

Chapter 2

Introduction to ANSYS Workbench

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the types of systems*
- *Understand different types of cells*
- *Understand the Graphic User Interface of the ANSYS Workbench*
- *Start a new project in ANSYS Workbench*
- *Add analysis systems to a project*
- *Set units for project*
- *Use ANSYS Workbench Help*

INTRODUCTION TO ANSYS Workbench

Welcome to the world of Computer Aided Engineering (CAE) with ANSYS Workbench. If you are a new user, you will be joining hands with thousands of users of this Finite Element Analysis software package. If you are familiar with the previous releases of this software, you will be able to upgrade your designing skills with tremendous improvement in this latest release.

ANSYS Workbench, developed by ANSYS Inc., USA, is a Computer Aided Finite Element Modeling and Finite Element Analysis tool. In the Graphical User Interface (GUI) of ANSYS Workbench, the user can generate any dimensional (1D, 2D, and 3D) and FE models, perform analysis, and generate results of analysis. You can perform a variety of tasks ranging from design assessment to finite element analysis to complete product optimization analysis by using ANSYS Workbench. ANSYS also enables you to combine the stand-alone analysis system into a project and to manage the project workflow.

The following is the list of analyses that can be performed by using ANSYS Workbench:

1. Coupled Field Harmonic
2. Coupled Field Modal
3. Coupled Field Static
4. Coupled Field Transient
5. Eigenvalue Buckling
6. Electric
7. Explicit Dynamics
8. Fluid Flow (CFX)
9. Fluid Flow (Fluent with Fluent Meshing)
10. Fluid Flow (Fluent)
11. Fluid Flow (Polyflow Classic)
12. Harmonic Acoustics
13. Harmonic Response
14. Hydrodynamic Diffraction
15. Hydrodynamic Response
16. LS-DYNA
17. LS-DYNA ACOUSTICS
18. LS-DYNA RESTART
19. Magnetostatic
20. Modal
21. Modal Acoustics
22. Particle Dynamics (Rocky)
23. Random Vibration
24. Response Spectrum
25. Rigid Dynamics
26. SPEOS
27. Static Acoustics
28. Static Structural
29. Steady-State Thermal
30. Structural Optimization.
31. Sub-structure Generation
32. Thermal-Electric

33. Transient Structural
34. Transient Thermal
35. Turbomachinery Fluid Flow

SYSTEM REQUIREMENTS

The following are minimum system requirements to ensure smooth functioning of ANSYS Workbench on your system:

- Operating System: Windows 64-bit (Windows 10 or Windows 11, Windows Server 2019/2022 (for enterprise setups))
- Platform: Intel i5 / i7 / i9 or AMD Ryzen equivalent
- Memory: 8 GB of RAM for all applications, 16 GB or higher recommended for running CFX and FLUENT
- Installation: Software installation via ANSYS Customer Portal download
- Graphics adapter: NVIDIA Quadro / RTX or AMD Radeon Pro (4 GB+ VRAM)
- Browser: Chrome/Microsoft edge 100 or higher, Firefox 100 or higher
- Display Resolution: 1920 × 1080 or higher

STARTING ANSYS Workbench 2026 R1

To start ANSYS Workbench 2026 R1, choose **Start > All Apps > Ansys 2026 R1 > Workbench 2026 R1** from the taskbar, refer to Figure 2-1. Alternatively, you can start ANSYS Workbench by double-clicking on its shortcut icon displayed on the desktop of your computer. After the necessary files are loaded and licenses are verified, the **Workbench** window will be displayed on the screen, as shown in Figure 2-2.

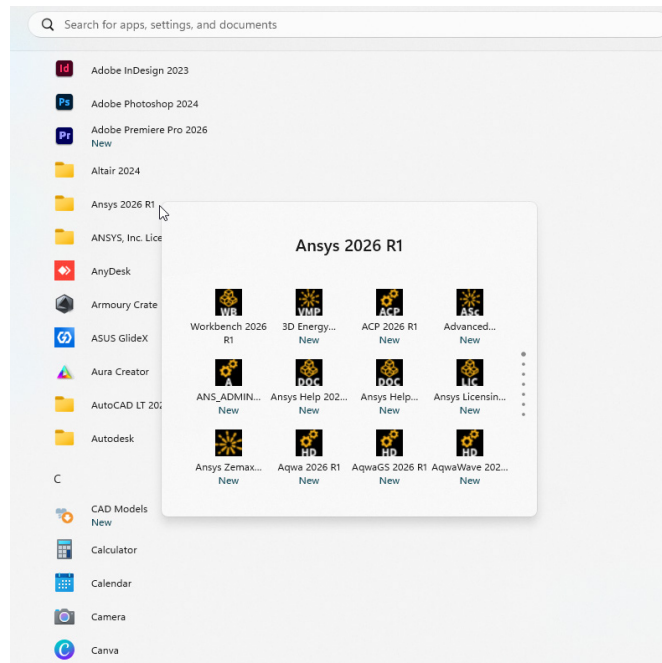


Figure 2-1 Starting ANSYS Workbench using the taskbar

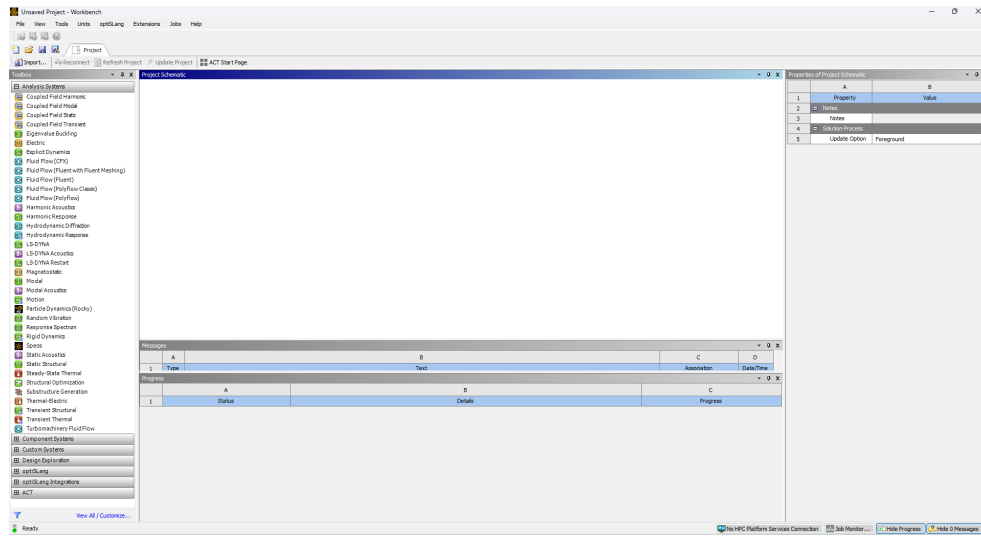


Figure 2-2 The Workbench window

The **Workbench** window helps streamline an entire project to be carried out in ANSYS Workbench 2026 R1. In this window, one can create, manage, and view the workflow of the entire project created by using Standard analysis systems. The **Workbench** window mainly consists of Menu bar, **Main** toolbar, the **Toolbox** window, **Project Schematic** window, and the **Status bar**, refer to Figure 2-3. Various components of the **Workbench** window are discussed next.

Toolbox Window

The **Toolbox** window is located on the left in the **Workbench** window. The **Toolbox** window lists the standard and customized templates or the individual analysis components that are used to create projects. To create a project, drag a particular analysis or component system from the **Toolbox** window and drop it into the **Project Schematic** window. Alternatively, double-click on a particular analysis or component system in the **Toolbox** window to add it to the **Project Schematic** window and to create the project.



Note

The double-click action always adds new system to the project, whereas dragging and dropping the system from the **Toolbox** window enables you to specify location of the new system in the **Project Schematic** window. Based on the specified location, you can create data sharing with the existing systems. You will learn more about sharing data between different systems in the **Project Schematic** window in later chapters.

The **Toolbox** window comprises five toolboxes: **Analysis Systems**, **Component Systems**, **Custom Systems**, **Design Exploration**, and **ACT**. The components of these toolboxes are discussed next.

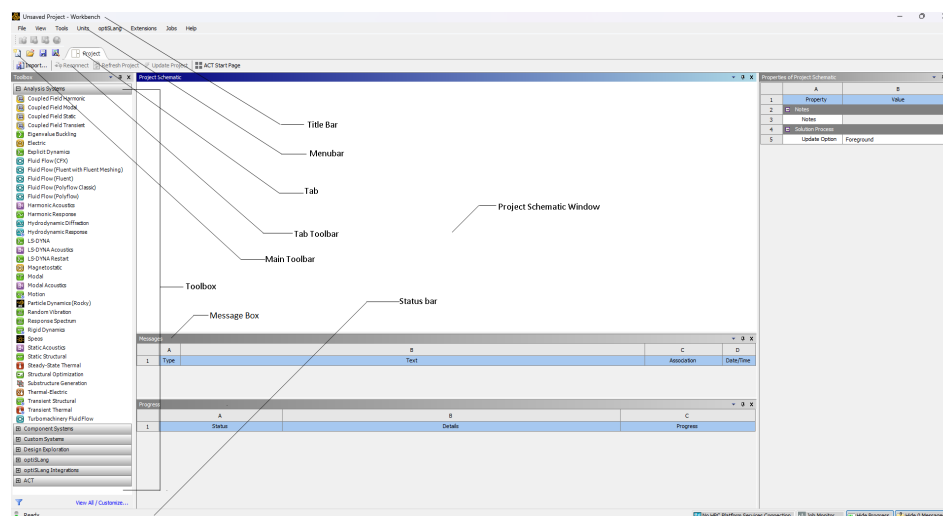


Figure 2-3 The components of the **Workbench** window

Analysis Systems Toolbox

The **Analysis Systems** toolbox is displayed expanded in the **Toolbox** window by default. It contains predefined templates for different types of analyses that can be carried out in ANSYS Workbench 2026 R1. Each predefined template consists of all the components that are used to perform a particular type of analysis. For example, the **Static Structural** analysis system of the **Analysis Systems** toolbox is used to carry out the Static Structural analysis. When you add this system in the **Project Schematic** window, it contains all the components that are necessary to carry out the Static Structural analysis. Figure 2-4 shows the **Analysis Systems** toolbox with different types of analysis systems available in ANSYS Workbench. These analysis systems are discussed next.

Coupled Field Harmonic

This analysis system, while performing the harmonic analysis, determines the steady-state response of a structure and the surrounding fluid medium to sinusoidally time-varying excited loads. The system supports piezoelectric coupling between electric and structural physics.

Coupled Field Modal

This analysis system, while performing the modal analysis, determines the frequencies and standing wave patterns within a structure and the surrounding fluid medium.

Coupled Field Static

This analysis system is used to determine the response of a structure subjected to steady loads that do not induce significant inertia and damping effects. Using this type of analysis, the displacements, stresses, strains,

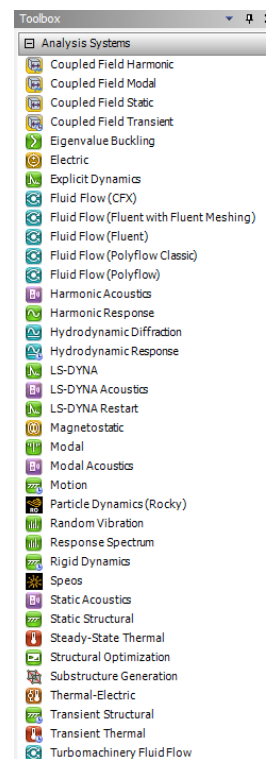


Figure 2-4 *The Analysis Systems toolbox displaying various analysis systems*

and forces caused by static loading conditions can be determined. The system supports 2D coupled structural-thermal physics.

Coupled Field Transient

This analysis system is used to determine the responses of a structure under the action of time dependent variables such as displacements, strains, stresses, and forces. The system supports 2D coupled structural-thermal physics.

Eigenvalue Buckling

Buckling is failure of structure which is caused when compressive stresses are less than the ultimate compressive stress of the material. Eigenvalue buckling is the theoretical behavior of an ideal elastic structure. However, geometric imperfections and nonlinearity prevent most structures from attaining their theoretical eigenvalue strength. To assess these effects, a nonlinear buckling is needed.

Electric

This analysis system is used to analyze steady-state electric conduction.

Explicit Dynamics

This analysis system is used to identify the dynamic response of a component under stress wave propagation, or time-dependent loads or impacts. It is also used for modal mechanical phenomena that are highly non-linear.

Fluid Flow (CFX)

This system allows users to carry out flow analysis of compressible and incompressible fluids. It is also used to analyze heat transfer in fluids.

Fluid Flow (Fluent with Fluent Meshing)

This system is used for simulating fluid flow, heat transfer, and related phenomena. Fluent Meshing provides automated, high-quality meshing for complex geometries. It is ideal for advanced CFD applications across industries like automotive, aerospace, and electronics.

Fluid Flow (Fluent)

Like Fluid Flow (CFX), Fluid Flow (Fluent) system is also used to carry out fluid flow analysis of compressible and incompressible fluids and their heat transfer analysis.

Fluid Flow (Polyflow Classic)

This analysis system is used to simulate polyflow application-specific capabilities.

