SOLIDWORKS® 2016

SOLIDWORKS Education Edition - Fundamentals of 3D Design and Simulation

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Document Number: PME-F3DDS100-ENG

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Introduction

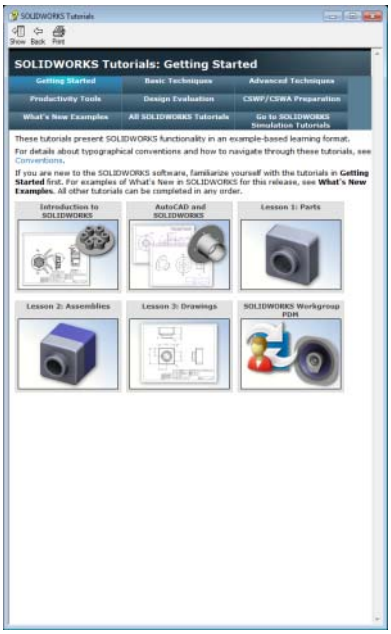
To the Teacher

The *SOLIDWORKS Education Edition - Fundamentals of 3D Design and Simulation* manual is designed to assist you in teaching SOLIDWORKS and SOLIDWORKS Simulation in an academic setting. This guide offers a competency-based approach to teaching 3D design concepts, analysis and techniques.

Qualified schools on subscription have access to the eBook at no cost to students. Contact your SOLIDWORKS Value Added Reseller to obtain access.

SOLIDWORKS Tutorials

The *SOLIDWORKS Education Edition - Fundamentals of 3D Design and Simulation* manual also supplements the SOLIDWORKS Tutorials.



Accessing the SOLIDWORKS Tutorials

To start the SOLIDWORKS Tutorials, click **Help, SOLIDWORKS Tutorials**. The SOLIDWORKS window is resized and a second window appears next to it with a list of the available tutorials. There are over 40 lessons in the SOLIDWORKS Tutorials. As you move the pointer over the links, an illustration of the tutorial will appear at the bottom of the window. Click the desired link to start that tutorial.

TIP: When you use SOLIDWORKS Simulation to perform analysis, click **Help**, **SOLIDWORKS Simulation**, **Tutorials** to access over 50 lessons and over 80 verification problems. Click **Tools**, **Add-ins** to activate SOLIDWORKS Simulation, SOLIDWORKS Motion, and SOLIDWORKS Flow Simulation.

Conventions

Set your screen resolution to 1280x1024 for optimal viewing of the tutorials.

The following icons appear in the tutorials:

Moves to the next screen in the tutorial.

Represents a note or tip. It is not a link; the information is below the icon. Notes and tips provide time-saving steps and helpful hints.

You can click most buttons that appear in the lessons to flash the corresponding SOLIDWORKS button.

Open File or **Set this option** automatically opens the file or sets the option.

A closer look at... links to more information about a topic. Although not required to complete the tutorial, it offers more detail on the subject.

Why did I... links to more information about a procedure, and the reasons for the method given. This information is not required to complete the tutorial.

Show me... demonstrates with a video.

Printing the SOLIDWORKS Tutorials

If you like, you can print the SOLIDWORKS Tutorials by following this procedure:

1. On the tutorial navigation toolbar, click **Show**.

   This displays the table of contents for the SOLIDWORKS Tutorials.

2. Right-click the book representing the lesson you wish to print and select **Print...** from the shortcut menu.

   The **Print Topics** dialog box appears.

3. Select **Print the selected heading and all subtopics**, and click **OK**.
4. Repeat this process for each lesson that you want to print.

My SOLIDWORKS

My.SolidWorks.com is a community website to share, connect, and learn everything about SOLIDWORKS. My SOLIDWORKS learning contains additional video lessons and individual learning paths for your students.

Certification Exams

The Certified SOLIDWORKS Associate(CSWA) - Academic program provides free certification exams for you or your students in a proctored setting. Achieving CSWA proves the fundamentals of engineering design competency. Employers verify students job ready credentials through our online virtual tester. Schools that provide two or more courses in SOLIDWORKS-based instruction can also apply to be a Certified SOLIDWORKS Professional(CSWP) - Academic Provider.

