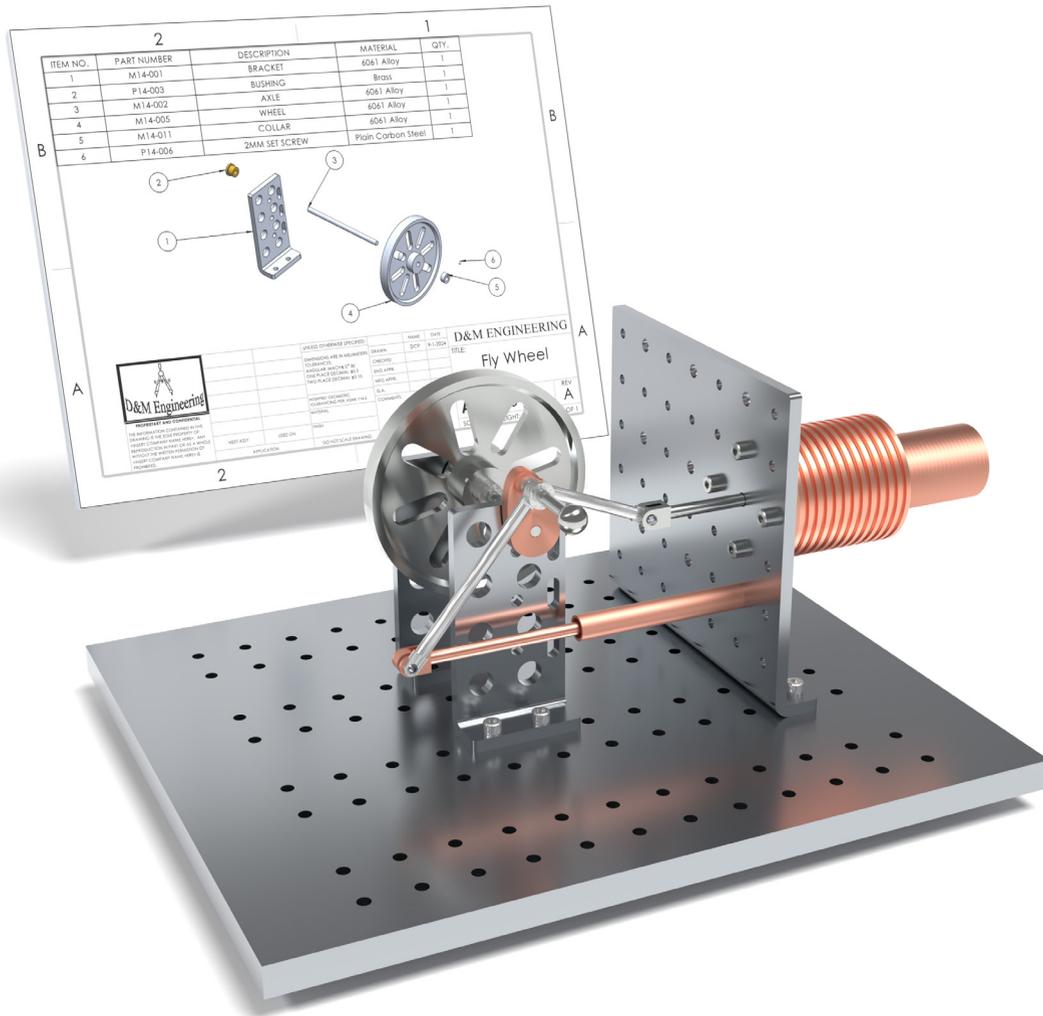


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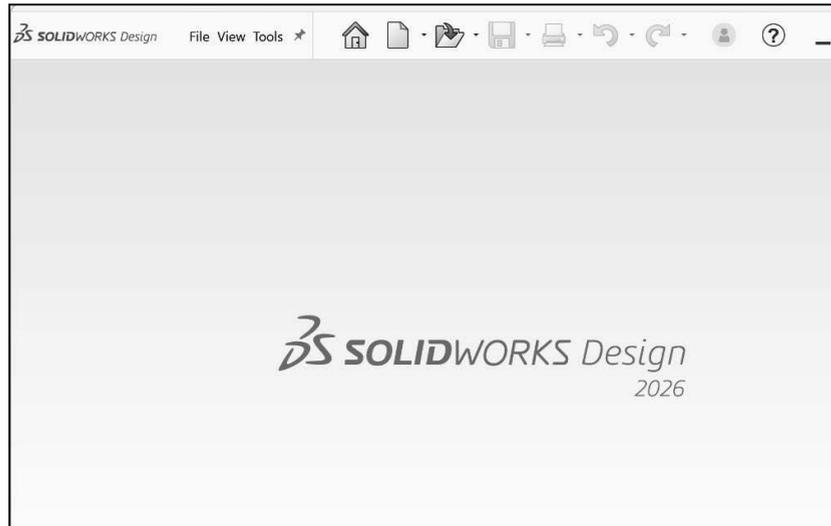
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Chapter 1

Overview of SOLIDWORKS® 2026 and the User Interface



Below are the desired outcomes and usage competencies based on the completion of Chapter 1.

Desired Outcomes:	Usage Competencies:
<ul style="list-style-type: none"> • A comprehensive understanding of the SOLIDWORKS® User Interface (UI) and CommandManager. 	<ul style="list-style-type: none"> • Ability to establish a SOLIDWORKS session. • Aptitude to utilize the following items: Menu bar toolbar, Menu bar menu, Drop-down menus, Context toolbars, Consolidated drop-down toolbars, System feedback icons, Task Pane, Confirmation Corner, Heads-up View toolbar and System and Document Properties. • Open a new and existing SOLIDWORKS part. Set units, precision and Drafting Standard. Address Y-up orientation. • Knowledge to zoom, rotate and maneuver a three-button mouse in the SOLIDWORKS Graphics window.

Notes:



Chapter 1 - Overview of SOLIDWORKS® 2026 and the User Interface

Chapter Objective

Chapter 1 introduces the user to the SOLIDWORKS User Interface, CommandManager, Document and System properties. Review and address: Menu bar toolbar, Menu bar menu, Drop-down menus, Context toolbars, Consolidated drop-down toolbars, System feedback icons, Confirmation Corner and Heads-up View toolbar.

Start a new SOLIDWORKS session. Open an existing part. View the created features and sketches using the Rollback bar. Examine design intent.

On the completion of this chapter, you will be able to:

- Utilize the Welcome - SOLIDWORKS dialog box.
- Establish a SOLIDWORKS session.
- Comprehend the SOLIDWORKS User Interface.
- Identify the default Reference Planes in the FeatureManager.
- Address Y-up orientation setting.
- Open a new and existing SOLIDWORKS part.
- Set Document Properties and System Properties.
- Utilize the FeatureManager Rollback bar.
- Locate Help and the SOLIDWORKS Tutorials.
- Zoom, rotate and maneuver a three-button mouse in the SOLIDWORKS Graphics window.

What is SOLIDWORKS®?

- SOLIDWORKS® is a design tool by Dassault Systèmes. The 3DEXPERIENCE® Platform is a technology platform that connects SOLIDWORKS to other design, simulation, intelligent information and collaboration Apps through a web browser.
- SOLIDWORKS® is a mechanical design automation software package used to build parts, assemblies and drawings that takes advantage of the familiar Microsoft® Windows graphical user interface.
- SOLIDWORKS is an easy to learn design and analysis tool (SOLIDWORKS Simulation, SOLIDWORKS Motion, SOLIDWORKS Flow Simulation, Sustainability, etc.), which makes it possible for designers to quickly sketch 2D and 3D concepts, create 3D parts and assemblies and detail 2D drawings.
- Model dimensions in SOLIDWORKS are associative between parts, assemblies and drawings. Reference dimensions are one-way associative from the part to the drawing or from the part to the assembly.
- This book is written for the beginner to intermediate user.

Start a SOLIDWORKS Session

Start a SOLIDWORKS session and familiarize yourself with the SOLIDWORKS User Interface. As you read and perform the tasks in this chapter, you will obtain a sense of how to use the book and the structure. Actual input commands or required actions in the chapter are displayed in bold.

The book does not cover starting a SOLIDWORKS session in detail for the first time. A default SOLIDWORKS installation presents you with several options. For additional information, visit <http://www.SOLIDWORKS.com>.

SOLIDWORKS provides the ability to specify workspace settings for display. Click **View, Workspace** from the main menu. View the options: **Simplified Interface, Default, Widescreen, Dual Monitor** and **Customize Menu**. The book covers the Default user interface.

Activity: Start a SOLIDWORKS Session.

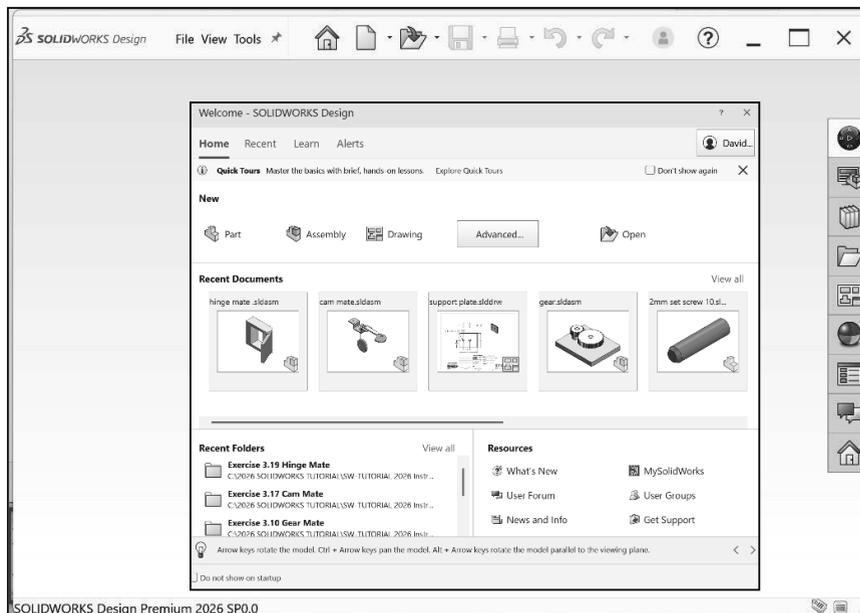
Start a SOLIDWORKS session.

- 1) Type **SOLIDWORKS Design 2026** in the Search window.
- 2) Click the **SOLIDWORKS Design 2026** application (or if available, **double-click** the SOLIDWORKS icon on the desktop). The Welcome - SOLIDWORKS Design dialog box is displayed by default.



The Welcome - SOLIDWORKS Design box provides a convenient way to open recent documents (Parts, Assemblies and Drawings), view recent folders, access SOLIDWORKS resources, and stay updated on SOLIDWORKS news.

- 3) **View** your options. Do not open a document at this time. **Note:** The Task Pane icon order or tab display may be different.



Home Tab

The Home tab lets you open new and existing documents, view recent documents and folders and access SOLIDWORKS resources (*Part, Assembly, Drawing, Advanced mode, Open*).

Recent Tab

The Recent tab lets you view a longer list of recent documents and folders. Sections in the Recent tab include *Documents, Folders and Document Groups*.

The Documents section includes thumbnails of documents that you have opened recently.

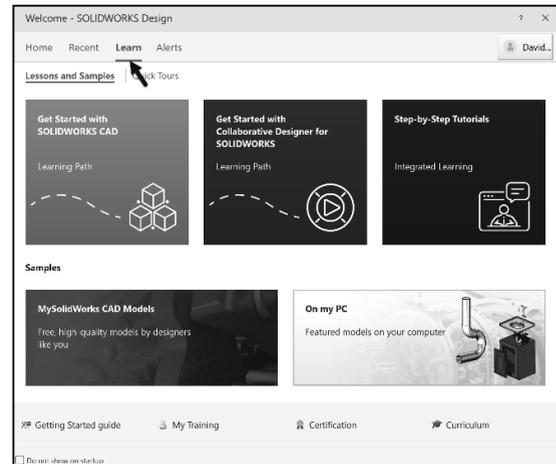
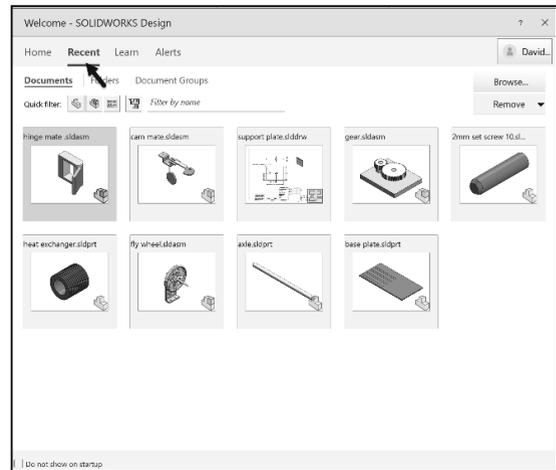
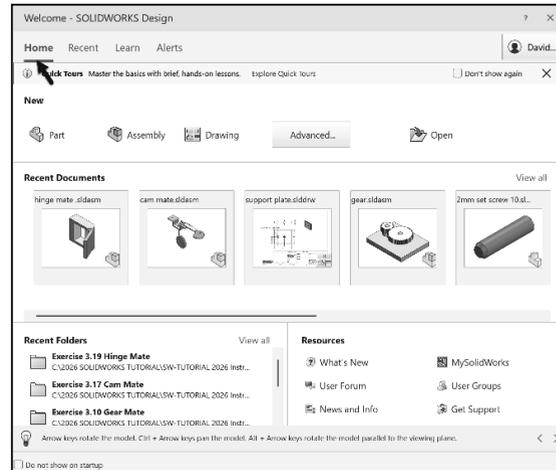
Click a thumbnail to open the document, or hover over a thumbnail to see the document location and access additional information about the document. When you hover over a thumbnail, the full path and last saved date of the document appear.

Learn Tab

The Learn tab lets you access instructional resources to help you learn more about the SOLIDWORKS software.

Sections in the Learn tab include:

- **Get Started with SOLIDWORKS CAD.**
- **Get Started with Collaborative Designer for SOLIDWORKS (Learning Path).**
- **Step-by-Step Tutorials (Integrated Learning).**
- **MySolidWorks CAD models.**
- **On my PC.**
- **Getting Started guide.**



- **My Training.** Open the My Training section at MySolidWorks.com.
- **Certification.** Open the SOLIDWORKS Certification Program section at solidworks.com. You will need to create an account with a password.
- **Curriculum.** Open the Curriculum section at solidworks.com. You will need to create an account with a password.