Harmonic Acoustics

Harmonic Acoustics is the prediction of structure borne noise and sound propagation that plays a vital role in the design of large variety of products, for example noise caused by vibrating structural components, transmission of sound through thin panels and the acoustic performance of piezoelectric devices. These types of acoustic wave propagation problems can be solved either in a coupled way in which the fluid and structural domains are solved simultaneously, or in an uncoupled way with the help of Harmonic Acoustics analysis system.

Harmonic Response

Harmonic Response is the response of a system under a sustained cyclic load. Harmonic Response analysis system is used to analyze a system working under periodic or sinusoidal loads. This analysis helps in determining whether a particular structure will be able to withstand resonance, fatigue, and other effects of forced vibration.

Hydrodynamic Diffraction

The **Hydrodynamic Diffraction** system is used in the aqua applications and determines the wave forces and structure motions in any types of wave pattern.

Hydrodynamic Response

The **Hydrodynamic Response** system is used to determine the response of a structure caused by ocean environment forces such as wind, wave, current in regular or irregular waves.

LS-DYNA

This analysis system is used for solving highly nonlinear, transient dynamic problems involving large deformations. It is particularly useful in simulations such as crash analysis, metal forming, and blast impacts. LS-DYNA handles complex contact interactions and material behaviors under extreme loading conditions.

LS-DYNA Acoustics

This system enables simulation of acoustic wave propagation in fluids and structures. It is used to study sound pressure levels, acoustic coupling, and noise transmission. Application of this system include cabin acoustics, speaker design, and underwater acoustics.

LS-DYNA Restart

This system allows continuation or modification of a previously run LS-DYNA simulation without restarting from the beginning. It is useful for refining results, making minor changes, or extending the analysis time. It helps save computational resources and time in long or complex simulations.

Magnetostatic

This analysis system is used to analyze the magnetic field developed due to the presence of a temporary or permanent magnet.

Modal

Modal analysis is the study of dynamic properties of a model, subjected to vibrations. Modal analysis system in ANSYS Workbench helps in determining the frequencies and mode shapes of a model.

Modal Acoustics

The **Modal Acoustics** system is used to determine frequencies and standing wave patterns within a structure and the surrounding fluid medium.

Particle Dynamics (Rocky)

This system is used to simulate the behavior of granular and discrete particle systems. It is ideal for analyzing particle flow, breakage, and interaction with equipment surfaces. Application of this system include conveyor transfer points, mills, and material handling systems.

Random Vibration

This analysis is carried out to determine the reaction of a structure or a component to changing frequencies of vibrations. Many components experience vibrations which are random in nature. This analysis system is used to determine the responses of structures that are exposed to such varying or random vibrations.

Response Spectrum

Response Spectrum analysis system is similar to Random Vibration analysis system and is used after a transient analysis is done.

Rigid Dynamics

Rigid Dynamics analysis system is used to determine the response of a rigid body or a mechanism consisting of rigid bodies. Response of a robot mechanism is an example of rigid body analysis.

SPEOS

This system performs optical simulation for evaluating lighting performance and visual appearance. It is used to analyze light propagation, reflection, and refraction in real-world conditions. Commonly applied in automotive lighting, displays, and ergonomics design.

Static Acoustics

This system is used as a base analysis (pre-stress environment) for future downstream Modal Acoustics or Harmonic Acoustics analyses where response of a structure is determined for a load that does not induce significant inertia and damping effects.

Static Structural

The Static Structural analysis system is used to determine the response of a structure subjected to static loading conditions. The loads in this case are assumed to produce no or negligible time based loading characteristics. Using this type of analysis, displacement, stresses, and deformations of structures under static loading conditions can be determined.

Steady-State Thermal

Steady-State Thermal analysis system is used to determine the temperature, thermal gradient, heat flow rates and heat fluxes under the influence of thermal loading which remains constant with time and are static in nature.

Structural Optimization

This system is used to improve the structural performance of components by minimizing weight, maximizing stiffness, or meeting specific design targets. It supports topology, shape, and size optimization under defined load and boundary conditions.

Substructure Generation

This system allows users to create a reduced representation (substructure or superelement) of a component for efficient inclusion in larger assemblies. It reduces computational time while maintaining accuracy in dynamic or static simulations.

Thermal-Electric

Thermal-Electric analysis system is used to simulate thermal and electric fields.

Transient Structural

Transient Structural analysis system is used to determine responses of structures under the action of time dependent variables. Using this analysis, time-varying displacement, stresses and strains can be determined.

Transient Thermal

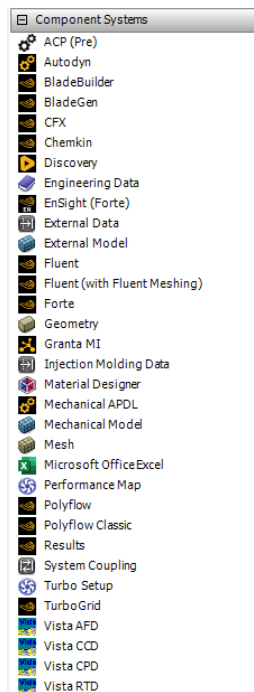
Transient Thermal analysis system is used to determine the temperature and other thermal variables of a structure that vary over time.

Turbomachinery Fluid Flow

The **Turbomachinery Fluid Flow** system contains cells for carrying out a study with ANSYS CFX with an added Turbo Mesh cell.

Component Systems Toolbox

By default, the **Component Systems** toolbox is displayed in collapsed state in the **Toolbox** window. To expand the **Component Systems** toolbox, click on the plus sign (+) located on the left of the **Component Systems** title bar. The components displayed in the **Component Systems** toolbox are the basic blocks of a project and form only a part of the analysis system, such as **Geometry** (used to create a model for analysis), **Mesh** (used to generate FEA model), **Results** (used to visualize the results of analysis in the desired form), and so on. Figure 2-5 shows the **Components Systems** toolbox with various components displayed in it.



*Figure 2-5 Partial view of the **Component Systems** toolbox*

Custom Systems Toolbox

By default, the **Custom Systems** toolbox is also displayed in collapsed state in the **Toolbox**. To expand this node, click on the plus sign (+) displayed on the left of the **Custom Systems** title bar; refer to Figure 2-6. The systems in the **Custom Systems** toolbox are used to carry out standard coupled analysis, in which the input and output data of one analysis are used as input for the next analysis. For example, the **Pre-Stress Modal** system is used to carry out Static Structural analysis followed by a Modal analysis. Similarly, the **FSI: Fluid Flow (CFX) -> Static Structural** custom system is used to carry out a Fluid Flow analysis in CFX followed by a Static Structural analysis.

To add a custom system to the **Project Schematic** window, double-click on it in the **Custom Systems** toolbox in the **Toolbox** window. Figure 2-7 shows **FSI: Fluid Flow (CFX) -> Static Structural** custom system added to the **Project Schematic** window. This figure illustrates two different systems sharing the same geometry. This type of sharing is done if a single project requires various analysis types for the same geometry. You will learn about adding systems to the **Project Schematic** window later in this chapter.

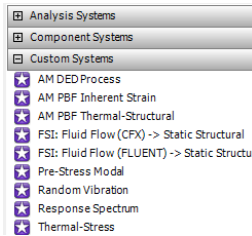


Figure 2-6 The Custom Systems toolbox

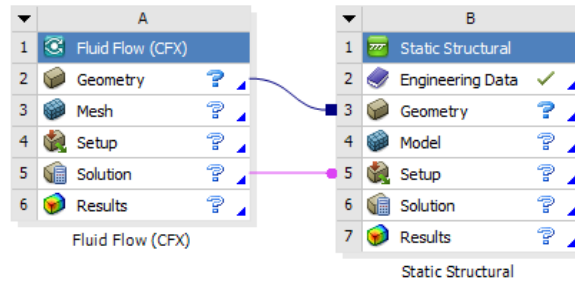


Figure 2-7 The FSI: Fluid Flow (CFX) -> Static Structural custom system added to the Project Schematic window

Design Exploration Toolbox

By default, the **Design Exploration** toolbox is displayed in collapsed state in the **Toolbox** window. Expand this toolbox by following the procedure discussed earlier. The options in the **Design Exploration** toolbox are used to explore a component, so that the design of the component can be further optimized by changing the design variables based on the performance of the product, refer to Figure 2-8.

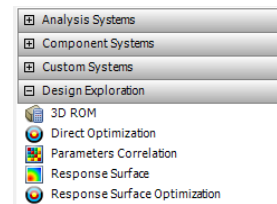


Figure 2-8 The Design Exploration toolbox

You can control the display of elements in the **Toolbox**. To do so, choose the **View All / Customize...** button displayed at the bottom of the **Toolbox** window; the **Toolbox Customization** window will be displayed, as shown in Figure 2-9. In this window, some of the check boxes are selected, indicating that the corresponding element will be displayed in the **Toolbox** window. Clear the check box corresponding to those elements that you do not want to be displayed in the **Toolbox** window.

Toolbox Customization				
	B	C	D	E
	Name	Physics	Solver Type	AnalysisType
1				
2	Analysis Systems			
3	☒ Coupled Field Harmonic	Multiphysics	Mechanical APDL	Harmonic
4	☒ Coupled Field Modal	Multiphysics	Mechanical APDL	Modal
5	☒ Coupled Field Static	Multiphysics	Mechanical APDL	Static
6	☒ Coupled Field Transient	Multiphysics	Mechanical APDL	Transient
7	☒ Eigenvalue Buckling	Structural	Mechanical APDL	Eigenvalue Buckling
8	☒ Eigenvalue Buckling (Sancetf)	Structural	Sancetf	Buckling
9	☒ Electric	Electric	Mechanical APDL	Steady-State Electric Conduction
10	☒ Explicit Dynamics	Structural	AUTODYN	Explicit Dynamics
11	☒ Fluid Flow (CFX)	Fluids	CFX	
12	☒ Fluid Flow (Fluent with Fluent Meshing)	Fluids	FLUENT	Any
13	☒ Fluid Flow (Fluent)	Fluids	FLUENT	Any
14	☒ Fluid Flow (Polyflow Classic)	Fluids	Polyflow	Any
15	☒ Fluid Flow (Polyflow)	Fluids	MatPro	Any
16	☒ Harmonic Acoustics	Multiphysics	Mechanical APDL	Harmonic
17	☒ Harmonic Response	Structural	Mechanical APDL	Harmonic Response
18	☒ Hydrodynamic Diffraction	Modal	Aqwa	Hydrodynamic Diffraction
19	☒ Hydrodynamic Response	Transient	Aqwa	Hydrodynamic Response
20	☒ LS-DYNA	Explicit	LS-DYNA@LS-DYNA	Structural
21	☒ LS-DYNA Acoustics	Multiphysics	LS-DYNA@Acoustic@LS-DYNA	Harmonic
22	☒ LS-DYNA Restart	Explicit	RestartLS-DYNA@LS-DYNA	Structural
23	☒ Magnetostatic	Electromagnetic	Mechanical APDL	Magnetostatic
24	☒ Modal	Structural	Mechanical APDL	Modal
25	☒ Modal (ABAQUS)	Structural	ABAQUS	Modal
26	☒ Modal (Sancetf)	Structural	Sancetf	Modal
27	☒ Modal Acoustics	Multiphysics	Mechanical APDL	Modal
28	☒ Motion	Structural	AnsysMotion@AnsysMotion	Transient
29	☒ Particle Dynamics (Rocky)	Any	Any	Any
30	☒ Random Vibration	Structural	Mechanical APDL	Random Vibration
31	☒ Response Spectrum	Structural	Mechanical APDL	Response Spectrum
32	☒ Rigid Dynamics	Structural	Rigid Body Dynamics	Transient
33	☒ Specs	Any	Any	Any
34	☒ Static Acoustics	Multiphysics	Mechanical APDL	Static
35	☒ Static Structural	Structural	Mechanical APDL	Static Structural
36	☒ Static Structural (ABAQUS)	Structural	ABAQUS	Static Structural
37	☒ Static Structural (Sancetf)	Structural	Sancetf	Static Structural
38	☒ Steady-State Thermal	Thermal	Mechanical APDL	Steady-State
39	☒ Steady-State Thermal (ABAQUS)	Thermal	ABAQUS	Steady-State
40	☒ Steady-State Thermal (Sancetf)	Thermal	Sancetf	Steady-State

Figure 2-9 Partial view of the *Toolbox Customization* window

Project Schematic Window

The **Project Schematic** window helps manage an entire project. It displays the workflow of entire analysis project. To add an analysis system to the **Project Schematic** window, drag the analysis system from the **Toolbox** window and drop it into the green-colored box displayed in the **Project Schematic** window, as shown in Figure 2-10 and 2-11. Alternatively, double-click on an analysis system in the **Toolbox** window to include it in the **Project Schematic** window. You can also add an analysis system to the **Project Schematic** window by using the shortcut menu displayed on right-clicking in the **Project Schematic** window. The procedure of adding an analysis system by using the shortcut menu is discussed later in this chapter.

In the **Project Schematic** window, when you click on the down arrow available at the top right corner, a flyout is displayed with various options to close, float, restore, minimize, and maximize the **Project Schematic** window, refer to Figure 2-12.