More information and to apply can be found at www.solidworks.com/cswa-academic.

Training Files

A complete set of the various files used throughout the course can be downloaded from the following website: www.solidworks.com/EDU_Fundamentals3DDesignSim

The files are organized by lesson number. The `Case Study` folder within each lesson contains the files you need when presenting the lessons. The `Exercises` folder contains any files that are required for doing the laboratory exercises.

Educator Resources link

The **Instructors Curriculum** link on the **SOLIDWORKS Resources** tab of the Task Pane includes substantial supporting materials to aid in your course presentation. Accessing this page requires a login account for the SOLIDWORKS Customer Portal. These supporting materials afford you flexibility in scope, depth, and presentation.

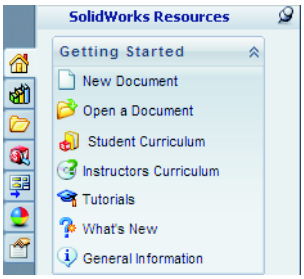
1. Start SOLIDWORKS.

   Using the **Start** menu, start the SOLIDWORKS application.

2. SOLIDWORKS Content.

   Click **SOLIDWORKS Resources** to open the SOLIDWORKS Resources Task Pane.

   Click on the **Instructors Curriculum** link which will take you to the SOLIDWORKS Customer Portal web page.



Prerequisites

Students attending this course are expected to have the following:

- Mechanical design experience.
- Experience with the Windows® operating system.
- Completed the online tutorials that are integrated in the SOLIDWORKS software. You can access the online tutorials by clicking **Help, Online Tutorial**.

Course Design Philosophy

This course is designed around a process- or task-based approach to training. A process-based training course emphasizes the processes and procedures you follow to complete a particular task. By utilizing case studies to illustrate these processes, you learn the necessary commands, options and menus in the context of completing a task.

A Note About Dimensions

The drawings and dimensions given in the lab exercises are not intended to reflect any particular drafting standard. In fact, sometimes dimensions are given in a fashion that would never be considered acceptable in industry. The reason for this is the labs are designed to encourage you to apply the information covered in class and to employ and reinforce certain techniques in modeling. As a result, the drawings and dimensions in the exercises are done in a way that complements this objective.

Conventions Used in this Book

This manual uses the following typographic conventions:

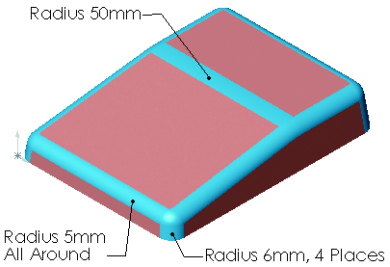
Convention	Meaning
Bold Sans Serif	SOLIDWORKS commands and options appear in this style. For example, **Features > Extruded Cut** means click the **Extruded Cut** icon on the **Features** tab of the CommandManager.
`Typewriter`	Feature names and file names appear in this style. For example, `Sketch1`.
17 Do this step	Double lines precede and follow sections of the procedures. This provides separation between the steps of the procedure and large blocks of explanatory text. The steps themselves are numbered in sans serif bold.

Windows 7

The screen shots in this manual were made using the SOLIDWORKS software running on Windows 7. If you are not using Windows 7, or if you have a different theme setting, you may notice slight differences in the appearance of the menus and windows. These differences do not affect the performance of the software.

Use of Color

The SOLIDWORKS user interface makes extensive use of color to highlight selected geometry and to provide you with visual feedback. This greatly increases the intuitiveness and ease of use of the SOLIDWORKS software. To take maximum advantage of this, the training manuals are printed in full color.



Also, in many cases, we have used additional color in the illustrations to communicate concepts, identify features, and otherwise convey important information. For example, we might show the result of a filleting operation with the fillets in a different color even though, by default, the SOLIDWORKS software would not display the results in that way.

Graphics and Graphics Cards

The SOLIDWORKS software sets a new standard with best-in-class graphics. The combination of a highly reflective material and the realism of **RealView Graphics** is an effective tool for evaluating the quality of advanced part models and surfaces.