When you install the software, if you do not install the Help Files or Example Files, the Tutorials and Samples links are unavailable.

Alerts Tab

The Alerts tab keeps you updated with SOLIDWORKS news.

Sections in the Alerts tab include Critical, Troubleshooting, and Technical.



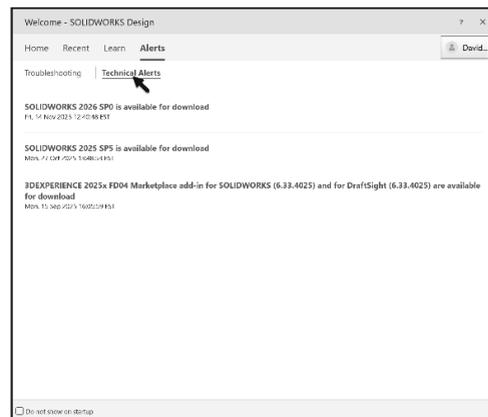
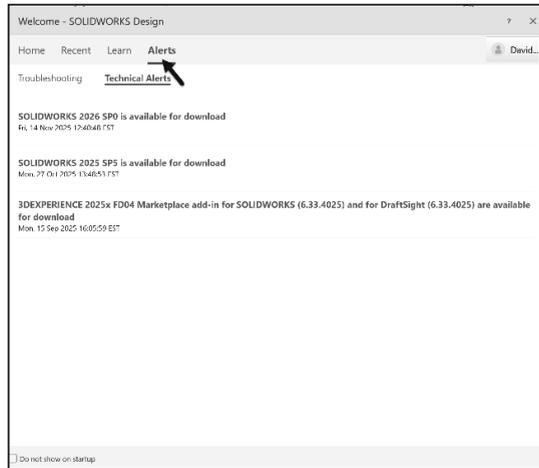
The Critical section does not appear if there are no critical alerts to display.

- **Troubleshooting.** Includes troubleshooting messages and recovered documents that used to be on the SOLIDWORKS Recovery tab in the Task Pane.



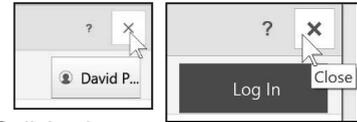
If the software has a technical problem and an associated troubleshooting message exists, the Welcome dialog box opens to the Troubleshooting section automatically on startup, even if you selected **Do not show at startup** in the dialog box.

- **Technical Alerts.** Open the contents of the SOLIDWORKS Support Bulletins RSS feed (Hotfixes, release news) at solidworks.com.

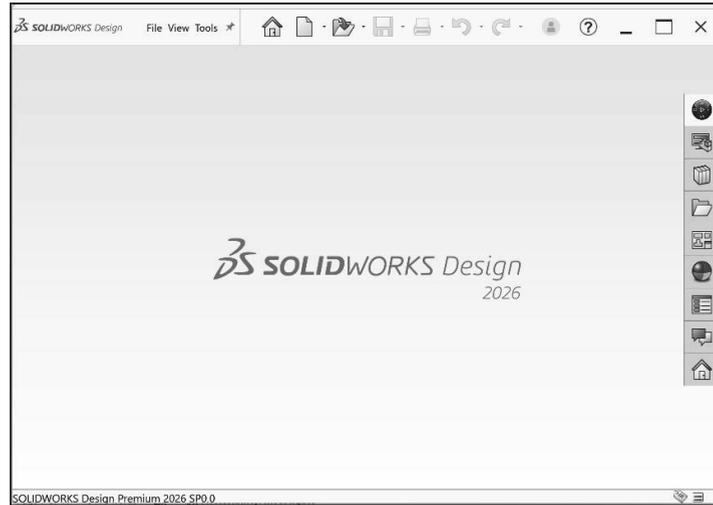


Close the Welcome - SOLIDWORKS dialog box.

- 4) Click **Close**  from the Welcome - SOLIDWORKS dialog box. The SOLIDWORKS Graphics window is displayed.
Note: You can also click outside the Welcome - SOLIDWORKS dialog box, in the Graphics window.

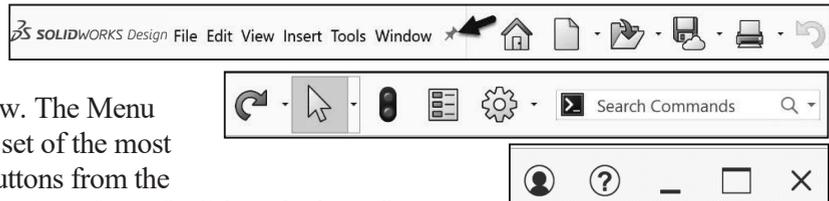


View the SOLIDWORKS Graphics window.



Menu Bar toolbar

The SOLIDWORKS (UI) is designed to make maximum use of the Graphics window. The Menu Bar toolbar contains a set of the most frequently used tool buttons from the Standard toolbar. **Note:** If you have the 3DEXPERIENCE platform, the SOLIDWORKS default Save icon will display additional commands.



The following default tools are available:

- **Welcome to SOLIDWORKS**  - Open the Welcome dialog box; **New**  - Create a new document; **Open**  - Open an existing document; **Save**  - Save an active document; **Print**  - Print an active document; **Undo**  - Reverse the last action; **Redo**  - Redoes the last action that you reverse; **Select**  - Select Sketch entities, components and more; **Rebuild**  - Rebuild the active part, assembly or drawing; **File Properties**  - Summary information on the active document; **Options**  - Change system options and Add-Ins for SOLIDWORKS; **Login**  - Login to SOLIDWORKS. You will need to create an account with a password; **Help**  - access to help, tutorials, updates and more.

Menu Bar menu (No model open)

SOLIDWORKS provides a context-sensitive menu structure. The menu titles remain the same for all

three types of documents

(Parts, Assemblies and

Drawings), but the menu

items change depending on which type of document is active. **Note:** If you have the

3DEXPERIENCE Platform, the SOLIDWORKS default Save icon will display additional commands.



Menu Bar menu (Model open)

The Pin  option displays the Menu bar toolbar and the Menu bar menu as illustrated.

Throughout the book, the Menu bar menu and the Menu bar toolbar are referred to as the

Menu bar. **Note:** If you have the 3DEXPERIENCE Platform, the SOLIDWORKS default

Save icon will display additional commands.



Drop-down menu (Open part document)

SOLIDWORKS takes advantage of the familiar Microsoft®

Windows user interface. Communicate with

SOLIDWORKS through drop-down menus, Context

sensitive toolbars, Consolidated toolbars or the

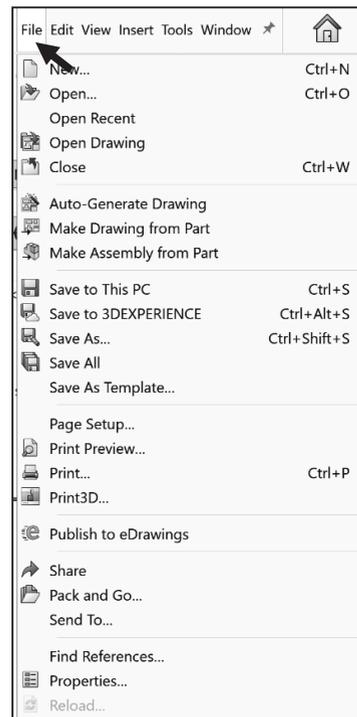
CommandManager tabs.

To close a SOLIDWORKS drop-down menu, press the Esc

key. You can also click any other part of the

SOLIDWORKS Graphics window or click another drop-

down menu.



Create a New Part Document

Activity: Create a new Part Document.

A part is a 3D model, which consists of features. What are features? Features are geometry building blocks. Most features either add or remove material. Some features do not affect material (Cosmetic Thread).

Features are created either from 2D or 3D sketched profiles or from edges and faces of existing geometry.

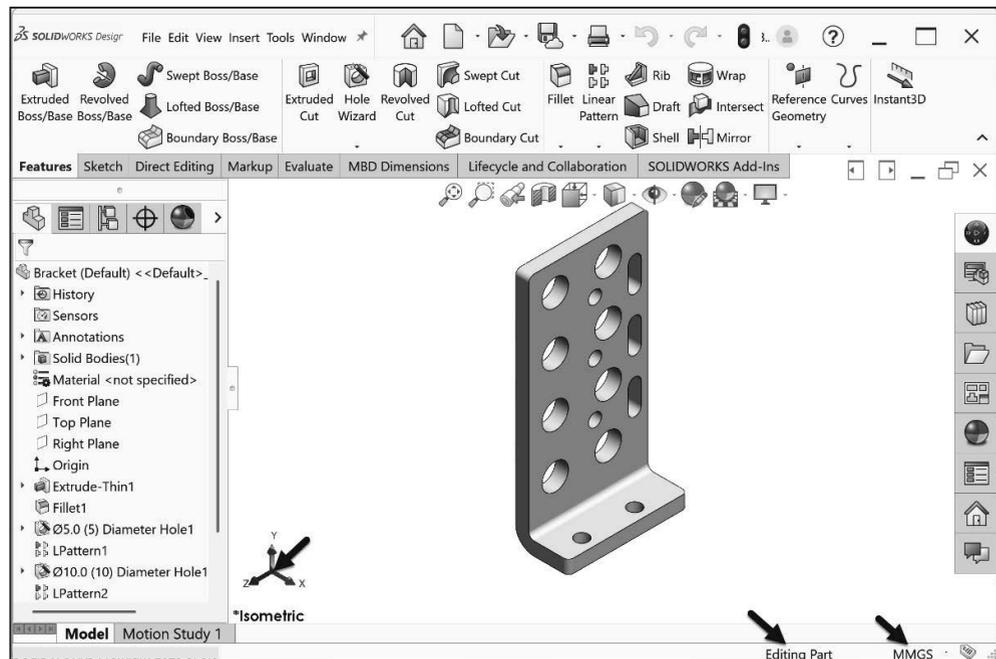
The first sketch of a part is called the Base Sketch. The Base sketch is the foundation for the 3D model. The book focuses on 2D sketches and 3D features.

Model about the origin; this provides a point of reference.

Features are individual shapes that combined with other features make up a part or assembly. Some features, such as bosses and cuts, originate as sketches. Other features, such as shells and fillets, modify a feature's geometry.

Note: When you create a part or assembly, you can choose a default part or assembly template for Y-up (Y-axis points upward) or Z-up (Z-axis points upward) orientation. In earlier releases, SOLIDWORKS only had a default Y-up orientation. All models in the book are written for a default Y-up orientation. View the Triad in the Graphics area for Y axis orientation.

 FeatureManager tabs, CommandManager tabs, Task Pane tabs and other tabs will vary depending on system setup, version and Add-ins.



There are two modes in the New SOLIDWORKS Document dialog box: Novice and Advanced. The Novice option is the default option with three templates.

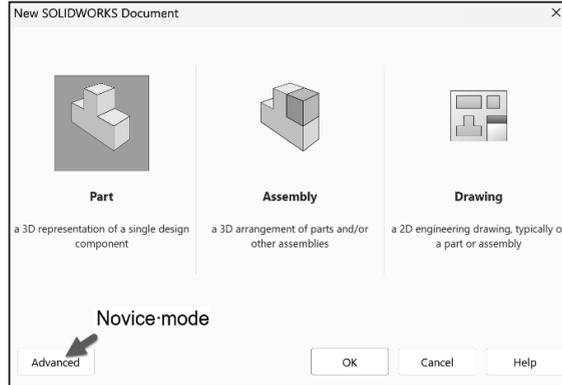


The Advanced mode contains access to additional templates and tabs that you create in System options.

Utilize the Advanced mode and accept the default Y-up orientation in this book.

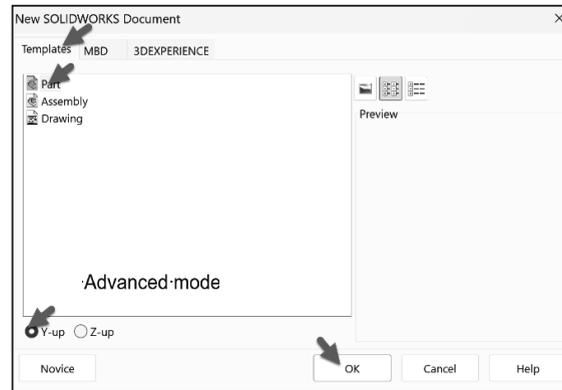
Create a new part.

- 5) Click **New**  from the Menu bar as illustrated. The New SOLIDWORKS Document dialog box is displayed.



Select the Advanced mode.

- 6) If needed, click the **Advanced** tab. The New SOLIDWORKS Document box is displayed.
- 7) Click the **Templates** tab. Accept the default Y-up orientation.
- 8) Click **Part**. Part is the default template from the New SOLIDWORKS Document dialog box.
- 9) Click **OK** from the New SOLIDWORKS Document dialog box.



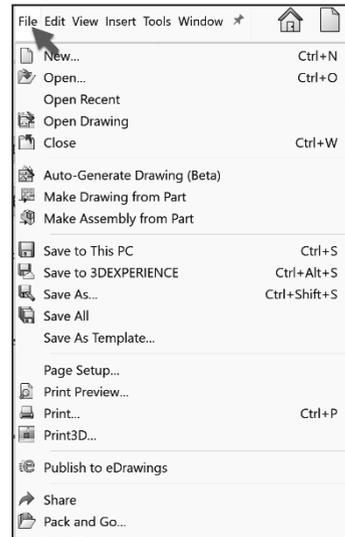
The Advanced mode remains selected for all new documents in the current SOLIDWORKS session. When you exit SOLIDWORKS, the Advanced mode setting is saved.

The default SOLIDWORKS installation contains two tabs in the New SOLIDWORKS Document dialog box: Templates and MBD.

The Templates tab corresponds to the default SOLIDWORKS templates.

The MBD tab corresponds to the templates utilized in the SOLIDWORKS (Model Based Definition). You may see a Tutorial tab depending on your version.

Note: 3DEXPERIENCE users can create templates for the 3DEXPERIENCE Platform directly from SOLIDWORKS. Use the 3DEXPERIENCE tab in the New SOLIDWORKS Document dialog box.



Part1 is displayed in the FeatureManager and is the name of the document. Part1 is the default part name.

