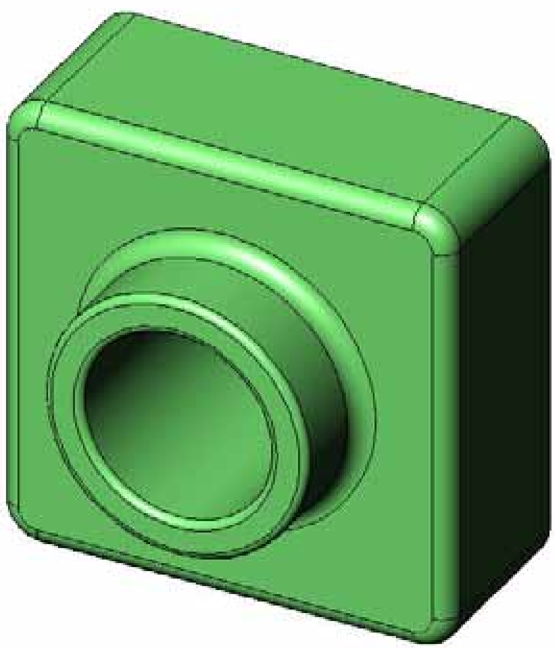
Engineering Design and Technology Series

Instructor’s Guide to Teaching SolidWorks® Software



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Document Number: PME0119-ENG

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Introduction

To the Teacher

Instructor's Guide to Teaching SolidWorks® Software and its supporting materials are designed to assist you in teaching SolidWorks in an academic setting. This guide offers a competency-based approach to teaching 3D design concepts and techniques.

Each lesson in *Instructor's Guide to Teaching SolidWorks Software* has corresponding pages in the *Student's Guide to Learning SolidWorks Software* (available as PDFs from the **Design Library** tab on the Task Pane. Expand **SolidWorks Content**, **SolidWorks Educator Curriculum**, **Curriculum**, **SolidWorks Student Guide**). *Instructor's Guide to Teaching SolidWorks Software* is annotated with discussion points, suggestions for class demonstrations, and explanatory information related to the exercises and projects. Also in this guide are answer keys for assessments, worksheets, and quizzes.

SolidWorks Tutorials

Instructor's Guide to Teaching SolidWorks Software is a companion resource and supplement for the SolidWorks Tutorials. Many of the exercises in *Student's Guide to Learning SolidWorks Software* use material from 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 static engineering analysis, click **Help**, **SolidWorks Simulation**, **Tutorials** to access over 20 lessons and over 35 verification problems. Click **Tools**, **Add-ins** to activate SolidWorks 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 toolbar 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.

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. You can use this course as is or you can select the pieces of it that meet your class needs. These supporting materials afford you flexibility in scope, depth, and presentation.

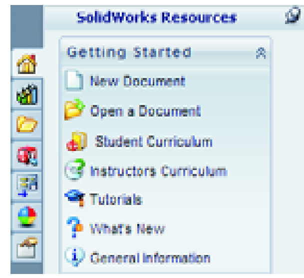
Before You Begin

If you have not done so already, copy the companion files for the lessons onto your computer before you begin this project.

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.

Click **Educator Resources**, under **Download**. Accessing this page requires a login account for the SolidWorks Customer Portal.

Here you will find the zip file containing the teacher companion files: **Teacher SolidWorks files**.

3 Download the zip file.

4 Open the zip file.

Browse to the folder where you saved the zip file in step **3** and double-click the zip file.

5 Click **Extract**.

Browse to the location where you want to save the files. The system automatically creates folders for the sample files in whatever location you specify. For example, you might want to save it in `My Documents`.

TIP: Remember the location of these files.

Using This Course

This course is not just this book. *Instructor's Guide to Teaching SolidWorks Software* is the focal point of the SolidWorks course — the road map for it. The supporting materials that are on the Educator Resources link and the SolidWorks Tutorials give you a lot of flexibility in how you present the course.

Learning 3D design is an interactive process. Students learn best when they can explore the practical applications of the concepts that they learn. This course has many activities and exercises that enable students to put design concepts into practice. Using the provided files, they can do so quickly.

The lesson plans for this course are designed to balance lecture and hands-on learning. There are also assessments and quizzes that give you additional measures of student progress.

Before Presenting the Lectures

- ❑ Verify that the SolidWorks software is loaded and running on your classroom/lab computers in accordance with your SolidWorks license.
- ❑ Download and unzip the files from the Educator Resources link.
- ❑ Print copies of *Student's Guide to Learning SolidWorks Software* for each student.
- ❑ Work through each of the labs yourself. This is not only to verify that you understand how they work, but to explore. Often there are different ways to accomplish a task.

Lesson Plans

Each lesson plan contains the following components:

- ❑ Goals of the Lesson — Clear objectives for the lesson.
- ❑ Before Beginning the Lesson — Prerequisites, if any, for the current lesson.
- ❑ Resources for This Lesson — Tutorials that correspond to the lesson.
- ❑ Review of Previous Lesson — Students reflect back on the material and models described in the previous lesson with questions and examples. Ask these questions of your students to reinforce concepts.
- ❑ Lesson Outline — Describes the major concepts explored in each lesson.
- ❑ Competencies — Lists the competencies that students develop as they learn the material presented in the lesson.
- ❑ In Class Discussion — Topics for discussion to explain some concepts in the lesson.
- ❑ Active Learning Exercises — Students create models. Some of these exercises are from *Student's Guide to Learning SolidWorks Software.* Most are from the SolidWorks Tutorials.
- ❑ 5-minute Assessments — These review the concepts developed in the outline of the lesson and the active learning exercises. Questions are presented in the *Student Workbook* and they may be answered in class or for homework. You can use the 5-minute assessment questions as verbal or written exercises. Space is provided in the *Student Workbook* for answers. These are check points for students before they move on to the additional exercises and projects.
- ❑ Additional Exercises and Projects — Additional exercises and projects are at the end of each lesson. These exercises and projects were developed from suggestions made by students and teachers.

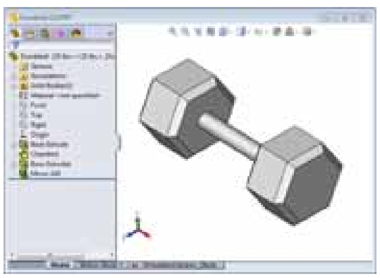
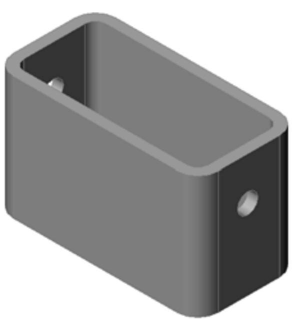
Note: Mathematics is also explored through a series of applied problems. For example: students design a coffee mug and determine how much liquid it holds. Does the answer make sense?

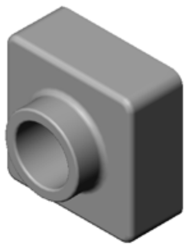
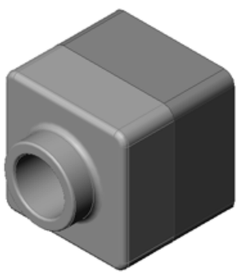
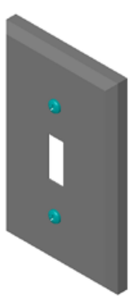
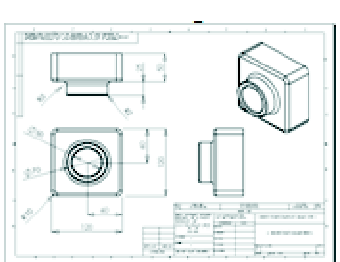
- ❑ More to Explore — Since students learn at different rates, some lessons also have advanced or related exercises that you can assign to all students or just students who have finished the other material of the lesson ahead of the class.

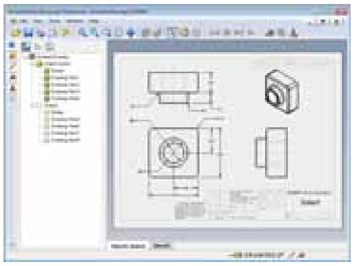
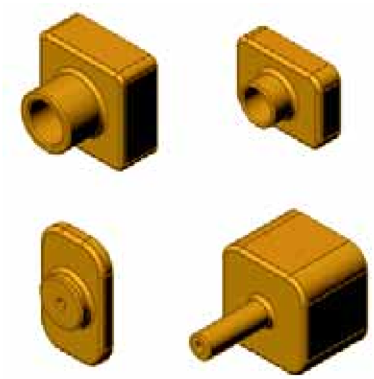
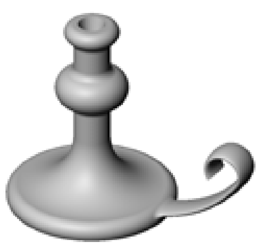
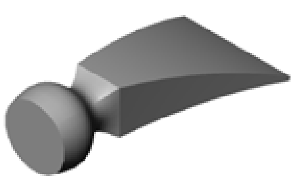
- Lesson Quizzes — Fill in the blank, true/false and short answer questions compose the lesson quizzes. The lesson quiz master and answer key are only available in the *Instructor's Guide to Teaching SolidWorks Software*.
- Lesson Summary — Quick recap of the main points of the lesson.
- Microsoft® PowerPoint® Slides — There are prepared Microsoft PowerPoint slides to explain each lesson. These slides are provided to you electronically on the Educator Resources link. These reproducible pages can also be used to create handouts.

Syllabus

Here is an overview of the material covered in each lesson:

Lesson	**Outcome for Students**	**Assessments**
Lesson 1: Using the Interface 	• Become familiar with Microsoft Windows • Become familiar with the SolidWorks user interface	• 5 minute assessment • Vocabulary worksheet • Lesson Quiz
Lesson 2: Basic Functionality 	• Develop an understanding of 3D modeling and recognition of an object in 3D space • Apply 2D sketch geometry, rectangle, circle, and dimensions • Understand 3D features that add and remove geometry including Extruded Base, Extruded Cut, Fillet and Shell • Create the Box part	• 5 minute assessment • Vocabulary worksheet • Lesson Quiz • Additional Exercises: Design a Switch Plate • Optional materials for Switch Plate: Cardboard, construction paper or foam board 120mmx80mm for each student, tape or glue, cutting tools, ruler • Optional materials for Box: For milled wood 100mmx60mmx50mm for each box. (Note: Cardboard sheets and tape can also be used)

Lesson	Outcome for Students	Assessments
Lesson 3: The 40-Minute Running Start 	• Reinforce the understanding of 3D features that add and remove geometry • Apply 2D sketch geometry, rectangle, circle, and dimensions • Create the Tutor1 part	• 5 minute assessment • Unit conversion worksheet • Material volume assessment • Lesson Quiz • Additional Exercises: Modifying the Tutor1 part • Additional Exercises: CD Jewel Case and Storage Box parts • Optional materials: cardboard or foam board, tape, wood (mill or precut pieces required) 29mmx17mmx18mm for each storage box
Lesson 4: Assembly Basics 	• Develop an understanding of 3D assembly modeling by combining Tutor1 part with Tutor2 part • Apply 2D sketch tools to offset geometry and project geometry to the sketch plane • Create Tutor2 part and Tutor assembly	• 5 minute assessment • Vocabulary worksheet • Lesson Quiz • Review of fasteners selection • Additional Exercises: Design a Switchplate assembly, Storage Box assembly, and Claw Mechanism assembly • Optional materials: screws for switchplate part, roughly 3.5mm diameter • A variety of fasteners to discuss design and manufacturing parameters for a product
Lesson 5: SolidWorks Toolbox Basics 	• Develop an understanding of SolidWorks Toolbox, a component library of standard parts • Understand how library components are utilized in an assembly • Modify SolidWorks Toolbox part definitions and create new parts for the Toolbox library	• 5 minute assessment • Vocabulary worksheet • Lesson Quiz • Assemble a standard Toolbox pan head screw to the switchplate • Additional Exercises: Add fasteners to the bearing block assembly • Optional materials: Variety of fasteners. For Switch Plate, #6-32 Pan Head
Lesson 6: Drawing Basics 	• Understand basic drawing concepts • Apply drawing standards to part and assembly drawings • Create a drawing template • Create Tutor1 drawing for part and assembly	• 5 minute assessment • Lesson Quiz • Additional Exercises: Create a drawing for Tutor2, the storage box, and the switchplate

Lesson	Outcome for Students	Assessments
Lesson 7: SolidWorks eDrawings Basics 	• Create eDrawings from existing SolidWorks files • View and manipulate eDrawings • Measure and markup eDrawings • Create animations of eDrawings to display multiple views	• 5 minute assessment • Vocabulary worksheet • Lesson Quiz • Additional Exercises: Create, explore and email eDrawings files
Lesson 8: Design Tables 	• Understand configurations • Develop a Design Table with Microsoft Excel to create families of parts • Explore how values in an Excel spreadsheet automatically change dimensions and features of an existing part to create multiple parts of different sizes	• 5 minute assessment • Lesson Quiz • Additional Exercises: Create a design table for Tutor2, the Tutor assembly, the storage box, and a cup • Optional materials: cups, beakers in different size and a ruler
Lesson 9: Revolve and Sweep Features 	• Understand 3D features that add and remove geometry including Revolve and Sweep • Apply 2D sketch tools such as ellipse, trim and centerline • Create the Candlestick part	• 5 minute assessment • Lesson Quiz • Additional Exercises: Create a candle and modify the switchplate • Optional materials: cup, beaker, candle and a ruler
Lesson 10: Loft Features 	• Understand the 3D Loft feature created from multiple profiles sketched on different planes • Create the Chisel part	• 5 minute assessment • Lesson Quiz • Additional Exercises: Create a bottle, a screwdriver, and a sports bottle • Optional materials: screwdriver and simple bottle

Lesson	**Outcome for Students**	**Assessments**
Lesson 11: Visualization 	• Understand how to apply materials, scenes, and lights to create a photorealistic images in JPEG format • Create an exploded view and develop an animation in AVI format	• 5 minute assessment • Lesson Quiz • Additional Exercises: Create a rendering of Tutor1, Tutor2 and Tutor assembly, create an exploded view, and create an animation of the nested slides assembly • Optional materials: digital photographs and images
Lesson 12: SolidWorks SimulationXpress 	• Understand basic concepts of stress analysis • Analyze parts to calculate factor of safety and maximum stress and displacement	• 5 minute assessment • Lesson Quiz • Additional Exercises: Analyze the storagebox and modify the storagebox to observe the effects on the maximum displacement

Supporting Course Materials

The following supporting course materials are provided to you via the Educators Resources link of the SolidWorks Customer Portal. Click the **Instructors Curriculum** link on the **SolidWorks Resources** tab of the Task Pane to access:

- ❑ *Student workbook* - An electronic version of the *Student's Guide to Learning SolidWorks Software*. It contains exercises, tutorials, projects, and worksheets. You can reproduce this book for use with your students.
- ❑ *Student SolidWorks files* - Parts, assemblies, and drawings that correspond to the activities and exercises in the *Student's Guide to Learning SolidWorks Software*.
- ❑ *Teacher SolidWorks files* - Parts, assemblies, and drawings that correspond to the activities and exercises in this guide.
- ❑ *Instructor guide* - A zip file that includes:
  - An electronic version of this guide.
  - An electronic version of *Student's Guide to Learning SolidWorks Software*.
  - Microsoft PowerPoint slides - These slides compliment the *Instructor's Guide to Teaching SolidWorks Software*. You can project these slides directly on a screen, reproduce these as student handouts, and modify them to suit your needs. These slides are available as .PPT and .PDF files.

Certified SolidWorks Associate (CSWA) Certification Program

The lessons, exercises, and projects in this course provide much of the background required for the Certified SolidWorks Associate (CSWA) Certification Program. The CSWA Certification Program provides the skills students need to work in the design and engineering fields. Successfully passing the CSWA Exam assessment proves competency in 3D CAD modeling technology, application of engineering principles, and recognition of global industry practices. Appendix A provides more information and a sample exam.

More Resources

The SolidWorks Education web site (http://www.solidworks.com/education) is a dynamic resource of information and updates for you. This site is focused on the needs of you — the instructor — and the resources that you need to modernize the way in which engineering design graphics is taught today.

The following table showcases many additional resources to help make the SolidWorks software easy to learn, use, and teach:

Curriculum and Community Resources for Educators and Students	
Curriculum Resources	
SolidWorks Instructor Guides - a collection of tutorials and projects that utilize SolidWorks design and analysis tools. Includes the documents, PowerPoint presentations, and movie files in reproducible format. Login account required on SolidWorks Customer Portal.	www.solidworks.com/curriculum
SolidWorks Student Guides - a collection of tutorials and projects that are available from within the SolidWorks Education Edition.	Select Help>Student Curriculum
Teacher Blog - a collection of lessons developed by teachers for teachers that use SolidWorks to reinforce concepts in science, technology, engineering and math concepts.	http://blogs.solidworks.com/teacher
Student Access - Allows students to access SolidWorks software outside the classroom or laboratory.	http://www.solidworks.com/studentaccess
SolidWorks Tutorials - Access a wide range of free, informative resources - full video tutorials, PDF guides, project files, and demo clips - designed to help you become a top SolidWorks user.	http://www.solidworks.com/tutorials
Community Resources	
3D Content Central - a library of part, assembly, drawing, blocks and macro files.	www.3DContentCentral.com
SolidWorks User Group Network - a independent community of local and regional SolidWorks users throughout the world.	www.swugn.org
SolidWorks Blog - the official SolidWorks blog and access to over 35 independent SolidWorks bloggers	http://blogs.solidworks.com
SolidWorks User Network - a comprehensive resource forum on specific product areas	http://forum.solidworks.com/
SolidWorks Sponsored Design Contests - SolidWorks supports thousands of students in design competitions in after school programs including FSAE/Formula Student teams, Robotics competitions, Technology competitions	www.solidworks.com/ SponsoredDesignContests

Curriculum and Community Resources for Educators and Students	
Textbooks - books based on SolidWorks software available from a variety of publishers	www.amazon.com www.delmarlearning.com www.g-w.com www.mcgrawhill.com www.prenhall.com www.schroff.com
Video - YouTube playlists for Formula SAE/Formula Student, Certified SolidWorks Associate Exam (CSWA) and SolidWorks Tutorials	www.youtube.com/solidworks
Certified SolidWorks Associate (CSWA) Exam Provider Program - The CSWA Provider Program is an engineering design competency based program that leads students to achieve certification through the Certified SolidWorks Associate Exam (CSWA) Exam. Used by industry as a recommended competency for job placement and used by academia for assessment and articulation agreements. A desk copy of the CSWA Exam Preparation Guide is available through www.schroff.com	CSWA Provider Application: www.solidworks.com/CSWAProvider Sample CSWA exam: www.solidworks.com/CSWA

1

Lesson 1: Using the Interface

Goals of This Lesson

- ❑ Become familiar with the Microsoft Windows® interface.
- ❑ Become familiar with the SolidWorks user interface.

Note: If your students are already experienced with the Microsoft Windows Graphical User Interface, you may wish to skip to the section of this lesson that familiarizes students with the SolidWorks user interface.

Before Beginning This Lesson

- ❑ Verify that Microsoft Windows is loaded and running on your classroom/lab computers.
- ❑ Verify that the SolidWorks software is loaded and running on your classroom/lab computers in accordance with your SolidWorks license.
- ❑ Load the lesson files from the Educator Resources link.

Outline of Lesson 1

- ❑ Active Learning Exercise — Using the Interface
  - Starting a Program
  - Exiting a Program
  - Opening an Existing File
  - Saving a File
  - Copying a File
  - Resizing Windows
  - SolidWorks Windows
  - Toolbars
  - Mouse Buttons
  - Context-sensitive Shortcut Menus
  - Getting Online Help
- ❑ Lesson Summary

The *Instructor's Guide to Teaching SolidWorks* provides additional examples, presentations, model files, and quizzes. Visit www.solidworks.com/customerportal for more.

Competencies for Lesson 1

Students develop the following competencies in this lesson:

- **Engineering**: Knowledge of an engineering design industry software application.
- **Technology**: Understand file management, copy, save, starting and exiting programs.