Each time you drag and drop an analysis system or an item into the **Project Schematic** window, a system is formed. Each system, consists of cells which are used to carry out various tasks within a system. You can add more than one systems in the **Project Schematic** window by dragging and dropping them from the **Toolbox** window, as per the requirement. After adding systems to the **Project Schematic** window, you can share the data available in the cells of one system with the corresponding cells of another system. A common example of systems sharing same kind of data among various cells of different systems is shown in Figure 2-13. The different types of systems connection links are discussed later in this chapter.

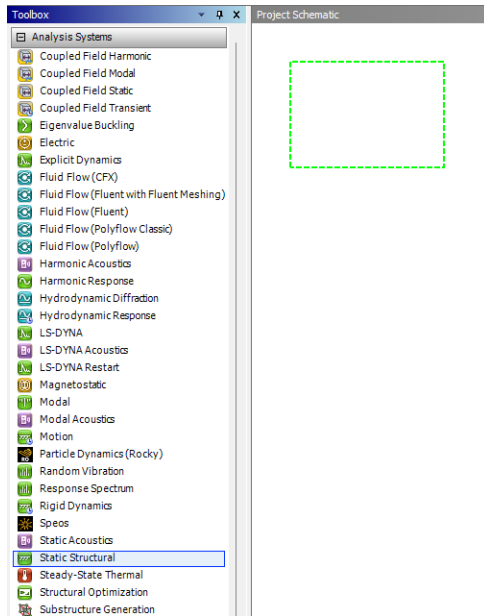


Figure 2-10 Dragging the **Static Structural** analysis system into the **Project Schematic** window

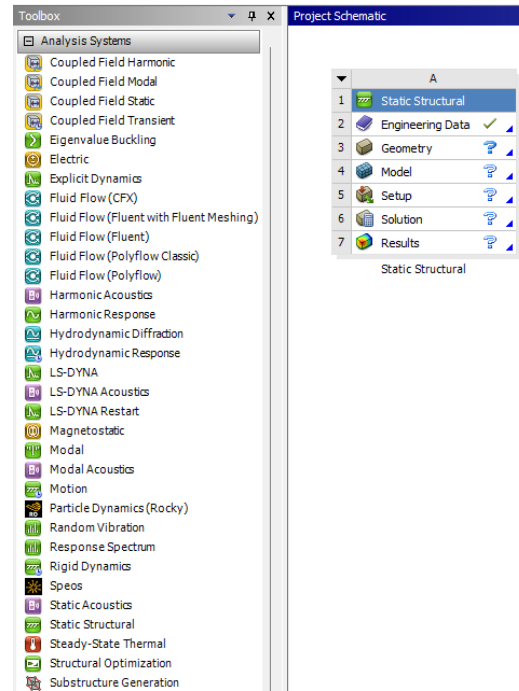


Figure 2-11 The **Static Structural** analysis system open in the **Project Schematic** window



Figure 2-12 Partial view of the **Project Schematic** window with the flyout displayed

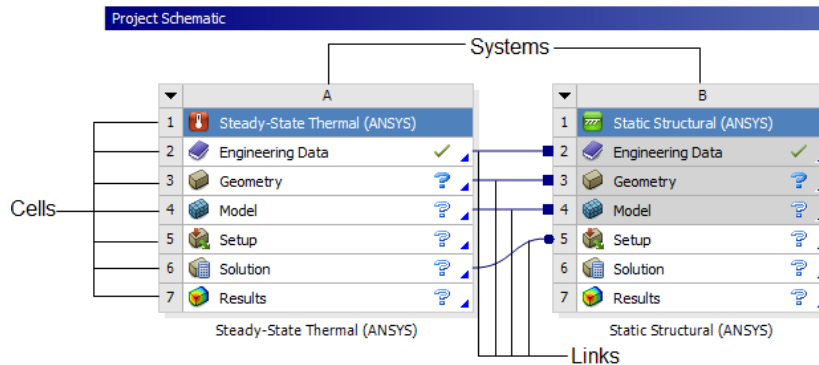


Figure 2-13 Partial view of the **Project Schematic** window showing sharing of cells among two different analysis systems



Note
You will learn about the **Project Schematic** window, systems, and cells in detail later in this chapter.

Menu Bar

Menu bar is located on the top of the **Workbench** window and contains various options such as **File**, **View**, **Tools**, and so on. These options enable you to control and manage the files of the current project. Figure 2-14 shows the Menu bar in the **Workbench** window. The options available in various menus will be discussed in detail later in this chapter.

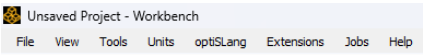


Figure 2-14 The Menu bar

Standard Toolbar

The **Standard** toolbar is a collection of the frequently used tools in ANSYS Workbench 2026 R1 and is shown in Figure 2-15. The tools available in the **Standard** toolbar are also available in the Menu bar.

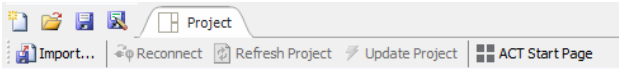


Figure 2-15 The Main and Tab toolbar

The various tools available in the **Standard** toolbar are **New**, **Open**, **Save**, **Save As**, **Import**, **Reconnect**, **Refresh Project**, **Update Project**, and so on. You can use these tools to create new projects, open existing ones, save a project, save a project to a different location with a different name, import a project from another source, refresh project status after changes are made to it, update a project to its latest status and so on.

Shortcut Menu

In ANSYS Workbench, you can invoke most of the tools by using a shortcut menu displayed on right-clicking. The shortcut menus displayed are context sensitive, that is, the context in the shortcut menus will change depending upon the place where you right-click to invoke it. You can right-click anywhere in the **Workbench** window to display a shortcut menu. Some of the options in a shortcut menu display an arrow on their right. This arrow indicates that one more menu will be displayed on choosing this option. Figure 2-16 shows the shortcut menu that is displayed by right-clicking on the **Geometry** cell of the **Static Structural** system in the **Project Schematic** window.

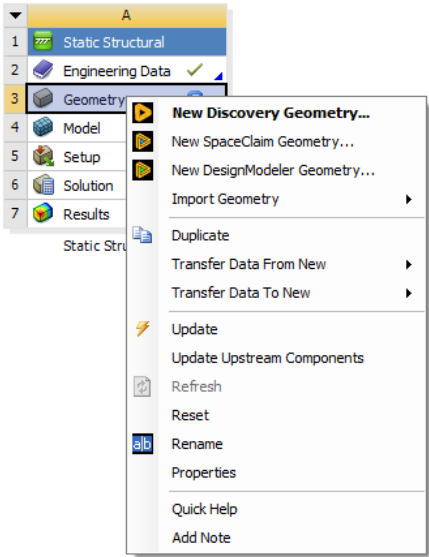


Figure 2-16 The shortcut menu displayed by right-clicking on the **Geometry** cell in the **Project Schematic** window

WORKING ON A NEW PROJECT

To start working on a new project, you need to add an appropriate analysis or component system to the **Project Schematic** window.

Evaluation Copy. Do not reproduce. For information visit www.cadcam.com

Adding a System to a Project

After starting a new project, it is necessary to define the tasks to be carried out in ANSYS Workbench 2026 R1. To start a new analysis, you need to add an analysis system to the **Project Schematic** window, as shown in Figure 2-17. There are many ways to add a system to a project. They are discussed next.

Adding a System by Drag and Drop

To add a system to a project by dragging and dropping, pick the required system template from the **Toolbox** window and then drag the cursor to the **Project Schematic** window; the green rectangular area of dash lines will be displayed, representing the location where the picked analysis system can be dropped. Move the cursor over the green rectangular area; the green rectangle will convert into a red rectangle of solid lines, refer to Figure 2-17. Drop the system in the red box; the system will be added to the project and will be displayed in the **Project Schematic** window.

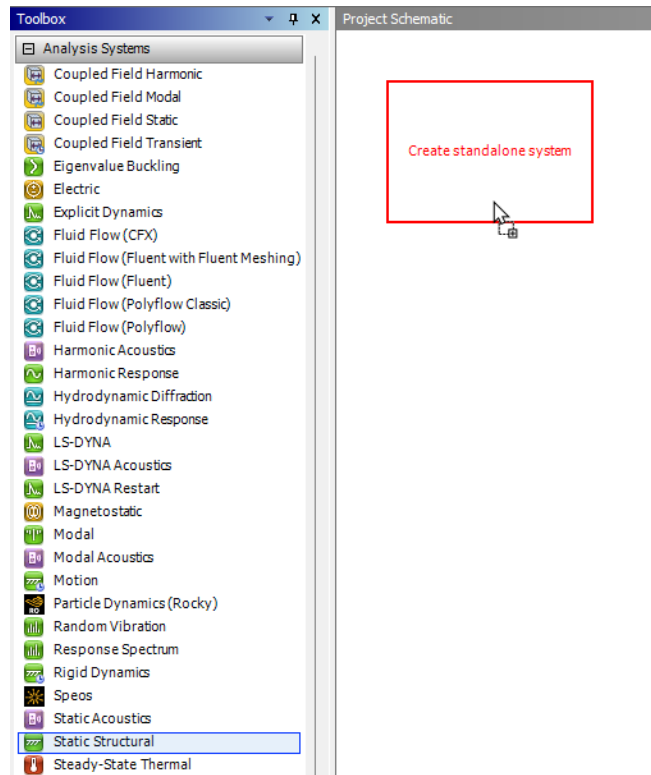


Figure 2-17 Adding an analysis system by using the drag and drop functionality



Note

After adding the first analysis system into the **Project Schematic** window, when you drag the next analysis system from the **Toolbox** window to add to the **Project Schematic** window, more than one green rectangular areas of dash lines will be displayed, representing the possible locations where you can drop analysis.

Adding a System by Double-clicking

You can also add an analysis system by double-clicking the left mouse button. To do so, double-click on the system that has to be added to the project; the system will be automatically added to the **Project Schematic** window.



Note

If an analysis system already exists in a project and then you double-click to add a new system, it will be added below the existing one.

Adding a System Using the Shortcut Menu

You can also add an analysis system by using the shortcut menu. To do so, right-click on the **Project Schematic** window; a shortcut menu will be displayed. Using this shortcut menu, you can add analysis, component, and custom systems to the **Project Schematic** window. To add an analysis system, choose the **New Analysis Systems** option from the shortcut menu; a flyout will be displayed. Choose the desired analysis system from the flyout to add it to the project, refer to Figure 2-18.

To add a new component system in the project, choose the **New Component Systems** option from the shortcut menu; a flyout will be displayed. Next, choose the desired component system from the flyout to add to the **Project Schematic** window.

Similarly, to add a new custom system or a new design exploration system into the **Project Schematic** window, choose the **New Custom Systems** or **New Design Exploration** option, respectively from their respective shortcut menu. Next, choose the desired option from the flyout to add to the **Project Schematic** window.

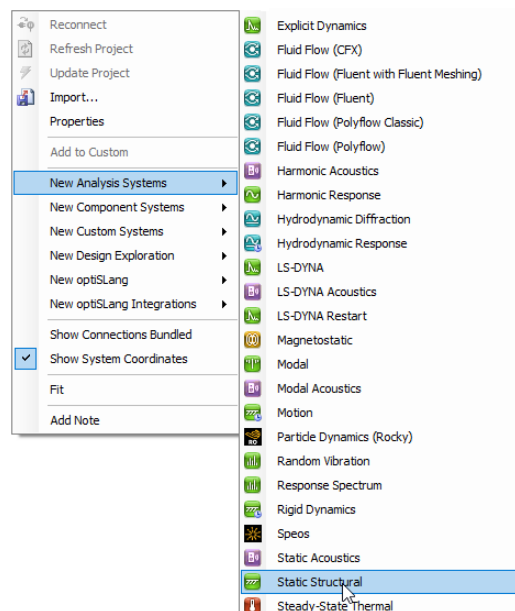


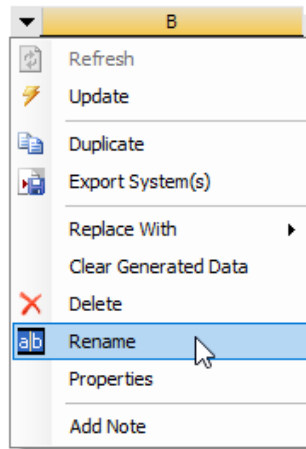
Figure 2-18 Choosing the *Static Structural* analysis system from the shortcut menu

**Tip**

After a system is added to the project, it is now important to define the cells that are displayed in respective systems. The most common types of cells that exist in a system are discussed later in this chapter.

RENAMING A SYSTEM

After a system is added to the **Project Schematic** window, its name will be highlighted at the bottom of the system. You can also rename an existing project by double-clicking on the name of the current project. Alternatively, click on the black down-arrow displayed at the upper left corner of the analysis system; a flyout will be displayed. Choose the **Rename** option from this flyout, refer to Figure 2-19; the name of the system will be highlighted. Specify a name to the system.