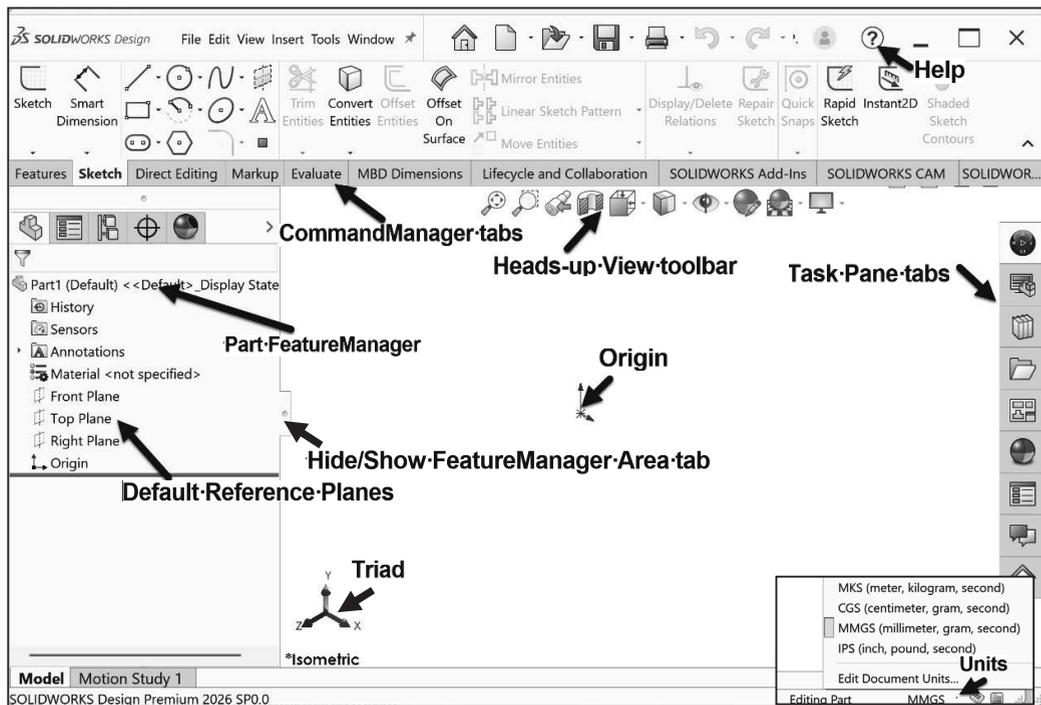
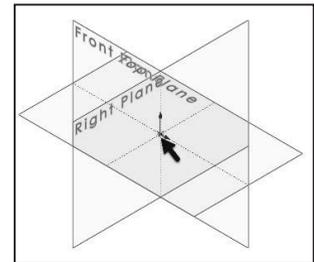
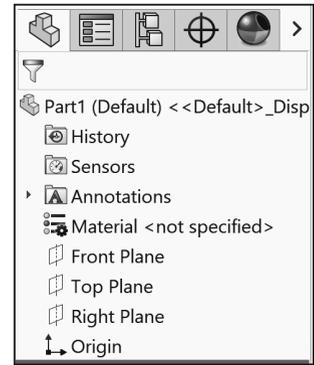
The Part Origin  is displayed in blue in the center of the Graphics window. The Origin represents the intersection of the three default reference planes: *Front Plane*, *Top Plane* and *Right Plane*.

Model about the origin; this provides a point of reference.

The positive X-axis is horizontal and points to the right of the Origin in the Front view. The positive Y-axis is vertical and points upward in the Front view. The FeatureManager contains a list of features, reference geometry, and settings utilized in the part.

 Edit the document units directly from the Graphics window as illustrated.

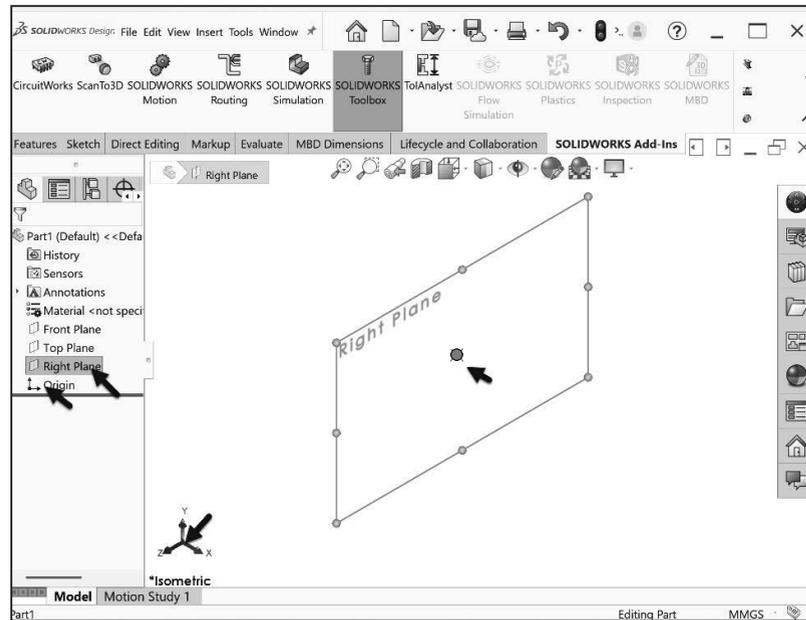
 CommandManager tabs, FeatureManager tabs, Task Pane tabs will vary depending on system setup, version and Add-ins.



SOLIDWORKS-release

View the Default Sketch Planes.

- 10) Click the **Front Plane** from the FeatureManager. The Front Plane is displayed.
- 11) Click the **Top Plane** from the FeatureManager. The Top Plane is displayed.
- 12) Click the **Right Plane** from the FeatureManager. The Right Plane is displayed.
- 13) Hover over the **Origin** from the FeatureManager. The Origin is the intersection of the Front, Top and Right Plane. The Origin point is displayed.
- 14) Click **inside** the Graphics window.

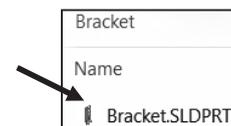
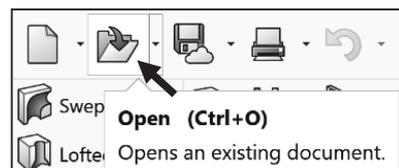


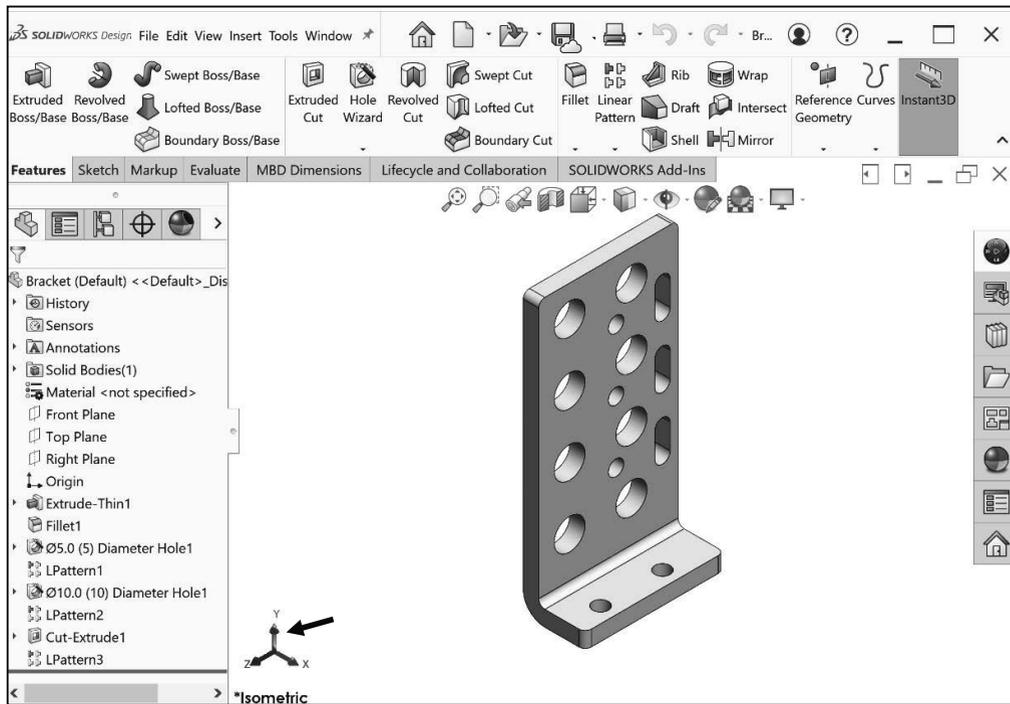
Download all model files from the SDC Publication website (www.SDCpublications.com/downloads/978-1-63057-778-0). Open the Bracket part. Review the features and sketches in the Bracket FeatureManager. Work directly from a local hard drive.

Activity: Download the SOLIDWORKS folder. Open the Bracket Part.

Download the SOLIDWORKS-MODELS 2026 folder. Open an existing SOLIDWORKS part.

- 15) **Download** the SOLIDWORKS-MODELS 2026 folder.
- 16) **Unzip** the folder. **Work** from the unzip folder.
- 17) Click **Open**  from the Menu bar menu.
- 18) Browse to the **SOLIDWORKS-MODELS 2026\Bracket** folder.
- 19) Double-click the **Bracket** part. The Bracket part is displayed in the Graphics window.





The FeatureManager design tree is located on the left side of the SOLIDWORKS Graphics window. The FeatureManager provides a summarized view of the active part, assembly, or drawing document. The tree displays the details on how the part, assembly or drawing document was created.

Use the FeatureManager rollback bar to temporarily roll back to an earlier state, to absorb features, roll forward, roll to previous, or roll to the end of the FeatureManager design tree.

You can add new features or edit existing features while the model is in the rolled-back state. You can save models with the rollback bar placed anywhere.

In the next section, review the features in the Bracket FeatureManager using the Rollback bar.

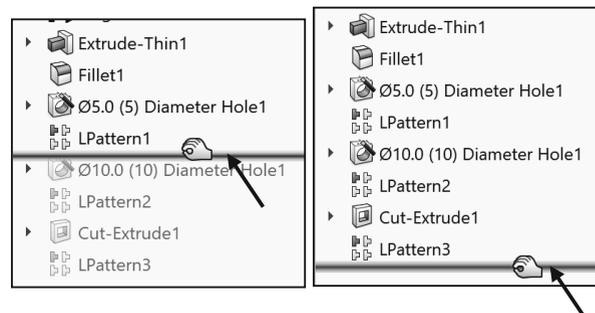
Activity: Use the FeatureManager Rollback Bar option.

Apply the FeatureManager Rollback Bar. Revert to an earlier state in the model.

- 20)** Place the **mouse pointer** over the rollback bar in the FeatureManager design tree as illustrated. The pointer changes to a hand . Note the provided information on the feature. This is called Dynamic Reference Visualization.

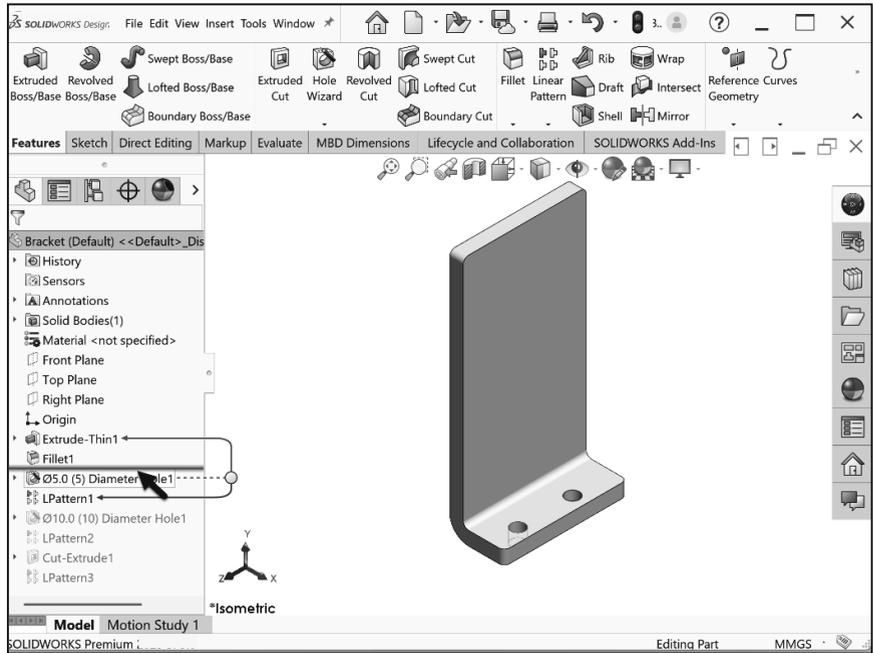
- 21)** Drag the **rollback bar** up the FeatureManager design tree until it is above the features you want rolled back, in this case 10.0 (10) Diameter Hole1.

- 22)** **Release** the mouse button.



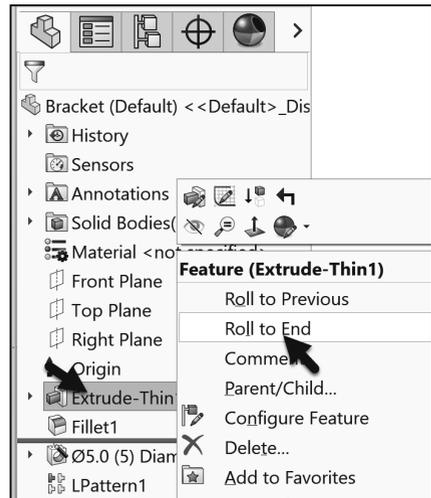
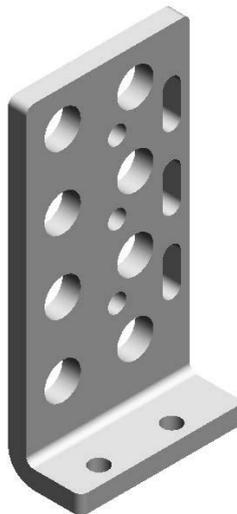
View the first feature in the Bracket Part.

- 23) Drag the **rollback bar** up the FeatureManager above Fillet1. View the results in the Graphics window. This is a great tool to re-engineer a part.



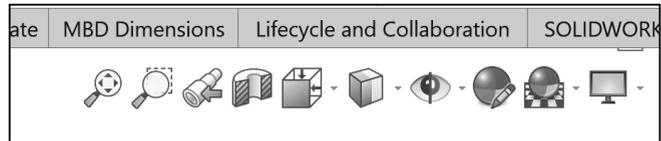
Return to the original Bracket Part FeatureManager.