RealView Graphics is hardware (graphics card) support of advanced shading in real time. For example, if you rotate a part, it retains its rendered appearance throughout the rotation.

Color Schemes

Out of the box, the SOLIDWORKS software provides several predefined color schemes that control, among other things, the colors used for highlighted items, selected items, sketch relation symbols, and shaded previews of features.

We have not used the same color scheme for every case study and exercise because some colors are more visible and clear than others when used with different colored parts.

In addition, we have changed the viewport background to plain white so that the illustrations reproduce better on white paper.

As a result, because the color settings on your computer may be different than the ones used by the authors of this book, the images you see on your screen may not exactly match those in the book.

User Interface Appearance

Throughout the development of the software, there have been some cosmetic User Interface changes, intended to improve visibility, that do not affect the function of the software. As a policy, dialog images in the manuals which exhibit no functional change from the previous version are not replaced. As such, you may see a mixture of current and “old” UI dialogs and color schemes.

Lesson 1
SOLIDWORKS Basics and the User Interface

Upon successful completion of this lesson, you will be able to:

- Describe the key characteristics of a feature-based, parametric solid modeler.
- Distinguish between sketched and applied features.
- Identify the principal components of the SOLIDWORKS user interface.
- Explain how different dimensioning methodologies convey different design intents.

What is the SOLIDWORKS Software?

SOLIDWORKS mechanical design automation software is a *feature-based*, *parametric solid modeling* design tool which takes advantage of the easy to learn Windows graphical user interface. You can create *fully associative* 3D solid models with or without *constraints* while utilizing automatic or user defined relations to capture *design intent*.

The italicized terms in the previous paragraph mean:

- **Feature-based**
  Just as an assembly is made up of a number of individual piece parts, a SOLIDWORKS model is also made up of individual constituent elements. These elements are called features.

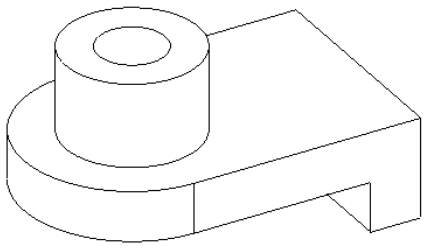
  When you create a model using the SOLIDWORKS software, you work with intelligent, easy to understand geometric features such as bosses, cuts, holes, ribs, fillets, chamfers, and drafts. As the features are created they are applied directly to the work piece.

  Features can be classified as either sketched or applied.

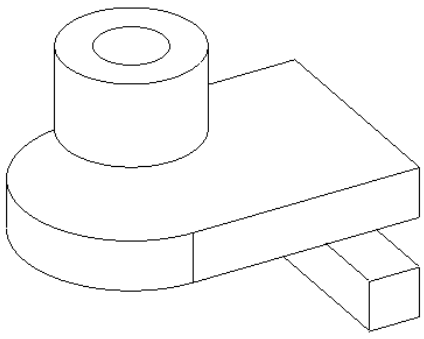
  - Sketched Features: Based upon a 2D sketch. Generally that sketch is transformed into a solid by extrusion, rotation, sweeping or lofting.
  - Applied Features: Created directly on the solid model. Fillets and chamfers are examples of this type of feature.

  The SOLIDWORKS software graphically shows you the feature-based structure of your model in a special window called the FeatureManager® design tree. The FeatureManager design tree not only shows you the sequence in which the features were created, it gives you easy access to all the underlying associated information. You will learn more about the FeatureManager design tree throughout this course.