*Figure 2-19 Choosing the **Rename** option from the shortcut menu*

DELETING A SYSTEM FROM A PROJECT

To delete a system from the **Project Schematic** window, right-click on its name displayed in the title bar; a shortcut menu will be displayed, refer to Figure 2-20. Choose **Delete** option from the shortcut menu; the **ANSYS Workbench** message box will be displayed, as shown in Figure 2-21. Choose the **OK** button from this message box; the selected system will be deleted from the project. Alternatively, click on the down-arrow displayed at the upper left corner of the system; a flyout will be displayed. Choose the **Delete** option from this flyout to delete the system from the project.

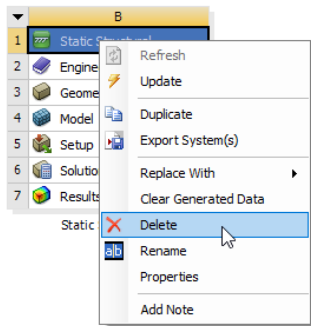


Figure 2-20 Choosing the **Delete** option from the shortcut menu

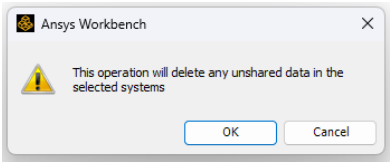


Figure 2-21 The **ANSYS Workbench** message box

DUPLICATING A SYSTEM IN A PROJECT

To duplicate a system, select the down arrow available at the top left corner of the selected system; a flyout is displayed, refer to Figure 2-19. Choose the **Duplicate** option from the flyout to duplicate it.



Note

When duplicating a system, all cells are copied. However, result data may not be retained or may require re-evaluation depending on the duplication method.

SAVING THE CURRENT PROJECT



Whenever you start a new analysis project, the title bar of the **Workbench** window displays **Unsaved Project - Workbench**. This indicates that the current project is not saved yet. To save the current project, choose the **Save Project** button from the **Main** toolbar. Alternatively, choose the **Save** option from the **File** menu; the **Save As** dialog box will be displayed, as shown in Figure 2-22. You can also invoke the **Save As** dialog box by pressing the CTRL and S keys together. In this dialog box, browse to the location where you want to save the current project and then specify its name in the **File name** edit box. Next, choose the **Save** button; the project will be saved at the specified location. After saving the project, the title bar of the **Workbench** window will display the name that you have specified while saving the project.

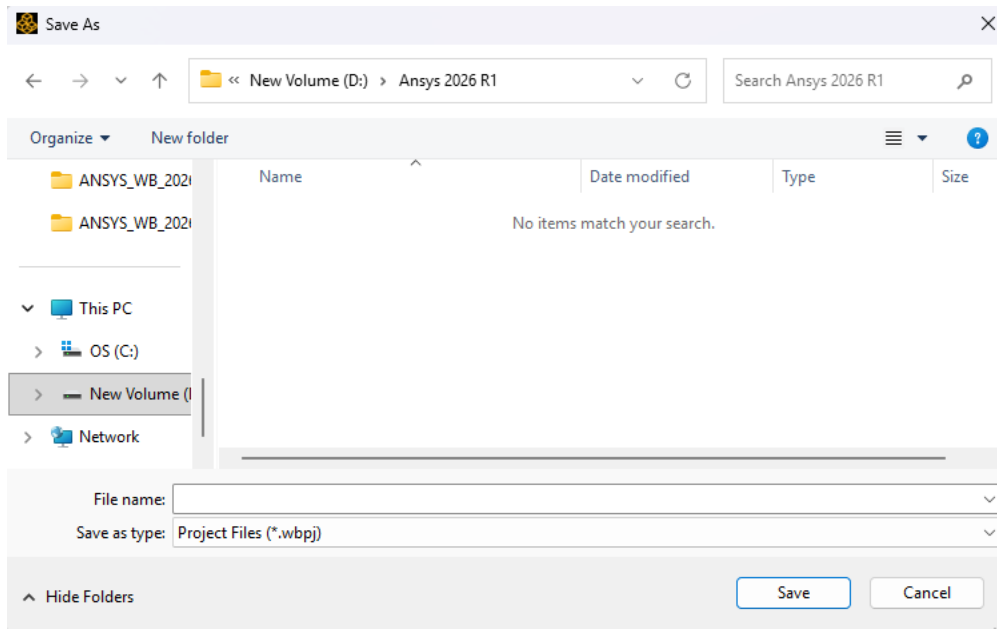


Figure 2-22 The Save As dialog box



Note

*If you have already saved the project, the **Save As** dialog box will not be displayed on choosing the **Save** button.*

If you want to save an opened project with a different name or at a different location, choose the **Save Project As** button from the **Main** toolbar; the **Save As** dialog box will be displayed, refer to Figure 2-22. Specify a new name and then choose the **Save** button from this dialog box; the same project will be saved with the new name and will become the current project.

OPENING A PROJECT



To open an existing project, choose the **Open Project, Archive or Script** button from the **Main** toolbar; the **Open** dialog box will be displayed, as shown in Figure 2-23. Browse to the location where the project file is saved, select the ***.wbproj** file, and then choose the **Open** button from this dialog box. The selected project file will be opened and its name will be displayed on the title bar of the **Workbench** window. Alternatively, choose the **Open** option from the **File** menu.

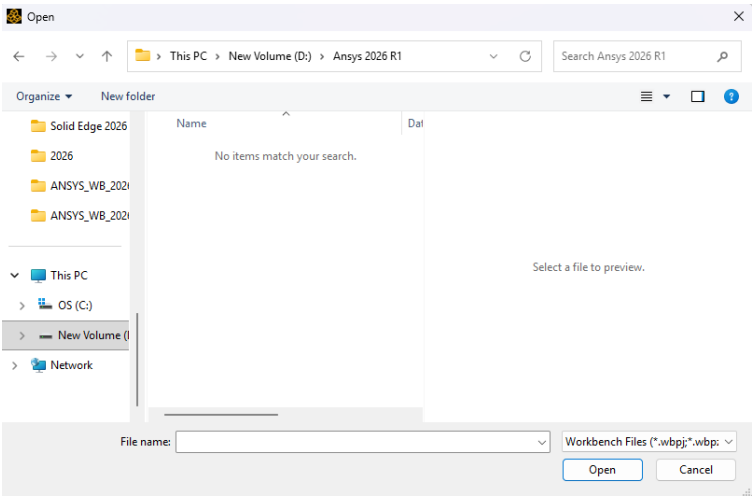


Figure 2-23 The *Open* dialog box

ARCHIVING THE PROJECT DATA

If you want to move the project data from one system to another, you can archive all project related data in a single zip file. This ZIP file contains all the files and folders required to open and run the project on another computer, including the project file (*.wbproj) and the associated project folder (*name_files*). To archive a project, choose **File > Archive** from the Menu bar; the **Save Archive** dialog box will be displayed, refer to Figure 2-24. By default, *name* is displayed as the name of the file in the **File name** edit box, where, *name* is the project name of the current project. If you want to assign a different name to the archive file, specify it in the **File name** edit box.

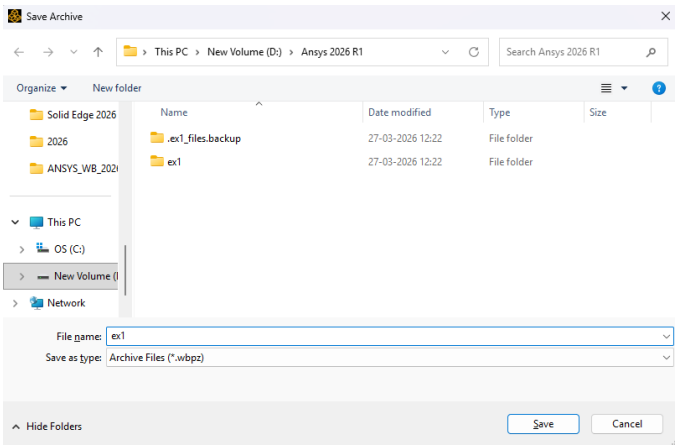


Figure 2-24 The *Save Archive* dialog box

Next, specify the location where you want to save the archive file and choose the **Save** button from the **Save Archive** dialog box; the **Archive Options** dialog box will be displayed, as shown in Figure 2-25. If you do not want to include the result and the solution file or some imported file that has not been created in the project, clear the respective check box from the **Archive Options** dialog box. Next, choose the **Archive** button from this dialog box; the zipped archive file will be created at the specified location. Transfer this zip file to any system and restore the project files.

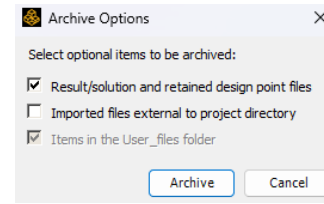


Figure 2-25 The *Archive Options* dialog box

UNITS IN ANSYS Workbench

In ANSYS Workbench, you can use any of the following predefined unit systems:

1. Metric (kg, m, s, °C, A, N, V) Unit System

Mass = Kilogram (kg)	Temperature = Degree Celsius (°C)
Length = Meter (m)	Current = Ampere (A)
Time = Second (s)	Force = Newton (N)
Voltage = Volts (V)	

2. Metric (tonne, mm, s, °C, mA, N, mV) Unit System

Mass = Tonne	Temperature = Degree Celsius (°C)
Length = Millimeter (mm)	Current = Milliampere (mA)
Time = Second (s)	Force = Newton (N)
Voltage = Millivolt (mV)	

3. U.S. Customary (lbm, in, s, °F, A, lbf, V) Unit System

Mass = Pound (lb)	Temperature = degree Fahrenheit (°F)
Length = Inch (in)	Current = Ampere (A)
Time = Second (s)	Force = Pound (lbf)
Voltage = Volts (V)	

4. SI (kg, m, s, K, A, N, V) Unit System

Mass = Kilogram (kg)	Temperature = Kelvin (K)
Length = Meter (m)	Current = Ampere (A)
Time = Second (s)	Force = Newton (N)
Voltage = Volts (V)	

5. U.S. Engineering (lb, in, s, R, A, lbf, V) Unit System

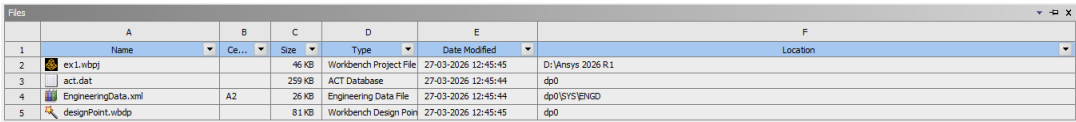
Mass = Pound (lb)	Temperature = Renkine (R)
Length = Inch (in)	Current = Ampere (A)
Time = Second (s)	Force = Pound (lbf)
Voltage = Volts (V)	

Metric (kg, m, s, °C, A, N, V) unit system is the default unit system that is assigned to a project. However, you can change the unit system for the current project or set the default unit system, to be assigned to any new project.

ANSYS WORKBENCH DATABASE AND FILE FORMATS

When you save a project, a project file is created with a name *name.wbpj*, where *name* can be any user specified name. In addition to the project file, a folder is also created with a name *name_files*. All other files relevant to the project are automatically saved in this folder under various sub folders such as, *dp0*, *user_files*, and so on.

In ANSYS Workbench, various files are created with different file extensions. To view all files associated with the current project, choose **View > Files** from the Menu bar; the **Files** window will be displayed, as shown in Figure 2-26.



	A	B	C	D	E	F
	Name	Ce...	Size	Type	Date Modified	Location
1	ex1.wbpj		46 KB	Workbench Project File	27-03-2026 12:45:45	D:\Ansys 2026 R1
2	act.dat		259 KB	ACT Database	27-03-2026 12:45:44	dp0
3	EngineeringData.xml	A2	26 KB	Engineering Data File	27-03-2026 12:45:44	dp0\SYSTEMGO
4	designPoint.vdbp		81 KB	Workbench Design Point	27-03-2026 12:45:45	dp0

Figure 2-26 The Files window

The **Files** window lists all files of the current project. The information displayed in the **File** window includes the following:

- 1. Name and extension of the files.
- 2. The reference of the analysis component cell with which the file is associated.
- 3. The file size and its location.

To open a folder containing the selected files, right-click on the corresponding cell and then choose the **Open Containing Folder** option from the shortcut menu displayed. To filter in any particular type of file extension, right-click on any cell to display a shortcut menu. Next, choose the **File Type Filter** option from the shortcut menu; the **File Type Filter** dialog box will be displayed, as shown in Figure 2-27. Select the check boxes displayed on the left of the desired file type from the **File Type Filter** dialog box; the list displayed in the **File** window will change dynamically based on the selection made in the **File Type Filter** dialog box. After selecting the desired file types, choose the **Ok** button from the **File Type Filter** dialog box to exit.