Active Learning Exercise — Using the Interface

Start the SolidWorks application, open a file, save the file, save the file with a new name, and review the basic user interface.

Starting a Program

1 Click the **Start** button  in the lower left corner of the window. The **Start** menu appears. The **Start** menu allows you to select the basic functions of the Microsoft Windows environment.

Note: Click means to press and release the left mouse button.

2 From the **Start** menu, click **All Programs, SolidWorks, SolidWorks**.

The SolidWorks application program is now running.

TIP: A desktop shortcut is an icon that you can double-click to go directly to the file or folder represented. The illustration shows the SolidWorks shortcut.

Exit the Program

To exit the application program, click **File**, **Exit** or click  on the main SolidWorks window.

Opening an Existing File

3 Double-click on the SolidWorks part file `Dumbell` in the `Lesson01` folder.

This opens the `Dumbell` file in SolidWorks. If the SolidWorks application program is not running when you double-click on the part file name, the system runs the SolidWorks application program and then opens the part file that you selected.

TIP: Use the left mouse button to double-click. Double-clicking with the left mouse button is often a quick way of opening files from a folder.

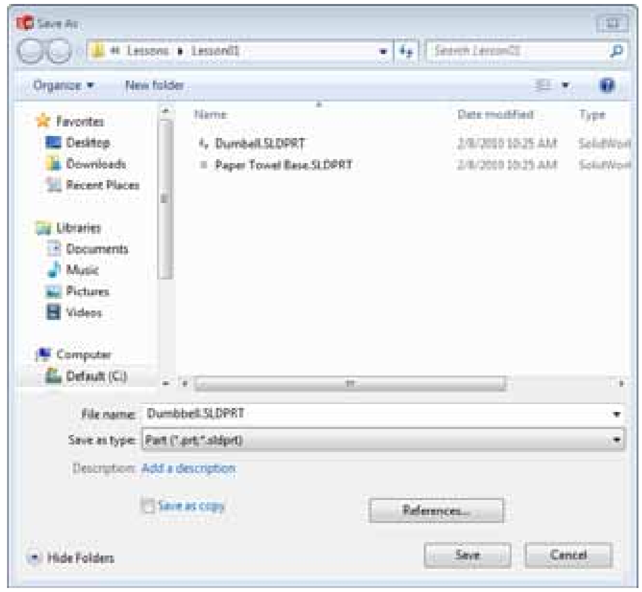
You could have also opened the file by selecting **File, Open**, and typing or browsing to a file name or by selecting a file name from the **File** menu in SolidWorks. SolidWorks lists the last several files that you had open.

Saving a File

4 Click **Save**  on the Standard toolbar to save changes to a file.

It is a good idea to save the file that you are working whenever you make changes to it.

Copying a File



Notice that `Dumbell` is not spelled correctly. It is supposed to have two "b's".

1 Click **File**, **Save As** to save a copy of the file with a new name.

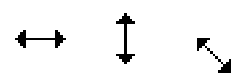
The **Save As** window appears. This window shows you in which folder the file is currently located, the file name, and the file type.

2 In the **File Name** field change the name to `Dumbbell` and click **Save**.

A new file is created with the new name. The original file still exists. The new file is an exact copy of the file as it exists at the moment that it is copied.

Resizing Windows

SolidWorks, like many applications, uses windows to show your work. You can change the size of each window.

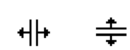


1 Move the cursor along the edge of a window until the shape of the cursor appears to be a two-headed arrow.

2 While the cursor still appears to be a two-headed arrow, hold down the left mouse button and drag the window to a different size.

3 When the window appears to be the size that you wish, release the mouse button.

Windows can have multiple panels. You can resize these panels relative to each other.



4 Move the cursor along the border between two panels until the cursor appears to be two parallel lines with perpendicular arrows.

5 While the cursor still appears to be two parallel lines with perpendicular arrows, hold down the left mouse button and drag the panel to a different size.

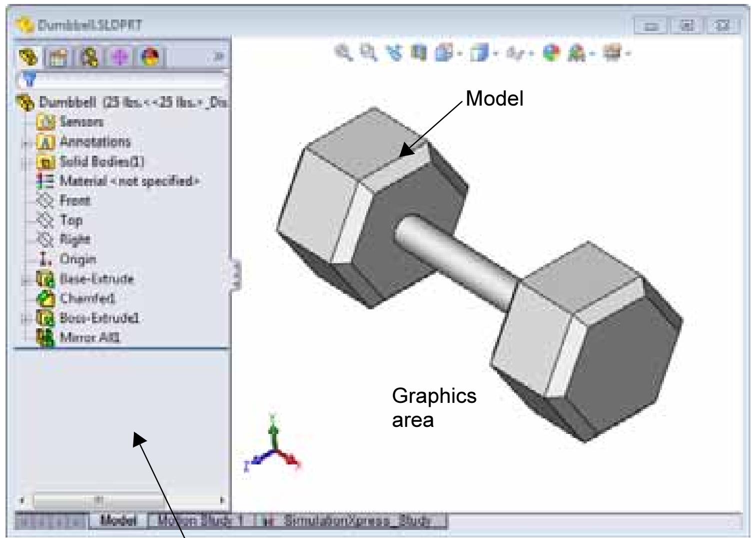
6 When the panel appears to be the size that you wish, release the mouse button.

SolidWorks Windows

SolidWorks windows have two panels. One panel provides non-graphic data. The other panel provides graphic representation of the part, assembly, or drawing.

The leftmost panel of the window contains the FeatureManager® design tree, PropertyManager and ConfigurationManager.

1 Click each of the tabs at the top of the left panel and see how the contents of the window changes.



Left panel displaying the FeatureManager design tree

The rightmost panel is the Graphics Area, where you create and manipulate the part, assembly, or drawing.

2 Look at the Graphics Area. See how the dumbbell is represented. It appears shaded, in color and in an isometric view. These are some of the ways in which the model can be represented very realistically.

Toolbars

Toolbar buttons are shortcuts for frequently used commands. You can set toolbar placement and visibility based on the document type (part, assembly, or drawing). SolidWorks remembers which toolbars to display and where to display them for each document type.

1 Click **View, Toolbars**.

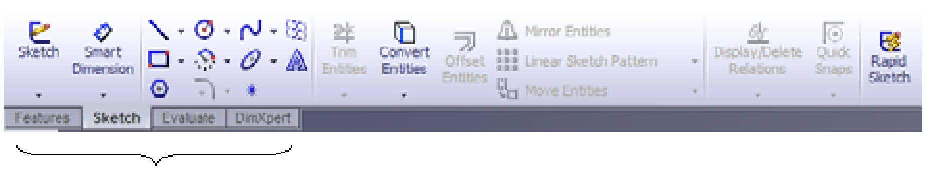
A list of all toolbars displays. The toolbars with their icon depressed or a check mark beside them are visible; the toolbars whose icons are not depressed or without a check mark are hidden.

2 Turn several toolbars on and off to see the commands.

CommandManager

The CommandManager is a context-sensitive toolbar that dynamically updates based on the toolbar you want to access. By default, it has toolbars embedded in it based on the document type.

When you click a button in the control area, the CommandManager updates to show that toolbar. For example, if you click **Sketch** in the control area, the sketch tools appear in the CommandManager.



control area

Use the CommandManager to access toolbar buttons in a central location and to save space for the graphics area.

Mouse Buttons

Mouse buttons operate in the following ways:

- **Left** – Selects menu items, entities in the graphics area, and objects in the FeatureManager design tree.
- **Right** – Displays the context-sensitive shortcut menus.
- **Middle** – Rotates, pans, and zooms the view of a part or an assembly, and pans in a drawing.

Shortcut Menus

Shortcut menus give you access to a wide variety of tools and commands while you work in SolidWorks. When you move the pointer over geometry in the model, over items in the FeatureManager design tree, or over the SolidWorks window borders, right-clicking pops up a shortcut menu of commands that are appropriate for wherever you clicked.

You can access the "more commands menu" by selecting the double-down arrows in the menu. When you select the double-down arrows or pause the pointer over the double-down arrows, the shortcut menu expands to offer more menu items.

The shortcut menu provides an efficient way to work without continually moving the pointer to the main pull-down menus or the toolbar buttons.

Getting Online Help

If you have questions while you are using the SolidWorks software, you can find answers in several ways:

- Click **Help** on the Standard toolbar.
- Click **Help, SolidWorks Help Topics** in the menu bar.
- While in a command, click **Help** in the dialog.

Lesson 1 — 5 Minute Assessment — Answer Key

Name: ______________________________Class: _________ Date:______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1. How do you open the file from Windows Explorer?

   Answer: Double-click on the file name.

2. How do you start the SolidWorks program?

   Answer: Click , **All Programs**, **SolidWorks**, **SolidWorks**.

3. What is the quickest way to start the SolidWorks program?

   Answer: Double-click the SolidWorks desktop shortcut (if one exists).

4. How do you copy a part within the SolidWorks program?

   Answer: Click **File, Save As** and assign a new name.

Lesson 1 — 5 Minute Assessment **REPRODUCIBLE**

Name: ______________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you open the file from Windows Explorer?

____

2 How do you start the SolidWorks program?

____

3 What is the quickest way to start the SolidWorks program?

____

4 How do you copy a part within the SolidWorks program?

____

Lesson 1 Vocabulary Worksheet — Answer Key

Name: ______________________________ Class: _________ Date: _______________

Fill in the blanks with the words that are defined by the clues.

1 Shortcuts for collections of frequently used commands: **toolbars**
2 Command to create a copy of a file with a new name: **File, Save As**
3 One of the areas that a window is divided into: **panel**
4 The graphic representation of a part, assembly, or drawing: **model**
5 Area of the screen that displays the work of a program: **window**
6 Icon that you can double-click to start a program: **desktop shortcut**
7 Action that quickly displays shortcut menus of frequently used or detailed commands: **right-click**
8 Command that updates your file with changes that you have made to it: **File, Save**
9 Action that quickly opens a part or program: **double-click**
10 The program that helps you create parts, assemblies, and drawings: **SolidWorks**
11 Panel of the SolidWorks window that displays a visual representation of your parts, assemblies, and drawings: **graphics area**

Lesson 1 Vocabulary Worksheet **REPRODUCIBLE**

Name: ______________________________Class: _________ Date:_______________

Fill in the blanks with the words that are defined by the clues.

1 Shortcuts for collections of frequently used commands: ________________________

2 Command to create a copy of a file with a new name: __________________________

3 One of the areas that a window is divided into: _______________________________

4 The graphic representation of a part, assembly, or drawing: ______________________

5 Area of the screen that displays the work of a program: _________________________

6 Icon that you can double-click to start a program: ______________________________

7 Action that quickly displays shortcut menus of frequently used or detailed commands:
____

8 Command that updates your file with changes that you have made to it: ____________
____

9 Action that quickly opens a part or program: _________________________________

10 The program that helps you create parts, assemblies, and drawings: _______________

11 Panel of the SolidWorks window that displays a visual representation of your parts, assemblies, and drawings: ____

Lesson 1 Quiz — Answer Key

Name: ________________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you start the SolidWorks application program?

Answer: Click , **All Programs**, **SolidWorks**, **SolidWorks**; or double-click on the SolidWorks desktop shortcut; or double-click on a SolidWorks file.

2 Which command would you use to create a copy of your file?

Answer: **File**, **Save As**

3 Where do you see a 3D representation of your model?

Answer: Graphics Area.

4 Look at the illustration (at right). What is this collection of frequently used commands called?

Answer: Toolbar

5 Which command would you use to preserve changes that you have made to a file?

Answer: **File**, **Save**

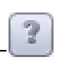
6 Circle the cursor that is used to resize a window.

Answer: 

7 Circle the cursor that is used to resize a panel.

Answer: 

8 Circle the button that is used to get online help.

Answer: 

Lesson 1 Quiz **REPRODUCIBLE**

Name: ______________________________ Class: _________ Date: ______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you start the SolidWorks application program?

____

____

2 Which command would you use to create a copy of your file? ____________________

3 Where do you see a 3D representation of your model? _________________________

4 Look at the illustration (at right). What is this collection of frequently used commands called?

____

5 Which command would you use to preserve changes that you have made to a file?

____

6 Circle the cursor that is used to resize a window.

7 Circle the cursor that is used to resize a panel.

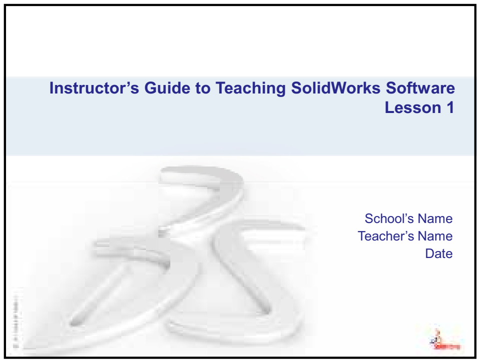
8 Circle the button that is used to get online help.

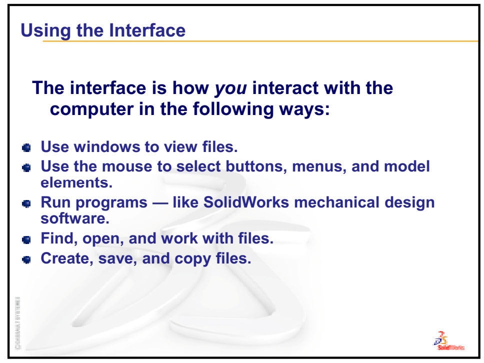
Lesson Summary

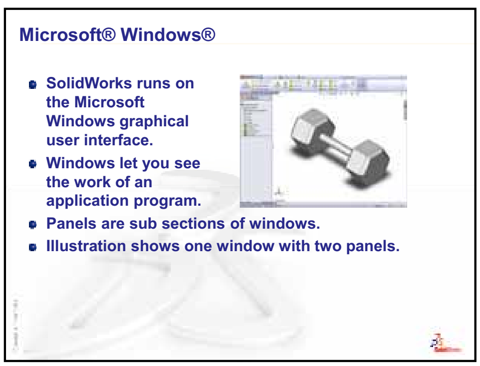
- The Start menu is where you go to start programs or find files.
- There are short cuts such as right-click and double-click that can save you work.
- **File, Save** allows you to save updates to a file and **File**, **Save As** allows you to make a copy of a file.
- You can change the size and location of windows as well as panels within windows.
- The SolidWorks window has a Graphics Area that shows 3D representations of your models.

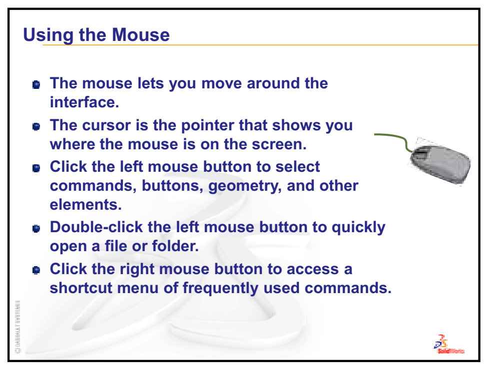
Thumbnail Images of PowerPoint Slides

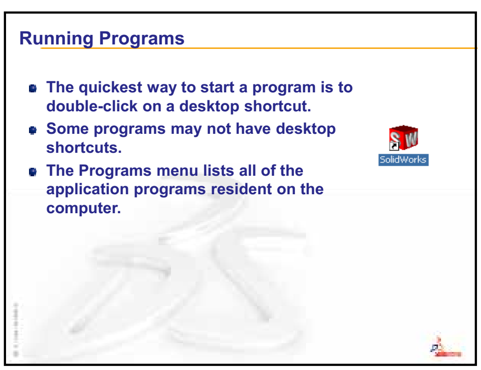
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

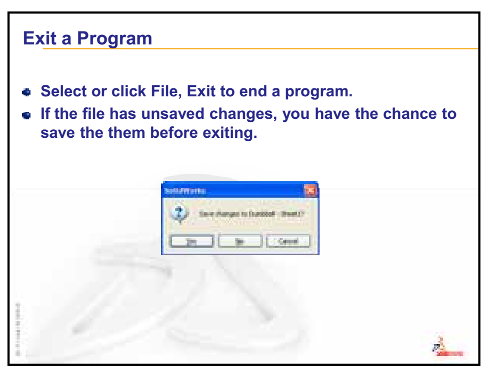












Opening a File

- The quickest way to open a file is to double-click on it.
- The File menu displays your most recently used files.

Saving and Copying Files

- Saving a file preserves the changes that you have made to it.

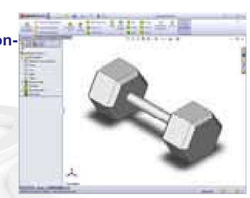
- Use File, Save As to copy a file.
- File, Save As creates an exact duplicate of the file as it existed at the moment that you copied it.

Resizing Windows

- Allows you to customize the appearance of your screen.
- View multiple files at the same time.
- Use ↔ ↕ ⤡ to change the size of a window.
- Use ⟛ ⟚ to change the size of panels within a window.

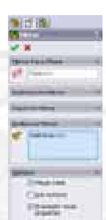
Using the SolidWorks Interface

- SolidWorks windows display graphic and non-graphic model data.
- Toolbars display frequently used commands.



Left Side of SolidWorks Window

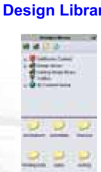
- FeatureManager® design tree
- Property Manager
- Configuration Manager



Right Side of SolidWorks Window

The Task Pane

- SolidWorks Resources
- Design Library



Right Side of SolidWorks Window

The Task Pane

- Toolbox

- File Explorer

Toolbars

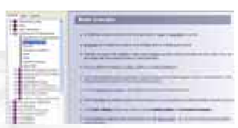
Buttons for frequently used commands.

- You can select the toolbars to display.
- Toolbars are displayed at the top and sides of the window.
- You can also access the toolbars from the CommandManager.

Getting Help

To view comprehensive online help:

- Click .
- Select Help, SolidWorks Help.
- Help displays in a separate window.

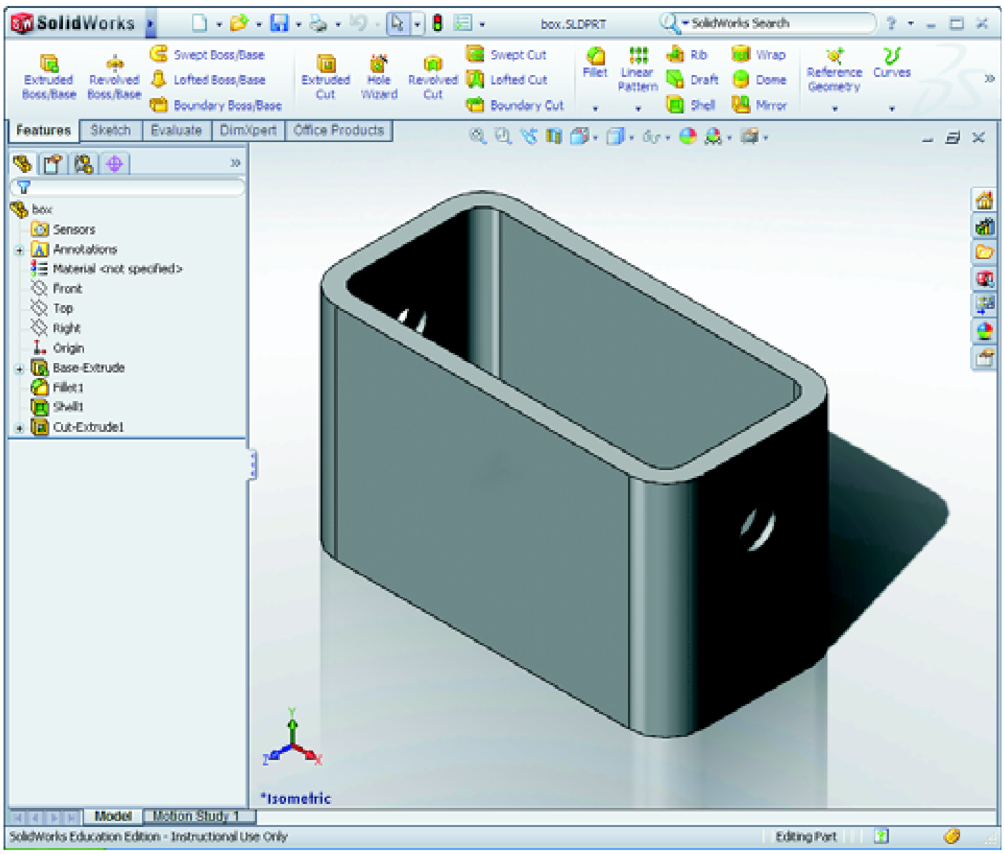


2

Lesson 2: Basic Functionality

Goals of This Lesson

- ❑ Understand the basic functionality of the SolidWorks software.
- ❑ Create the following part:



Before Beginning This Lesson

Complete Lesson 1: Using the Interface.

