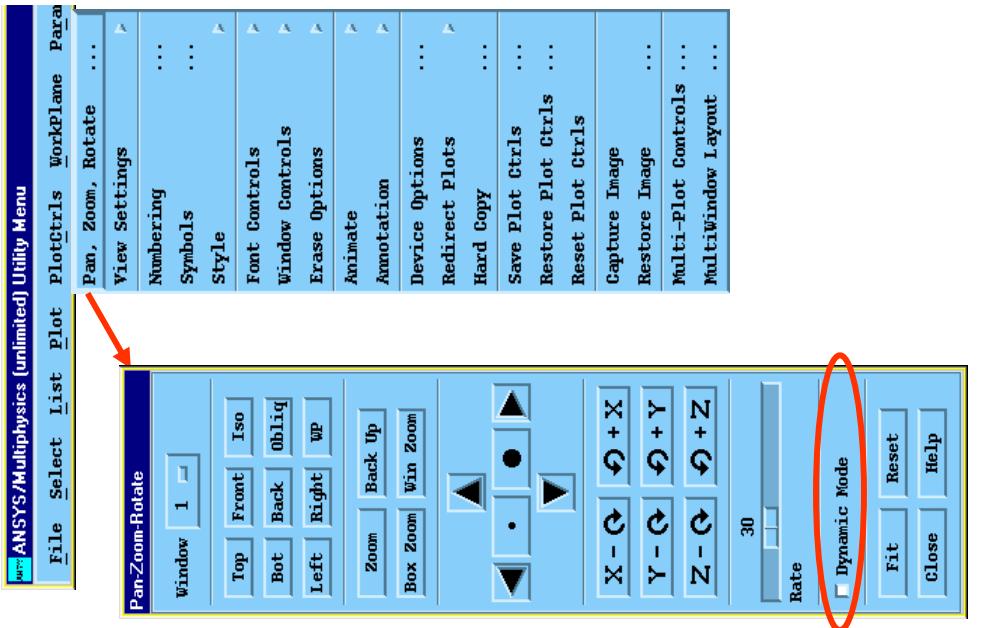


ANSYS Basics

...Graphics & Picking

- If you don't want to hold down the Control key, you can use the **Dynamic Mode** setting in the Pan-Zoom-Rotate dialog box.
 - The same mouse button assignments apply.

- On 3-D graphics devices, you can also dynamically orient the light source. Useful for different light source shading effects.

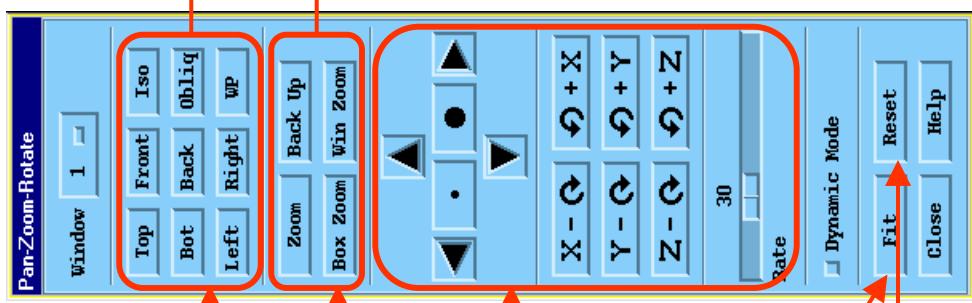


ANSYS Basics

...Graphics & Picking

- Other functions in the Pan-Zoom-Rotate dialog box:

- Preset views
- Zoom-in on specific regions of the model
- Pan, zoom, or rotate in discrete increments (as specified by the Rate slider)
- Rotation is about the screen X, Y, Z coordinates.
- Fit the plot to the window
- Reset everything to default