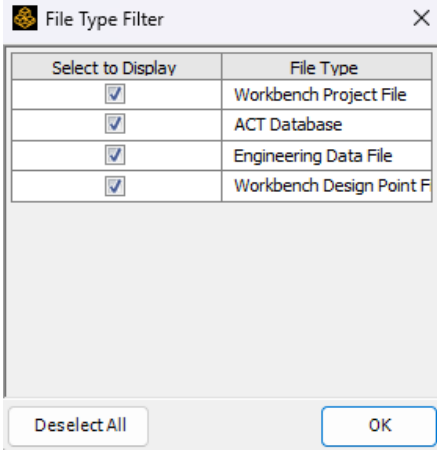


Figure 2-27 The File Type Filter dialog box

The following are some of the main file extensions used in ANSYS:

- *.wbpj = ANSYS Workbench project database file
- *.engd = Engineering Data
- *.agdb = DesignModeler file
- *.fedb = FE Modeler files
- *.cldb = Meshing file
- *.mechdb= Mechanical file
- *.rsx = Mesh Morpher

Evaluation Copy. Do not reproduce. For information visit www.cadcam.com

*.ad = ANSYS AUTODYN
 *.dxd = Design Exploration
 *.bgd = BladeGen
 *.db = Mechanical APDL database file
 *.cas, *.dat, *.msh = FLUENT files
 *.cfx, *.def, *.res, *.mdef, and *.mres = CFX files
 *.cndb = CFX-Mesh files

CHANGING THE UNIT SYSTEMS

You can change the unit system being used in the current project. To do so, choose the **Units** menu from the Menu bar; a menu with various options will be displayed. Select the desired unit system from the menu; a tick mark will be displayed on the left of the selected unit system, refer to Figure 2-28.

In addition to the unit systems displayed in the Units menu, you can customize to add some more unit systems by using the **Unit Systems** dialog box. To invoke this dialog box, choose **Units > Unit Systems** from the Menu bar; the **Unit Systems** dialog box will be displayed, as shown in Figure 2-29. All unit systems supported by ANSYS Workbench will be displayed under the **Unit System** column in the **Unit Systems** dialog box. If you select a unit system in this column, the units used for measuring various quantities will be displayed on the right pane in this dialog box.

ANSYS Workbench supports nineteen main unit systems. However, by default only five Main unit systems are displayed under the **Units** menu. This is because the rest of the unit systems are suppressed in the **Unit Systems** dialog box, by default. To unsuppress a unit system, clear the corresponding check box under the **Suppress** column in the **Unit Systems** dialog box. Now onward, the unsuppressed unit system will be displayed in the **Units** menu.

You can change the default unit system that is assigned to a new project. To do so, select the radio button corresponding to the desired unit system, in the **Default** column in the **Unit Systems** dialog box. Now onward, the specified unit system will become the default unit system for new projects. Select the radio button corresponding to the desired unit system under the **Active Project** column to make it the unit system for the current project.

If the Main unit systems given in ANSYS Workbench do not suit your requirement, you can customize a unit system according to your requirement by using the **Unit Systems** dialog box. To do so, select the unit system that closely fits your requirement under the **Unit System** column and then choose the **Duplicate** button from the **Unit Systems** dialog box; a new unit system will be added under the **Unit System** column with the default name *Custom Unit System*. Rename this system. Select the newly defined unit system; the corresponding measuring units will be displayed on the right pane. Click on the down-arrow displayed on the right of the units under the **Unit** column; a drop-down list will be displayed with all feasible units for the quantity to be measured. Select the desired units from this drop-down list. You can also export custom units for the use of other users or import an already saved unit system by using the **Export** or **Import**

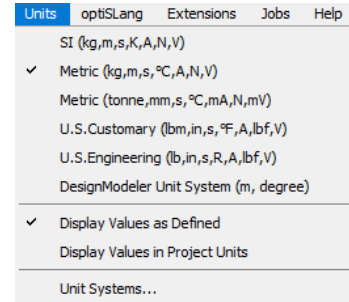


Figure 2-28 The Units menu

button, from the **Unit Systems** dialog box. The imported or exported unit system files are saved in *.xml* format. To delete the selected customized unit system, choose the **Delete** button from the **Unit Systems** dialog box.

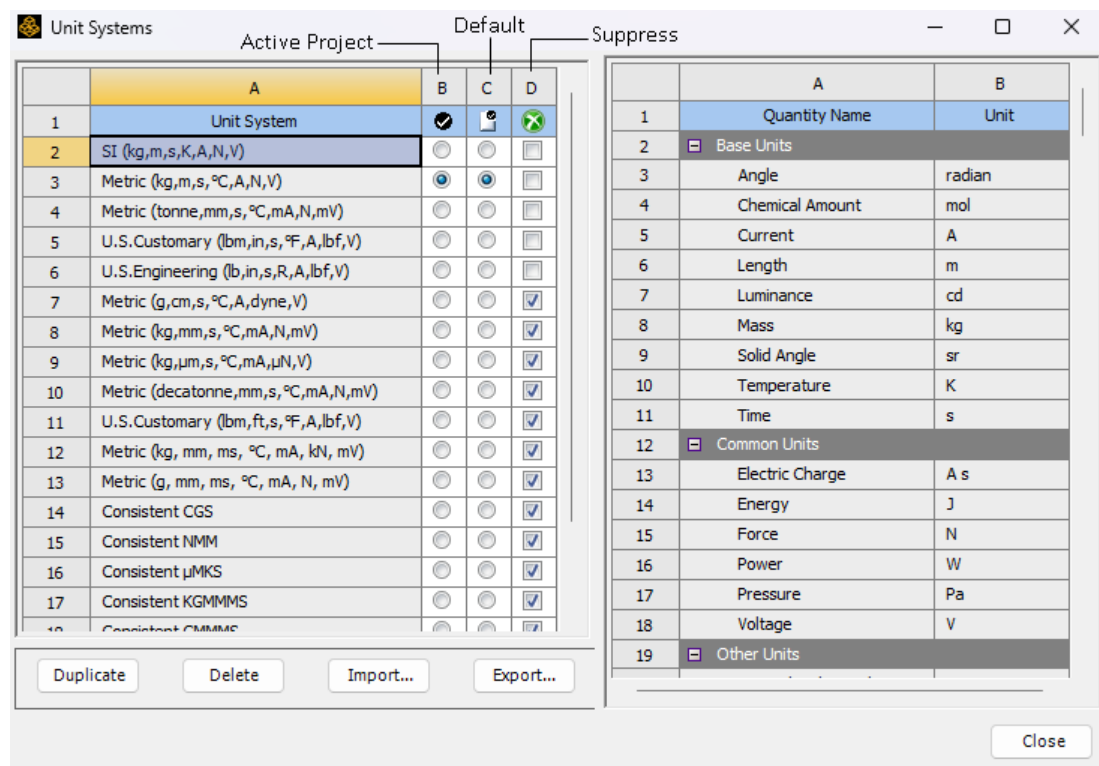


Figure 2-29 The Unit Systems dialog box

COMPONENTS OF A SYSTEM

An item that is added from the **Toolbox** window to the **Project Schematic** window is known as system and the constituent elements of the system are known as cells. Each cell of a system plays an important role in carrying out a project and are discussed next.

**Engineering Data
Setup**

**Geometry
Solution**

**Model/Mesh
Results**

Engineering Data Cell

The **Engineering Data** cell is used to define the material to be used in the analysis. To define the material, double-click on the **Engineering Data** cell; the workspace corresponding to this the **Engineering Data** cell will be displayed, as shown in Figure 2-30. Alternatively, you can also invoke the Engineering Data workspace to specify the material by right-clicking on this cell and then choosing the **Edit** option from the shortcut menu displayed.

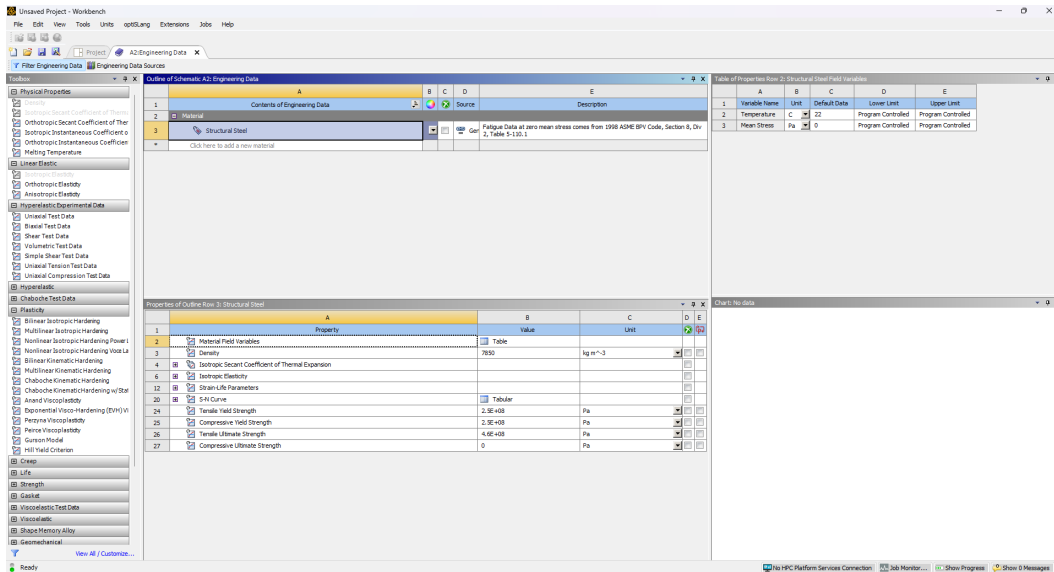


Figure 2-30 The Engineering Data workspace



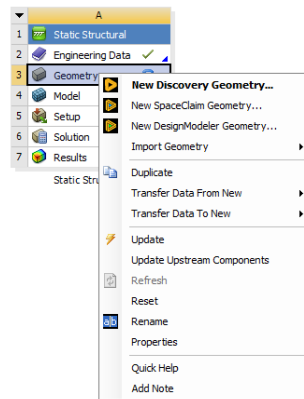
Note

When you right-click on **Engineering Data** cell, the **Edit** option will be highlighted in the shortcut menu displayed. This indicates that this is the most suitable action that can be taken. You can select an option from the shortcut menu as per the requirement. However, when you double-click on the **Engineering Data** cell, the most suitable action will be initiated directly.

Geometry Cell

The **Geometry** cell is used to create, edit, or import the geometry that is used for analysis. To create a geometry for analysis, right-click on this cell and then choose either the **New SpaceClaim Geometry** or **New DesignModeler Geometry** option from the shortcut menu displayed, refer to Figure 2-31. Choose the **New SpaceClaim Geometry** option in the shortcut menu; the **SpaceClaim** window will be opened. Alternatively, if you double-click on the **Geometry** cell; the **SpaceClaim** window will be displayed which is used to create geometry. You can choose the **New DesignModeler Geometry** option which uses Parasolid geometry modeling kernel to create geometry. You can also choose the **New Discovery Geometry** option from the shortcut menu, which opens the ANSYS Discovery interface. Discovery provides an intuitive and interactive environment for creating and editing geometry.

Some other important options in the shortcut menu that are specific to the **Geometry** cell are discussed next. These options will only be available if you have not defined a geometry. The **Import Geometry** option is used to import any existing geometry to the current analysis system. You can also import the geometry created in other CAD packages. The **Edit Geometry** option will only be displayed if a geometry is already associated with the current analysis system. When you choose this option, the geometry opens in the **SpaceClaim** window for modification. The **Replace Geometry** option is used to replace the existing geometry that is associated with the analysis system with another geometry.



*Figure 2-31 The shortcut menu displayed on right-clicking on the **Geometry** cell*



Note

To exit the **SpaceClaim** window, choose **File > Close SpaceClaim** from the Menu bar.

Model Cell

The **Model** cell is available for mechanical analysis systems and is used to open the ANSYS Mechanical interface, where the geometry is discretized into elements, boundary conditions and loads are applied, and the analysis is solved. On double-clicking on this cell, the **Mechanical** window will be displayed. This cell is associated with the ANSYS Mechanical interface.

Mesh Cell

The **Mesh** cell is available in various analysis systems and is used to generate the mesh for the geometry. On double-clicking on this cell, the **Meshing** window will be displayed. This cell is associated with the **Meshing** window.

Setup Cell

The **Setup** cell is used to define the analysis settings, including boundary conditions and physical parameters, depending on the type of analysis. This cell is associated with the appropriate solver interface, such as **ANSYS Mechanical** or **ANSYS Fluent**.

Solution Cell

The **Solution** cell is used to compute the solution based on the settings and conditions defined in the preceding cells. This cell is also associated with the **Mechanical** workspace.

Results Cell

The **Results** cell is used to display the results of the analysis in the user specified formats. This cell is associated with the post-processing interface of the selected analysis system.

STATES OF A CELL IN AN ANALYSIS SYSTEM

All cells display some visual symbols on their right to indicate their current status. This helps the user understand the next step in the analysis process as well as understanding the overall status of an analysis, refer to Figure 2-32. The explanation of these symbols is given in Table 2-1.

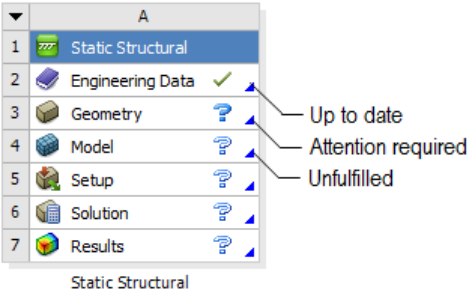


Figure 2-32 Symbols for cell status

Table 2-1 Symbols and their meaning

Symbol	Meaning	Function
	Attention required symbol	indicates an immediate requirement of action for a cell and a user cannot proceed further without fixing the cell
	Unfulfilled	indicates that the previous cells do not have sufficient or appropriate data
	Up to date	indicates that the cell is up-to-date and the data in the cell is ready to be shared with other cells
	Refresh required	indicates that the data of the previous cells has been changed since last update and you need to refresh the cell with this symbol.
	Input changes pending	indicates that the data of the current cell will change on updation, if changes are done in the previous cells
	Update required	indicates that the input data of the cell has been changed and the output data needs to be updated
	Refresh failed	indicates that the last refresh process for the cell has failed
	Update failed, update needed	indicates that the last update process for the cell has failed and you need to update it again
	Update failed, attention needed	indicates that the last update process has failed and the cell needs immediate attention.