Access a wide range of free, informative resources - full video tutorials, PDF guides, project files, and demo clips - designed to help you become a top SolidWorks user. Visit http://www.solidworks.com/tutorials.

Review of Lesson 1: Using the Interface

The interface is how *you* interact with the computer in the following ways:

- ❑ Use windows to view files.
- ❑ Use the mouse to select buttons, menus, and model elements.
- ❑ Run programs — like SolidWorks mechanical design software.
- ❑ Find, open, and work with files.
- ❑ Create, save, and copy files.
- ❑ SolidWorks runs on the Microsoft Windows graphical user interface.
- ❑ The mouse lets you move around the interface.
- ❑ The quickest way to open a file is to double-click on it.
- ❑ Saving a file preserves the changes that you have made to it.
- ❑ SolidWorks windows display graphic and non-graphic model data.
- ❑ Toolbars display frequently used commands.

Outline of Lesson 2

- ❑ In Class Discussion — The SolidWorks Model
- ❑ Active Learning Exercise — Creating a Basic Part
  - Create a New Part Document
  - Overview of the SolidWorks Window
  - Sketch a Rectangle
  - Add Dimensions
  - Changing the Dimension Values
  - Extrude the Base Feature
  - View Display
  - Save the Part
  - Round the Corners of the Part
  - Hollow Out the Part
  - Extruded Cut Feature
  - Open a Sketch
  - Sketch the Circle
  - Dimension the Circle
  - Extrude the Sketch
  - Rotate the View
  - Save the Part
- ❑ In Class Discussion — Describing the Base Feature
- ❑ Exercises and Projects — Designing a Switch Plate
- ❑ More to Explore — Modifying a Part
- ❑ Lesson Summary

Competencies for Lesson 2

Students develop the following competencies in this lesson:

- ❑ **Engineering**: Develop a 3D part based on a selected plane, dimensions, and features. Apply the design process to develop the box or switch plate out of cardboard or other material. Develop manual sketching techniques by drawing the switch plate.
- ❑ **Technology**: Apply a windows based graphical user interface.
- ❑ **Math**: Understand units of measurement, adding and subtracting material, perpendicularity, and the x-y-z coordinate system.

In Class Discussion — The SolidWorks Model

SolidWorks is design automation software. In SolidWorks, you sketch ideas and experiment with different designs to create 3D models. SolidWorks is used by students, designers, engineers, and other professionals to produce simple and complex parts, assemblies, and drawings.

The SolidWorks model is made up of:

- Parts
- Assemblies
- Drawings

A part is a single 3D object made up of features. A part can become a component in an assembly, and it can be represented in 2D in a drawing. Examples of parts are bolt, pin, plate, and so on. The extension for a SolidWorks part file name is .SLDPRT. Features are the *shapes* and *operations* that construct the part. The Base feature is the first feature that is created.The Base feature is the foundation of the part.

An assembly is a document in which parts, features, and other assemblies (sub-assemblies) are mated together. The parts and sub-assemblies exist in documents separate from the assembly. For example, in an assembly, a piston can be mated to other parts, such as a connecting rod or cylinder. This new assembly can then be used as a sub-assembly in an assembly of an engine. The extension for a SolidWorks assembly file name is .SLDASM.

A drawing is a 2D representation of a 3D part or assembly. The extension for a SolidWorks drawing file name is .SLDDRW.

Active Learning Exercises — Creating a Basic Part

Use SolidWorks to create the box shown at the right.

The step-by-step instructions are given below.

Create a New Part Document

1 Create a new part. Click **New** on the Standard toolbar.

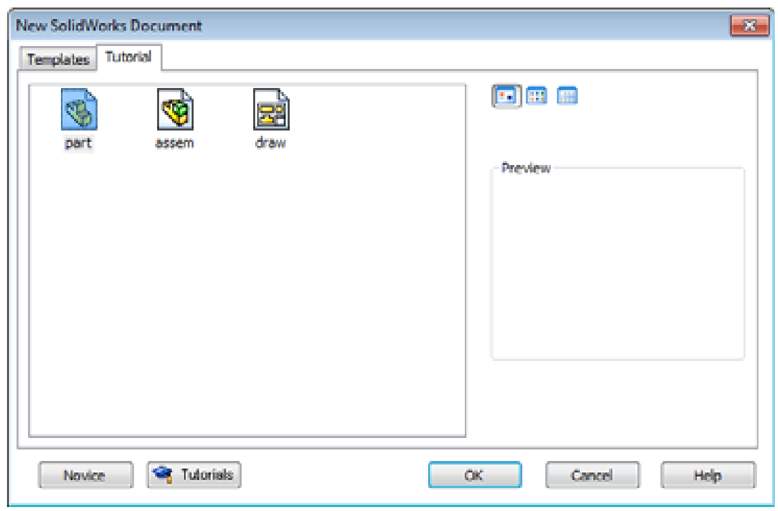
The **New SolidWorks Document** dialog box appears.

2 Click the **Tutorial** tab.

3 Select the **Part** icon.

4 Click **OK**.

A new part document window appears.



Base Feature

The Base feature requires:

- Sketch plane – `Front` (default plane)
- Sketch profile – 2D Rectangle
- Feature type – Extruded boss feature

Open a Sketch

1 Click to select the `Front` plane in the FeatureManager design tree.

2 Open a 2D sketch. Click **Sketch** on the Sketch toolbar.

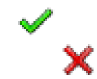
Confirmation Corner

When many SolidWorks commands are active, a symbol or a set of symbols appears in the upper right corner of the graphics area. This area is called the **Confirmation Corner**.

Sketch Indicator

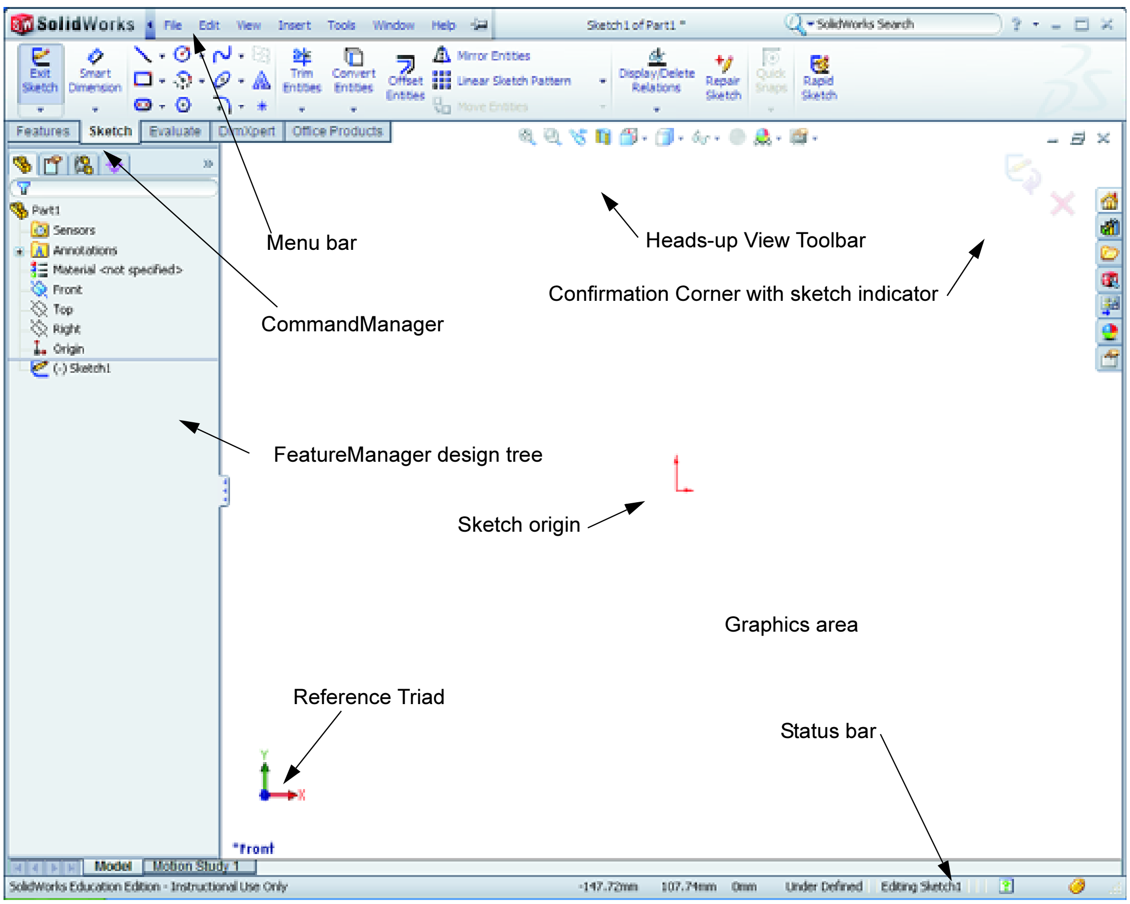
When a sketch is active, or open, a symbol appears in the confirmation corner that looks like the **Sketch** tool. It provides a visual reminder that you are active in a sketch. Clicking this symbol exits the sketch saving your changes. Clicking the red X exits the sketch discarding your changes.

When other commands are active, the confirmation corner displays two symbols: a check mark and an X. The check mark executes the current command. The X cancels the command.



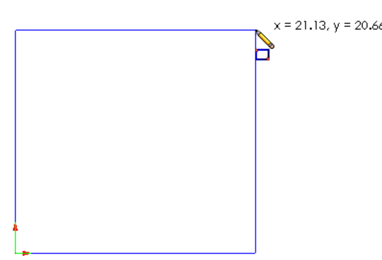
Overview of the SolidWorks Window

- A sketch origin appears in the center of the graphics area.
- **Editing Sketch1** appears in the status bar at the bottom of the screen.
- `Sketch1` appears in the FeatureManager design tree.
- The status bar shows the position of the pointer, or sketch tool, in relation to the sketch origin.

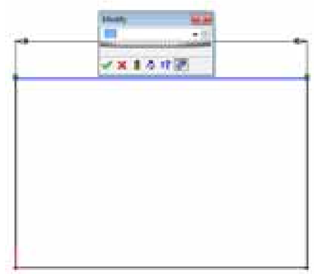


Sketch a Rectangle

1. Click **Corner Rectangle** on the Sketch toolbar.
2. Click the sketch origin to start the rectangle.
3. Move the pointer up and to the right, to create a rectangle.
4. Click the mouse button again to complete the rectangle.



Add Dimensions



1 Click **Smart Dimension** on the Dimensions/Relations toolbar.

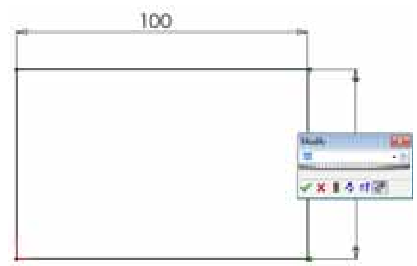
The pointer shape changes to .

2 Click the top line of the rectangle.

3 Click the dimension text location above the top line.

The **Modify** dialog box is displayed.

4 Enter **100**. Click or press **Enter**.



5 Click the right edge of the rectangle.

6 Click the dimension text location. Enter **65**. Click .

The top segment and the remaining vertices are displayed in black. The status bar in the lower-right corner of the window indicates that the sketch is fully defined.

Changing the Dimension Values

The new dimensions for the `box` are 100mm x 60mm. Change the dimensions.

1 Double-click **65**.

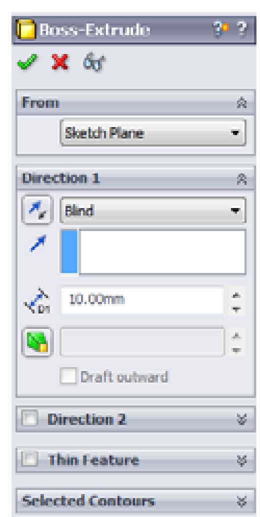
The **Modify** dialog box appears.

2 Enter **60** in the **Modify** dialog box.

3 Click .

Extrude the Base Feature.

The first feature in any part is called the *Base Feature*. In this exercise, the base feature is created by extruding the sketched rectangle.



1 Click **Extruded Boss/Base** on the Features toolbar.

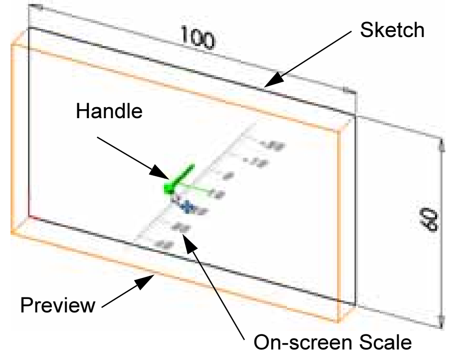
TIP: If the Features toolbar is not visible (active), you may also access the feature commands from the CommandManager.

The **Extrude** PropertyManager appears. The view of the sketch changes to trimetric.

2 Preview graphics.

A preview of the feature is shown at the default depth.

Handles appear that can be used to drag the preview to the desired depth. The handles are colored magenta for the active direction and gray for inactive direction. A callout shows the current depth value.

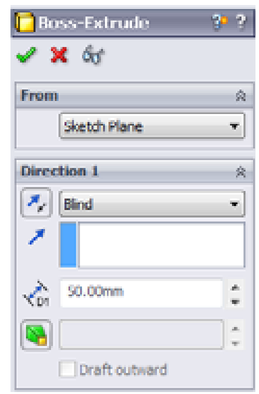


The cursor changes to . If you want to create the feature now, click the right mouse button. Otherwise, you can make additional changes to the settings. For example, the depth of extrusion can be changed by dragging the dynamic handle with the mouse or by setting a value in the PropertyManager.

3 Extrude feature settings.

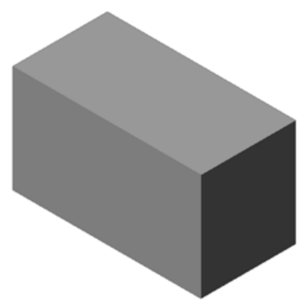
Change the settings as shown.

- End Condition = **Blind**
- (Depth) = **50**



4 Create the extrusion. Click **OK** .

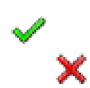
The new feature, `Boss-Extrude1`, is displayed in the FeatureManager design tree.



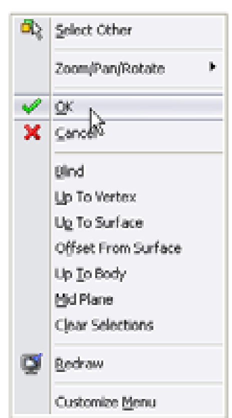
TIP:

The **OK** button on the PropertyManager is just one way to complete the command.

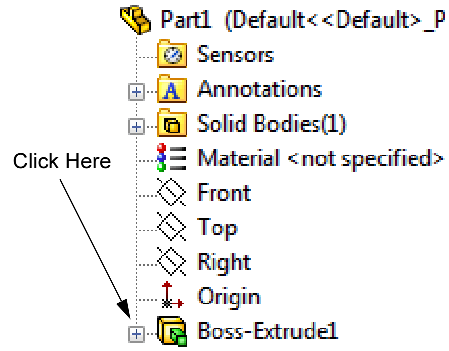
A second method is the set of **OK/Cancel** buttons in the confirmation corner of the graphics area.



A third method is the right-mouse shortcut menu that includes **OK**, among other options.



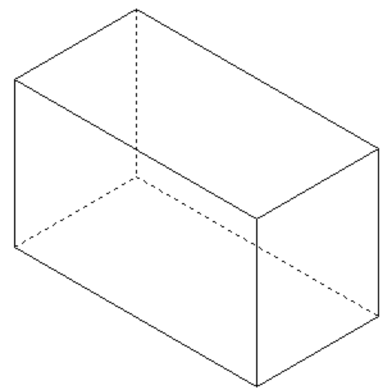
5 Click the plus sign ⊞ beside `Extrude1` in the FeatureManager design tree. Notice that `Sketch1` — which you used to extrude the feature — is now listed under the feature.



View Display

Change the display mode. Click **Hidden Lines Visible** on the View toolbar.

Hidden Lines Visible enables you to select hidden back edges of the box.



Save the Part

1 Click **Save** on the Standard toolbar, or click **File, Save.**

The **Save As** dialog box appears.

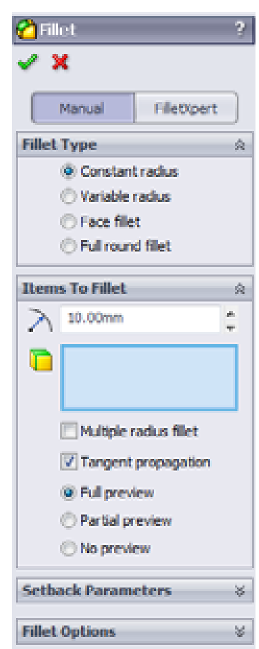
2 Type `box` for the filename. Click **Save**.

The `.sldprt` extension is added to the filename.

The file is saved to the current directory. You can use the Windows browse button to change to a different directory.

Round the Corners of the Part

Round the four corner edges of the `box`. All rounds have the same radius (10mm). Create them as a single feature.



1 Click **Fillet** on the Features toolbar.

The **Fillet** PropertyManager appears.

2 Enter **10** for the **Radius.**

3 Select **Full preview**.

Leave the remaining settings at their default values.

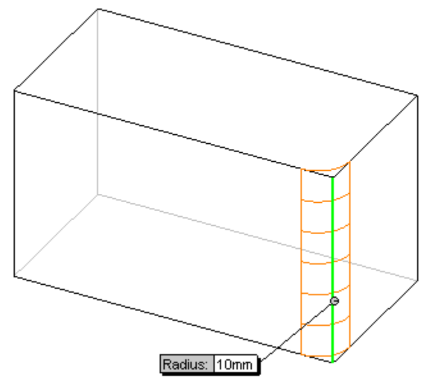
4 Click the first corner edge.

The faces, edges, and vertices are highlighted as you move the pointer over them.

When you select the edge, a callout Radius: 10mm appears.

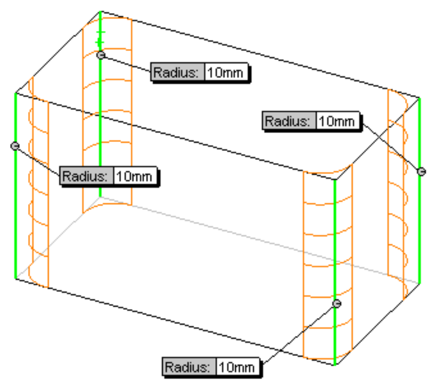
5 Identify selectable objects. Notice how the pointer changes shapes:

Edge: Face: Vertex:



6 Click the second, third and fourth corner edges.

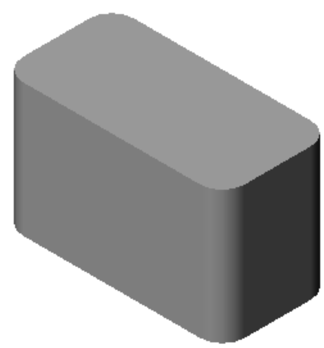
Note: Normally, a callout only appears on the *first* edge you select. This illustration has been modified to show callouts on each of the four selected edges. This was done simply to better illustrate which edges you are supposed to select.



7 Click **OK** .

`Fillet1` appears in the FeatureManager design tree.