- 24) Right-click **Extrude-Thin1** in the FeatureManager. The Pop-up Context toolbar is displayed.
- 25) Click **Roll to End**. View the results in the Graphics window.



Heads-up View toolbar

SOLIDWORKS provides the user with numerous view options. One of the most useful tools is the Heads-up View toolbar displayed in the Graphics window when a document is active.



In the next section, apply the following tools: Zoom to Fit, Zoom to Area, Zoom out, Rotate and select various view orientations from the Heads-up View toolbar.

Activity: Utilize the Heads-up View toolbar.

Zoom to Fit the model in the Graphics window.

- 26)** Click the **Zoom to Fit**  icon. The tool fits the model to the Graphics window.

Zoom to Area on the model in the Graphics window.

- 27)** Click the **Zoom to Area**  icon. The Zoom to Area  icon is displayed.

Zoom in on the top left hole.

- 28)** **Window-select** the top left corner as illustrated. View the results.

De-select the Zoom to Area tool.

- 29)** Click the **Zoom to Area**  icon.

Fit the model to the Graphics window.

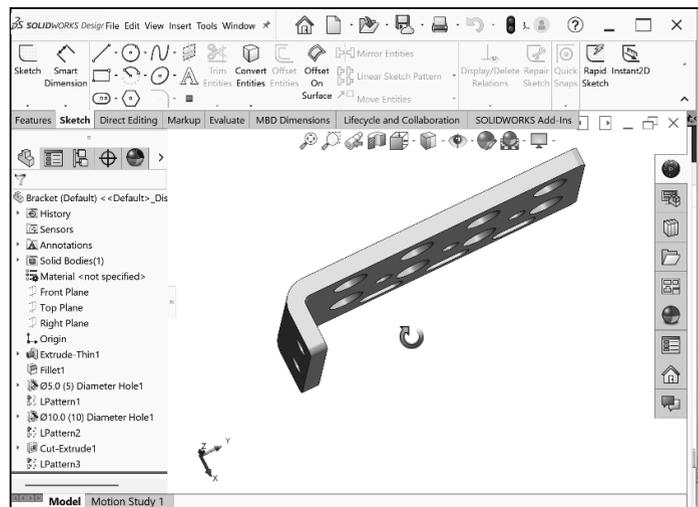
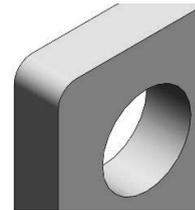
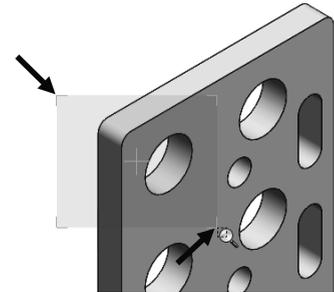
- 30)** Press the **f** key.

Rotate the model.

- 31)** Hold the **middle mouse button** down.

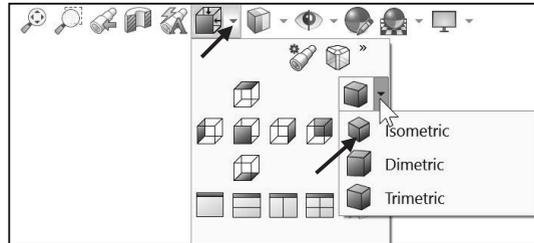
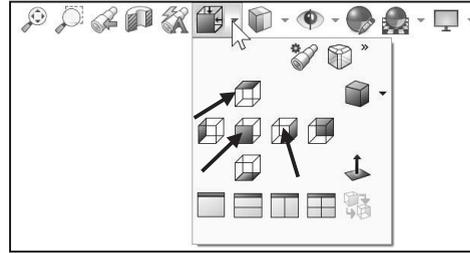
- 32)** Drag **upward** , **downward** , to the **left**  and to the **right**  to rotate the model in the Graphics window.

- 33)** **Release** the middle mouse button.



Display a few Standard Views.

- 34) Click **inside** the Graphics window.
- 35) Click **Front**  from the drop-down Heads-up view toolbar. The model is displayed in the Front view.
- 36) Click **Right**  from the drop-down Heads-up view toolbar. The model is displayed in the Right view.
- 37) Click **Top**  from the drop-down Heads-up view toolbar. The model is displayed in the Top view.

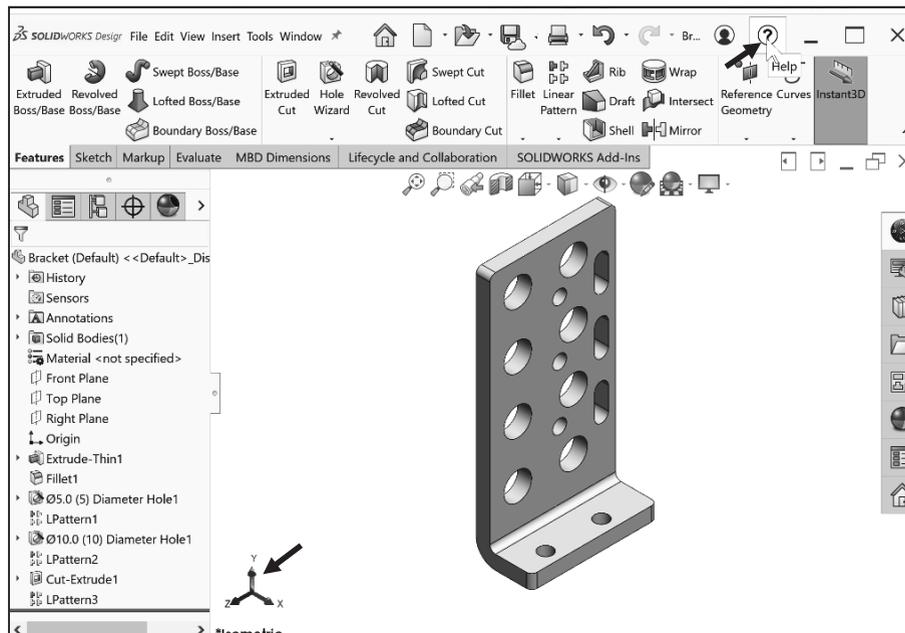


Display an Isometric view of the model.

- 38) Click **Isometric**  from the drop-down Heads-up view toolbar as illustrated. Note your options. View the results in the Graphics window.

SOLIDWORKS Help

Help in SOLIDWORKS is context-sensitive and in HTML format. Help is accessed in many ways, including Help buttons in all dialog boxes and PropertyManager and Help  on the Standard toolbar for SOLIDWORKS Help.



- 39) Click  from the Standard toolbar.
- 40) Click  **Help** from the drop-down menu. The SOLIDWORKS Home Page is displayed by default. View your options.

 SOLIDWORKS Web Help is active by default under Help in the Main menu.

Close Help. Return to the SOLIDWORKS Graphics window.

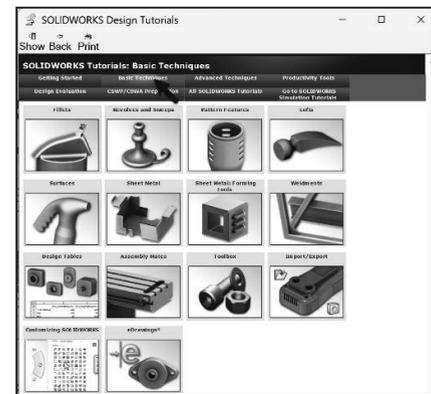
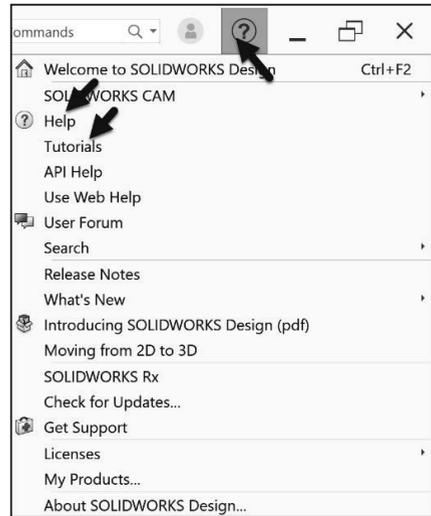
- 41) Close  SOLIDWORKS Home.

SOLIDWORKS Tutorials

Display and explore the SOLIDWORKS Tutorials.

- 42) Click  from the Standard toolbar.
- 43) Click **Tutorials**. The SOLIDWORKS Tutorials are displayed. The SOLIDWORKS Tutorials are presented by category.
- 44) Click the **Getting Started** category. The Getting Started category provides lessons on parts, assemblies, and drawings.
- 45) Click the **Basic Techniques** category. View the provided information.

In the next section, exit SOLIDWORKS Tutorials. View additional user interface tools.



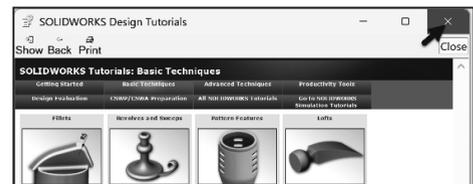
Activity: Close the SOLIDWORKS Tutorials.

Close SOLIDWORKS Tutorials.

- 46) Close  SOLIDWORKS Tutorials.

User Interface Tools

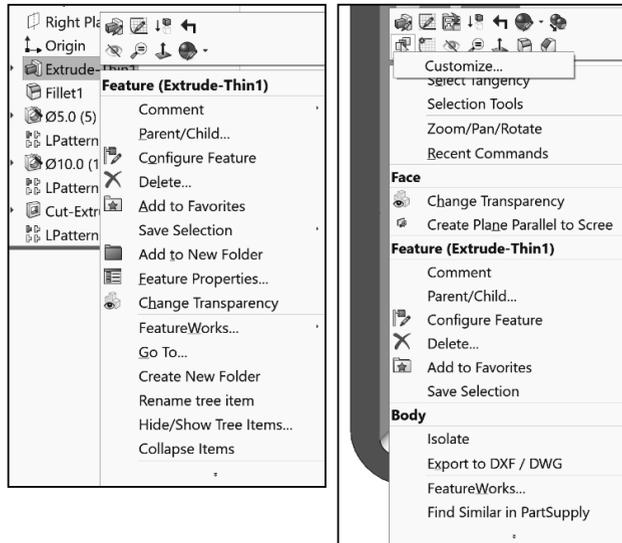
The book utilizes additional areas of the SOLIDWORKS User Interface. Explore an overview of these tools in the next section.



Right-click

Right-click in the Graphics window on a model, or in the FeatureManager on a feature or sketch to display the Context-sensitive toolbar. If you are in the middle of a command, this toolbar displays a list of options specifically related to that command.

Right-click an empty space in the Graphics window of a part or assembly, and a selection context toolbar above the shortcut menu is displayed. This provides easy access to the most commonly used selection tools.



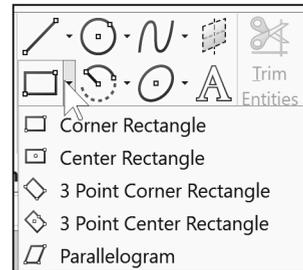
Consolidated toolbar

Similar commands are grouped together in the CommandManager. For example, variations of the Rectangle sketch tool are grouped in a single fly-out button as illustrated.

If you select the Consolidated toolbar button without expanding:

For some commands such as Sketch, the most commonly used command is performed. This command is the first listed and the command shown on the button.

For commands such as rectangle, where you may want to repeatedly create the same variant of the rectangle, the last used command is performed. This is the highlighted command when the Consolidated toolbar is expanded.

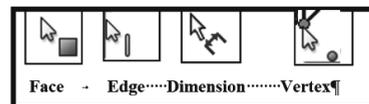


System feedback icon

SOLIDWORKS provides system feedback by attaching a symbol to the mouse pointer cursor.

The system feedback symbol indicates what you are selecting or what the system is expecting you to select.

As you move the mouse pointer across your model, system feedback is displayed in the form of a symbol, riding next to the cursor as illustrated. This is a valuable feature in SOLIDWORKS.



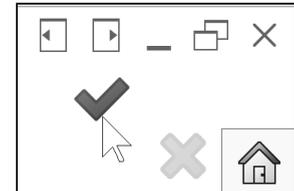
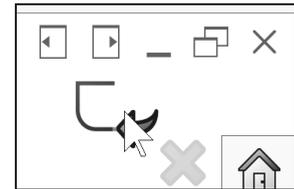
Confirmation Corner

When numerous SOLIDWORKS commands are active, a symbol or a set of symbols is displayed in the upper right-hand corner of the Graphics window. This area is called the Confirmation Corner.

When a sketch is active, the confirmation corner box displays two symbols. The first symbol is the sketch tool icon. The second symbol is a large red X. These two symbols supply a visual reminder that you are in an active sketch. Click the sketch symbol icon to exit the sketch and to save any changes that you made.

When other commands are active, the confirmation corner box provides a green check mark and a large red X. Use the green check mark to execute the current command. Use the large red X to cancel the command.

 Confirm changes you make in sketches and tools by using the D keyboard shortcut to move the OK and Cancel buttons to the pointer location in the Graphics window.

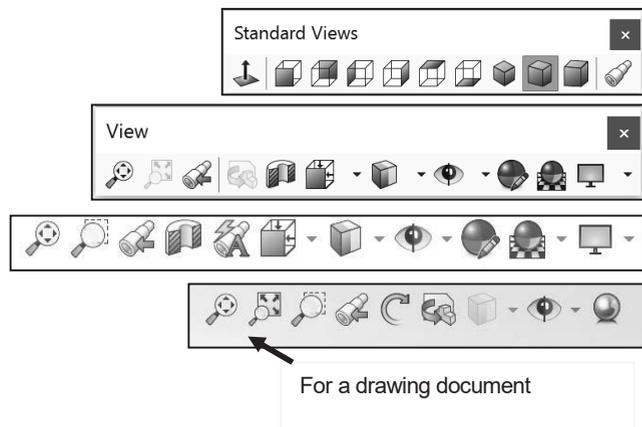


Heads-up View toolbars

SOLIDWORKS provides the user with numerous view options from the Standard Views, View and Heads-up View toolbars.

The Heads-up View toolbar is a transparent toolbar that is displayed in the Graphics window when a document is active.