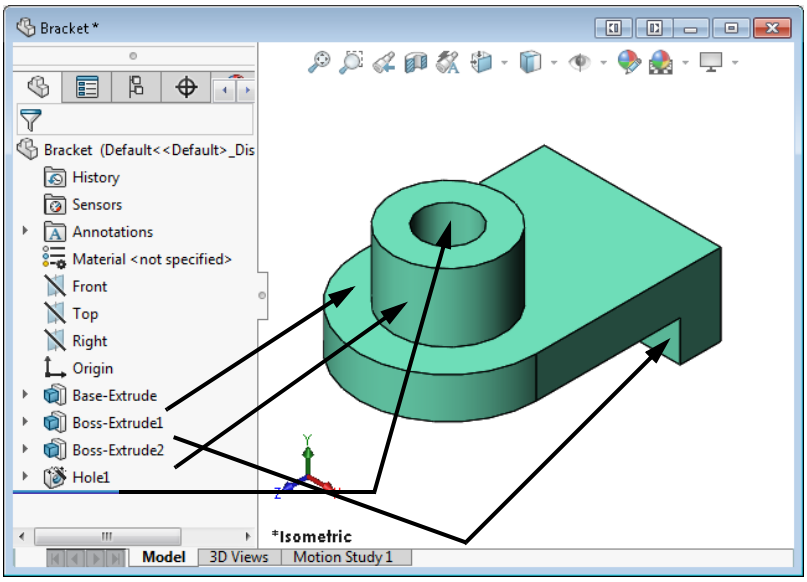
To illustrate the concept of feature-based modeling, consider the part shown at the right:



This part can be visualized as a collection of several different features – some of which add material, like the cylindrical boss, and some which remove material, like the blind hole.



If we were to map the individual features to their corresponding listing in the FeatureManager design tree, it would look like this:



- **Parametric**
  The dimensions and relations used to create a feature are captured and stored in the model. This not only enables you to capture your design intent, it also enables you to quickly and easily make changes to the model.
  - Driving Dimensions: These are the dimensions used when creating a feature. They include the dimensions associated with the sketch geometry, as well as those associated with the feature itself. A simple example of this would be a feature like a cylindrical boss. The diameter of the boss is controlled by the diameter of the sketched circle. The height of the boss is controlled by the depth to which that circle was extruded when the feature was made.
  - Relations: These include such information as parallelism, tangency, and concentricity. Historically, this type of information has been communicated on drawings via feature control symbols. By capturing this in the sketch, SOLIDWORKS enables you to fully capture your design intent up front, in the model.
- **Solid Modeling**
  A solid model is the most complete type of geometric model used in CAD systems. It contains all the wire frame and surface geometry necessary to fully describe the edges and faces of the model. In addition to the geometric information, it has the information called topology that relates the geometry together. An example of topology would be which faces (surfaces) meet at which edge (curve). This intelligence makes operations such a filleting as easy as selecting an edge and specifying a radius.

- **Fully Associative**
  A SOLIDWORKS model is fully associative to the drawings and assemblies that reference it. Changes to the model are automatically reflected in the associated drawings and assemblies. Likewise, you can make changes in the context of the drawing or assembly and know that those changes will be reflected back in the model.
- **Constraints**
  Geometric relationships such as parallel, perpendicular, horizontal, vertical, concentric, and coincident are just some of the constraints supported in SOLIDWORKS. In addition, equations can be used to establish mathematical relationships among parameters. By using constraints and equations, you can guarantee that design concepts such as through holes or equal radii are captured and maintained.
- **Design Intent**
  The final italicized term is design intent. This subject is worthy of its own section, as follows.

Design Intent

In order to use a parametric modeler like SOLIDWORKS efficiently, you must consider the design intent before modeling. Design intent is your plan as to how the model should behave when it is changed. The way in which the model is created governs how it will be changed. Several factors contribute to how you capture design intent:

- **Automatic (sketch) Relations**
  Based on how geometry is sketched, these relations can provide common geometric relationships between objects such as parallel, perpendicular, horizontal, and vertical.
- **Equations**
  Used to relate dimensions algebraically, they provide an external way to force changes.
- **Added Relations**
  Added to the model as it is created, relations provide another way to connect related geometry. Some common relations are concentric, tangent, coincident, and collinear.
- **Dimensioning**
  Consider your design intent when applying dimensions to a sketch. What are the dimensions that should drive the design? What values are known? Which are important for the production of the model? The way dimensions are applied to the model will determine how the geometry will change if modifications are made.

  Consider the design intent in the following examples.