8 Click **Shaded** on the View toolbar



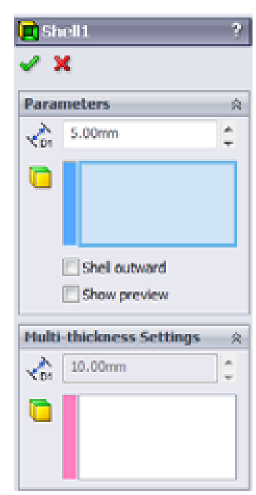
Hollow Out the Part

Remove the top face using the Shell feature.

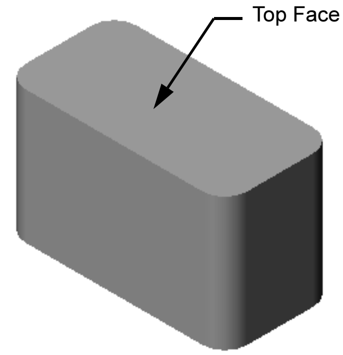
1 Click **Shell** on the Features toolbar.

The **Shell** PropertyManager appears.

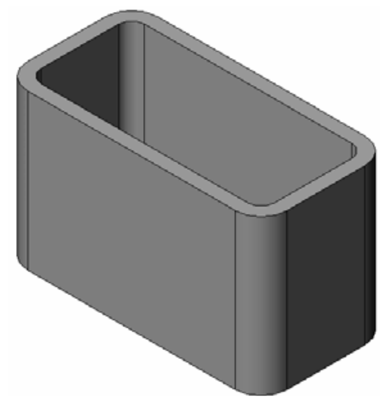
2 Enter **5** for **Thickness**.



3 Click the top face.



4 Click ✓.



Extruded Cut Feature

The Extruded Cut feature removes material. To make an extruded cut requires a:

- Sketch plane – In this exercise, the face on the right-hand side of the part.
- Sketch profile – 2D circle

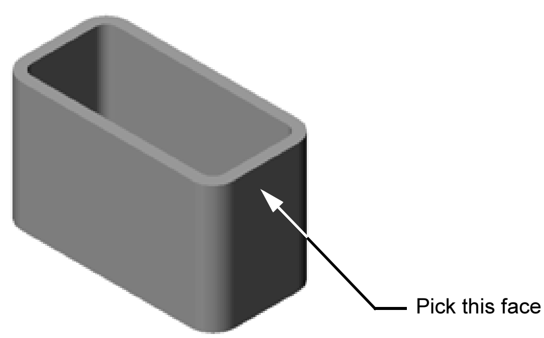
Open a Sketch

1 To select the sketch plane, click the right-hand face of the `box`.

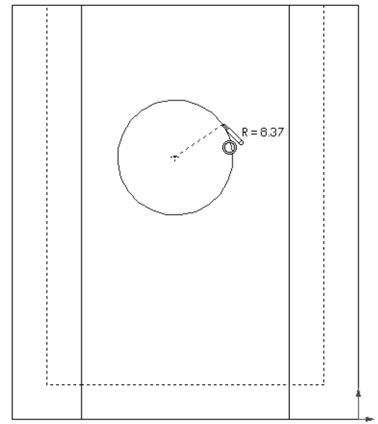
2 Click **Right** on the Standard Views toolbar.

The view of the `box` turns. The selected model face is facing you.

3 Open a 2D sketch. Click **Sketch** on the Sketch toolbar.

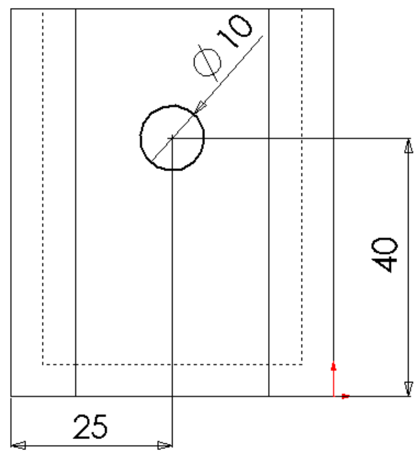


Sketch the Circle



1 Click **Circle** on the Sketch Tools toolbar.
2 Position the pointer where you want the center of the circle. Click the left mouse button.
3 Drag the pointer to sketch a circle.
4 Click the left mouse button again to complete the circle.

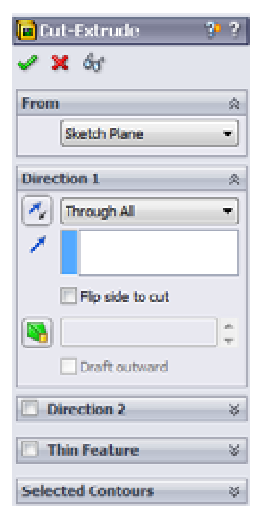
Dimension the Circle



Dimension the circle to determine its size and location.

1 Click **Smart Dimension** on the Dimensions/ Relations toolbar.
2 Dimension the diameter. Click on the circumference of the circle. Click a location for the dimension text in the upper right corner. Enter **10**.
3 Create a horizontal dimension. Click the circumference of the circle. Click the left most vertical edge. Click a location for the dimension text below the bottom horizontal line. Enter **25**.
4 Create a vertical dimension. Click the circumference of the circle. Click the bottom most horizontal edge. Click a location for the dimension text to the right of the sketch. Enter **40**.

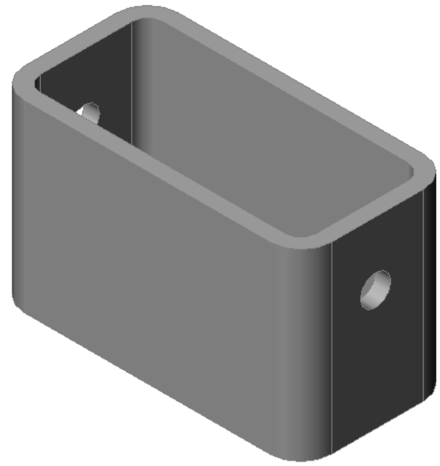
Extrude the Sketch



1 Click **Extruded Cut** on the Features toolbar.
The **Extrude** PropertyManager appears.
2 Select **Through All** for the end condition.
3 Click .

4 Results.

The cut feature is displayed.



Rotate the View

Rotate the view in the graphics area to display the model from different angles.

1 Rotate the part in the graphics area. Press and hold the middle mouse button. Drag the pointer up/down or left/right. The view rotates dynamically.

2 Click **Isometric** on the Standard Views toolbar.

Save the Part

1 Click **Save** on the Standard toolbar.

2 Click **File, Exit** on the Main menu.

Lesson 2 — 5 Minute Assessment — Answer Key

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you start a SolidWorks session?

Answer: Click . Click All Programs. Click the SolidWorks folder. Click the SolidWorks application.

2 Why do you create and use Document Templates?

Answer: Document Templates contain the units, grid and text settings for the model. You can create Metric and English templates each with different settings.

3 How do you start a new Part Document?

Answer: Click the **New** icon. Select a part template.

4 What features did you use to create the `box`?

Answer: Extruded Boss, Fillet, Shell, and Extruded Cut.

5 True or False. SolidWorks is used by designers and engineers.

Answer: True.

6 A SolidWorks 3D model consists of _________ _________ ________.

Answer: Parts, assemblies and drawings.

7 How do you open a sketch?

Answer: Click the Sketch icon on the Sketch toolbar.

8 What does the Fillet feature do?

Answer: The Fillet feature rounds sharp edges.

9 What does the Shell feature do?

Answer: The Shell feature removes material from the selected face.

10 What does the Cut-Extrude feature do?

Answer: The Cut-Extrude feature removes material.

11 How do you change a dimension value?

Answer: Double-click on the dimension. Enter the new value in the **Modify** dialog box.

Lesson 2 — 5 Minute Assessment **REPRODUCIBLE**

Name: ______________________________Class: _________ Date:______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you start a SolidWorks session?

____

____

2 Why do you create and use Document Templates?

____

____

3 How do you start a new Part Document?

____

4 What features did you use to create the `box`?

____

5 True or False. SolidWorks is used by designers and engineers.

____

6 A SolidWorks 3D model consists of _________ _________ ________.

____

7 How do you open a sketch?

____

8 What does the Fillet feature do?

____

9 What does the Shell feature do?

____

10 What does the Cut-Extrude feature do?

____

11 How do you change a dimension value?

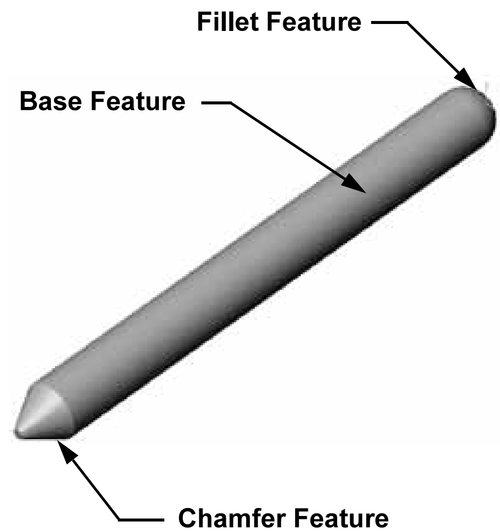
____

In Class Discussion — Describing the Base Feature

Pick up a pencil. Ask the students to describe the base feature of the pencil. How would you create the additional features for the pencil?

Answer

- ❑ Sketch a circular 2D profile.
- ❑ Extrude the 2D sketch. This creates the base feature which is named `Extrude1`.
- ❑ Select one circular edge on the base feature. Create a fillet feature. The fillet feature removes sharp edges. The fillet feature creates the eraser for the pencil.
- ❑ Select the other circular edge on the base feature. Create a chamfer feature. The chamfer feature creates the point for the pencil.



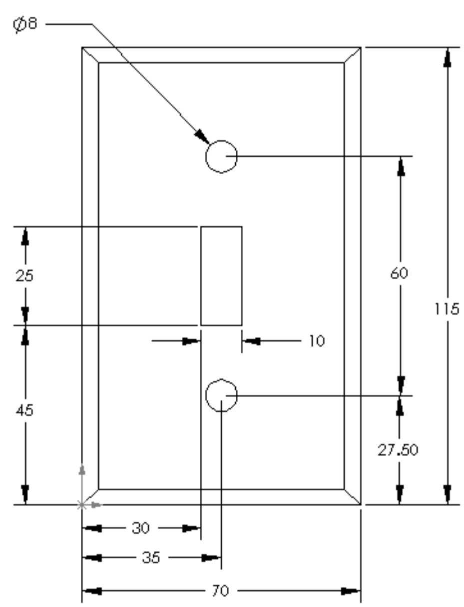
Exercises and Projects — Designing a Switch Plate

Switch plates are required for safety. They cover live electrical wires and protect people from electric shock. Switch plates are found in every home and school.

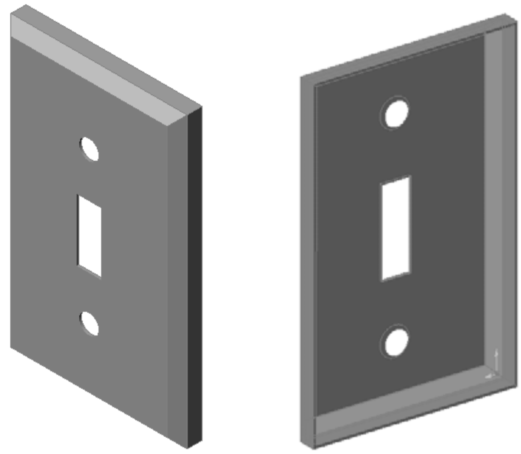
⚠ Caution: Do not use metal rulers near switch plates attached to a live wall outlet.

Tasks

1. Measure a single light plate switch cover.
   Answer: Overall a single switch plate is approximately 70mm x 115mm x 10mm. The switch cut-out is approximately 10mm x 25mm.
2. Using paper and pencil, manually sketch the light plate switch cover.
3. Label the dimensions.
4. What is the base feature for the light plate switch cover?
   Answer: It is an extruded boss feature.



5 Create a simple single light switch cover using SolidWorks. The filename for the part is `switchplate`.



6 What features are used to develop the `switchplate`?

Answer: The extruded boss, chamfer, shell and extruded cut features are used to create the `switchplate`.

- The order in which the features are created is important.

  First – create the base feature.

  Second – create the chamfer feature.

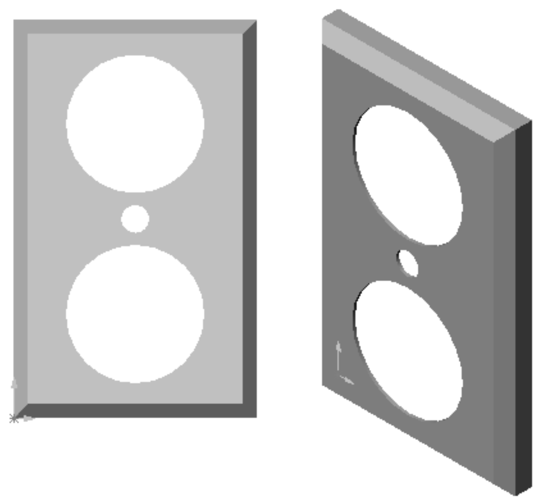
  Third – create the shell feature.

  Fourth – create the cut feature for the switch hole.

  Fifth – create the cut feature for the screw holes.

- The file `switchplate.sldprt` is found in `Lessons\Lesson2` in the `SolidWorks Teacher Tools` folder.

7 Create a simplified duplex outlet cover plate. The filename for the part is `outletplate`.



Answer: The `outletplate.sldprt` file is found in `Lessons\Lesson2` in the `SolidWorks Teacher Tools` folder.

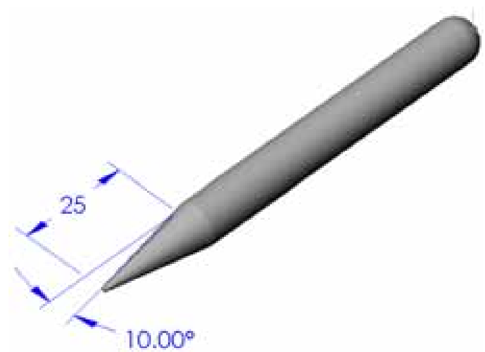
8 Save the parts. They will be used in later lessons.

More to Explore — Modifying a Part

Many pencils have a longer, sharper point than the one shown earlier. How can this be accomplished?

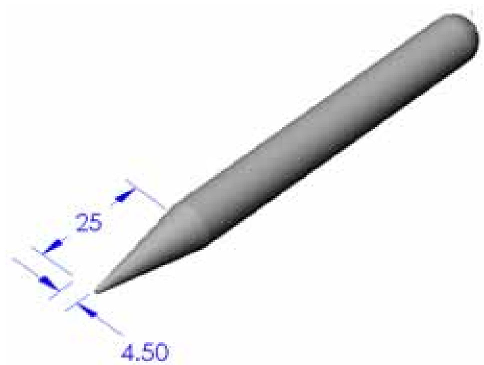
Answer

Answers will vary. One possibility is:



- ❑ Double-click chamfer feature, either in the FeatureManager design tree or the graphics area.
- ❑ Change the angle to **10°**.
- ❑ Change the distance to **25mm**.
- ❑ Click **Rebuild** on the Standard toolbar to rebuild the part.

Another possibility is:



- ❑ Edit the definition of the chamfer feature.
- ❑ Change the **Type** option to **Distance-Distance**.
- ❑ Set the **Distance1** value to **25mm**.
- ❑ Set the **Distance2** value to **4.5mm**.
- ❑ Click **OK** to rebuild the chamfer feature.

Lesson 2 Vocabulary Worksheet — Answer Key

Name: ______________________________Class: _________ Date:_______________

Fill in the blanks with the words that are defined by the clues.

1 The corner or point where edges meet: **vertex**
2 The intersection of the three default reference planes: **origin**
3 A feature used to round off sharp corners: **fillet**
4 The three types of documents that make up a SolidWorks model: **parts, assemblies, drawings**
5 A feature used to hollow out a part: **shell**
6 Controls the units, grid, text, and other settings of the document: **template**
7 Forms the basis of all extruded features: **sketch**
8 Two lines that are at right angles (90°) to each other are: **perpendicular**
9 The first feature in a part is called the **base** feature.
10 The outside surface or skin of a part: **face**
11 A mechanical design automation software application: **SolidWorks**
12 The boundary of a face: **edge**
13 Two straight lines that are always the same distance apart are: **parallel**
14 Two circles or arcs that share the same center are: **concentric**
15 The shapes and operations that are the building blocks of a part: **features**
16 A feature that adds material to a part: **boss**
17 A feature that removes material from a part: **cut**
18 An implied centerline that runs through the center of every cylindrical feature: **axis**

Lesson 2 Vocabulary Worksheet REPRODUCIBLE

Name: ______________________________Class: _________ Date:_______________

Fill in the blanks with the words that are defined by the clues.

1 The corner or point where edges meet: ______________________________

2 The intersection of the three default reference planes:__________________

3 A feature used to round off sharp corners:___________________________

4 The three types of documents that make up a SolidWorks model: ___________

5 A feature used to hollow out a part: ______________________________

6 Controls the units, grid, text, and other settings of the document:___________

7 Forms the basis of all extruded features: ___________________________

8 Two lines that are at right angles (90°) to each other are: _______________

9 The first feature in a part is called the ____________ feature.

10 The outside surface or skin of a part: ______________________________

11 A mechanical design automation software application:__________________

12 The boundary of a face: ______________________________

13 Two straight lines that are always the same distance apart are:_____________

14 Two circles or arcs that share the same center are:____________________

15 The shapes and operations that are the building blocks of a part: ___________

16 A feature that adds material to a part: ___________________________

17 A feature that removes material from a part: _______________________

18 An implied centerline that runs through the center of every cylindrical feature:_______

Lesson 2 Quiz — Answer Key

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 You build parts from features. What are features?

Answer: Features are the shapes (bosses, cuts and holes) and the operations (fillets, chamfers and shells) that are use to build a part.

2 Name the features that are used to create the `box` in Lesson 2.

Answer: Extruded Boss, Fillet, Shell and Extruded Cut.

3 How do you begin a new part document?

Answer: Click the **New** tool or click **File, New**. Select a part template.

4 Give two examples of shape features that require a sketched profile.

Answer: Shape features are Extruded Boss, Extruded Cut, and Hole.

5 Give two examples of operation features that require a selected edge or face.

Answer: Operation features are Fillet, Chamfer and Shell.

6 Name the three documents that make up a SolidWorks model.

Answer: Parts, assemblies and drawings

7 What is the default sketch plane?

Answer: The default sketch plane is `Front`.

8 What is a plane?

Answer: A plane is a flat 2D surface.

9 How do you create an extruded boss feature?

Answer: Select a sketch plane. Open a new sketch. Sketch the profile. Extrude the profile perpendicular to the sketch plane.

10 Why do you create and use document templates?

Answer: Document templates contain the units, grid and text settings for the model. You can create Metric and English templates, each with different settings.