Figure 2-33 shows an analysis system in which the geometry of the analysis model has been changed after performing the complete analysis. Therefore, the refresh required symbol is displayed against the **Model** cell.

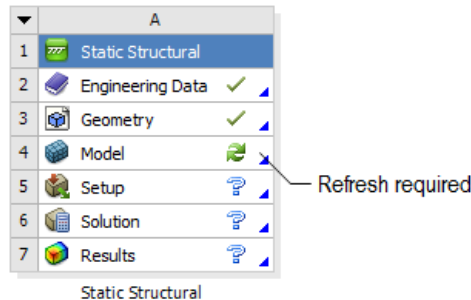
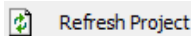


Figure 2-33 The Static Structural analysis system with the refresh required symbol annotated

REFRESHING AND UPDATING A PROJECT



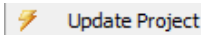
Refresh Project

Refresh is a process in which ANSYS Workbench reads the entire modified data. To refresh the entire project, choose the **Refresh Project** button from the **Tab** toolbar. Alternatively, choose **Tools > Refresh Project** from the Menu bar.



Note

When you refresh the entire project, the output data will not be recalculated based on the modifications made in the project.



Update Project

Update is a process in which ANSYS Workbench refreshes the input data and recalculates the output data. To update the entire project, choose the **Update Project** button from the **Tab** toolbar. Alternatively, choose **Tools > Update Project** from the Menu bar.

You can also refresh or update a particular cell by choosing the respective option from the shortcut menu that is displayed on right-clicking on that particular cell. These options will only be available in the shortcut menu if that particular cell needs refreshing or updating.

If you refresh a particular cell, you can get information about its effect on the cells below the specified cell, and if needed, you can make the required changes before updating the cell. Refreshing a cell before updating is useful in complex analysis systems, where updating can take more time compared to refreshing.

ADDING SECOND SYSTEM TO A PROJECT

The second or the subsequent system can be added in the **Project Schematic** window in two ways, either as a stand-alone system or as a connected system that shares data with the existing analysis system. In a project, multiple stand-alone systems are preferred in case of performing analysis for various components of an assembly. In this way, analysis of all components of an assembly can be kept and managed in a single analysis project. The connected analysis systems

are preferred more for conducting coupled analysis, where data from the first analysis system can be used or shared with the next analysis system.

As discussed earlier, to add a stand-alone system, double-click on the analysis system template in the **Toolbox** window; the new analysis system will get added in the **Project Schematic** window below the existing analysis system, as shown in Figure 2-34. Alternatively, use the shortcut menu or the drag and drop method to add a stand-alone system to the **Project Schematic** window.

Adding Connectors

There are two ways in which you can add connectors between cells of different systems in the **Project Schematic** window. In the first method, drag the cell from one system and drop it onto the corresponding cell of another system; connectors will be added between the systems, which indicates that the data is shared. In the second method, drag the system from the **Toolbox** window to the **Project Schematic** window and move the cursor over the cell of the existing system with which you want to share the data; a red rectangle will be displayed on the right of the existing system with text written inside the rectangle. The text will give you information about the cells with which the data will be shared by the new system, as shown in Figure 2-35. Drop the selected analysis system over the desired cell of the existing analysis system; the new system will be added to the right of the existing analysis system and will share data from the specified cells. Figure 2-36 shows a new analysis system, the **Eigenvalue Buckling** analysis system, dropped over the **Solution** cell of the **Static Structural** analysis system. Note that the new system will automatically share the necessary data from the cells that are above the cell over which you dropped the new analysis system.

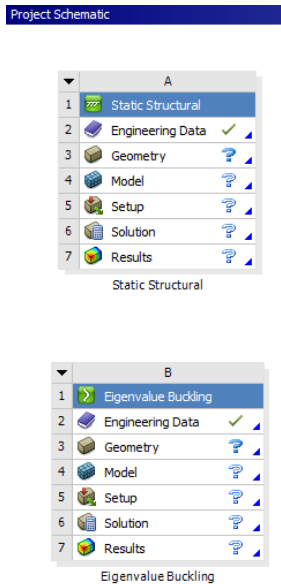


Figure 2-34 Two stand-alone systems added in the **Project Schematic** window

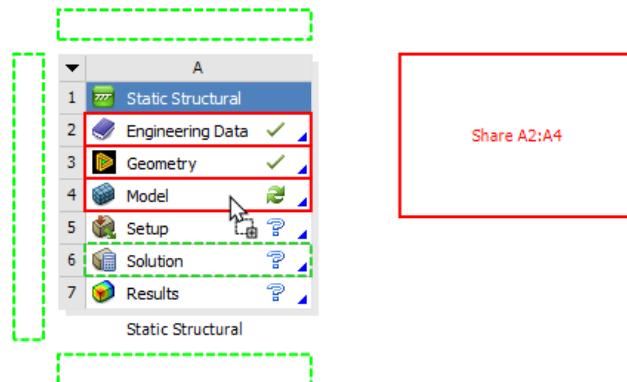


Figure 2-35 Text reference of the cells to be shared

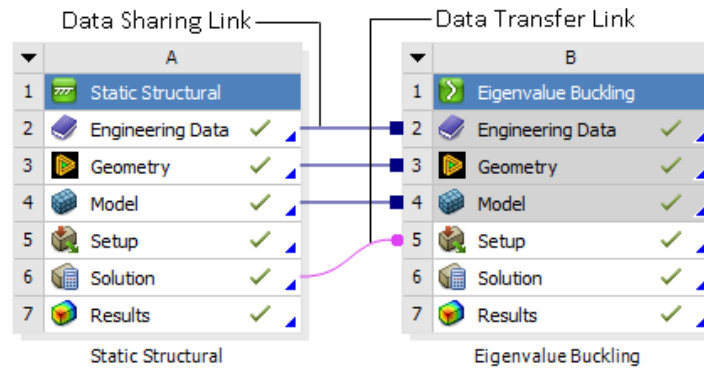


Figure 2-36 The Eigenvalue Buckling analysis system added as a connected system to the Static Structural analysis system

Connector/Link Types

Links connecting systems represent data sharing or data transfer between the cells of systems in the **Project Schematic** window. ANSYS Workbench supports two different types of connectors, as shown in Figure 2-36. In the first type, line with a square on its right (target) side is used to share data when the inputs and outputs of the two connected cells are identical. Shared data links can only be created between two cells of the same type. In the second type, a curve with a circle on its right (target) side is used to transfer data when the output of one cell is used as the input to the connected cell. Transfer data links are usually created between two cells of different types.

Deleting Connectors/Links

In most cases, you can delete links by right-clicking the link and choosing **Delete** from the shortcut menu. After deleting the link, the downstream system retains the current data, allowing the cells to be edited independently.

In some cases (for example, links between two mechanical geometry cells), you cannot delete shared data links. In such cases, the linked cells in the downstream system have a gray background, refer to Figure 2-36.

SPECIFYING A GEOMETRY FOR ANALYSIS

In ANSYS Workbench, geometry can be specified in two different ways. In the first method, you can import a geometry created by using any other solid modeling software. In the second method, you can create a new geometry in the **SpaceClaim** or **DesignModeler** application of ANSYS Workbench. The procedure to create a new geometry is described next.

Creating a Geometry

When a new system is added in the **Project Schematic** window, the next step is to create the geometry. To create a new geometry, right-click on the **Geometry** cell of the system; a shortcut menu will be displayed, refer to Figure 2-37. Choose **New Discovery Geometry**, **New SpaceClaim Geometry** or **New DesignModeler Geometry** from the shortcut menu; the status bar will display the message **Starting Discovery Geometry**, **Starting SpaceClaim** or **Starting DesignModeler**.

If you choose **DesignModeler**, then after sometime, the **DesignModeler** window will be displayed, refer to Figure 2-38.

Next, in the **DesignModeler** window, create a model according to your design requirements by using the tools available in this window. These tools will be discussed in detail in later chapters. After creating the model, close this window. On doing so, the model will be automatically saved in the ANSYS Workbench database. Also, a green color check mark will be displayed on the right of the **Geometry** cell of the analysis system in the **Project Schematic** window, indicating that the geometry requirement is satisfied for the current system.

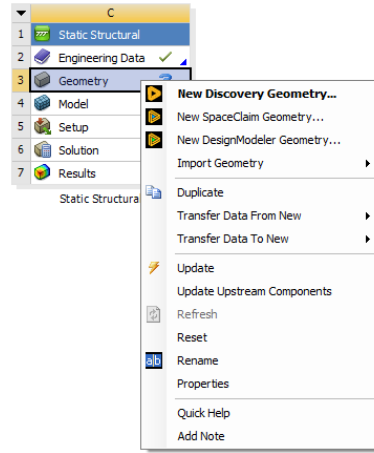


Figure 2-37 The shortcut menu displayed on right-clicking on the **Geometry** cell

After the geometry is specified for the analysis, you need to fulfill the requirement of other cells in the system. The cell types in the system are context specific and are solely dependent on the type of system selected.

In a system tree, you need to update the cells from top to bottom. For example, in a Static Structural analysis system, you need to satisfy the requirement of the **Geometry** cell before you move to the **Model** cell.

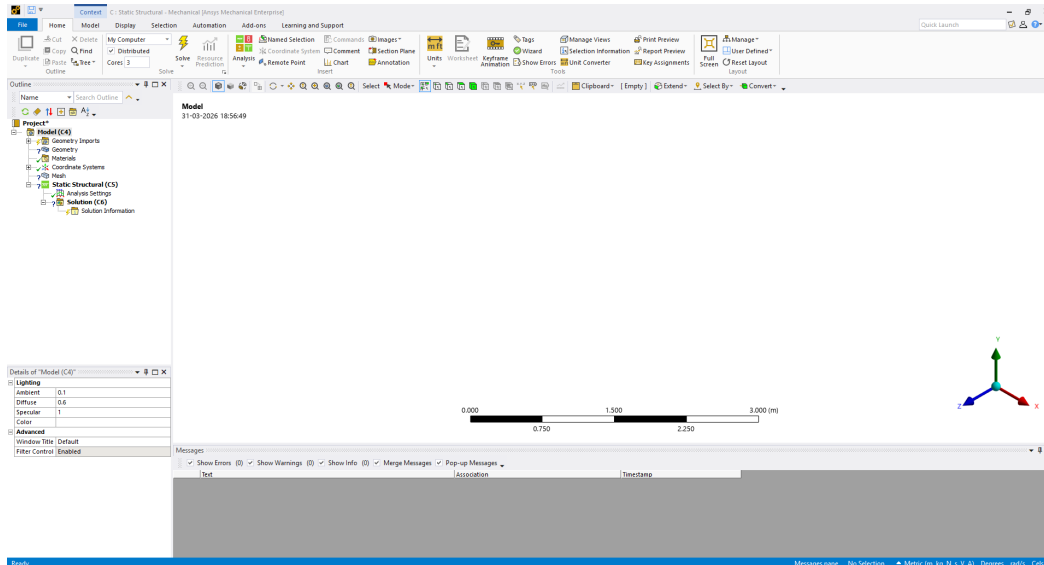


Figure 2-38 A typical **DesignModeler** window



Note

The detailed procedure of working through analysis component cells in an analysis system is given in later chapters.

USING HELP IN ANSYS WORKBENCH

In ANSYS Workbench, there are different ways in which you can access help. These methods are discussed next.

ANSYS Workbench Help

You can get online help and documentation while working on ANSYS Workbench. To access the help, choose ANSYS Workbench Help from the Help menu of the Menu bar; you will be redirected to ANSYS help web page of the Ansys website(ansys.com), as shown in Figure 2-39. This Help page is divided into two parts: Navigation pane (left pane) and Document pane (right pane). The Navigation pane consists of the Contents and the Document pane contains the information related to the topic selected in the Navigation pane. When a particular topic is selected in the left pane, corresponding information or help is displayed in the Document pane. You can open multiple help web pages on your browser by clicking at the desired topics. Apart from the ANSYS Help website, ANSYS Workbench offers two more ways to access help and they are discussed next.