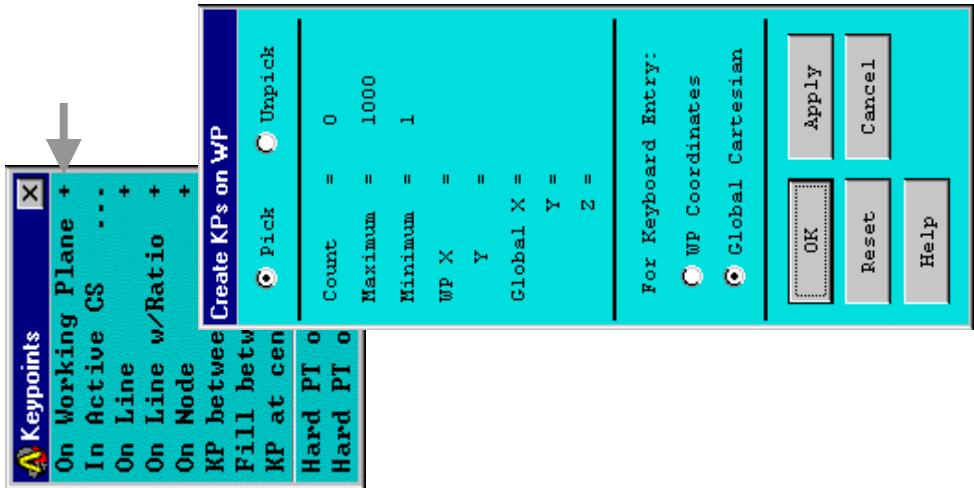


Front	+Z view, from (0,0,1)
Back	-Z view (0,0,-1)
Top	+Y view (0,1,0)
Bot	-Y view (0,-1,0)
Right	+X view (1,0,0)
Left	-X view (-1,0,0)
Iso	Isometric (1,1,1)
Oblique	Oblique (1,2,3)
WP	Working plane view
Zoom	By picking center of a square
Box Zoom	By picking two corners of a box
Win Zoom	Same as Box Zoom, but box is proportional to window.
Back Up	“Unzoom” to previous zoom.

...Graphics & Picking

Picking

- Picking allows you to identify model entities or locations by clicking in the Graphics Window.
- A picking operation typically involves the use of the mouse and a picker menu. It is indicated by a + sign on the menu.
- For example, you can create keypoints by picking locations in the Graphics Window and then pressing OK in the picker.

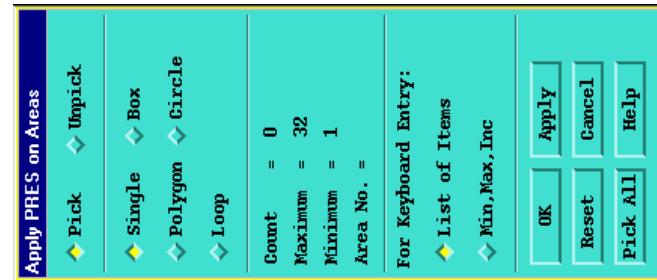


ANSYS Basics

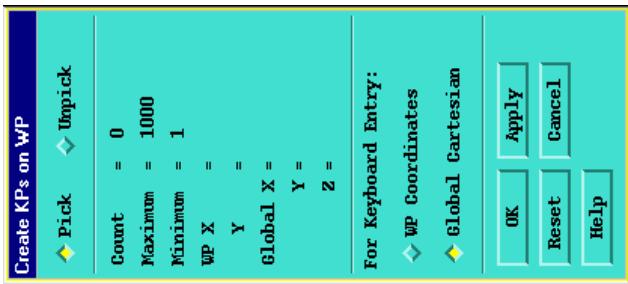
...Graphics & Picking

Two types of picking:

- **Retrieval picking**
 - Picking existing entities for a subsequent operation.
 - Allows you to enter entity numbers in the Input Window.
 - Use the *Pick All* button to indicate all entities.
- **Locational picking**
 - Locating coordinates of a point, such as a keypoint or node.
 - Allows you to enter coordinates in the Input Window.



Example of Retrieval Picker



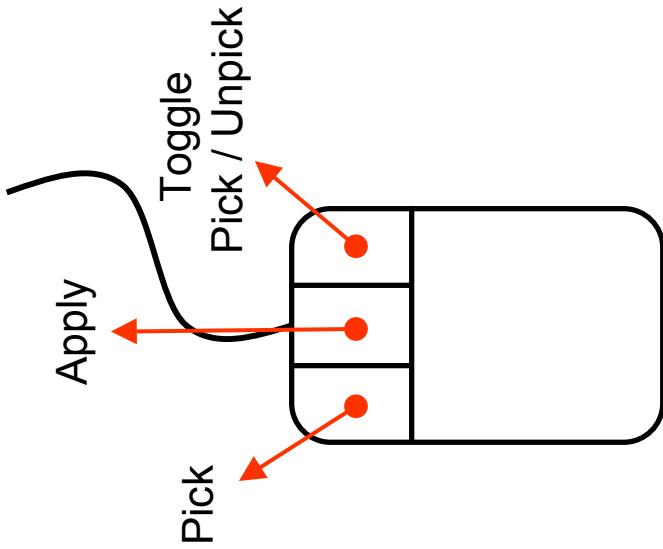
For Keyboard Entry:



Example of Locational Picker

Mouse button assignments for picking:

- **Left mouse button picks (or unpicks)** the entity or location closest to the mouse pointer. Pressing and dragging allows you to “preview” the item being picked (or unpicked).
- **Middle mouse button does an Apply.** Saves the time required to move the mouse over to the Picker and press the Apply button. Use Shift-Right button on a two-button mouse.
- **Right mouse button toggles between** pick and unpick mode.



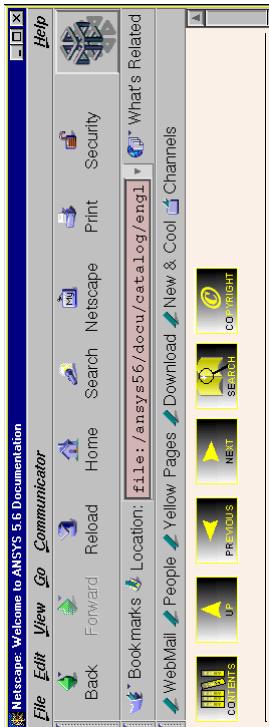
- **ANSYS uses an HTML-based documentation system to provide extensive on-line help.**
- You can get help on:
 - **ANSYS commands**
 - **element types**
 - **analysis procedures**
 - **special GUI “widgets” such as Pan-Zoom-Rotate**

- There are several ways to start the help system:
 - Launcher > Help System
 - Utility Menu > Help > Help Topics
 - Any dialog box > Help
 - Type HELP, name in the Input Window. Name is a command or element name.

ANSYS Basics

...On-Line Help

- Pressing the Help System button on the launcher brings up two browser windows:
 - a *document* window containing the help information.
 - a *navigational* window containing the Table of Contents and Index.



Welcome to ANSYS 5.6 Documentation

Welcome to ANSYS 5.6 Documentation –

HTML Online Documentation

The online documentation for ANSYS is provided as a set of HTML files and ANSYS uses Netscape on UNIX systems and either Netscape or Internet Explorer on Microsoft Windows to display the HTML files. This topic explains the various features of the HTML online documentation and how to use its navigational controls.

Sections in this topic include the following:

- [Accessing ANSYS Documentation](#)
- [Using the Document Window](#)
- [Using the Navigational Window](#)
- [Using the Search Window](#)