Lesson 2 Quiz

REPRODUCIBLE

Name: ________________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 You build parts from features. What are features? ______________________________

____

2 Name the features that are used to create the `box` in Lesson 2. ____________________

____

3 How do you begin a new part document? ____________________________________

____

4 Give two examples of shape features that require a sketched profile. _______________

____

5 Give two examples of operation features that require a selected edge or face. ________

____

6 Name the three documents that make up a SolidWorks model. ____________________

____

7 What is the default sketch plane? ____

____

8 What is a plane? ____

____

9 How do you create an extruded boss feature? _______________________________

____

10 Why do you create and use document templates? ___________________________

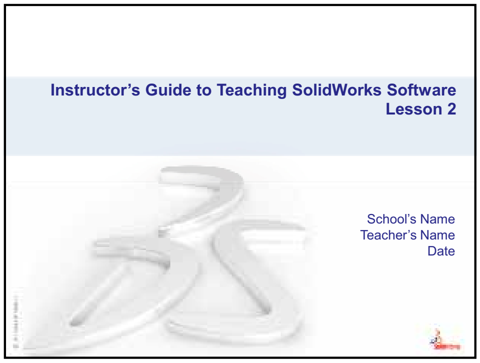
____

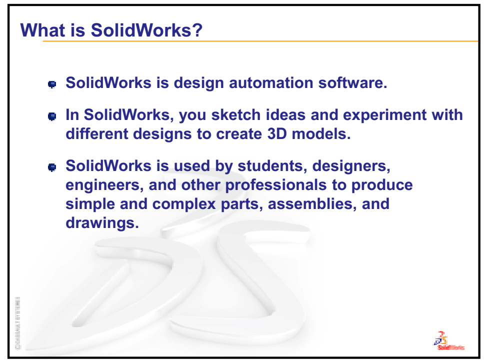
Lesson Summary

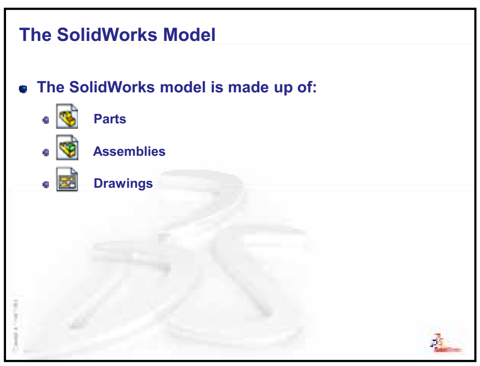
- SolidWorks is design automation software.
- The SolidWorks model is made up of:
  Parts
  Assemblies
  Drawings
- Features are the building blocks of a part.

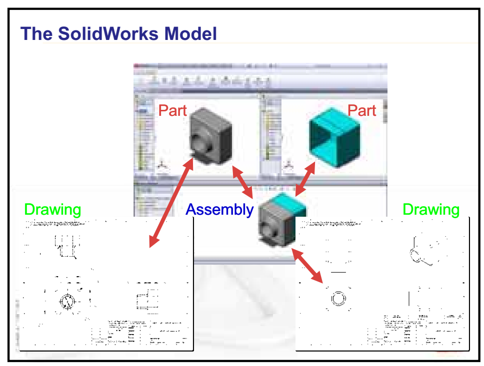
Thumbnail Images of PowerPoint Slides

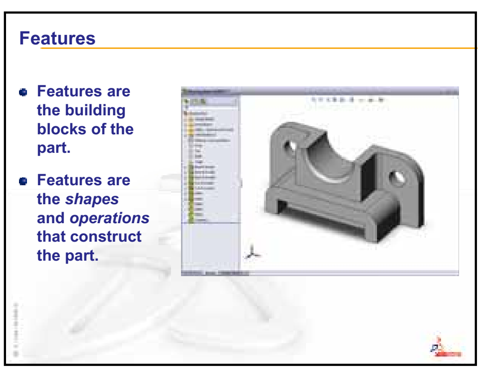
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

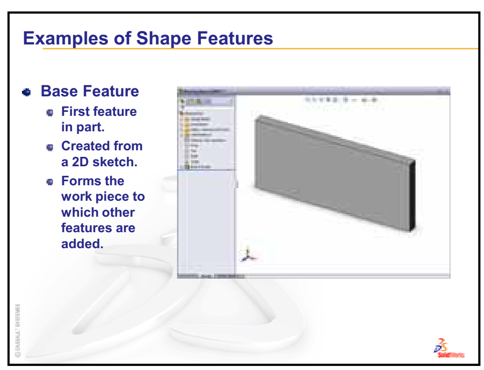






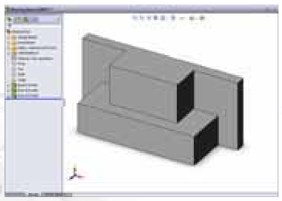






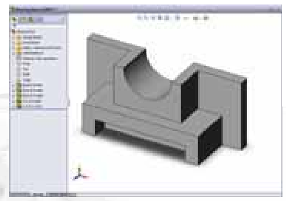
Examples of Shape Features

- Boss feature
  - Adds material to part.
  - Created from 2D sketch.



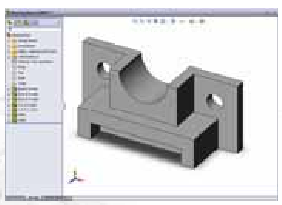
Examples of Shape Features

- Cut feature
  - Removes material from part.
  - Created from 2D sketch.



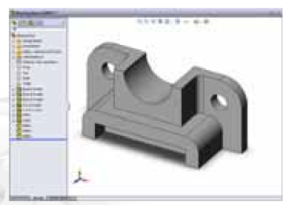
Examples of Shape Features

- Hole feature
  - Removes material.
  - Works like more intelligent cut feature.
  - Corresponds to process such as counter-sink, thread, counter-bore.



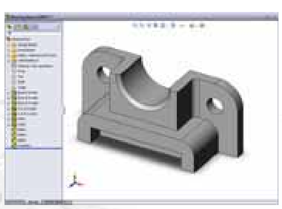
Examples of Shape Features

- Fillet feature
  - Used to round off sharp edges.
  - Can remove or add material.
    - Outside edge (convex fillet) removes material.
    - Inside edge (concave fillet) adds material.



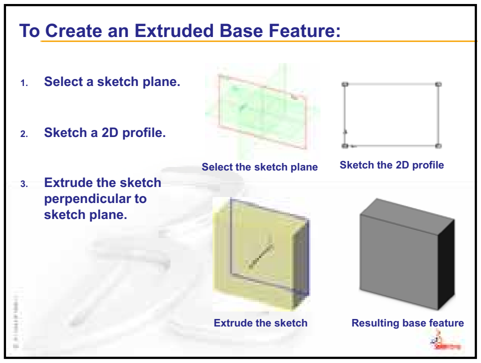
Examples of Shape Features

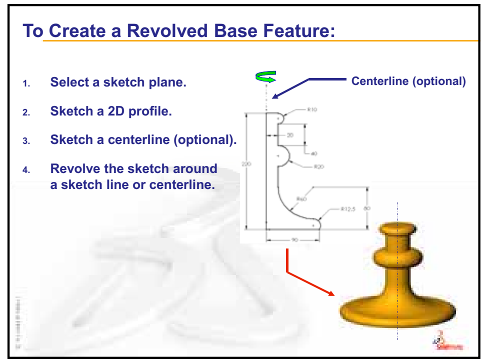
- Chamfer feature
  - Similar to a fillet.
  - Bevels an edge rather than rounding it.
  - Can remove or add material.

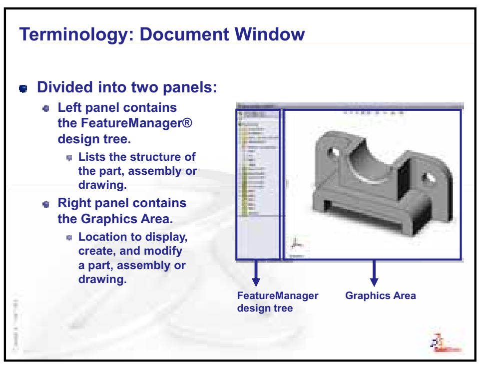


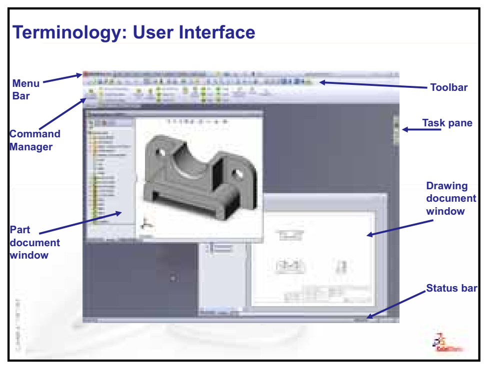
Sketched Features & Operation Features

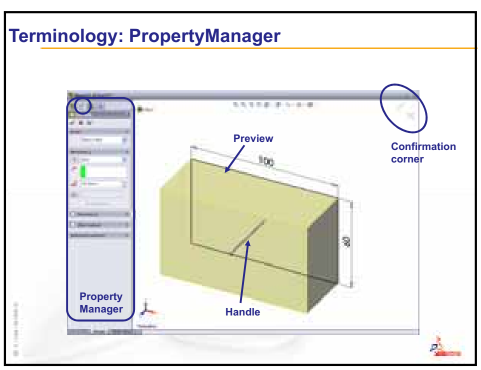
- Sketched Features
  - Shape features have sketches.
  - Sketched features are built from 2D profiles.
- Operation Features
  - Operation features do not have sketches.
  - Applied directly to the work piece by selecting edges or faces.

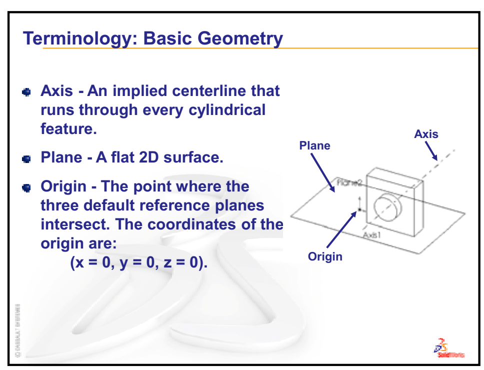

To Create an Extruded Base Feature:
1. Select a sketch plane.
2. Sketch a 2D profile.
3. Extrude the sketch perpendicular to sketch plane.
Select the sketch plane
Sketch the 2D profile
Extrude the sketch
Resulting base feature


To Create a Revolved Base Feature:
1. Select a sketch plane.
2. Sketch a 2D profile.
3. Sketch a centerline (optional).
4. Revolve the sketch around a sketch line or centerline.
Centerline (optional)


Terminology: Document Window
Divided into two panels:
Left panel contains the FeatureManager® design tree.
Lists the structure of the part, assembly or drawing.
Right panel contains the Graphics Area.
Location to display, create, and modify a part, assembly or drawing.
FeatureManager design tree
Graphics Area

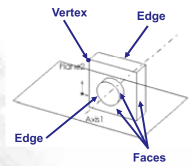

Terminology: User Interface
Menu Bar
Toolbar
Command Manager
Task pane
Drawing document window
Part document window
Status bar


Terminology: PropertyManager
Preview
Confirmation corner
Property Manager
Handle


Terminology: Basic Geometry
Axis - An implied centerline that runs through every cylindrical feature.
Plane - A flat 2D surface.
Origin - The point where the three default reference planes intersect. The coordinates of the origin are:
(x = 0, y = 0, z = 0).
Axis
Plane
Origin

Terminology: Basic Geometry

- Face – The surface or "skin" of a part. Faces can be flat or curved.
- Edge – The boundary of a face. Edges can be straight or curved.
- Vertex – The corner where edges meet.



Features and Commands

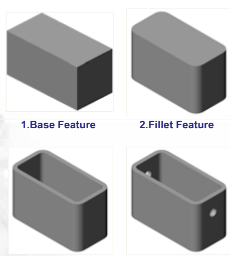
Base feature

- The Base feature is the first feature that is created.
- The Base feature is the foundation of the part.
- The Base feature geometry for the box is an extrusion.
- The extrusion is named Extrude1.

Features and Commands

Features used to build the *box* are:

- Extruded Base feature
- Fillet feature
- Shell feature
- Extruded Cut feature

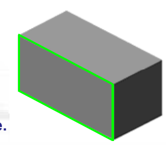


1.Base Feature
2.Fillet Feature
3.Shell Feature
4.Cut Feature

Features and Commands

To create the extruded base feature for the *box*:

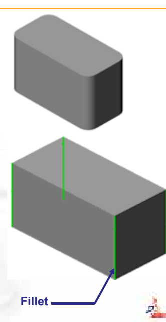
- Sketch a rectangular profile on a 2D plane.
- Extrude the sketch.
- By default extrusions are perpendicular to the sketch plane.



Features and Commands

Fillet feature

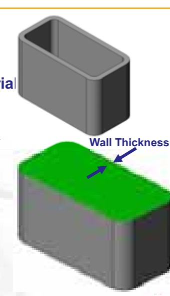
- The fillet feature rounds the edges or faces of a part.
- Select the edges to be rounded. Selecting a face rounds all the edges of that face.
- Specify the fillet radius.



Features and Commands

Shell feature

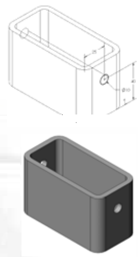
- The shell feature removes material from the selected face.
- Using the shell feature creates a hollow box from a solid box.
- Specify the wall thickness for the shell feature.



Features and Commands

To create the extruded cut feature for the *box*:

- Sketch the 2D circular profile.
- Extrude the 2D Sketch profile perpendicular to the sketch plane.
- Enter Through All for the end condition.
- The cut penetrates through the entire part.

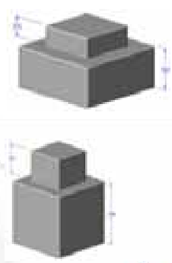


Dimensions and Geometric Relationships

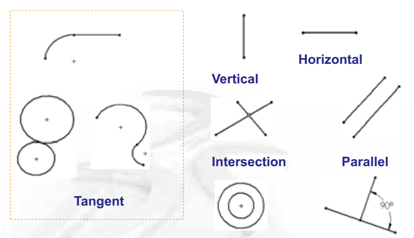
- Specify dimensions and geometric relationships between features and sketches.
- Dimensions change the size and shape of the part.
- Mathematical relationships between dimensions can be controlled by equations.
- Geometric relationships are the rules that control the behavior of sketch geometry.
- Geometric relationships help capture design intent.

Dimensions

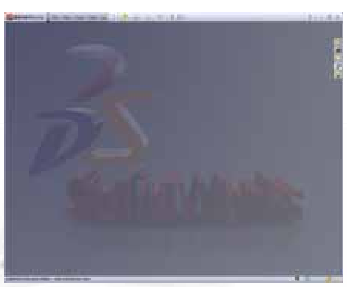
- Dimensions
  - Base depth = 50 mm
  - Boss depth = 25 mm
- Mathematical relationship
  - Boss depth = Base depth ÷ 2



Geometric Relationships

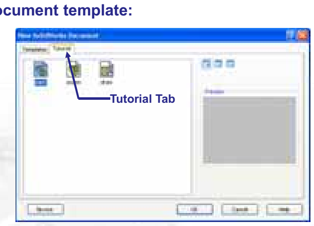


The SolidWorks Window



Creating New Files Using Templates

- Click New on the Standard toolbar.
- Select a document template:
  - Part
  - Assembly
  - Drawing

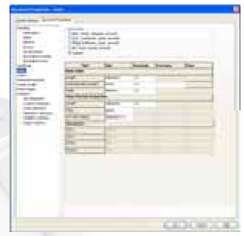


Document Templates

- Document Templates control the units, grid, text, and other settings for the model.
- The Tutorial document templates are required to complete the exercises in the *Online Tutorials*.
- The templates are located in the Tutorial tab on the <u>New SolidWorks Document</u> dialog box.
- Document properties are saved in templates.

Document Properties

- Accessed through the Tools, Options menu.
- Control settings like:
  - Units: English (inches) or Metric (millimeters)
  - Grid/Snap Settings
  - Colors, Material Properties and Image Quality

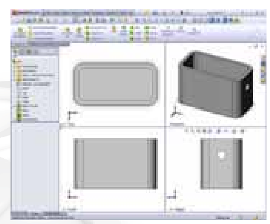


System Options

- Accessed through the Tools, Options menu.
- Allow you to customize your work environment.
- System options control:
  - File locations
  - Performance
  - Spin box increments

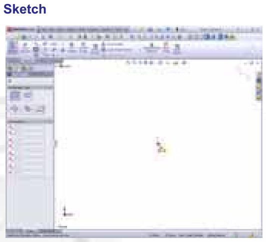
Multiple Views of a Document

- Click the view pop-up menu.
- Select an icon. The viewport icons include:
  - Single View
  - Two View (horizontal and vertical)
  - Four View



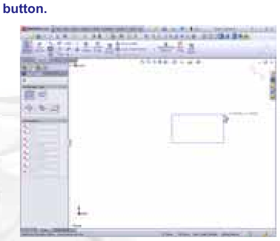
Creating a 2D Sketch

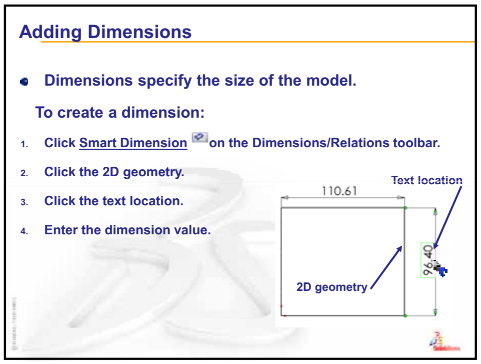
1. Click <u>Sketch</u> on the Sketch toolbar.
2. Select the Front plane as a sketch plane.
3. Click <u>Rectangle</u> on the Sketch Tools toolbar.
4. Move the pointer to the Sketch Origin.



Creating a 2D Sketch

5. Click the left mouse button.
6. Drag the pointer up and to the right.
7. Click the left mouse button again.



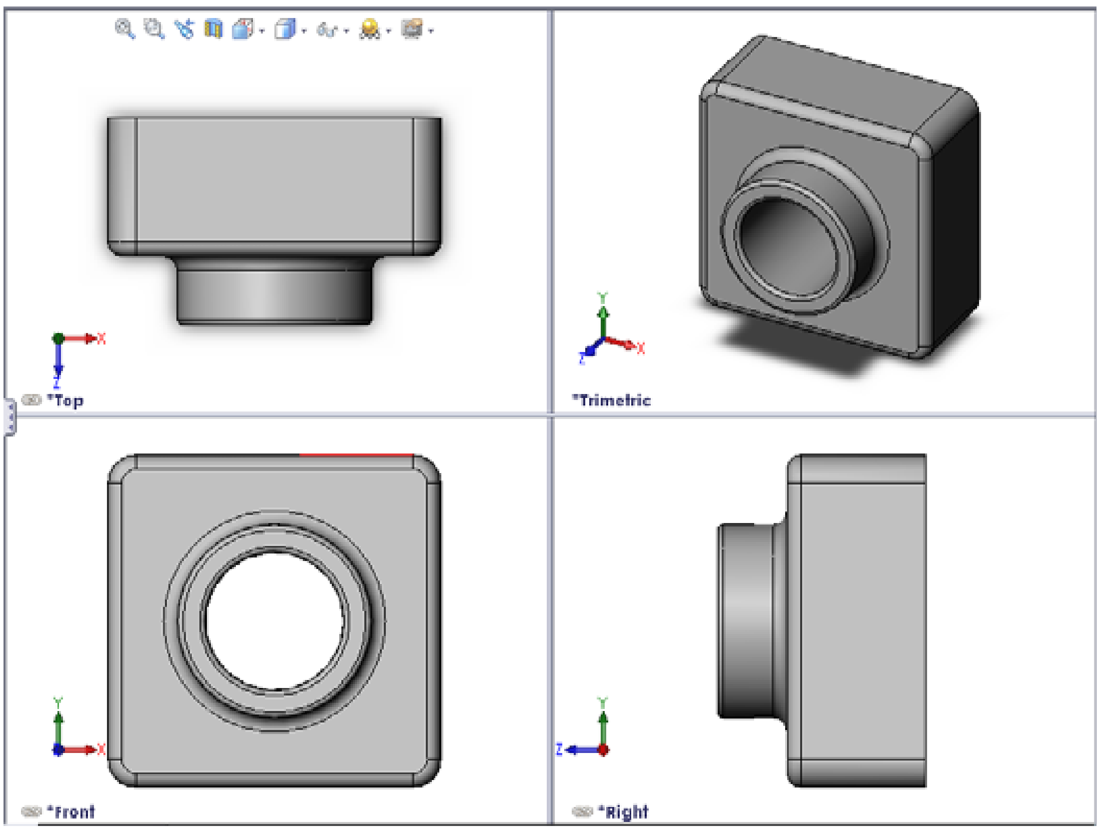

Adding Dimensions
Dimensions specify the size of the model.
To create a dimension:
1. Click Smart Dimension on the Dimensions/Relations toolbar.
2. Click the 2D geometry.
3. Click the text location.
4. Enter the dimension value.
Text location
110.61
96.40
2D geometry

3

Lesson 3: The 40-Minute Running Start

Goals of This Lesson

Create and modify the following part:



Before Beginning This Lesson

Complete Lesson 2: Basic Functionality.