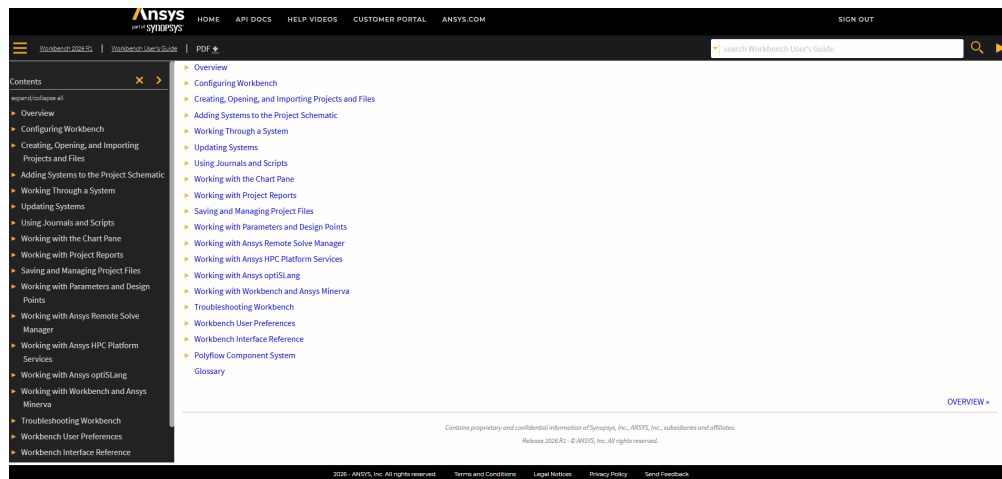


Figure 2-39 The ANSYS Help website

Quick Help

Quick Help is available for all the cells of an analysis or component system in the **Project Schematic** window. Click on the blue inclined arrow at the bottom right corner of any cell; a flyout will be displayed with a short description on the current status of the cell, refer to Figure 2-40. This flyout contains some relevant links that when clicked, will directly open the related document in the **ANSYS Help** window.

Context Sensitive Help

Context sensitive help lists the help topics of the **ANSYS Help** window, related to the currently active option. To open the **ANSYS Help** window, choose **Help > Show Context Help** from the Menu bar; the **Sidebar Help** window will be displayed on the right of the **Project Schematic** window, as shown in Figure 2-41. Alternatively, press the F1 key to open the **Sidebar Help** window.

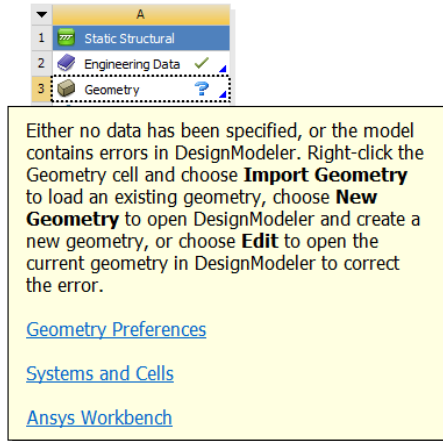


Figure 2-40 The Quick Help

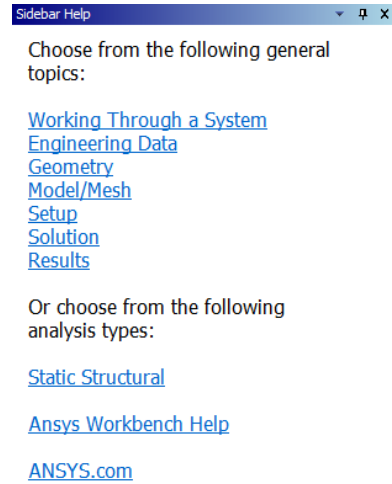


Figure 2-41 The Sidebar Help window

EXITING ANSYS WORKBENCH

To exit the **Workbench** window, choose **File > Exit** from the Menu bar. If the current project is not saved or if you have made any changes after the project was last saved, then on choosing the **Exit** option, the **ANSYS Workbench** message box will be displayed, as shown in Figure 2-42. Choose the **Yes** button from this message box to save the changes made in the current project and exit the **Workbench** window. Choose the **No** button to discard changes and close the **ANSYS Workbench** window. Choose the **Cancel** button from this message box to cancel the exit process.

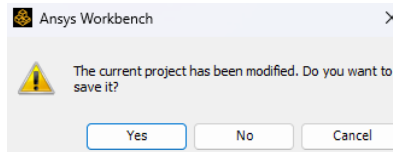


Figure 2-42 The ANSYS Workbench message box

TUTORIAL

The following tutorial is available in video format. Follow the instructions given in the preface to access the videos if you have not already accessed them. Once your registration is complete, you will have access to all video tutorials included in this book.

Tutorial 1

In this tutorial, you will start ANSYS Workbench and add a new Static Structural analysis system to the **Project Schematic** window. After adding the analysis system, save the project with the name `c02_ansWB_tut01` at the location `C:\ANSYS_WB\c02\Tut01`.

(Expected time: 15 min)

The following steps are required to complete this tutorial:

- a. Create the project folder.
- b. Start ANSYS Workbench 2026 R1.
- c. Add the Static Structural analysis system to the **Project Schematic** window.
- d. Save the project and exit the ANSYS Workbench session.

Creating the Project Folder

Before you start ANSYS Workbench, you need to create a project folder in which you will save all your projects.

1. Create a folder with the name **ANSYS_WB** at the location *C:*.

You will save all your projects in the folder *ANSYS_WB*.

Starting ANSYS Workbench 2026 R1

First, you need to start ANSYS Workbench and then add the analysis system to the project.

1. Choose **Start > Ansys 2026 R1 > Workbench 2026 R1** (Windows 10) from the Start menu; the **Workbench** window is displayed.
2. Choose the **Save** button from the **Main** toolbar; the **Save As** dialog box is displayed. 
3. Browse to the location *C:\ANSYS_WB* and then create a folder with the name **c02**.
4. Now, create another folder with the name **Tut01** under the *c02* folder and then open the newly created folder by double-clicking on it.
5. Enter **c02_ansWB_tut01** in the **File name** edit box and then choose the **Save** button from the **Save As** dialog box; the project is saved with the specified name.

Adding Static Structural Analysis System to the Project

Next, you need to add the **Static Structural** analysis system to the project.

1. Double-click on the **Static Structural** option displayed in the **Analysis Systems** toolbox in the **Toolbox** window; the **Static Structural** system is added to the project and is displayed in the **Project Schematic** window.



Note

*The **Static Structural** analysis system can also be added by dragging it from the **Toolbox** window and then dropping it on the **Project Schematic** window. You can also add it by using the options in the shortcut menu that is displayed on right-clicking in the **Project Schematic** window.*

2. Once the project is added to the **Project Schematic** window, its name is highlighted at the bottom of the analysis system in blue. If it is not highlighted, double-click on the name and enter **c02_ansWB_tut01**, as shown in Figure 2-43. The analysis system is renamed.

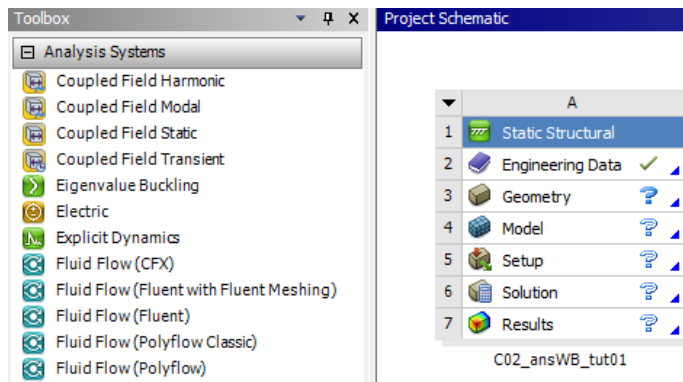


Figure 2-43 The *Static Structural* analysis system added to the project

Saving the Project and Exiting ANSYS Workbench

Now, you need to save the project and exit ANSYS Workbench.

1. Choose the **Save** button to save the project with the name *c02_ansWB_tut01*.
2. Choose **Exit** from the **File** menu; the current ANSYS Workbench session is closed

Overview of FEA Simulation

Tutorial 2 provides an overview of the finite element analysis (FEA) simulation workflow in ANSYS Workbench. In this tutorial, you will perform a complete static structural analysis by creating the geometry, generating the mesh, applying the boundary and loading conditions, solving the finite element model, and evaluating the results. The primary objective of this tutorial is to familiarize you with the overall simulation workflow in ANSYS Workbench. The detailed procedures for geometry creation, mesh generation, application of boundary and loading conditions, and result evaluation and post-processing are discussed in the subsequent chapters.

Tutorial 2

In this tutorial, you will create the model of an I-section cantilever beam and perform a Static Structural analysis. The beam has a flange width of 50 mm and flange/web thickness of 10 mm. The left end of the beam is fixed, and a downward force is applied at the free end. After solving the analysis, you will evaluate the Total Deformation of the beam.

(Expected time: 1.5 hr)

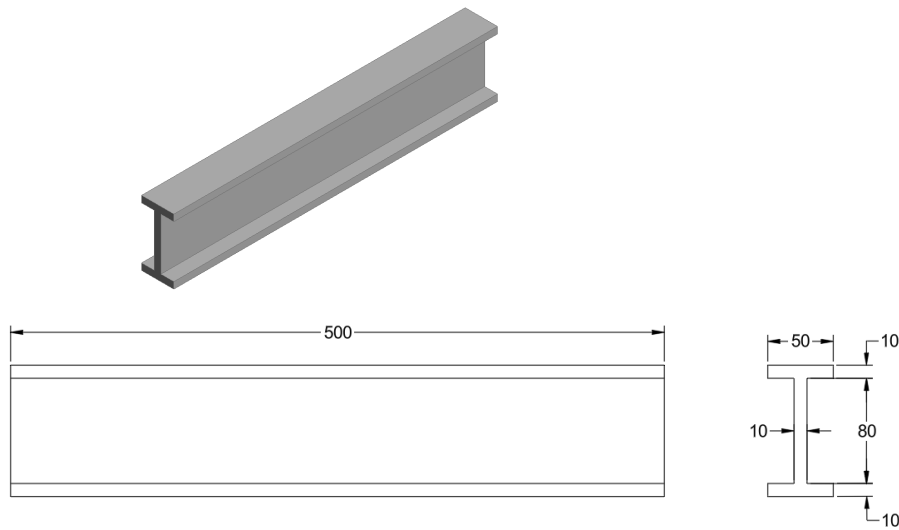


Figure 2-44 The I-Section cantilever beam with dimensions and boundary and loading conditions

The following steps are required to complete this tutorial:

- Start a new project and create the model.
- Generate the default mesh.
- Set the boundary and loading conditions.
- Solve the model.
- Evaluate the total deformation.
- Save the project.

Starting a New Project and Creating the Model

The first step is to start a new project in the **Workbench** window.

- Start ANSYS Workbench.
- Choose the **Save Project** button from the **Standard** toolbar; the **Save As** dialog box is displayed. 
- In this dialog box, enter **c02_ansWB_Tut02** in the **File name** field and then save the file in the location: *C:\ANSYS_WB\c02\Tut02*
- Double-click on **Static Structural** in the **Toolbox** window; the **Static Structural** analysis system is added in the **Project Schematic** window.
- Rename the **Static Structural** analysis system to **I- Section Cantilever**.
- Right-click on the **Geometry** cell of this component system to display a shortcut menu. Next, choose **New DesignModeler Geometry** from the shortcut menu; the **Starting DesignModeler** message is displayed in the status bar. After sometime, the **DesignModeler** window is displayed.

7. Choose the **Millimeter** option from the **Units** menu of the **Menu** bar. Now, create the sketch on the XY plane and extrude, as shown in Figure 2-45. For dimension of the model, refer to Figure 2-44.



Note

The detailed procedure for creating geometry in ANSYS Workbench using DesignModeler is discussed in Chapters 3 and 4.

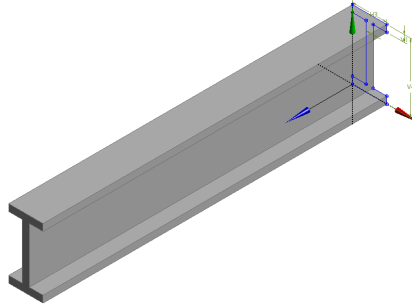


Figure 2-45 Model created on the XY plane

8. Exit the **DesignModeler** window to display the **Workbench** window.

Generating the Mesh

After the model is created in the **DesignModeler** window, you need to generate the mesh for the model in the **Mechanical** window.

1. In the **Project Schematic** window, double-click on the **Model** cell in the **Cantilever** analysis system; the **Mechanical** window is displayed.
2. In the **Outline** pane, right-click on **Mesh**; a shortcut menu is displayed. Choose the **Generate Mesh** option from it to generate the mesh with default settings, as shown in Figure 2-46.

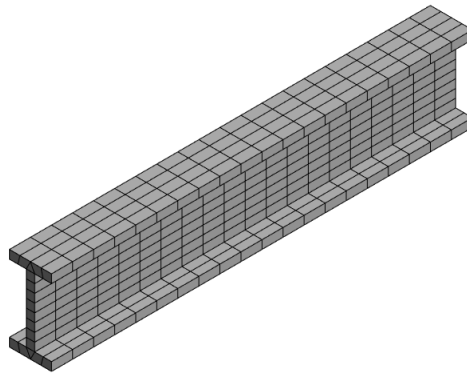


Figure 2-46 Mesh generated with default mesh controls

- Expand the **Statistics** node in the **Details of “Mesh”** window to display the total number of elements created. On doing so, you will find that the total number of elements created is 360.



Setting the Boundary and Loading Conditions

After the mesh is generated, you need to set the boundary and loading conditions under which the analysis will be performed.