Accessing ANSYS Documentation

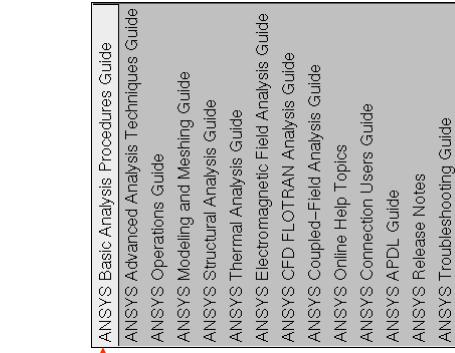
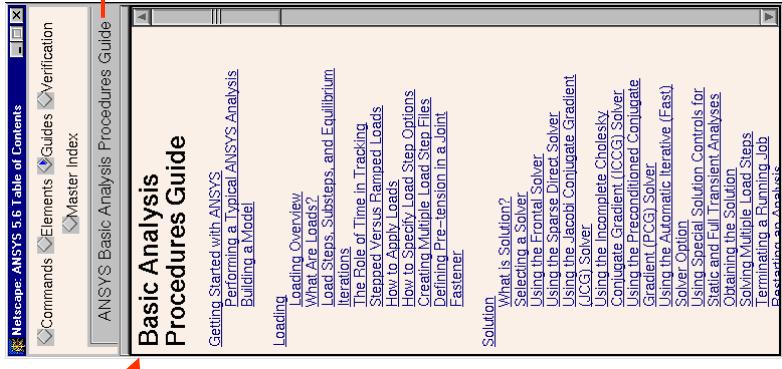
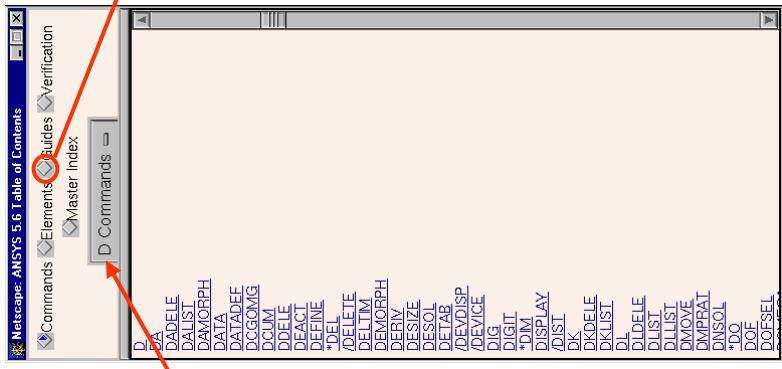
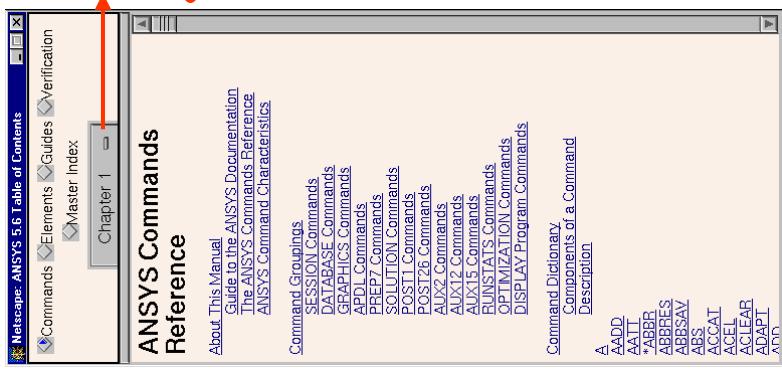
To access the table of contents and this topic, choose the Help Topics entry under Help in the **Utility Menu** (as shown in the following figure).



ANSYS Basics

...On-Line Help

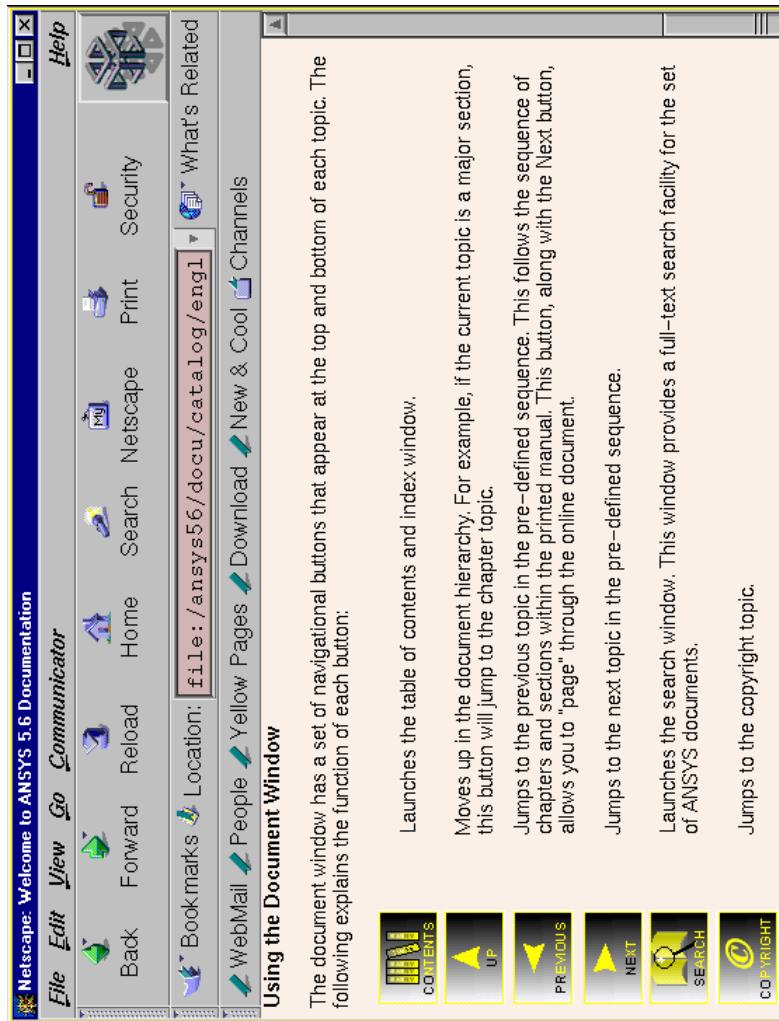
- Use the navigation window to choose the desired chapter or section within a particular document.