Resources for This Lesson

This lesson plan corresponds to *Getting Started: Lesson 1 – Parts* in the SolidWorks Tutorials. For more information, see “SolidWorks Tutorials” on page v.

SolidWorks education suite contains 80 tutorials in engineering design, sustainability, simulation and analysis.

Review of Lesson 2: Basic Functionality

Questions for Discussion

1 A SolidWorks 3D model consists of three documents. Name the three documents.

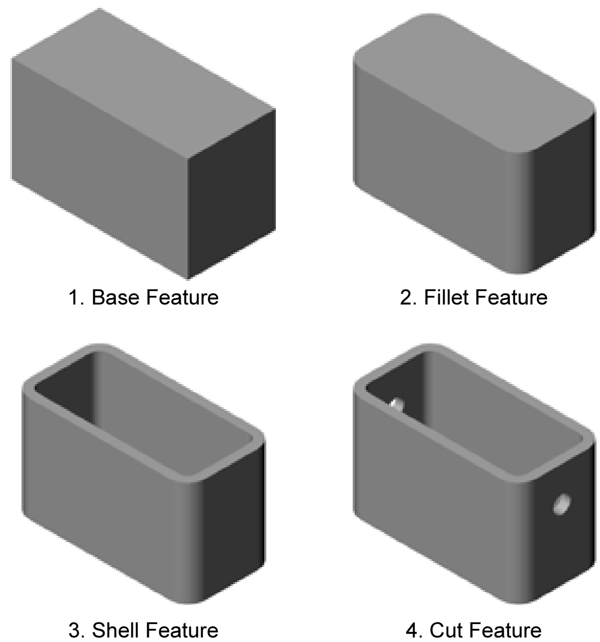
Answer: Part, Assembly and Drawing.

2 Parts are built from features. What are features?

Answer: Features are the shapes (bosses, cuts and holes) and the operations (fillets, chamfers and shells) that you use to build a part.

3 Name the features that are used to create the `box` in Lesson 1.

Answer: Extruded Boss, Fillet, Shell, and Extruded Cut.



4 What is the base feature of the box?

Answer: The base feature is the first feature of the box. The base feature is the foundation of the part. The base feature geometry for the box is an extrusion. The extrusion is named `Extrude1`. The base feature represents the general shape of the box.

5 Why did you use the Fillet feature?

Answer: The fillet feature rounds the sharp edges and faces. The result of using the fillet feature created the rounded edges of the box.

6 Why did you use the Shell feature?

Answer: The shell feature removes material. The result of using the shell feature created a hollow block from a solid block.

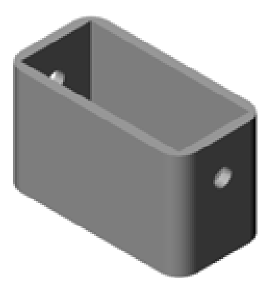
7 How do you create the Base feature?

Answer: To create a solid Base feature:

- Sketch a rectangular profile on a flat 2D plane.
- Extrude the profile perpendicular to the sketch plane.

8 What would have happened if the Shell feature was created before the Fillet feature?

Answer: The inside corners of the box would be sharp instead of rounded.



Outline of Lesson 3

- ❑ In Class Discussion — Base Features
- ❑ Active Learning Exercise — Create a Part
- ❑ Exercises and Projects — Modifying the Part
  - Converting Dimensions
  - Calculating the Modification
  - Modifying the Part
  - Calculating Material Volume
  - Calculating the Volume of the Base Feature
- ❑ Exercises and Projects — Creating a CD Jewel Case and Storage Box
  - Measuring the CD Jewel Case
  - Rough Sketch of the Jewel Case
  - Calculate the Overall Case Capacity
  - Calculate the Outside Measurements of the CD Storage Box
  - Creating the CD Jewel Case and Storage Box
- ❑ More to Explore — Modeling More Parts
- ❑ Lesson Summary

Competencies for Lesson 3

Students develop the following competencies in this lesson:

- ❑ **Engineering**: Utilize 3D features to create a 3D part. Create a pencil sketch of a profile for chalk and an eraser.
- ❑ **Technology**: Work with a common music/software case and determine the size of a CD container.
- ❑ **Math**: Apply concentric relations (same center) between circles. Understand conversion from millimeters to inches in an applied project. Apply width, height, and depth to a right prism (box).
- ❑ **Science**: Calculate volume of a right prism (box).

In Class Discussion — Base Features

- ❑ Select a simple object in the classroom, a piece of chalk or board eraser.
- ❑ Ask the students to describe the Base feature of these objects.
- ❑ How would you create the additional features for these objects?

<u>Answer</u>

Chalk:

- ❑ Sketch a circular 2D profile.
- ❑ Extrude the 2D profile. The extruded 2D profile creates the Base feature. The Base feature is named `Extrude1`.
- ❑ Select the circular edge on the Base feature. Create a Fillet feature. The Fillet feature removes sharp edges.

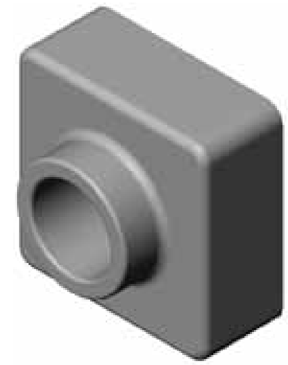
Note: You would probably not want to use the Fillet feature for a new piece of chalk.

Board Eraser:

- ❑ Sketch a rectangular 2D profile.
- ❑ Extrude the 2D profile. The extruded 2D profile creates the Base feature.
- ❑ Select the 4 corners on the Base feature. Create a Fillet feature to remove the sharp edges.

Active Learning Exercises — Create a Part

Follow the instructions in *Getting Started: Lesson 1 – Parts* of the SolidWorks Tutorial. In this lesson you will create the part shown at the right. The part name is `Tutor1.sldprt`.



Lesson 3 — 5 Minute Assessment — Answer Key

Name: ________________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What features did you use to create `Tutor1`?

Answer: Extruded Boss, Fillet, Shell and Extruded Cut.

2 What does the Fillet feature do?

Answer: The Fillet feature rounds sharp edges and faces.

3 What does the Shell feature do?

Answer: The Shell feature removes material from the selected face.

4 Name three view commands in SolidWorks.

Answer: Zoom to Fit, Rotate View, and Pan.

5 Where are the display buttons located?

Answer: The display buttons are located on the View toolbar.

6 Name the three SolidWorks default planes.

Answer: `Front`, `Top`, and `Right`.

7 The SolidWorks default planes correspond to what principle drawing views?

Answer:

- `Front` = Front or Back view
- `Top` = Top or Bottom view
- `Right` = Right or Left view

8 True or False. In a fully defined sketch, geometry is displayed in black.

Answer: True.

9 True or False. It is possible to make a feature using an over defined sketch.

Answer: False.

10 Name the primary drawing views used to display a model.

Answer: Top, Front, Right and Isometric views.

Lesson 3 — 5 Minute Assessment **REPRODUCIBLE**

Name: ______________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What features did you use to create `Tutor1`?

____

2 What does the Fillet feature do?

____

3 What does the Shell feature do?

____

4 Name three view commands in SolidWorks.

____

5 Where are the display buttons located?

____

6 Name the three SolidWorks default planes.

____

7 The SolidWorks default planes correspond to what principle drawing views?

____

____

8 True or False. In a fully defined sketch, geometry is displayed in black.

____

9 True or False. It is possible to make a feature using an over defined sketch.

____

10 Name the primary drawing views used to display a model.

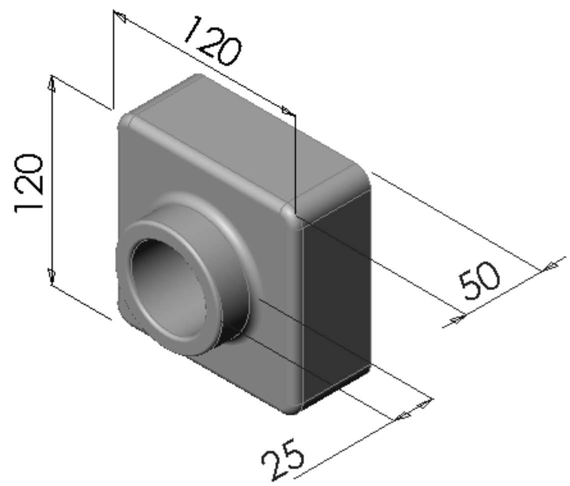
____

Exercises and Projects — Modifying the Part

Task 1 — Converting Dimensions

The design for `Tutor1` was created in Europe. `Tutor1` will be manufactured in the US. Convert the overall dimensions of `Tutor1` from millimeters to inches.

Note: Units are in Millimeters



Given:

- ❑ Conversion: 25.4 mm = 1 inch
- ❑ Base width = 120 mm
- ❑ Base height = 120 mm
- ❑ Base depth = 50 mm
- ❑ Boss depth = 25 mm

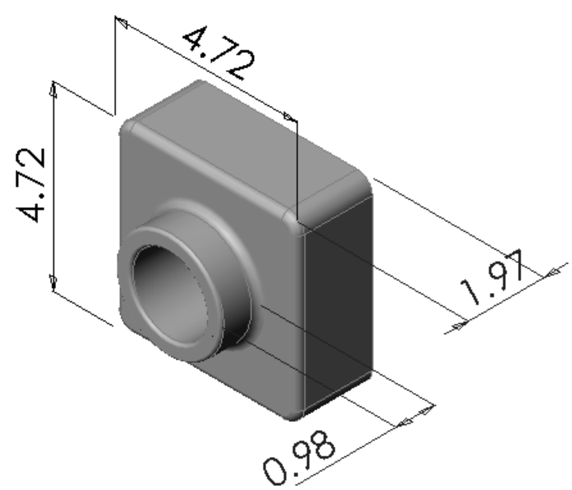
Answer:

- ❑ Overall depth = Base depth + Boss depth
  Overall depth = 1.97” + 0.98” = 2.95”
- ❑ Overall dimensions = Base width **x** Base height **x** Depth
  Overall dimensions = 4.72” **x** 4.72” **x** 2.95”

In Class Demonstration:

SolidWorks supports both metric and English units. Demonstrate the software conversion from metric to English units.

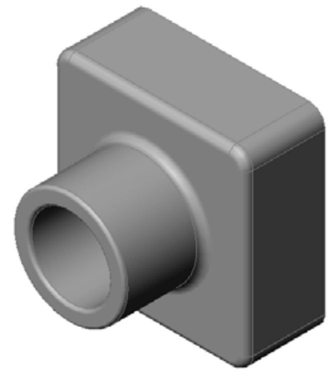
Note: Units are in Inches



1. Click **Tools, Options**.
2. Click the **Document Properties** tab.
3. Click **Units**.
4. Change **Unit system** to **Custom** and select **inches** for **Length**. Click **OK**.
5. Double-click the `Tutor1` features to display the dimensions.
   - Base width = 4.72”
   - Base height = 4.72”
   - Base depth = 1.97”
   - Boss depth = 0.98”
6. Change the **Length** of the part back to **Millimeters** for the next task.

Task 2 — Calculating the Modification

The current overall depth of `Tutor1` is 75 mm. Your customer requires a design change. The new required overall depth is 100 mm. The Base depth must remain fixed at 50 mm. Calculate the new Boss depth.



Given:

- ❑ New overall depth = 100 mm
- ❑ Base depth = 50 mm

Answer:

- ❑ Overall depth = Base depth + Boss depth
  Boss depth = Overall depth - Base depth
  Boss depth = 100mm - 50 mm
  Boss depth = 50 mm

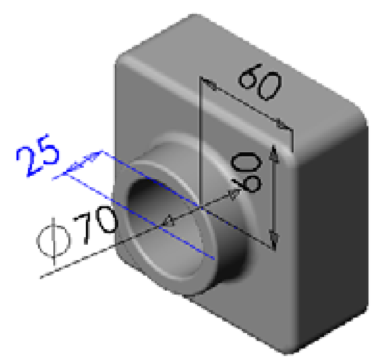
Task 3 — Modifying the Part

Using SolidWorks, modify `Tutor1` to meet the customer's requirements. Change the depth of the Boss feature such that the overall depth of the part equals 100 mm.

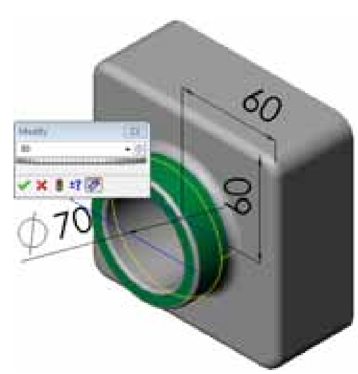
Save the modified part under a different name.

Answer:

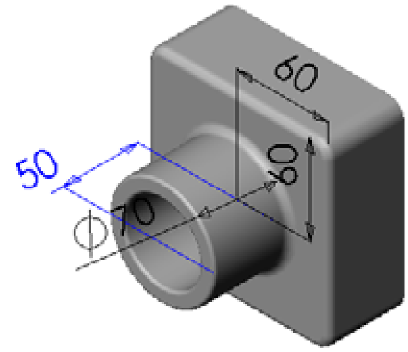
1 Double-click on the `Extrude2` feature.



2 Double-click on the **25 mm** depth dimension.
3 In the **Modify** dialog, enter the value **50mm**.
4 Press **Enter**.



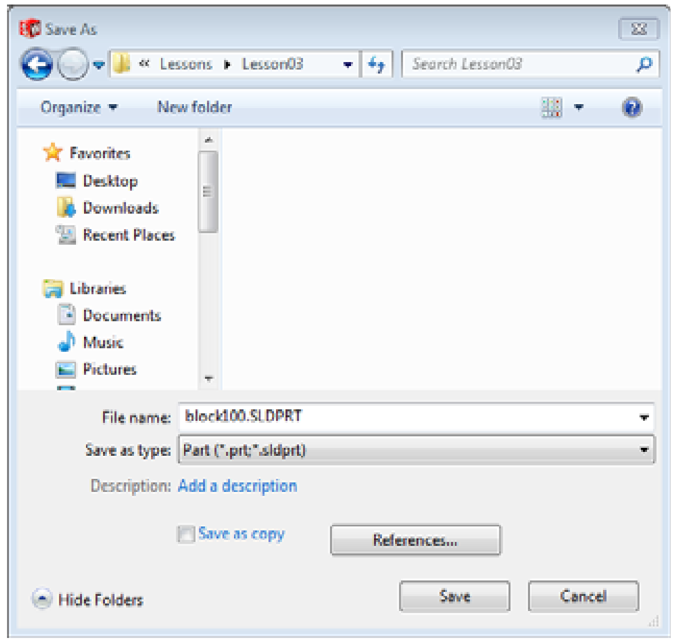
5 Click **Rebuild**.



6 Click **File, Save As** to create `block100`.

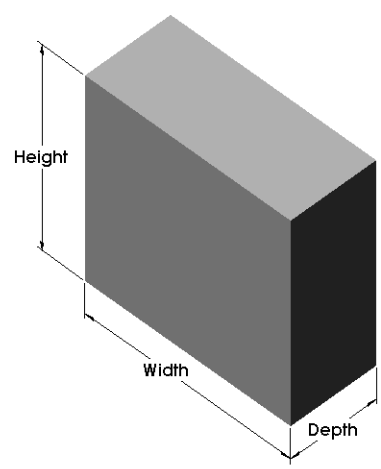
When you use **File, Save As**, you save a copy of the document with a new name or path. You can create a new folder in the **Save As** dialog box if needed. After you use **File, Save As**, you are working in the *new* document. The original document is closed without saving.

If you click the **Save as copy** check box you will save a copy of the document, with a new name, *without* replacing the active document. You continue to work in the original document.



Task 4 — Calculating Material Volume

Material volume is an important calculation for designing and manufacturing parts. Calculate the volume of the Base feature in mm^3 for `Tutor1`.



Answer:

- Volume = Width **x** Height **x** Depth
  Volume = 120mm **x** 120mm **x** 50mm = 720,000 mm^3

Task 5 — Calculating the Volume of the Base feature

Calculate the volume of the Base feature in cm^3.

Given:

- 1cm = 10mm

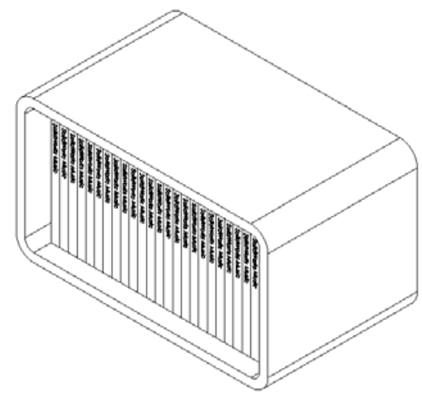
Answer:

- Volume = Width **x** Height **x** Depth
  Volume = 12cm **x** 12cm **x** 5cm = 720cm^3

Exercises and Projects — Creating a CD Jewel Case and Storage Box

You are part of a design team. The project manager has provided the following design criteria for a CD storage box:

- The CD storage box is constructed of a polymer (plastic) material.
- The storage box must hold 25 CD jewel cases.
- The title of the CD must be visible when the jewel case is positioned in the storage box.
- The wall thickness of the storage box is 1cm.
- On each side of the storage box, there must be 1cm clearance between the jewel case and the inside of the box.
- There must be 2cm clearance between the top of the CD cases and the inside of the storage box.
- There must be 2cm clearance between the jewel cases and the front of the storage box.

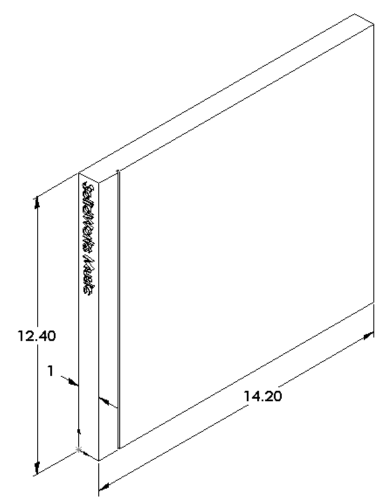


Task 1 — Measuring the CD Jewel Case

Measure the width, height, and depth of one CD jewel case. What are the measurements in centimeters?

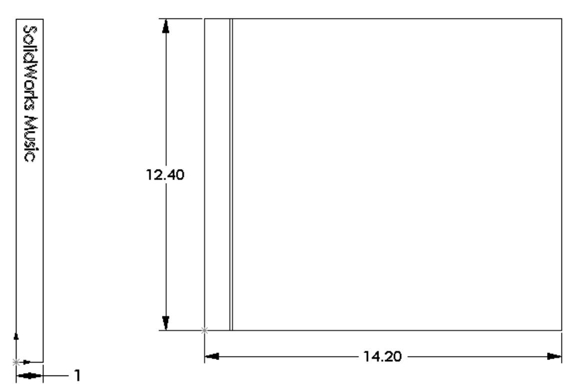
Answer:

Approximately 14.2cm **x** 12.4cm **x** 1cm



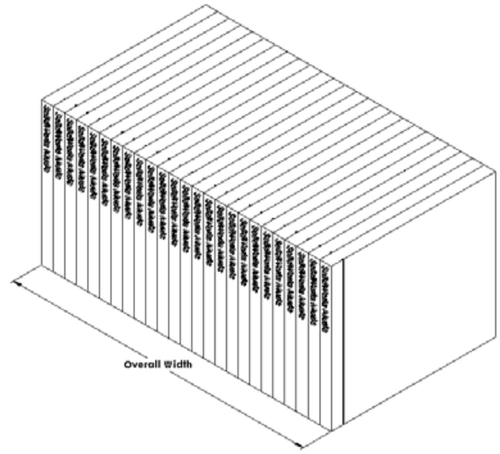
Task 2 — Rough Sketch of the Jewel Case

Using paper and pencil, manually sketch the CD jewel case. Label the dimensions.