- Select the **Static Structural** node in the **Outline** pane; the **Details of “Static Structural (A5)”** window is displayed. Also, the **Environment** contextual tab is displayed, refer to Figure 2-47.
- In the **Environment** contextual tab, choose the **Fixed** tool from the **Structural** group, refer to Figure 2-47; the **Fixed Support** is attached to the **Outline** pane. Also, the **Details of “Fixed Support”** window is displayed.

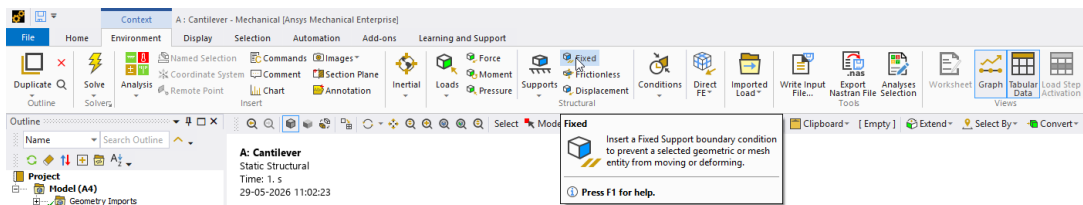


Figure 2-47 Choosing the **Fixed** tool in the **Environment** contextual tab



Note

- The detailed procedure for generating and controlling the mesh in ANSYS Mechanical is discussed in Chapter 7 and 8.
- The detailed procedure for defining analysis settings in a Static Structural analysis system is discussed in Chapter 9.
- The detailed procedure for applying boundary and loading conditions in a Static Structural analysis system is discussed in Chapters 9, 10 and 11.
- Select the **Geometry** selection box in the **Scope** node of the **Details of “Fixed Support”** window to display the **Apply** and **Cancel** buttons, if they are not already displayed.
- Choose the **Face** tool from the **Select** toolbar to select the face to apply fixed support. Next, select the face of the model; the face turns green, as shown in Figure 2-48.
- Choose the **Apply** button in the **Geometry** selection box to confirm the selection of the face for Fixed support; the color of the face turns violet and a flag is attached to the face, as shown in Figure 2-49.

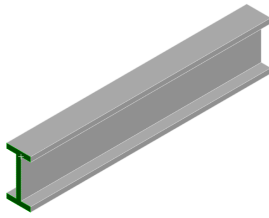


Figure 2-48 Face selected

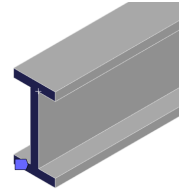


Figure 2-49 The violet color face of the model displaying the Fixed support

After the boundary is defined for the model, you need to define the load for which the analysis is to be carried out.

6. In the **Environment** contextual tab, choose the **Force** tool from the **Structural** group, refer to Figure 2-50; the **Force** is attached to the **Outline** pane. Also, the **Details of “Force”** window is displayed.

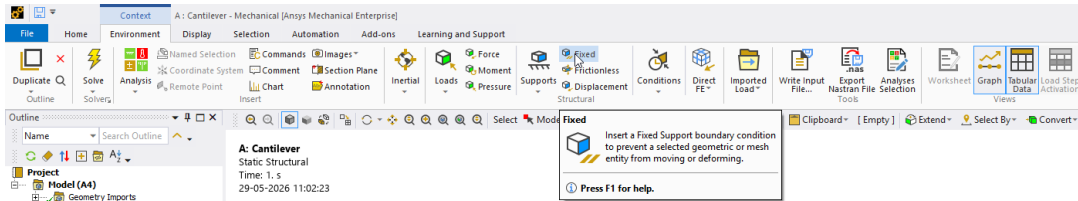


Figure 2-50 Choosing the **Force** tool in the **Environment** contextual tab

7. In the **Details of “Force”** window, select the **Geometry** selection box to display the **Apply** and **Cancel** buttons, if they are not already displayed.
8. Choose the **Edge** tool from the **Select** toolbar to select an edge from the graphics screen.
9. In the graphics screen, select the edge, just opposite to the face on which you have applied the Fixed support, refer to Figure 2-51.

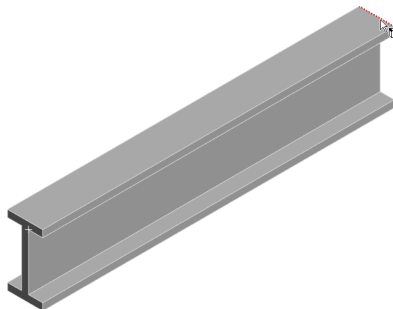


Figure 2-51 Selecting the edge for applying a force load

10. Choose the **Apply** button from the **Geometry** selection box in the **Details of “Force”** window; the edge is selected for applying the Force load.

**Note**

In this tutorial, the direction of application of load is downward.

11. In the **Magnitude** edit box, enter **500** as the magnitude of Force load.

The **Magnitude** edit box is used to specify the magnitude of a vector quantity.

12. Select the **Direction** selection box to display the **Apply** and **Cancel** buttons. Next, select the edge of the model, as shown in Figure 2-52.

The **Direction** selection box is used to specify the direction of a vector quantity.

13. Choose the **Flip** toggle button available in the graphics screen, refer to Figure 2-53, to specify the direction of load as downward.

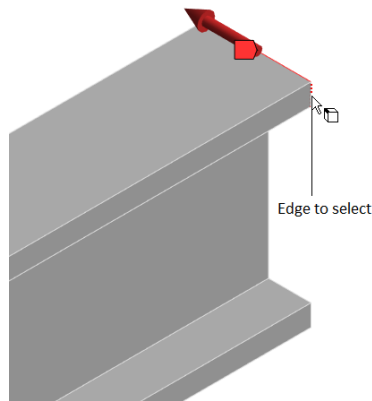


Figure 2-52 Selecting the edge to specify the direction for applying the load

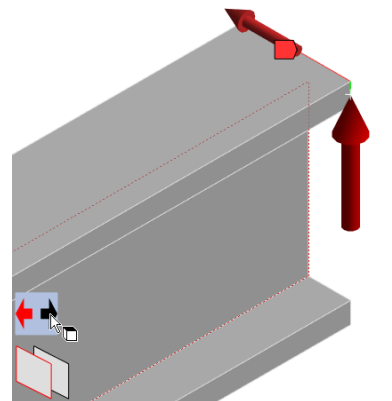


Figure 2-53 Choosing the **Flip** button from the Graphics screen

14. Choose the **Apply** button in the **Direction** selection box in the **Details of “Force”** window.

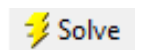
Now, the preprocessing part is complete. Next, you need to work on the solution part of the analysis.

Solving and Post-Processing the Finite Element Model

After the boundary and loading conditions are specified for the analysis, you need to evaluate the results that are of importance in the case of a particular analysis. You can view the response of the model under the given boundary and loading conditions. The various results that can be evaluated are: Deformation, Stress, Strain, Energy, and Linearized Stress.

To evaluate the results in this analysis, follow the procedure explained next.

1. Select the **Solution** node in the **Outline** pane; the **Solution** contextual tab is displayed. Also, the **Details of “Solution”** window is displayed, refer to Figure 2-54.



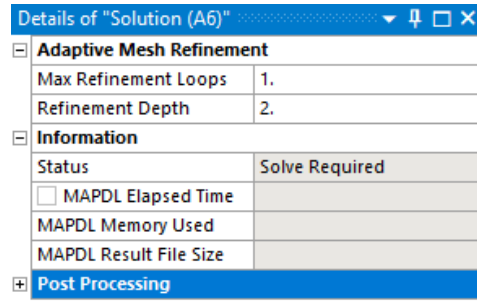


Figure 2-54 The Details of "Solution" window



Note

The detailed procedure for working with the Solution node, inserting result objects, solving the finite element model, and evaluating the analysis results in ANSYS Mechanical is discussed in Chapter 9, 10, and 11.

- Choose the **Total** tool from the **Deformation** drop-down in the **Solution** contextual tab, as shown in Figure 2-55; **Total Deformation** is attached under the **Solution** node. Also, the **Details of "Total Deformation"** window is displayed.

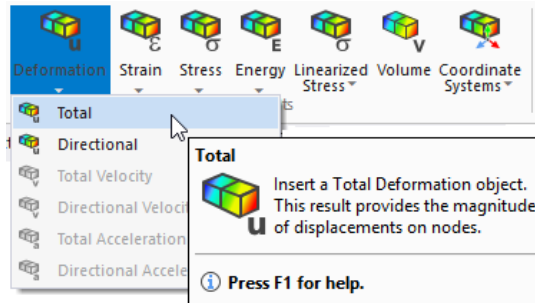


Figure 2-55 Choosing the **Total** tool from the **Deformation** drop-down

- Choose the **Solve** tool from the **Standard** toolbar; the **ANSYS Workbench Solution Status** dialog box is displayed and after sometime, a green tick mark is placed before **Total Deformation**, indicating that they are evaluated.
- Select **Total Deformation** in the **Outline** pane; the corresponding contours are displayed in the Graphics screen, as shown in Figure 2-56.

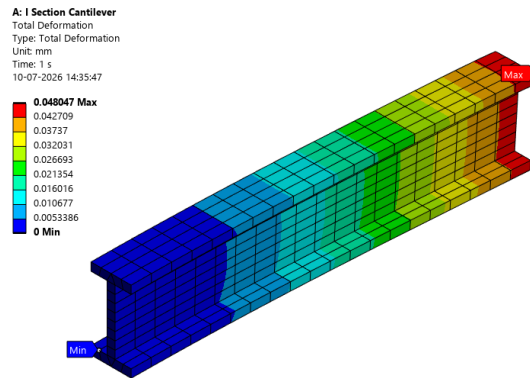


Figure 2-56 Total deformation contour in the model

Saving the Project and Exiting ANSYS Workbench

Now, you need to save the project and exit ANSYS Workbench.

1. Choose the **Save** button to save the project with the name *c02_ansWB_tut02*.
2. Choose **Exit** from the **File** menu; the current ANSYS Workbench session is closed

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. The archived project files are saved in _____ format.
2. To view all files related to the current project, choose _____ from the Menu bar.
3. Press the _____ key to open the **Sidebar Help** window.
4. In the **Workbench** window, you can change the units by using the options available in the _____ menu.
5. You can add an analysis system to the **Project Schematic** window by double clicking on the desired component in the **Analysis Systems** toolbox. (T/F)
6. The geometry for an analysis can not be imported from an external software package. (T/F)
7. The **Results** cell is used to display the contents of an analysis system. (T/F)
8. If you choose an option that has 3 dots (...) on its right then a dialog box corresponding to that option is displayed. (T/F)

Review Questions

Answer the following questions:

1. Which one of the following unit systems is assigned to a project by default?
 - (a) Metric (kg,m,s,°C,A,N,V)
 - (b) SI (kg,m,s,K,A,N,V)
 - (c) U.S. Engineering (lbm,in,s,R,A,lbf,V)
 - (d) U.S. Customary (lbm,in,s,°F,A,lbf,V)
2. An ANSYS project file is saved with _____ extension.
3. The _____ environment is used to create the geometry to be used in an analysis.
4. The customized analysis systems are added under the _____ toolbox in the **Toolbox** window.
5. To create a connected system for coupled analysis, double-click on an analysis system in the **Toolbox**. (T/F)
6. The *.wbpj file is the default extension for the project files in ANSYS Workbench 2026 R1. (T/F)
7. A project consists of systems and a system consists of cells. (T/F)
8. You can specify a unit system from the Menu bar. (T/F)
9. The **Engineering Data** cell is used to define the geometry to be used in an analysis project. (T/F)

EXERCISE

Exercise 1

Start ANSYS Workbench and add a new thermal analysis system to the project, as shown in Figure 2-44. Save the project with the name *c02_exr01* at the location *C:\ANSYS_WB\c02\Exr*:

(Expected time: 10 min)

Hint: Use the **Thermal-Stress** analysis system displayed under the **Custom Systems** toolbox in the **Toolbox**.

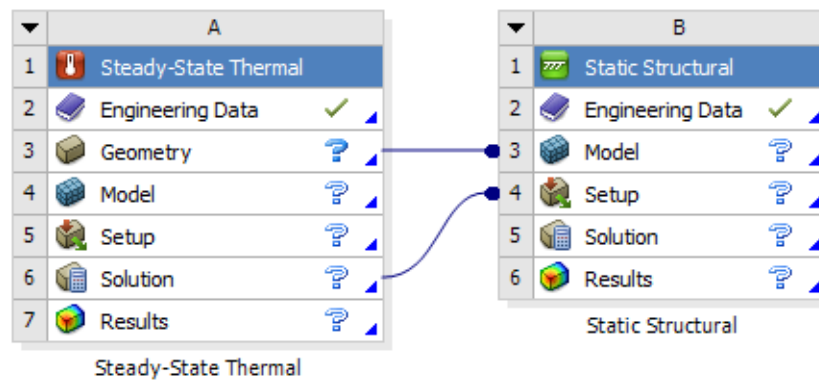


Figure 2-44 The *Thermal-Stress* analysis system added to the *Project Schematic* window

Answers to Self-Evaluation Test

1. *.zip, 2. View > Files, 3. F1, 4. Units, 5. T, 6. F, 7. F, 8. T