ANSYS Basics

...On-Line Help

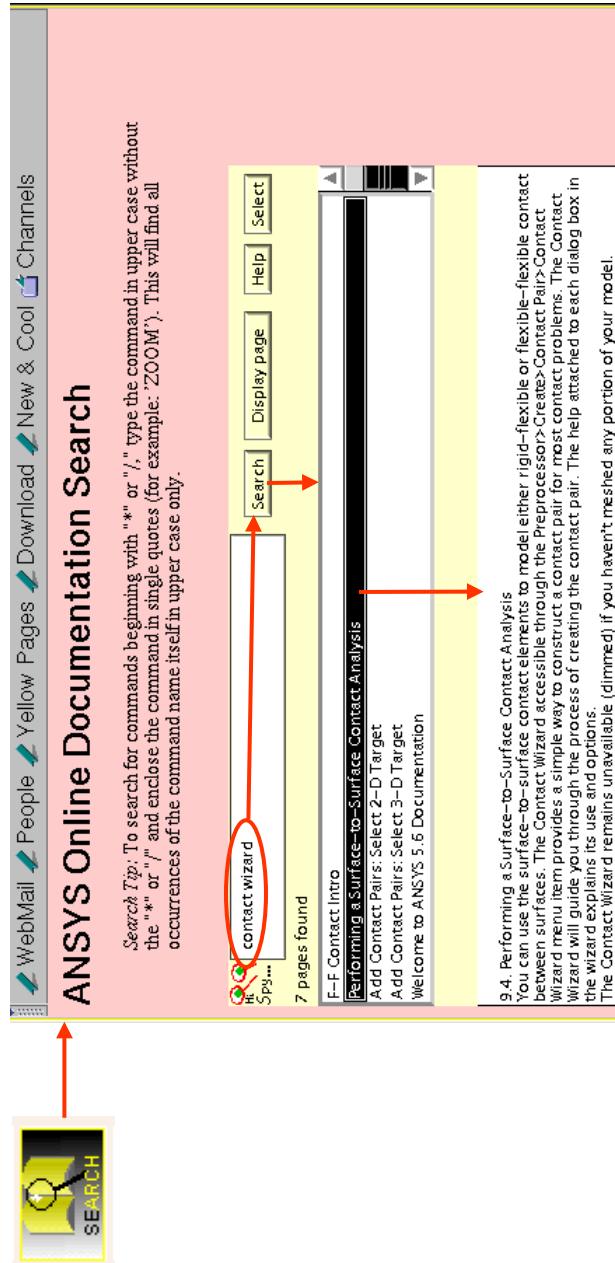
- In the document window, use the standard Back and Forward buttons on your browser, or the navigational buttons that appear at the top and bottom of each topic.



ANSYS Basics

...On-Line Help

- The **Search button** in the document window allows you to search for words or phrases.
- Click on **Using the Search Window** in the start-up help page (file:/ansys56/docu/catalog/english/ansyshelp/toc.html) for details on how to use the search facility.



9.4. Performing a Surface-to-Surface Contact Analysis

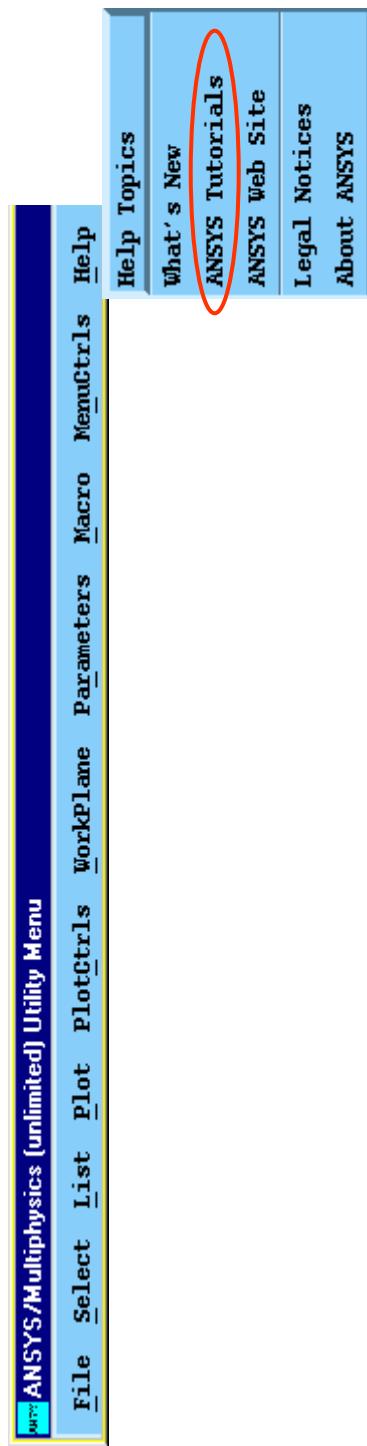
You can use the surface-to-surface contact elements to model either rigid-flexible or flexible-flexible contact between surfaces. The Contact Wizard is accessible through the Preprocessor>Create>Contact Pair>Contact Wizard menu item. It provides a simple way to construct a contact pair for most contact problems. The Contact Wizard will guide you through the process of creating the contact pair. The help attached to each dialog box in the wizard explains its use and options.