Task 3 — Calculate the Overall Case Capacity

Calculate the overall size of 25 stacked CD jewel cases. Record the overall width, height and depth.



Given:

- CD jewel case width = 1cm
- CD jewel case height = 12.4cm
- CD jewel case depth = 14.2cm

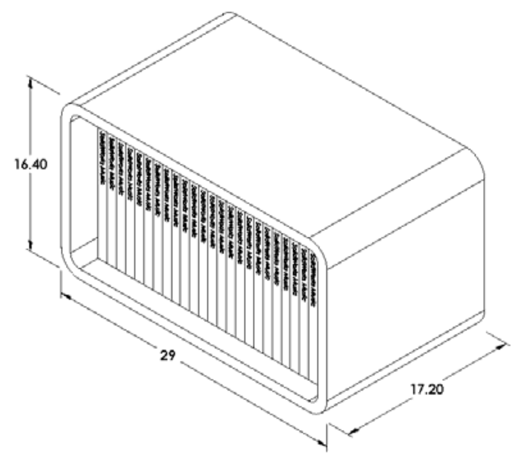
Answer:

- Overall width of 25 CD jewel cases = 25 **x** 1cm = 25cm
- Overall size for 25 CD jewel cases = Overall width **x** CD case height **x** CD case depth
  Overall size for 25 CD jewel cases = 25cm **x** 12.4cm **x** 14.2cm

Task 4 — Calculate the Outside Measurements of the CD Storage Box

Calculate the overall *outside* measurements of the CD storage box. The box requires a clearance to insert and position the CD jewel cases. Add a 2cm clearance to the overall width (1cm on each side) and 2cm to the height. The wall thickness is equal to 1cm.

Answer:



- Clearance = 2cm
- Wall thickness = 1cm
- Wall thickness is applied to both sides of the width and height dimensions. Wall thickness is applied to one side of the depth dimension.
- CD storage box width = Overall width of 25 CD jewel cases + Clearance + Wall thickness + Wall thickness
  CD storage box width = 25cm + 2cm + 1cm + 1cm = 29cm
- CD storage box height = CD case height + Clearance + Wall thickness + Wall thickness
  CD storage box height = 12.4cm + 2cm + 1cm + 1cm = 16.4cm
- CD storage box depth = CD case depth + Clearance + Wall thickness
  CD storage box depth = 14.2cm + 2cm + 1cm = 17.2cm
- Overall size CD storage box = Storage box width **x** Storage box height **x** Storage box depth
  Overall size CD storage box = 29cm **x** 16.4cm **x** 17.2cm

Task 5 — Creating the CD Jewel Case and Storage Box

Create two parts using SolidWorks.

- ❑ Model a CD jewel case. You should use the dimensions you obtained in Task 1. Name the part `CD case`.

Note: A real CD jewel case is an assembly of several parts. For this exercise, you will make a simplified representation of a jewel case. It will be a single part that represents the overall outside dimensions of the jewel case.

- ❑ Design a storage box to hold 25 CD jewel cases. The fillets are 2 cm. Name the part `storagebox`.
- ❑ Save both parts. You will use them to make an assembly at the end of the next lesson.

More to Explore — Modeling More Parts

Description

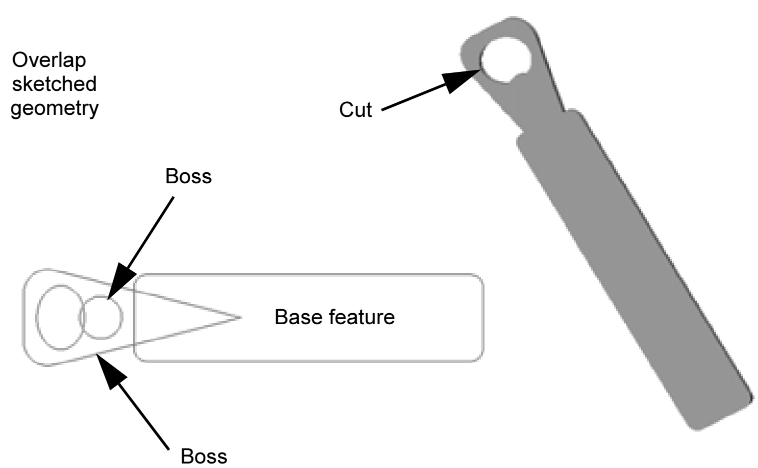
Look at the following examples. The files are in the `Lessons\Lesson03` folder in `SolidWorks Teacher Tools`. There are at least three features in each example. Identify the 2D Sketch tools used to create the shapes. You should:

- ❑ Consider how the part should be broken down into individual features.
- ❑ Focus on creating sketches that represent the desired shape. You do not need to use dimensions. Concentrate on the shape.
- ❑ Also, experiment and create your own designs.

Note: Each new sketch should overlap an existing feature.

Task 1 — Explore

`bottleopener.sldprt`



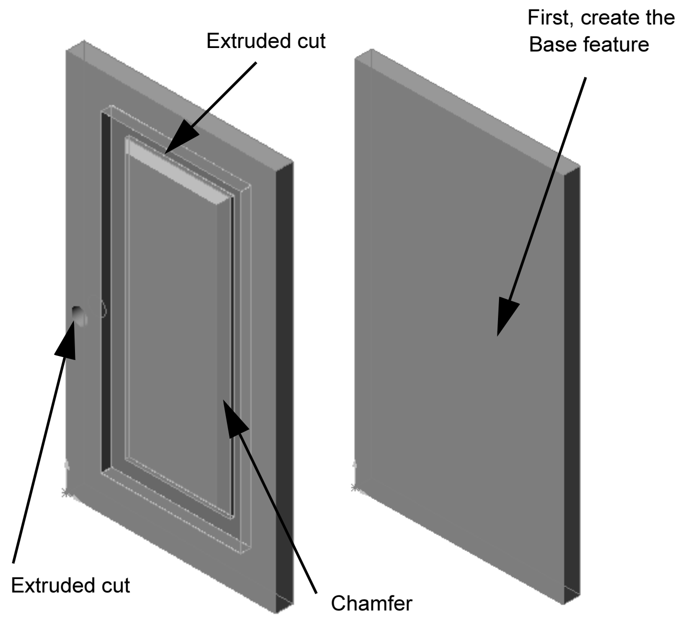
Answer:

- ❑ The features used to create the bottle opener are:
  - Base feature - Sketch a rectangle with rounded corners to create the handle.
  - Extruded boss - Sketch a triangle with rounded corners to create the head.
  - Extruded cut - Sketch an ellipse to create the hole.
  - Extruded boss - Sketch a circle to create the hook tab.

Task 2 — Explore `door.sldprt`

Answer:

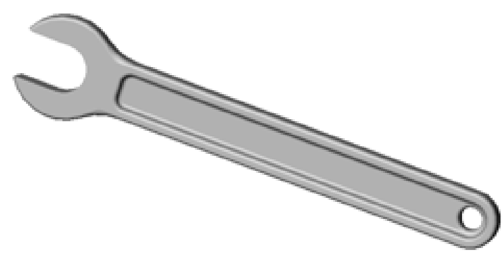
- The features used to create the door are:
  - Base feature - Sketch a rectangle to create the door.
  - Extruded cut - Sketch a circle to create the door hole.
  - Extruded cut - Sketch two rectangles to create the panel.
  - Chamfer - Select the middle face.



Task 3 — Explore `wrench.sldprt`

Answer:

- The features used to create the wrench are:
  - Base feature - Sketch a rectangle then round one end to create the handle.
  - Shell - Select the top face to create the recess in the handle.
  - Extruded boss - Sketch a circle to create the head.
  - Extruded cut - Sketch a slot with one rounded end to create the opening.
  - Extruded cut - Sketch the circle to create the hole in the handle.
  - Fillet - Select faces and edges to round the handle and outside edges of the head.
  - Chamfer - Select the two leading inside edges of the opening.



Lesson 3 Quiz — Answer Key

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you begin a new part document?

Answer: Click the **New** icon. Select a part template.

2 How do you open a sketch?

Answer: Select the desired sketch plane. Click the **Sketch** icon on the Sketch toolbar.

3 What is the Base feature?

Answer: The base feature is the first feature of a part. It is the foundation of the part.

4 What color is the geometry of a fully defined sketch?

Answer: Black

5 How can you change a dimension value?

Answer: Double-click on the dimension. Enter the new value in the **Modify** dialog box.

6 What is the difference between an extruded boss feature and an extruded cut feature?

Answer: The boss feature adds material. The cut feature removes material.

7 What is a fillet feature?

Answer: The Fillet feature rounds the edges or faces of a part at a specified radius.

8 What is a shell feature?

Answer: The shell feature removes material by hollowing out the part.

9 Name four types of geometric relations you can add to a sketch?

Answer: The Geometric Relations you can add to a Sketch are: horizontal, vertical, collinear, coradial, perpendicular, parallel, tangent, concentric, midpoint, intersection, coincident, equal, symmetric, fix, pierce and merge points.

10 What is a section view?

Answer: A section view shows the part as if it were cut into two pieces. This displays the internal structure of the model.

11 How do you create multiple views of a part?

Answer: To create multiple views of a part, drag one or both of the split boxes at the corners of the window to create panes. Adjust the pane size. Change the view orientation in each pane.

Lesson 3 Quiz

REPRODUCIBLE

Name: ______________________________ Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you begin a new part document? ____________________________________

____

2 How do you open a sketch? ____

____

3 What is the Base feature? ____

____

4 What color is the geometry of a fully defined sketch? ________________________

____

5 How can you change a dimension value? __________________________________

____

6 What is the difference between an extruded boss feature and an extruded cut feature?

____

7 What is a fillet feature? ____

____

8 What is a shell feature? ____

____

9 Name four types of geometric relations you can add to a sketch? ______________

____

10 What is a section view? ____

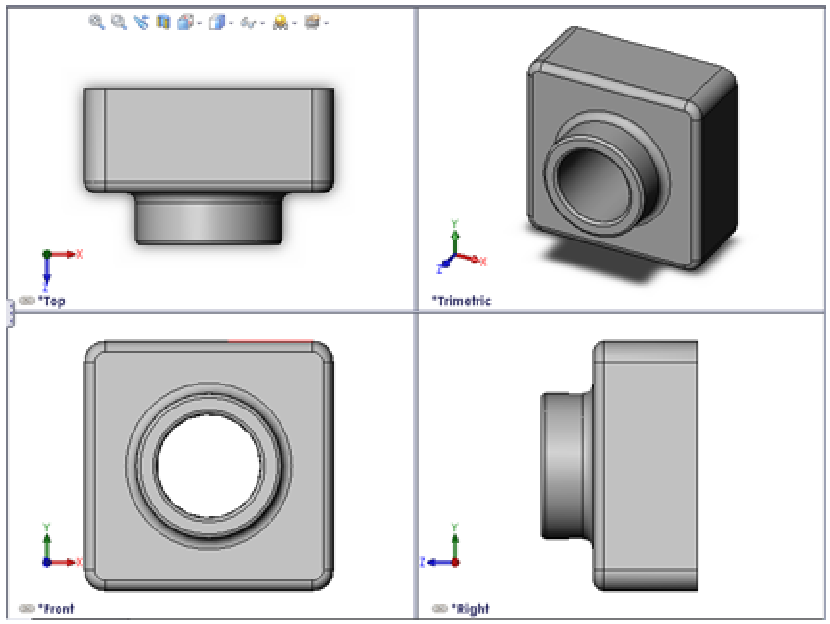
____

11 How do you create multiple views of a part? ______________________________

____

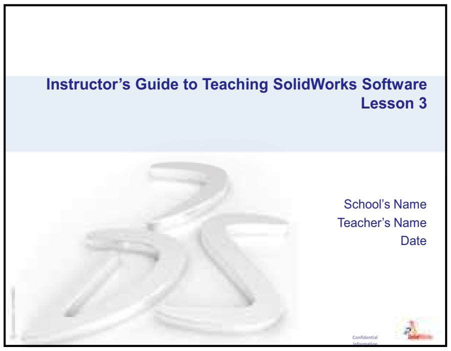
Lesson Summary

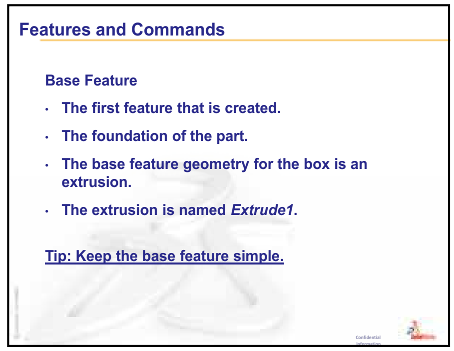
- ❑ The Base Feature is the first feature that is created — the foundation of the part.
- ❑ The Base Feature is the workpiece to which everything else is attached.
- ❑ You can create an Extruded Base Feature by selecting a sketch plane and extruding the sketch perpendicular to sketch plane.
- ❑ A Shell Feature creates a hollow block from a solid block.
- ❑ The views most commonly used to describe a part are:
  Top
  Front
  Right
  Isometric or
  Trimetric

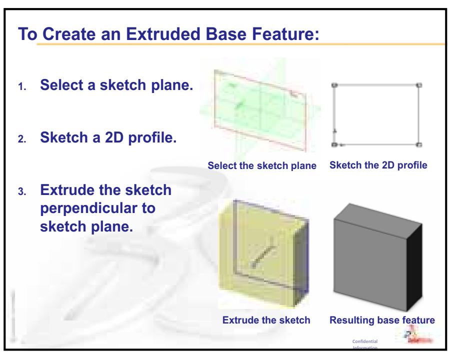


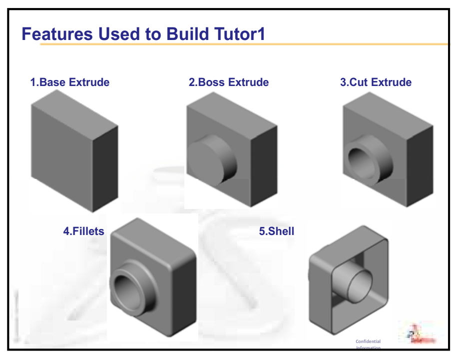
Thumbnail Images of PowerPoint Slides

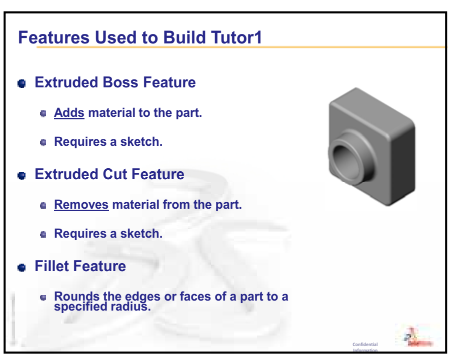
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

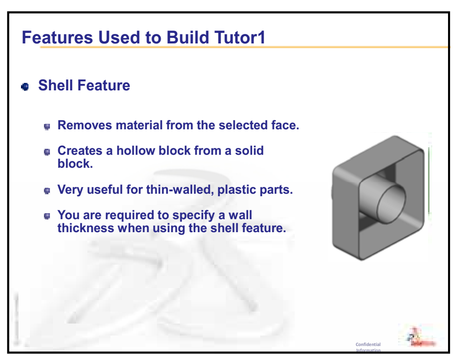


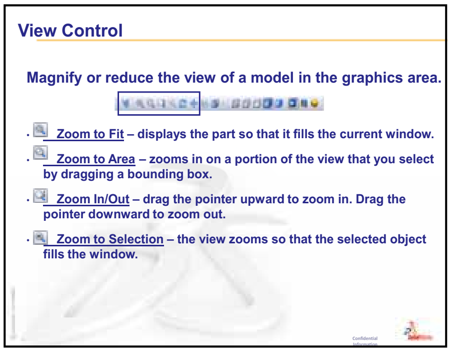


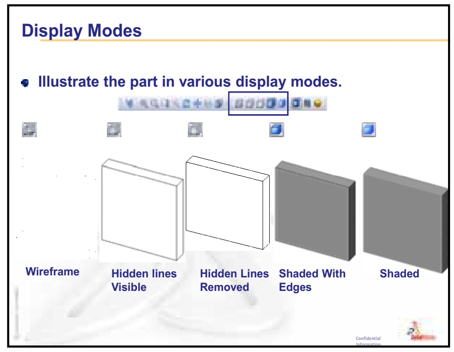


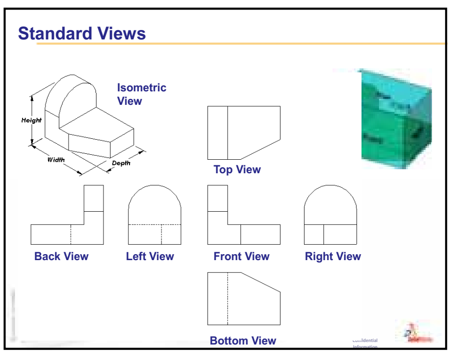


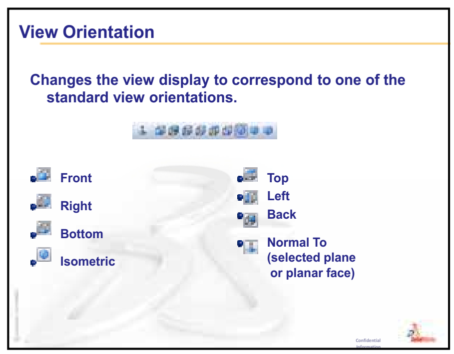


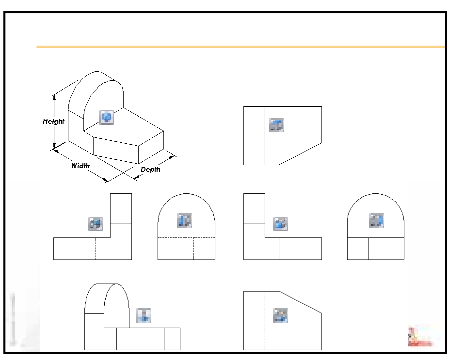


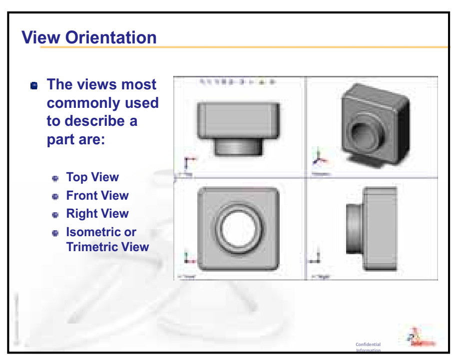

View Control
Magnify or reduce the view of a model in the graphics area.
Zoom to Fit – displays the part so that it fills the current window.
Zoom to Area – zooms in on a portion of the view that you select by dragging a bounding box.
Zoom In/Out – drag the pointer upward to zoom in. Drag the pointer downward to zoom out.
Zoom to Selection – the view zooms so that the selected object fills the window.