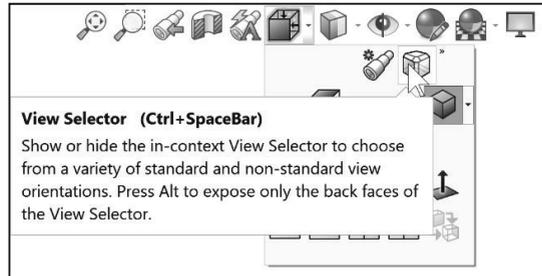
You can hide, move or modify the Heads-up View toolbar. To modify the Heads-up View toolbar, right-click on a tool and select or deselect the tools that you want to display.



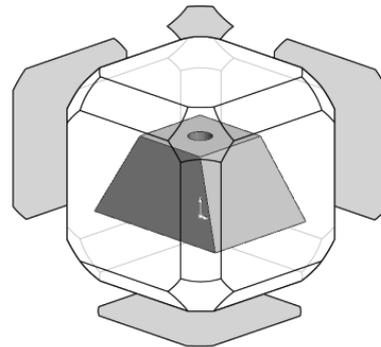
The following views are available. **Note:** available views are document dependent.

- *Zoom to Fit*  : Fit the model to the Graphics window.
- *Zoom to Area*  : Zoom to the areas you select with a bounding box.
- *Previous View*  : Display the previous view.

- **Section View** : Display a cutaway of a part or assembly, using one or more cross section planes.
- **Dynamic Annotation Views** : Only available with SOLIDWORKS MBD. Control how annotations are displayed when you rotate a model.



The Orientation dialog has an option to display a view cube (in-context View Selector) with a live model preview. This helps the user to understand how each standard view orientates the model. With the view cube, you can access additional standard views. The views are easy to understand and they can be accessed simply by selecting a face on the cube.



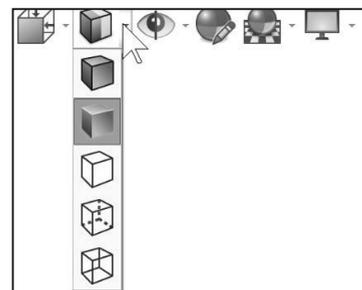
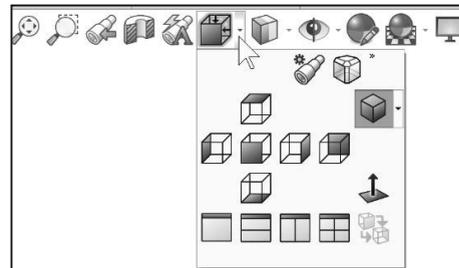
To activate the Orientation dialog box, press (Ctrl + spacebar) or click the View Orientation  icon from the Heads-up View toolbar. The active model is displayed in the View Selector in an Isometric orientation (default view).

Click the View Selector icon in the Orientation dialog box to show or hide the in-context View Selector.

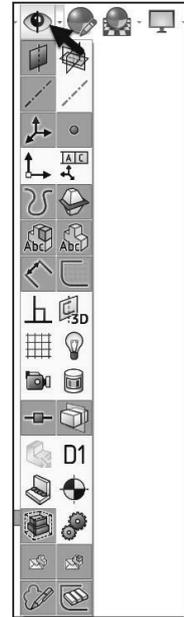
 Press **Ctrl + spacebar** to activate the View Selector.

 Press the **spacebar** to activate the Orientation dialog box.

- **View Orientation box** : Select a view orientation or the number of viewports. The options are: *Top, Left, Front, Right, Back, Bottom, Single view, Two view - Horizontal, Two view - Vertical, Four view*. Click the drop-down arrow to access Axonometric views: *Isometric, Dimetric and Trimetric*.
- **Display Style** : Display the style for the active view. The options are: *Wireframe, Hidden Lines Visible, Hidden Lines Removed, Shaded, Shaded With Edges*.

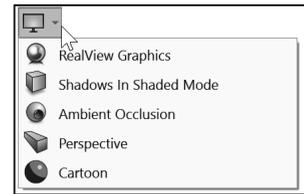


- *Hide/Show Items* : Select items to hide or show in the Graphics window. The available items are document dependent. Note the View Center of Mass  icon.
- *Edit Appearance* : Edit the appearance of entities of the model.
- *Apply Scene* : Apply a scene to an active part or assembly document. View the available options.
- *View Setting* : Select the following settings: *RealView Graphics*, *Shadows In Shaded Mode*, *Ambient Occlusion*, *Perspective* and *Cartoon*.
- *Rotate view* : Rotate a drawing view. Input Drawing view angle and select the ability to update and rotate center marks with view.
- *3D Drawing View* : Dynamically manipulate the drawing view in 3D to make a selection.

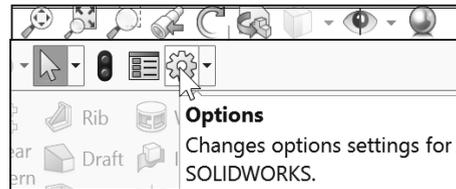


To display a grid for a part, click Options , Document Properties tab. Click Grid/Snaps, check the Display grid box.

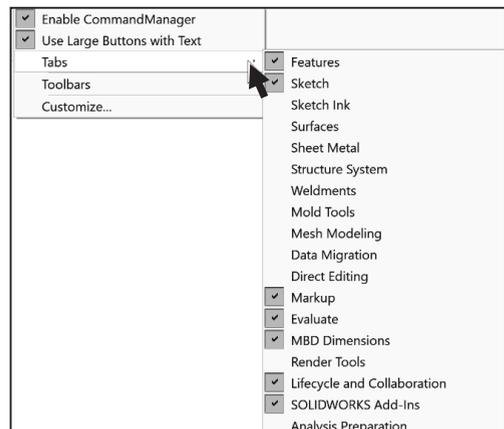
 Add a custom view to the Heads-up View toolbar. Press the space key. The Orientation dialog box is displayed. Click the New View  tool. The Name View dialog box is displayed. Enter a new named view. Click OK.



Use commands to display information about the triad or to change the position and orientation of the triad. Available commands depend on the triad's context.



 Save space in the CommandManager, limit your CommandManager tabs. Right-click on a CommandManager tab. Click Tabs. View your options to display CommandManager tabs.



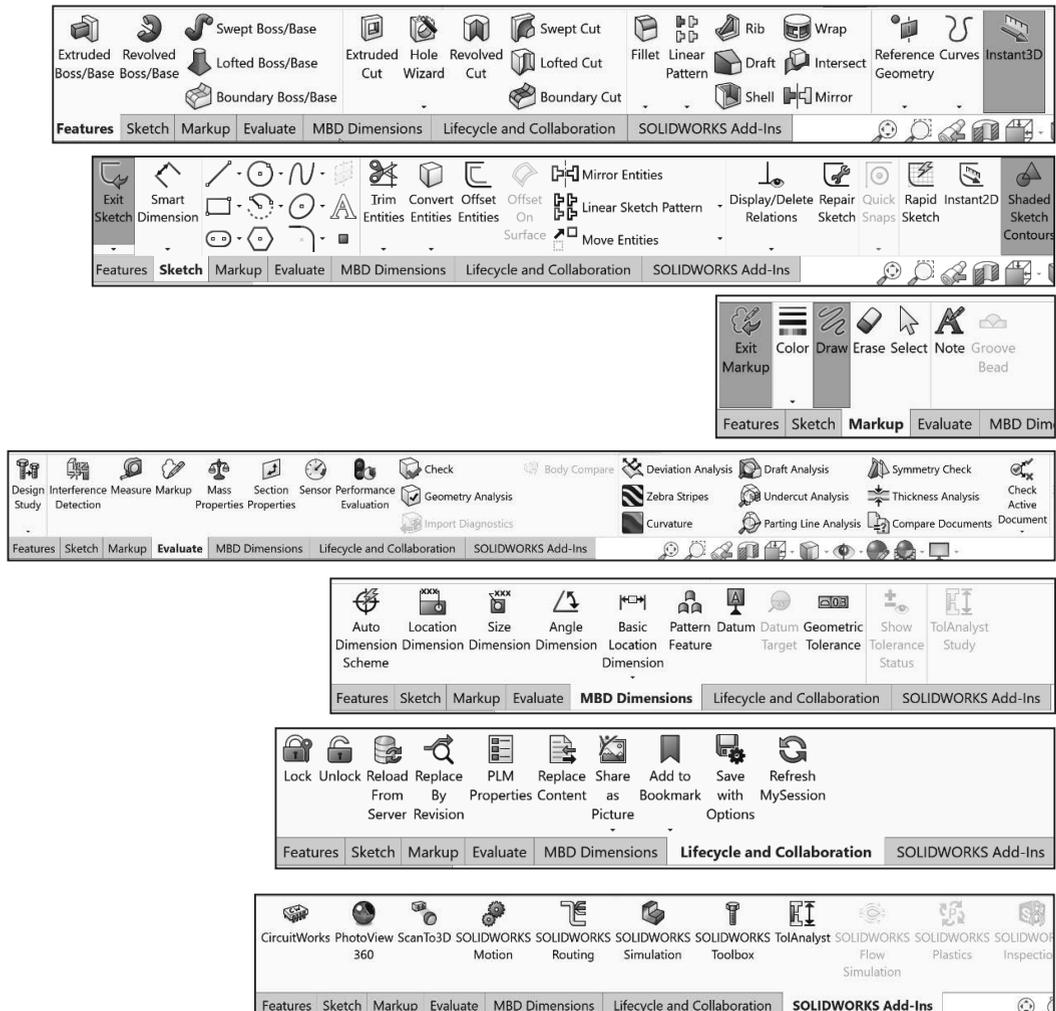
SOLIDWORKS CommandManager

The SOLIDWORKS CommandManager is a Context-sensitive toolbar. By default, it has toolbars embedded in it based on your active document type. When you click a tab below the CommandManager, it updates to display that toolbar. For example, if you click the Sketch tab, the Sketch toolbar is displayed.



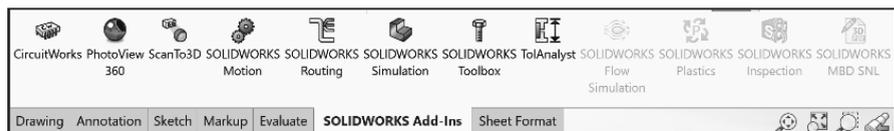
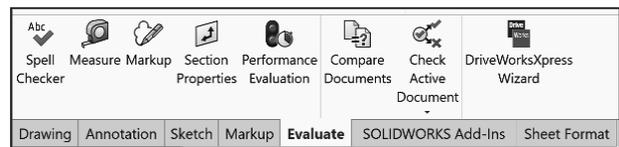
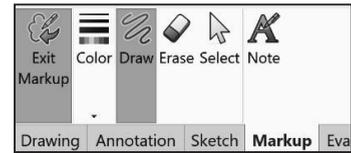
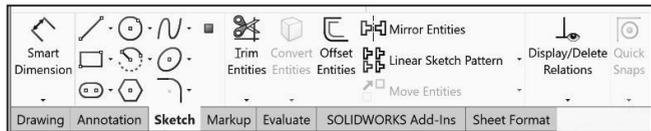
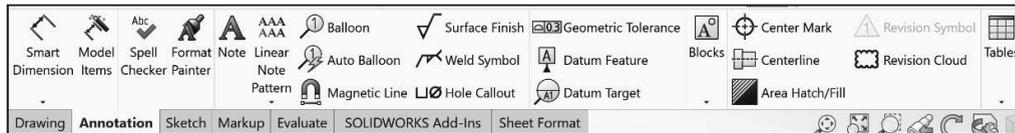
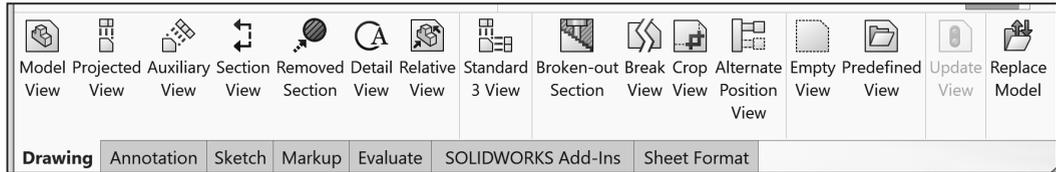
The Lifecycle and Collaboration tab is displayed with all active tools if you purchased 3DEXPERIENCE education roles and are logged in.

Below is an illustrated CommandManager for a **Part** document. CommandManager tabs will vary depending on system setup and Add-ins.

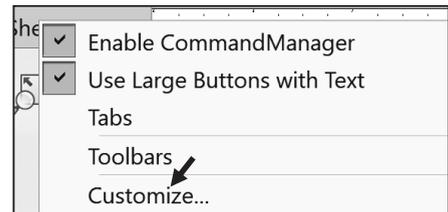


The SOLIDWORKS CommandManager is a Context-sensitive toolbar that automatically updates based on the toolbar you want to access. By default, it has toolbars embedded in it based on your active document type. The available tools are feature and document dependent.

Below is an illustrated CommandManager for a **Drawing** document. CommandManager tabs will vary depending on system setup and Add-ins.

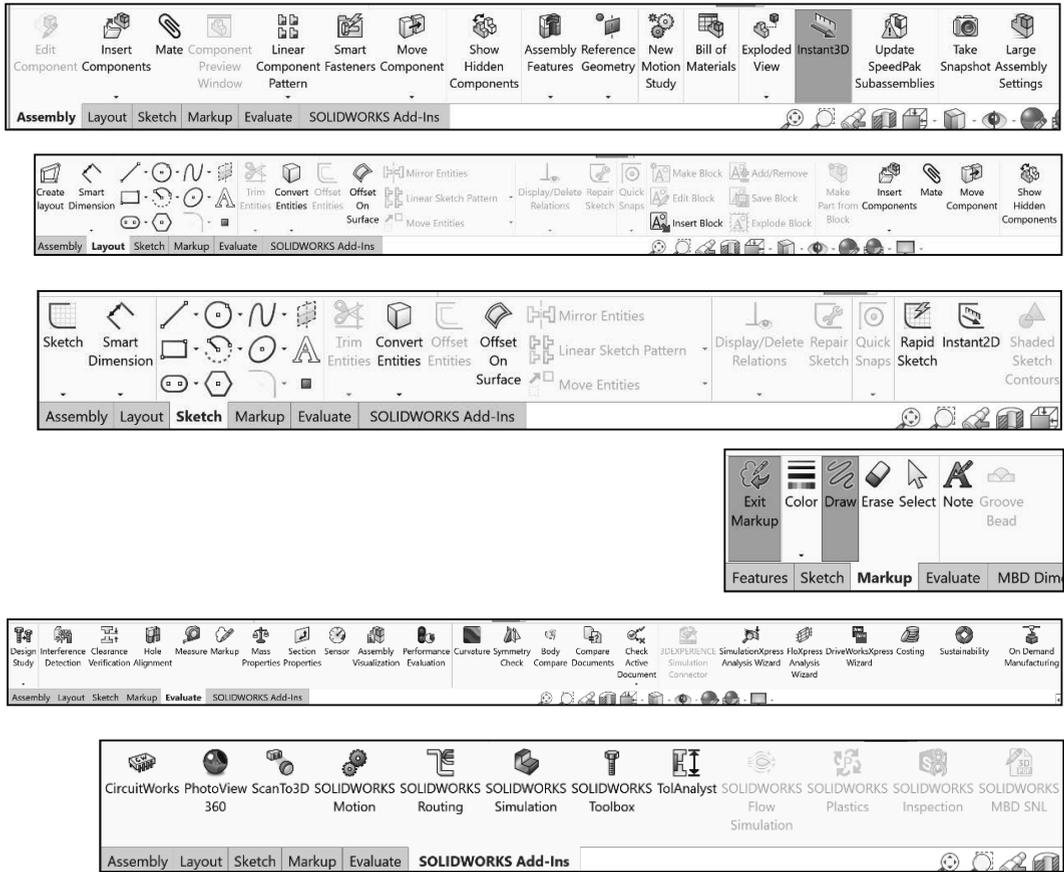


 To add a custom tab, right-click on a tab in the CommandManager and click Customize. You can also select to add a blank tab and populate it with custom tools from the Customize dialog box.