The Contact Wizard remains unavailable (dimmed) if you haven't meshed any portion of your model.

ANSYS Basics

...On-Line Help

- ANSYS also provides an HTML-based on-line tutorial.
- The tutorial consists of detailed instructions for a set of problems solved in ANSYS.
- To access the tutorial, click on Utility Menu > Help > ANSYS Tutorials.



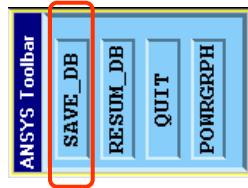
E. The Database & Files

- The term **ANSYS database** refers to the data ANSYS maintains in memory as you build, solve, and postprocess your model.
- The database stores both your input data and ANSYS results data:
 - Input data -- information you must enter, such as model dimensions, material properties, and load data.
 - Results data -- quantities that ANSYS calculates, such as displacements, stresses, strains, and reaction forces.

... The Database & Files

Save and Resume

- Since the database is stored in the computer's memory (RAM), it is good practice to save it to disk frequently so that you can restore the information in the event of a computer crash or power failure.
- The **SAVE** operation copies the database from memory to a file called the database file (or db file for short).
 - The easiest way to do a save is to click on Toolbar > SAVE_DB
 - Or use:
 - Utility Menu > File > Save as Jobname.db
 - Utility Menu > File > Save as...
 - SAVE command



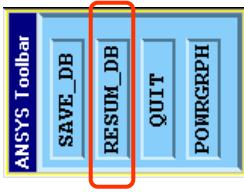
... The Database & Files

- To restore the database from the db file back into memory, use the RESUME operation.

– Toolbar > RESUME_DB

– Or use:

- Utility Menu > File > Resume Jobname.db
- Utility Menu > File > Resume from...
- RESUME command



- The default file name for SAVE and RESUME is *jobname.db*, but you can choose a different name by using the “Save as” or “Resume from” functions.

... The Database & Files

- **Notes on SAVE and RESUME:**
 - Choosing the *Save as* or *Resume from* function **does NOT** change the current *jobname*.
 - If you **save to the default file name and a *jobname.db* already exists**, ANSYS will first copy the “old” file to *jobname.dbb* as a back-up.
 - The *db* file is simply a “snapshot” of what **is** in memory at the time the save is done.

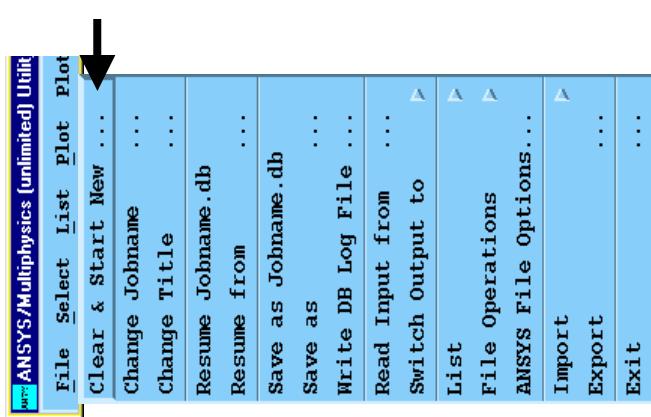
... The Database & Files

- **Tips on SAVE and RESUME:**
 - Periodically **save the database as you progress through an analysis.** ANSYS does **NOT** do automatic saves.
 - You should definitely **SAVE the database before attempting an unfamiliar operation (such as a Boolean or meshing) or an operation that may cause major changes (such as a delete).**
 - RESUME can then be used as an “undo” if you don’t like the results of that operation.
 - **SAVE is also recommended before doing a solve.**

Clearing the Database

- The Clear Database operation allows you to “zero out” the database and start fresh. It is similar to exiting and re-entering ANSYS.

- Utility Menu > File > Clear & Start New
- Or use the /CLEAR command.



ANSYS Basics

... The Database & Files

Files

- ANSYS writes and reads **several files** during an analysis. File names are of the format *jobname.ext*.
 - Jobname
 - A name you choose while starting ANSYS, up to 32 characters. Defaults to *file*.
 - Can be changed within ANSYS with the /FILNAME command (Utility Menu > File > Change Jobname).
 - Extension
 - Identifies the contents of the file, such as *db* for database.
 - Usually assigned by ANSYS.

... The Database & Files

- **Typical files:**

jobname.log: Log file, ASCII.

- Contains a log of every command issued during the session.
- If you start a second session with the same jobname in the same working directory, ANSYS will append to the previous log file (with a time stamp).

jobname.err: Error file, ASCII.

- Contains all errors and warnings encountered during the session.

jobname.db, *.dbb*: Database file, binary.

- Compatible across all platforms.

jobname.rst, *.rth*, *.rmg*, *.rfi*: Results files, binary.

- Contains results data calculated by ANSYS during solution.
- Compatible across all platforms.

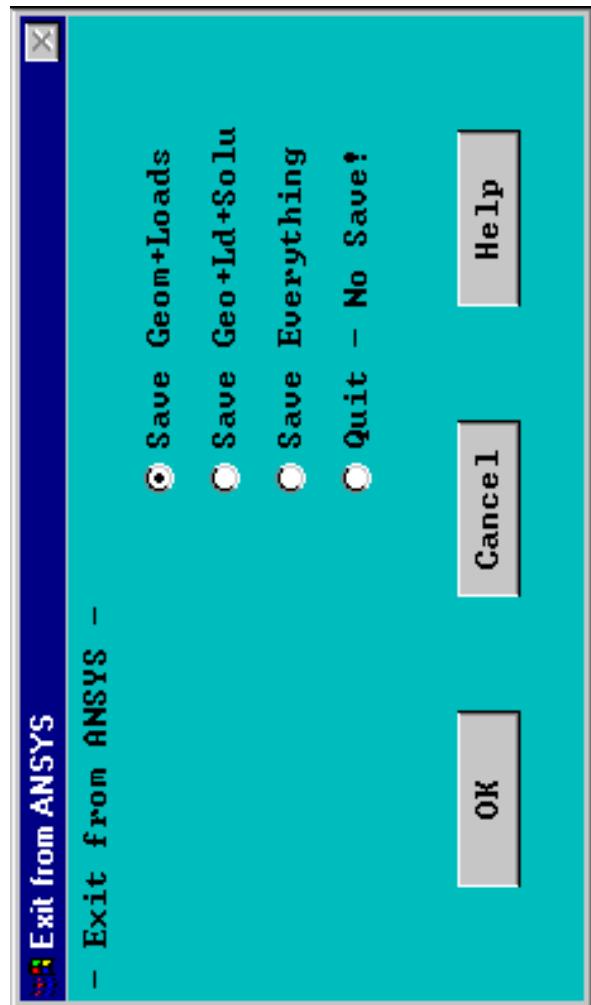
File Management Tips

- Run each **analysis** project in a **separate working directory**.
- Use different **jobnames** to differentiate **various analysis runs**.
- You should **keep the following files after any ANSYS analysis:**
 - **log file** (**.log**), **database file** (**.db**), **results files**, **load step files**, if any (**.s01**, **.s02**, ...), **physics files** (**.ph1**, **.ph2**, ...).
- Use **/DELETE** or Utility Menu > File > ANSYS File Options to **automatically delete files no longer needed by ANSYS** during that session.

ANSYS Basics

F. Exiting ANSYS

- **Three ways to exit ANSYS:**
 - Toolbar > QUIT
 - Utility Menu > File > Exit
 - **Use the /EXIT command in the Input Window**



- Refer to your *Workshop Supplement* for instructions on:

[W1. Introductory Workshop](#)