The SOLIDWORKS CommandManager is a context-sensitive toolbar that automatically updates based on the toolbar you want to access. By default, it has toolbars embedded in it based on your active document type. The available tools are feature and document dependent.

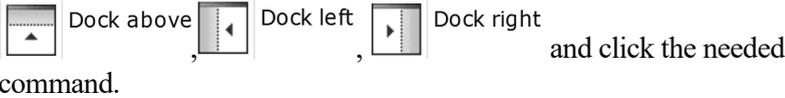
Below is an illustrated CommandManager for an **Assembly** document. CommandManager tabs will vary depending on system setup and Add-ins.



Float the CommandManager. Drag the Features, Sketch or any CommandManager tab. Drag the CommandManager anywhere on or outside the SOLIDWORKS window.

To dock the CommandManager, perform one of the following:

While dragging the CommandManager in the SOLIDWORKS window, move the pointer over a docking icon -



Double-click the floating CommandManager to revert the CommandManager to the last docking position.

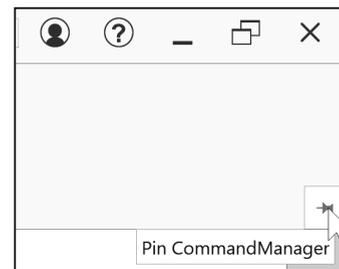
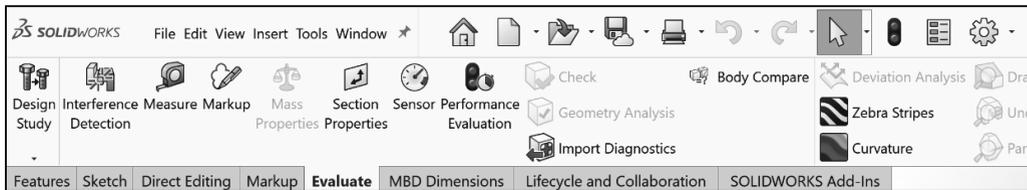
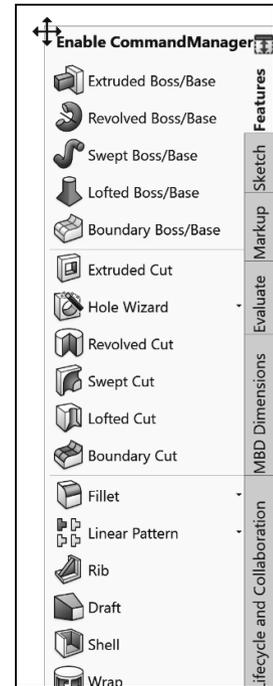
Collapse the CommandManager

Collapse the CommandManager to only display the available tabs. No tools are shown.

Click the Collapsed CommandManager arrow to the right of the active CommandManager tab as illustrated. Only the tabs are displayed.

To display the CommandManager tools back, click on a CommandManager tab. Click the Pin CommandManager as illustrated.

Note: If you have the 3DEXPERIENCE Platform, the SOLIDWORKS default Save icon will display additional commands.



Part FeatureManager Design Tree

The Part FeatureManager consists of various tabs:

- *FeatureManager design tree*  tab.
- *PropertyManager*  tab.
- *ConfigurationManager*  tab.
- *DimXpertManager*  tab.
- *DisplayManager*  tab.
- *CAM FeatureManager tree*  tab.
- *CAM Operation tree*  tab.
- *CAM Tools tree*  tab.



Click the  direction arrows to expand or collapse the FeatureManager design tree.

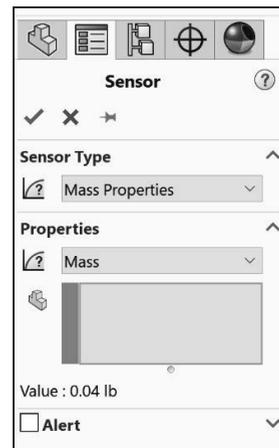
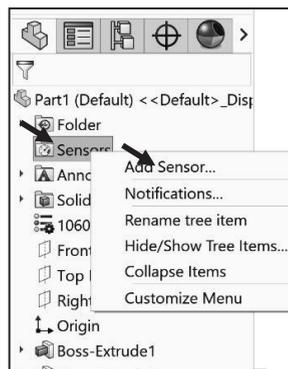
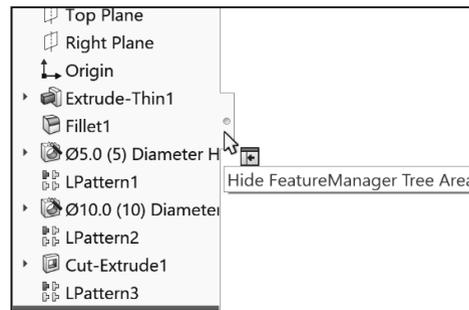
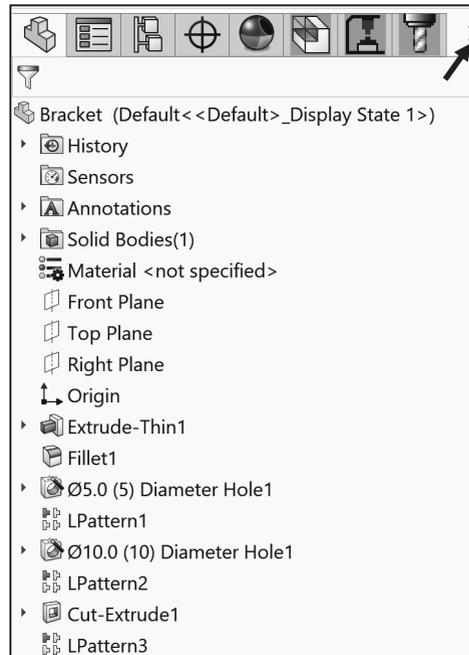
CommandManager and FeatureManager tabs and folder files will vary depending on your SOLIDWORKS applications and Add-ins.

Select the Hide/Show FeatureManager Area tab

 as illustrated to enlarge the Graphics window for modeling.



The Sensors tool  located in the FeatureManager monitors selected properties in a part or assembly and alerts you when values deviate from the specified limits. There are five sensor types: Simulation Data, Mass properties, Dimensions, Measurement and Costing Data.



Various commands provide the ability to control what is displayed in the FeatureManager design tree.

1. Show or Hide FeatureManager items.

 Click **Options**  from the Menu bar. Click **FeatureManager** from the System Options tab. Customize your FeatureManager from the Hide/Show tree Items dialog box.

2. Filter the FeatureManager design tree. Enter information in the filter field. You can filter by *Type of features, Feature names, Sketches, Folders, Mates, User-defined tags* and *Custom properties*.

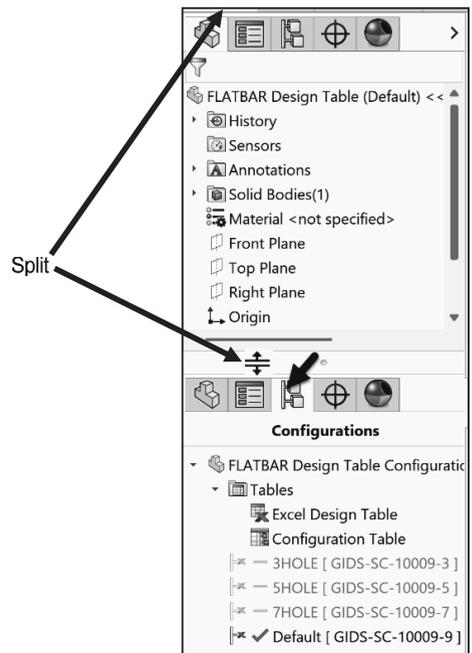
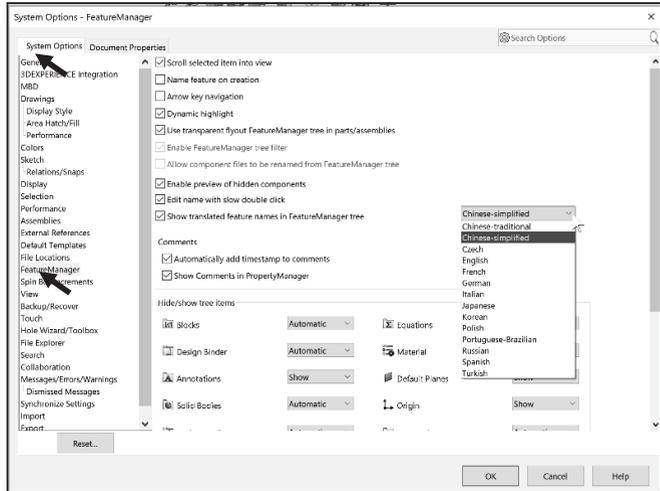
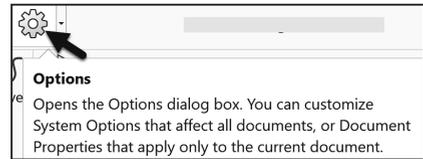
 Tags are keywords you can add to a SOLIDWORKS document to make them easier to filter and to search. The Tags  icon is located in the bottom right corner of the Graphics window.

 Collapse all items in the FeatureManager, **right-click** and select **Collapse items**, or press the **Shift + C** keys.

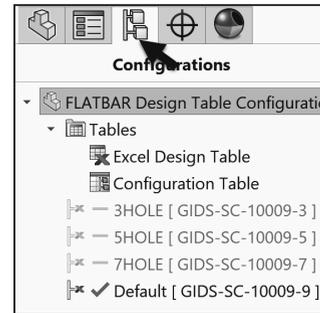
The FeatureManager design tree and the Graphics window are dynamically linked. Select sketches, features, drawing views, and construction geometry in either pane.

Split  the FeatureManager design tree and either display two FeatureManager instances, or combine the FeatureManager design tree with the ConfigurationManager or PropertyManager.

Move between the FeatureManager design tree, PropertyManager, ConfigurationManager, DimXpertManager, DisplayManager and others by selecting the tab at the top of the menu.



The ConfigurationManager  tab is located to the right of the PropertyManager  tab. Use the ConfigurationManager to create, select and view multiple configurations of parts and assemblies.



The icons in the ConfigurationManager denote whether the configuration was created manually or with a design table.

The DimXpertManager  tab provides the ability to insert dimensions and tolerances manually or automatically. The

options are: **Auto Dimension Scheme** , **Auto Pair Tolerance** , **Basic**, **Location Dimension** , **Basic Size Dimension** , **General Profile Tolerance** , **Show Tolerance Status** , **Copy Scheme** , **Import Scheme** , **TolAnalyst Study**  and **Datum Target** .

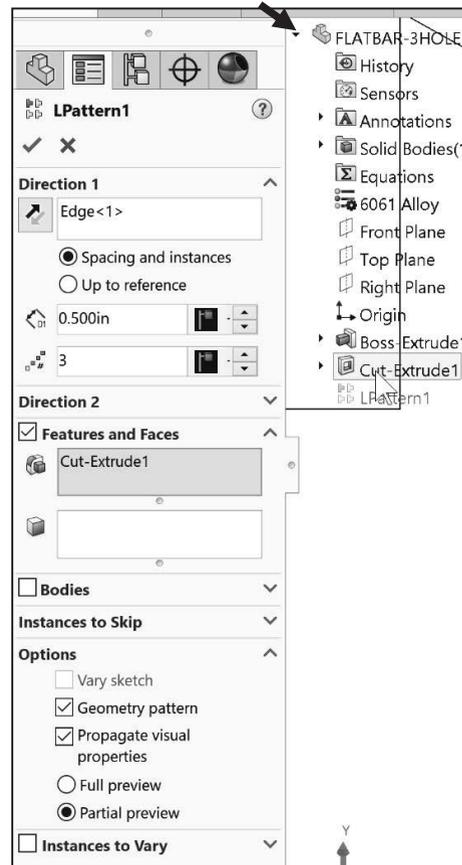
 TolAnalyst is available in SOLIDWORKS Premium.

Fly-out FeatureManager

The fly-out FeatureManager design tree provides the ability to view and select items in the PropertyManager and the FeatureManager design tree at the same time.

Throughout the book, you will select commands and command options from the drop-down menu, fly-out FeatureManager, Context toolbar, or from a SOLIDWORKS toolbar.

 Another method for accessing a command is to use the accelerator key. Accelerator keys are special keystrokes, which activate the drop-down menu options. Some commands in the menu bar and items in the drop-down menus have an underlined character.



Task Pane

The Task Pane is displayed when a SOLIDWORKS session starts. You can show, hide, and reorder tabs in the Task Pane. You can also set a tab as the default so it appears when you open the Task Pane, pin or unpin to the default location. **Note:** The Task Pane icon order and tab display may be different.

The Task Pane contains the following default tabs:

- *3DEXPERIENCE*  .
- *3DEXPERIENCE files on This PC*  .
- *Design Library*  .
- *File Explorer*  .
- *View Palette*  .
- *Appearances, Scenes and Decals*  .
- *Custom Properties*  .
- *SOLIDWORKS Resources*  .
- *SOLIDWORKS User Forum*  .

 Additional tabs are displayed with Add-Ins.



3DEXPERIENCE

The 3DEXPERIENCE  tab provides access to the 3DEXPERIENCE Platform. The 3DEXPERIENCE Platform provides downloadable content (DLC) from a secure cloud location within your SOLIDWORKS session.

The platform enables you to collaborate with students and teams on and off campus through a browser.

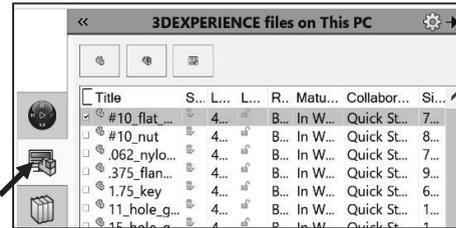
Think of the 3DEXPERIENCE Platform simply as a SOLIDWORKS Add-In on your desktop. Experience a single modeling environment. Access the platform within the SOLIDWORKS Task Pane.



Create and register your **3DEXPERIENCE** ID (credentials) to the platform. Go to <https://solidworks.com/academic-cloud> to create and register a new ID. **Note:** In an academic environment, it is strongly recommended to use your school’s assigned email address.

3DEXPERIENCE files on This PC

The 3DEXPERIENCE files on This PC  tab displays the models on your PC which were saved to the 3DEXPERIENCE platform. There are three filters: **Parts**, **Assemblies** and **Drawings**.



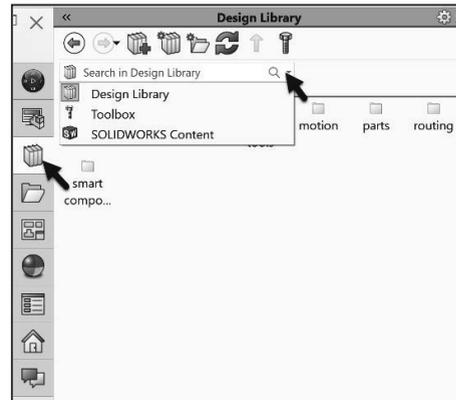
Design Library

The Design Library  contains reusable parts, assemblies, and other elements including library features.

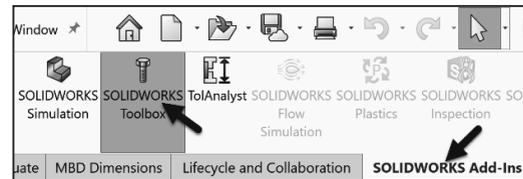
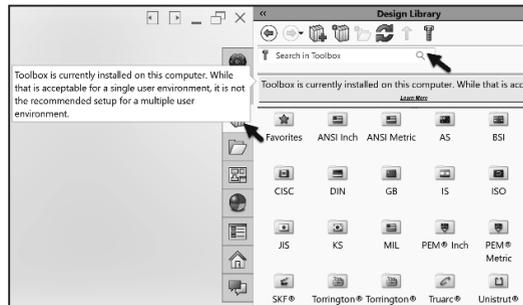
The Design Library tab contains default selections. Each default selection contains additional sub categories.

The default selections are:

- *Design Library.*
- *Toolbox (Add-In).*
- *SOLIDWORKS Content (Internet access required).*

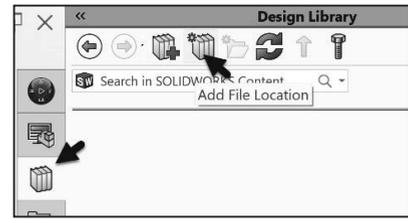


Activate the SOLIDWORKS Toolbox. Click SOLIDWORKS Toolbox from the SOLIDWORKS Add-Ins tab.



To access the Design Library folders in a non-network environment, click Add File Location  and browse to the needed path.

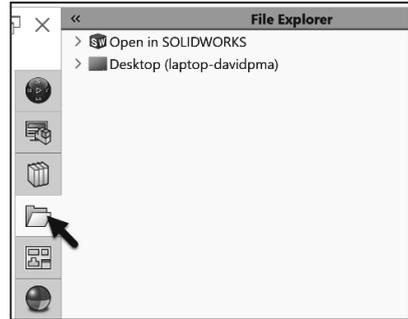
Paths may vary depending on your SOLIDWORKS version and window setup. In a network environment, contact your IT department for system details.



File Explorer

File Explorer  tab duplicates Windows Explorer from your local computer and displays:

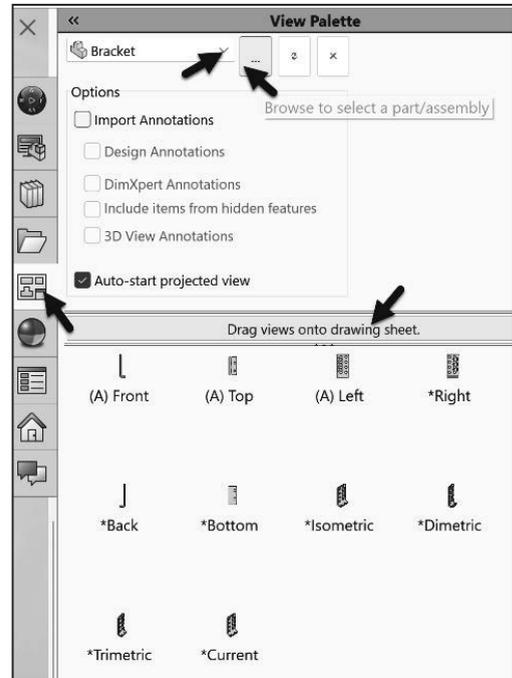
- *Open in SOLIDWORKS.*
- *Desktop.*



View Palette

The View Palette  tab provides the ability to insert drawing views of an active document, or click the Browse button to locate the desired document.

Click and drag the view from the View Palette into an active drawing sheet to create a drawing view.

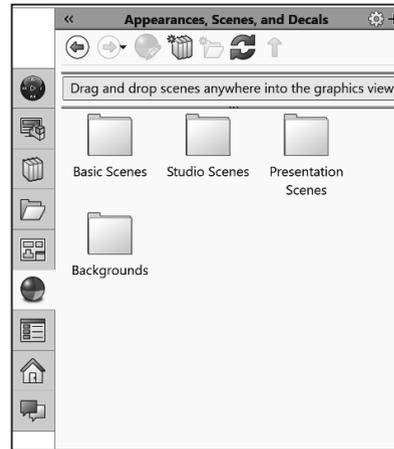


Appearances, Scenes, and Decals

Appearances, Scenes, and Decals  tab provides a simplified way to display models in a photo-realistic setting using a library of Appearances, Scenes, and Backgrounds.

An appearance defines the visual properties of a model, including color and texture. Appearances do not affect physical properties, which are defined by materials.

Scenes provide a visual backdrop behind a model. In SOLIDWORKS they provide reflections on the model. PhotoView 360 is an Add-in. Drag and drop a selected appearance, scene or decal on a feature, surface, part or assembly.



Custom Properties

The Custom Properties  tab provides the ability to enter custom and configuration specific properties directly into SOLIDWORKS files.

SOLIDWORKS Resources

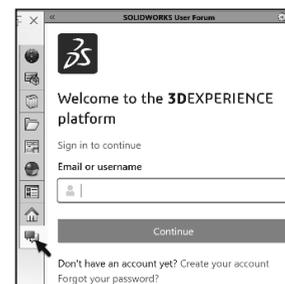
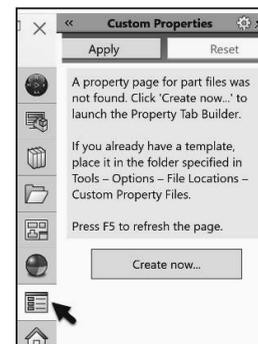
The SOLIDWORKS Resources  icon displays the following default selections:

- *Welcome to SOLIDWORKS.*
- *SOLIDWORKS Tools.*
- *Online Resources.*
- *Subscription Services.*

Other user interfaces are available during the initial software installation selection: *Machine Design, Mold Design, Consumer Products Design, etc.*

SOLIDWORKS User Forum

The SOLIDWORKS User Forum  icon displays information on general news and user information, user groups, and access to the 3DEXPERIENCE Platform Dashboard. **Note:** You need to be logged into the 3DEXPERIENCE Platform.



Dynamic Reference Visualization (Parent/Child)

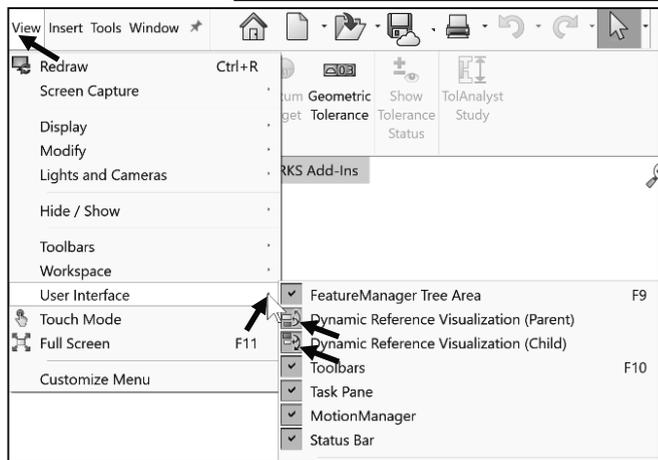
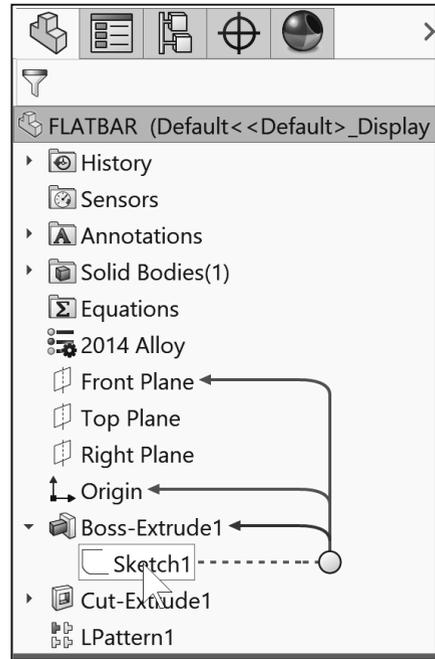
Dynamic Reference Visualization provides the ability to view the parent/child relationships between items in the FeatureManager design tree.

When you hover over a feature with references in the FeatureManager design tree, arrows display showing the relationships.

If a reference cannot be shown because a feature is not expanded, the arrow points to the feature that contains the reference and the actual reference appears in a text box to the right of the arrow.

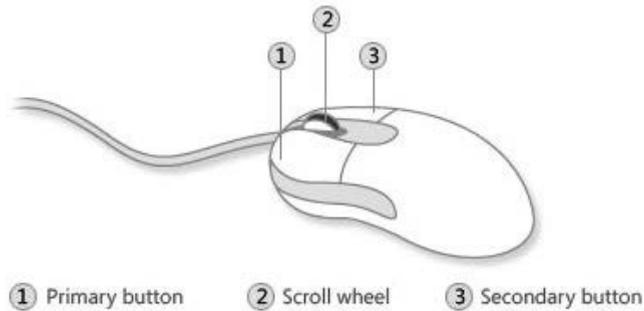
Use Dynamic reference visualization for a part, assembly and mates.

To display the Dynamic Reference Visualization, click **View, User Interface, Dynamic Reference Visualization Parent/Child** from the Main menu bar.



Mouse Movements

A mouse typically has two buttons: a primary button (usually the left button) and a secondary button (usually the right button). Most mice also include a scroll wheel between the buttons to help you scroll through documents and to Zoom in, Zoom out and rotate models in SOLIDWORKS. It is highly recommended that you use a mouse with at least a Primary, Scroll and Secondary button.



Single-click

To click an item, point to the item on the screen, and then press and release the primary button (usually the left button). Clicking is most often used to select (mark) an item or open a menu. This is sometimes called single-clicking or left-clicking.

Double-click

To double-click an item, point to the item on the screen, and then click twice quickly. If the two clicks are spaced too far apart, they might be interpreted as two individual clicks rather than one double-click. Double-clicking is most often used to open items on your desktop. For example, you can start a program or open a folder by double-clicking its icon on the desktop.

Right-click

To right-click an item, point to the item on the screen, and then press and release the secondary button (usually the right button). Right-clicking an item usually displays a list of things you can do with the item. Right-click in the open Graphics window or on a command in SOLIDWORKS, and additional pop-up context is displayed.

Scroll wheel

Use the scroll wheel to zoom-in or to zoom-out of the Graphics window in SOLIDWORKS. To zoom-in, roll the wheel backward (toward you). To zoom-out, roll the wheel forward (away from you).

Save a SOLIDWORKS Document as Previous Version

Save SOLIDWORKS parts, assemblies and drawings created or saved in the latest version of SOLIDWORKS as fully functional documents to a previous version of SOLIDWORKS.

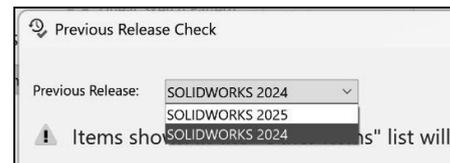
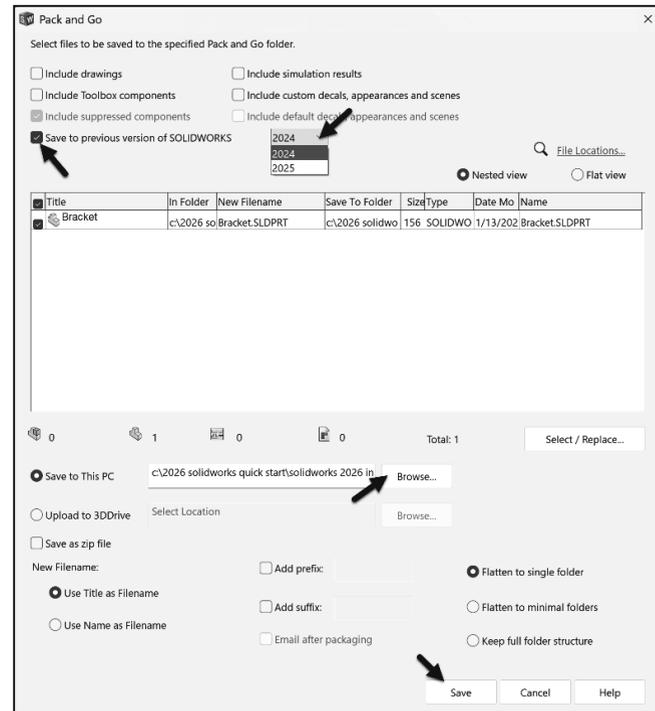
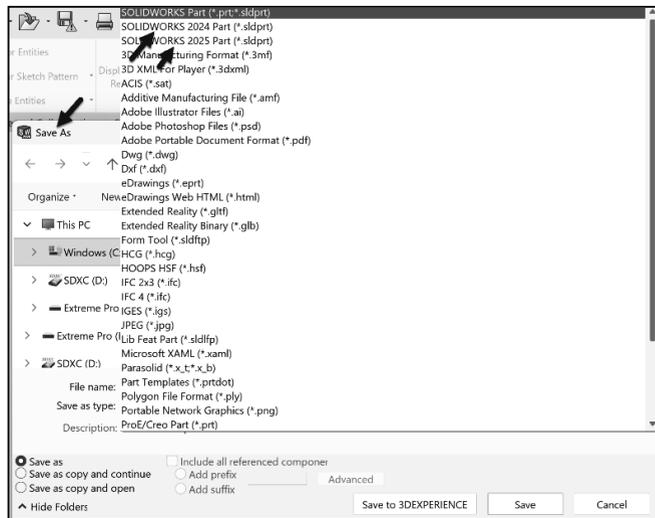
Save documents back to the previous two versions. **Note:** Pack and Go also supports this functionality.

You can save SOLIDWORKS 2026 files as SOLIDWORKS 2025 or SOLIDWORKS 2024 versions.

You are required to either modify the name or save to a different file folder.

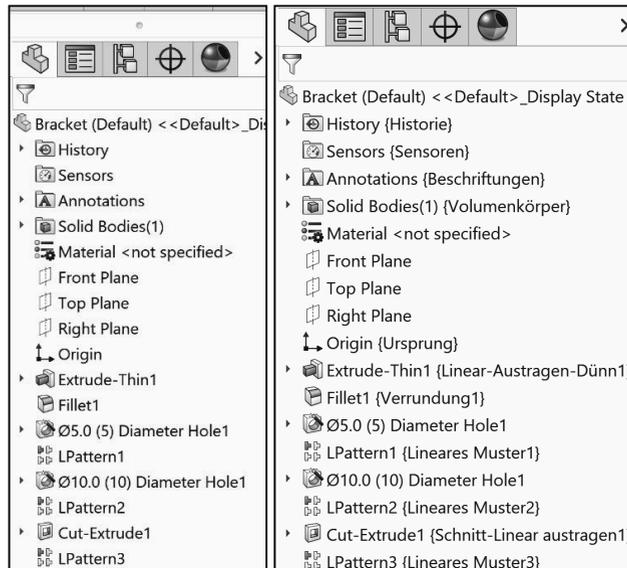
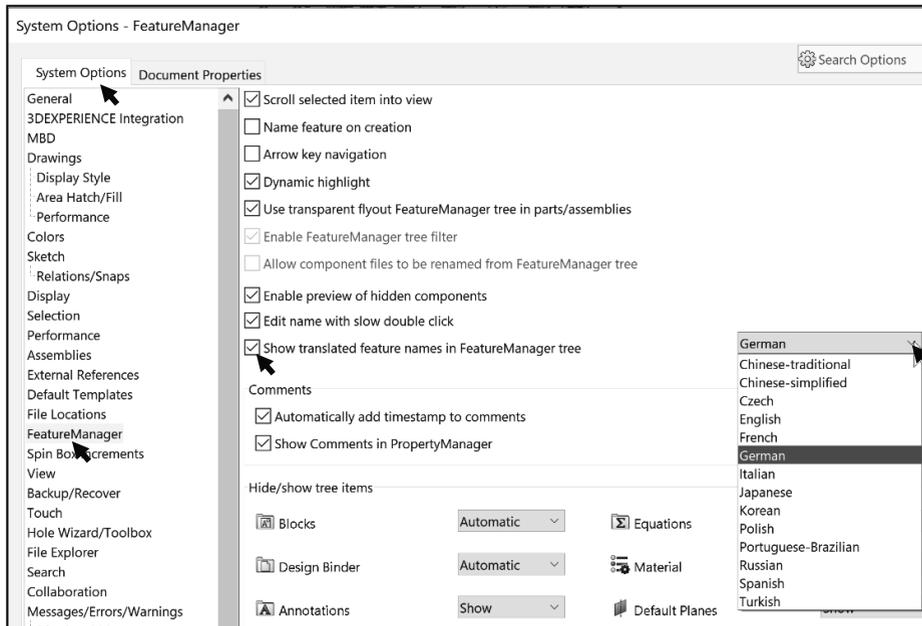
Note: SOLIDWORKS users must have an active subscription license to access this functionality. 3DEXPERIENCE users are active subscribers by default.

To check the Previous Release. Click Tools > Evaluate from the Main menu. Click Previous Release Check. View the results.



Translate Feature Names in the FeatureManager Tree

SOLIDWORKS provides the ability to display the FeatureManager in English and a second language. Check Options from the Main menu. Click the System Options tab. Click the FeatureManager folder. Check the Show translated feature names in the FeatureManager tree box. Click the drop-down menu to view the available languages. This is a great way to share your work in various languages.



Share and Send To option

The Share option provides the ability to share a part, assembly or drawing directly to a person or team for design review and comments.

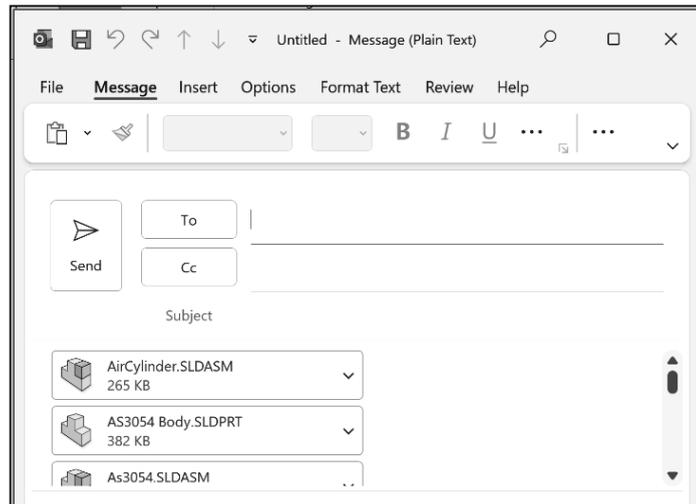
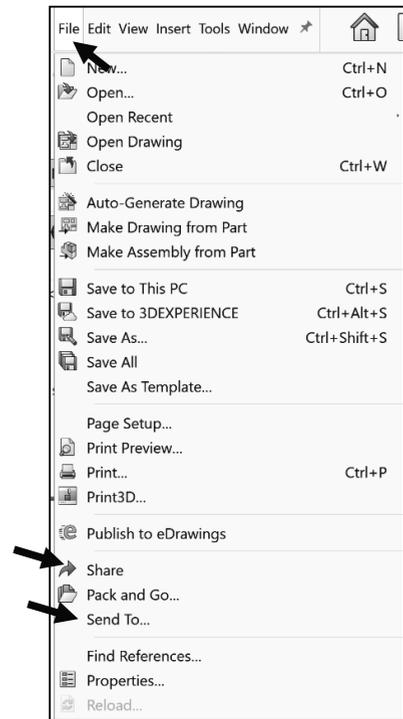
The Share option (Share files on 3DDrive) requires access to the **3DEXPERIENCE** Platform. The **3DEXPERIENCE** Platform provides downloadable content (DLC) from a secure cloud location within your SOLIDWORKS session. The platform enables you to collaborate with students and teams on and off campus through a browser.

Secure links, not files, can be created to share information with your team.

To use the Share option, you need to purchase a **3DEXPERIENCE** education offer from your VAR.

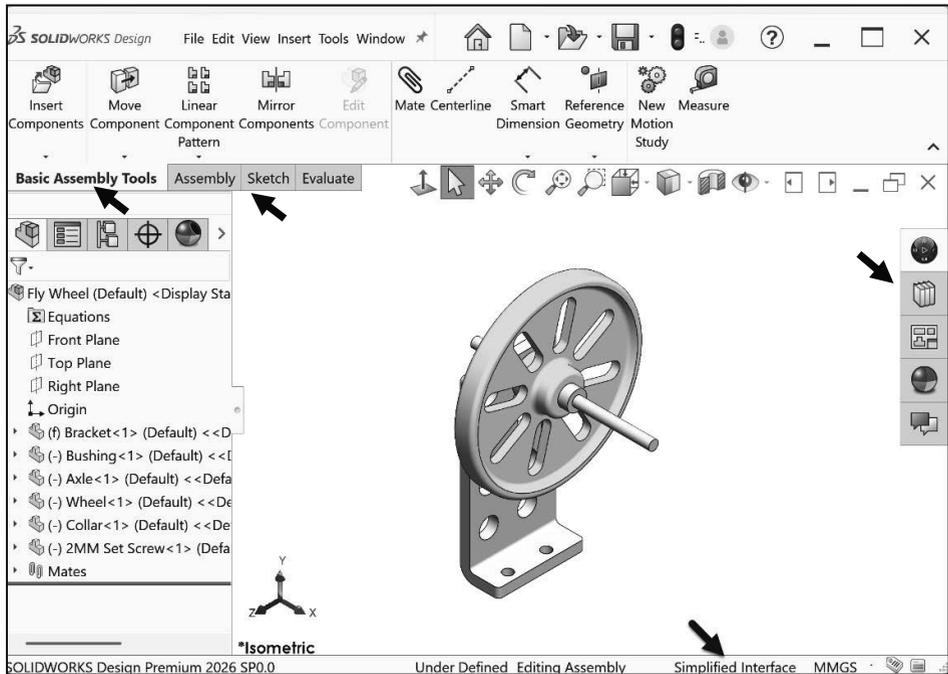
Create and register your **3DEXPERIENCE** ID (credentials) to the platform. An internet connection is required.

The Send To option provides the ability to email a part, assembly or drawing directly to a person or team.



Simplified User Interface

SOLIDWORKS provides the ability to specify workspace settings for display. Click **View**, **Workspace** from the main menu. View the options: **Simplified Interface**, **Default**, **Widescreen**, **Dual Monitor** and **Customize Menu**.



The Simplified Interface option displays an abridged user interface. The Graphics window includes basic user interface elements tailored to the type of document that you open.

The Default Interface arranges the SOLIDWORKS window with the CommandManager at the top and PropertyManager on the left.

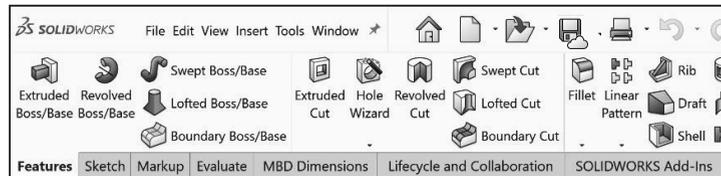
Note: If you have the 3DEXPERIENCE platform, the SOLIDWORKS default Save icon will display additional commands.

Summary

The SOLIDWORKS (UI) is designed to make maximum use of the Graphics window for your model. Displayed toolbars and commands are kept to a minimum.

The SOLIDWORKS User Interface and CommandManager consist of the following main options: Menu bar toolbar, Menu bar menu, Drop-down menus, Context toolbars, Consolidated fly-out menus, System feedback icons, Confirmation Corner and Heads-up View toolbar.

The Part CommandManager controls the display of tabs: *Features, Sketch, Markup, Evaluate, MBD Dimensions, Lifecycle and Collaboration* and various *SOLIDWORKS Add-Ins*.

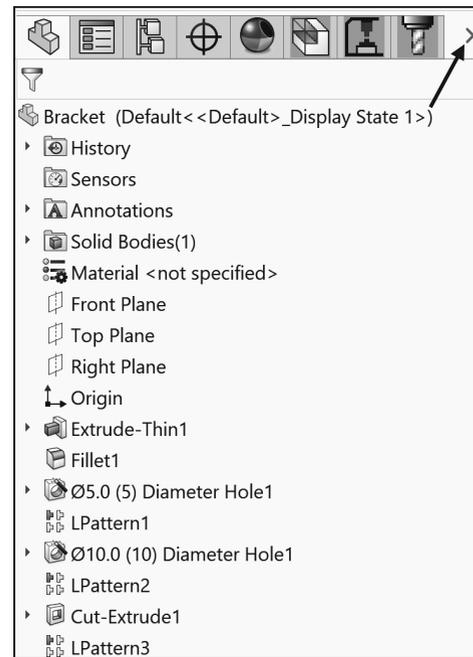


The FeatureManager consists of various tabs:

- *FeatureManager design tree*  tab.
- *PropertyManager*  tab.
- *ConfigurationManager*  tab.
- *DimXpertManager*  tab.
- *DisplayManager*  tab.
- *CAM FeatureManager tree*  tab.
- *CAM Operation tree*  tab.
- *CAM Tools tree*  tab.



Click the  direction arrows to expand or collapse the FeatureManager design tree.

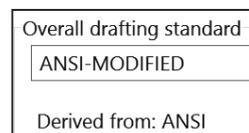


CommandManager, FeatureManager, and file folders will vary depending on system set-up, Add-ins and 3DEXPERIENCE Roles.

You learned about creating a new SOLIDWORKS part and opening an existing SOLIDWORKS part along with using the Rollback bar to view the sketches and features.



If you modify a document property from an Overall drafting standard, a modify message is displayed as illustrated.





Use the Search box, in the upper left corner of the Materials dialog box, to search through the entire materials library.

Templates are part, drawing and assembly documents which include user-defined parameters. Open a new part, drawing or assembly. Select a template for the new document.

In Chapter 2, establish a SOLIDWORKS session. Learn about 2D Sketching and 3D features.

Create a new part. Create the Wheel for the Fly Wheel sub-assembly.

Utilize the Fly Wheel sub-assembly in the final Stirling Engine assembly.

Apply the following sketch and feature tools: Circle, Line Centerline, Centerpoint Straight Slot, Mirror Entities, Extruded Boss, Extruded Cut, Revolved Boss, Circular Pattern, Hole Wizard and Fillet.

Incorporate design change into a part using proper design intent, along with applying multiple geometric relations: Coincident, Vertical, Horizontal, Tangent and Midpoint and feature and sketch modifications.

Utilize the Material, Mass Properties, Appearance and Measure tool on the Wheel.