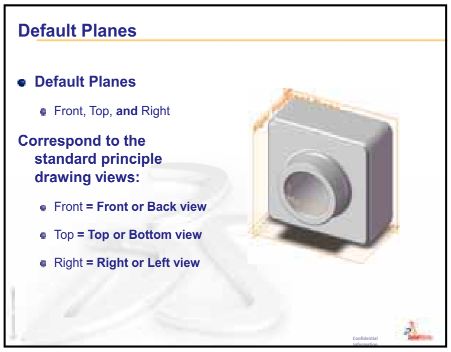

Display Modes
Illustrate the part in various display modes.
Wireframe
Hidden lines Visible
Hidden Lines Removed
Shaded With Edges
Shaded

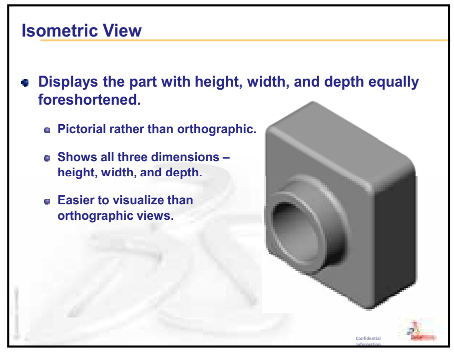

Standard Views
Isometric View
Height
Width
Depth
Top View
Back View
Left View
Front View
Right View
Bottom View

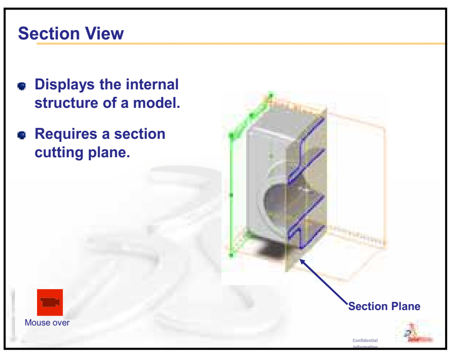

View Orientation
Changes the view display to correspond to one of the standard view orientations.
Front
Right
Bottom
Isometric
Top
Left
Back
Normal To (selected plane or planar face)

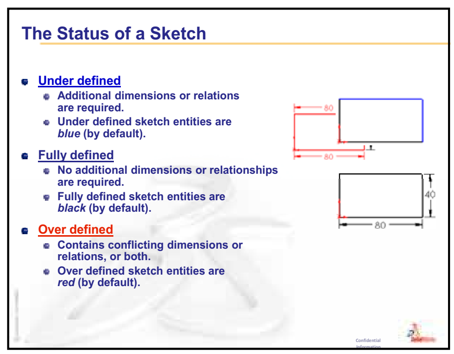

Height
Width
Depth

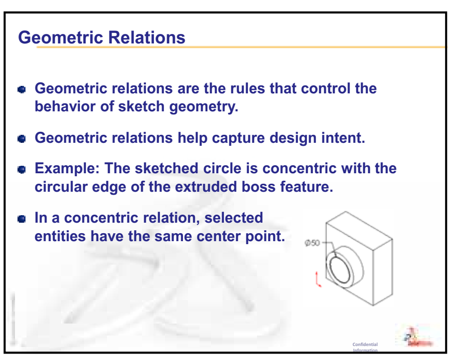

View Orientation
The views most commonly used to describe a part are:
Top View
Front View
Right View
Isometric or Trimetric View

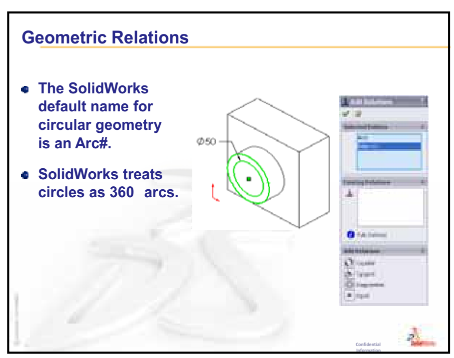

Default Planes
Default Planes
Front, Top, and Right
Correspond to the standard principle drawing views:
Front = Front or Back view
Top = Top or Bottom view
Right = Right or Left view


Isometric View
Displays the part with height, width, and depth equally foreshortened.
Pictorial rather than orthographic.
Shows all three dimensions – height, width, and depth.
Easier to visualize than orthographic views.


Section View
Displays the internal structure of a model.
Requires a section cutting plane.
Section Plane
Mouse over


The Status of a Sketch
Under defined
Additional dimensions or relations are required.
Under defined sketch entities are blue (by default).
Fully defined
No additional dimensions or relationships are required.
Fully defined sketch entities are black (by default).
Over defined
Contains conflicting dimensions or relations, or both.
Over defined sketch entities are red (by default).
80
40


Geometric Relations
Geometric relations are the rules that control the behavior of sketch geometry.
Geometric relations help capture design intent.
Example: The sketched circle is concentric with the circular edge of the extruded boss feature.
In a concentric relation, selected entities have the same center point.
Ø50

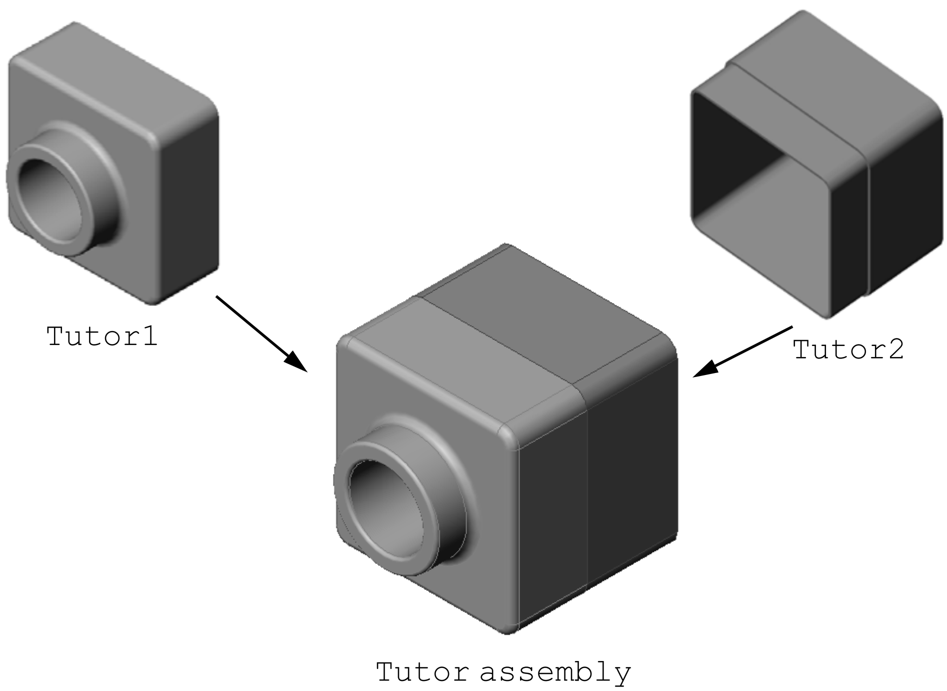

Geometric Relations
The SolidWorks default name for circular geometry is an Arc#.
SolidWorks treats circles as 360 arcs.
Ø50

4

Lesson 4: Assembly Basics

Goals of This Lesson

- Understand how parts and assemblies are related.
- Create and modify the part `Tutor2` and create the `Tutor` assembly.



Before Beginning This Lesson

Complete the `tutor1` part in Lesson 3: The 40-Minute Running Start.

Resources for This Lesson

This lesson plan corresponds to *Getting Started: Lesson 2– Assemblies* in the SolidWorks Tutorials.

Additional information about assemblies can be found in the *Building Models: Assembly Mates* lesson in the SolidWorks Tutorials.

www.3dContentCentral.com contains 1000's of model files, industry supplier components, and multiple file formats.

Review of Lesson 3: The 40-Minute Running Start

Questions for Discussion

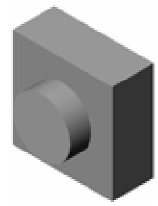
1 A SolidWorks 3D model consists of three documents. Name the three documents.

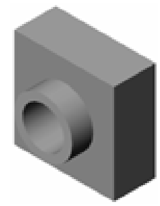
Answer: Part, Assembly and Drawing.

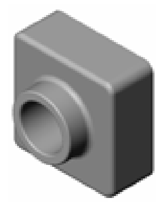
2 Name the features that were used to create `tutor1` in Lesson 3.

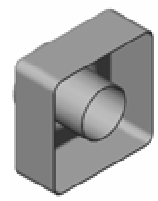
Answer: Review the PowerPoint slides in Lesson 3. The features are shown here.


1. Base Extrude

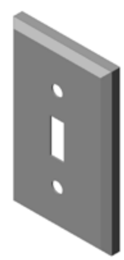

2. Boss Extrude

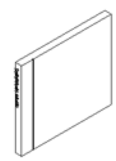

3. Cut Extrude

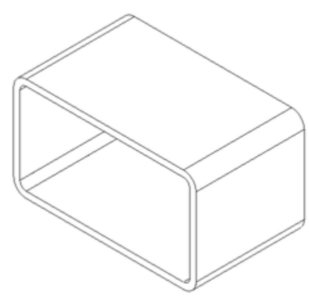

4. Fillets


5. Shell

3 Discuss any questions on the creation of the `switchplate`, `cdcase`, and `storagebox`.







Outline of Lesson 4

- ❑ In Class Discussion — Exploring an Assembly
- ❑ In Class Discussion — Size, Fit, and Function
- ❑ Active Learning Exercises — Creating an Assembly
- ❑ Exercises and Projects — Creating the Switchplate Assembly
  - Modifying Feature Size
  - Designing a Fastener
  - Creating an Assembly
- ❑ Exercises and Projects — Creating CD Storage Box Assembly
  - Component Patterns
- ❑ Exercises and Projects - Assembling a Mechanical Claw
  - Smart Mates
  - Circular Component Pattern
  - Dynamic Assembly Motion
- ❑ Lesson Summary

Competencies for Lesson 4

Students develop the following competencies in this lesson:

- ❑ **Engineering**: Evaluate the current design and incorporate design changes that result in an improved product. Review fastener selection based on strength, cost, material, appearance, and ease of assembly during installation.
- ❑ **Technology**: Review different materials and safety in the design of an assembly.
- ❑ **Math**: Apply angular measurements, axes, parallel, concentric and coincident faces, and linear patterns.
- ❑ **Science**: Develop a volume from a profile revolved around an axis.

In Class Discussion — Exploring an Assembly

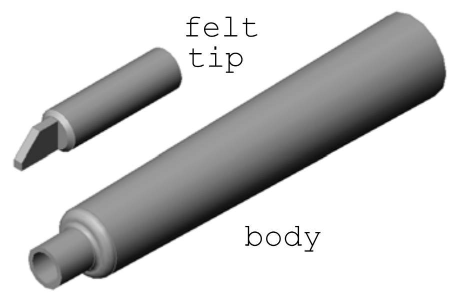
- ❑ Show your students a white board marker or highlighter.
- ❑ Ask the students to describe the marker in terms of features and components.

Answer

There are four visible major components on the marker. They are: `body`, `felt tip`, `end plug`, and `cap`.

Discussion

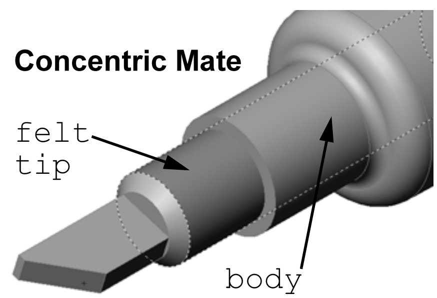
What are the mates required to complete the assembly between the `felt tip` and the `body`?



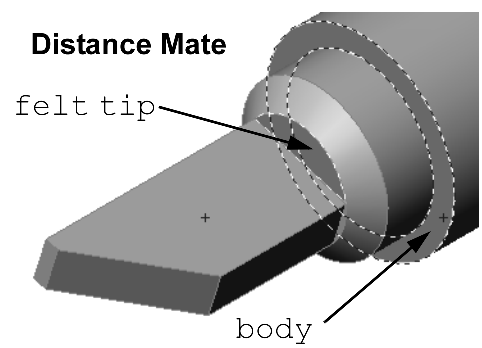
Answer

The assembly is named `Marker`. The `Marker` requires three mates to fully define the assembly. The three mates are:

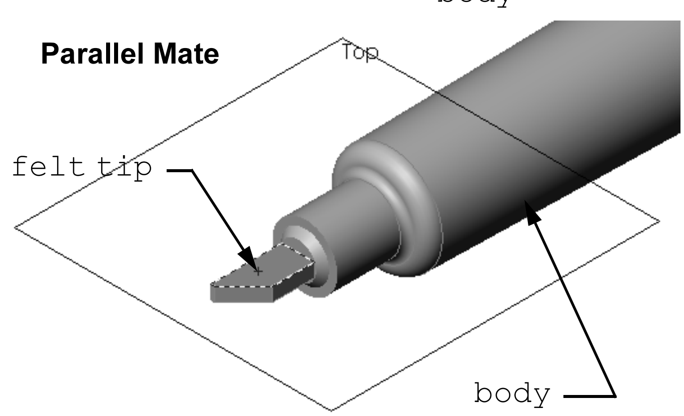
- ❑ **Concentric Mate** between a cylindrical face of the `body` and a cylindrical face of the `felt tip`.



- ❑ **Distance Mate** between the front face of the `body` and the flat front face of the `felt tip`.



- ❑ **Parallel Mate** between the `Top` plane of the `body` and the flat face of the `felt tip`. The `Marker` assembly is now fully defined.



Note: The completed assembly is in the `Lessons\Lesson04` folder in `SolidWorks Teacher Tools`.

In Class Discussion — Size, Fit, and Function

A 3.5mm fastener cannot be inserted into a 3.5mm hole without great difficulty. The 3.5mm dimension is a nominal dimension. The nominal dimension is approximately the size of the feature which corresponds to a common fraction or whole number. One example of a nominal dimension that your students might know is a wooden 2x4. A 2x4 is not 2 inches by 4 inches. It is $1^1/_2$ inches by $3^1/_2$ inches.

Tolerance is the difference between the maximum and minimum variation of a nominal dimension and the actual manufactured dimension. For example, a design might call for a 4mm hole. When the product is manufactured, the actual diameter of the hole will vary depending on many factors such as the method used to make the hole or tool wear. A dull drill makes a different size hole than a sharp one.

A designer must take tolerances into account when designing a product. For example, if the hole is at the small end of its tolerance range and the fastener that goes into the hole is at the large end of its tolerance range, will they still go together? This assembly relationship between a fastener and the hole is called fit. The fit is defined as the tightness or looseness between two components. There are three major types of fits:

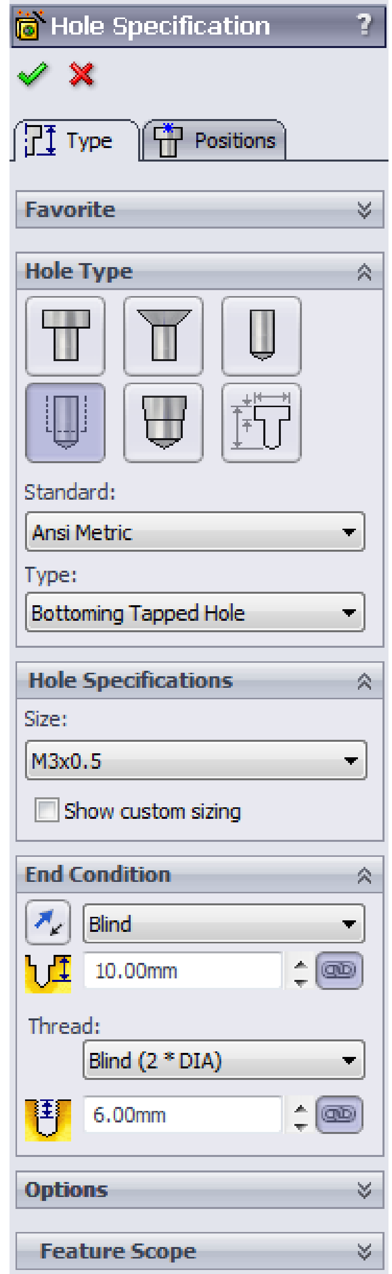
- Clearance fit – The shaft diameter of the fastener is less than the hole diameter of the plate.
- Interference fit – The shaft diameter of the fastener is larger than the hole diameter of the plate. The difference between the shaft diameter and the hole diameter is called interference.
- Transition fit – Clearance or interference can exist between the shaft of the fastener and the hole diameter of the plate.

Present additional examples to explain fit and tolerance from your experience or from text books such as:

- Bertoline et. al. Fundamentals of Graphics Communications, Irwin, 1995.
- Earle, James, Engineering Design Graphics, Addison Wesley 1999.
- Jensel et al. Engineering Drawing and Design, Glencoe, 1990.

The Hole Wizard

Show your students the Hole Wizard. Show how the Hole Wizard uses the size of the fastener and the desired amount of clearance to create the correct size hole.



Fastener Selection

Fastener selection is a vast topic. Selecting the correct fastener for a particular application involves many considerations. Discuss some of the following factors that will influence selecting the right fastener for a particular job:

- Strength: Will the fastener be strong enough for the intended application? Fasteners that fail under a load can lead to problems ranging from unhappy customers to product liability lawsuits to injury or even death.
- Material: This is related to strength, cost, and appearance. However, the appropriate material is also important in its own right. For example, fasteners used in marine applications (boats) must be made of a corrosion resistant material such as stainless steel.
- Cost: All other things being equal, a manufacturer would want to use the lowest cost fastener.
- Appearance: Is the fastener visible to the consumer or is it hidden inside the product? Some fasteners serve a decorative purpose in addition to their functional purpose of holding things together.
- Ease of assembly: Today many products are being designed to snap together without fasteners. Why? Because even with automatic assembly equipment, fasteners add a great deal of expense to a product.
- Special considerations: Some fasteners have special characteristics. For example, some are designed with special heads that allow them to be installed but not removed. One application for this type of fastener would be road signs, to make them vandal proof.

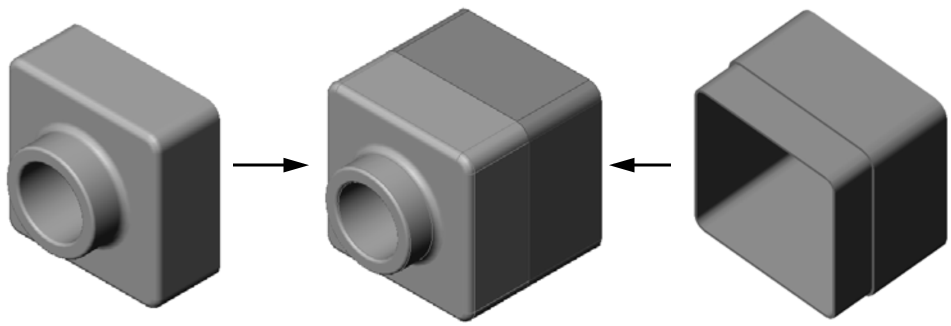
Invite designers and engineers from local industries into your classroom to discuss the area of fastener selection.

Active Learning Exercises — Creating an Assembly

Follow the instructions in *Getting Started: Lesson 2– Assemblies* in the SolidWorks Tutorials. In this lesson you will first create `Tutor2`. Then you will create an assembly.

Note: For `Tutor1.sldprt`, use the sample file provided in the `\Lessons\Lesson04` folder to ensure the correct dimensions.

For `Tutor2.sldprt`, the tutorial instructs you to create a fillet with a 5mm radius. You must modify the radius of the fillet to 10mm to mate properly with the `Tutor1.sldprt`.



`Tutor1` `Tutor assembly` `Tutor2`

Lesson 4 — 5 Minute Assessment — Answer Key

Name: ______________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What features did you use to create `Tutor2`?

Answer: Extruded base/boss, fillet, shell and extruded cut.

2 What two sketch tools did you use to create the extruded cut feature?

Answer: The two sketch tools used to create the extruded cut are **Convert Entities** and **Offset Entities**.

3 What does the **Convert Entities** sketch tool do?

Answer: The **Convert Entities** sketch tool creates one or more curves in a sketch by projecting geometry onto the sketch plane.

4 What does the **Offset Entities** sketch tool do?

Answer: The **Offset Entities** sketch tool creates a curve from a selected edge at a specified distance.

5 In an assembly, parts are referred to as _______________.

Answer: In an assembly, parts are referred to as components.

6 True or False. A fixed component is free to move.

Answer: False.

7 True or False. Mates are relationships that align and fit components together in an assembly.

Answer: True.

8 How many components does an assembly contain?

Answer: An assembly contains two or more components.

9 What mates are required for the `Tutor assembly`?

Answer: Three **Coincident Mates** are required for the `Tutor assembly`.

Lesson 4 — 5 Minute Assessment REPRODUCIBLE

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What features did you use to create `Tutor2`?

____

2 What two sketch tools did you use to create the extruded cut feature?

____

____

3 What does the **Convert Entities** sketch tool do?

____

4 What does the **Offset Entities** sketch tool do?

____

5 In an assembly, parts are referred to as _______________.

____

6 True or False. A fixed component is free to move.

____

7 True or False. Mates are relationships that align and fit components together in an assembly.

____

8 How many components does an assembly contain?

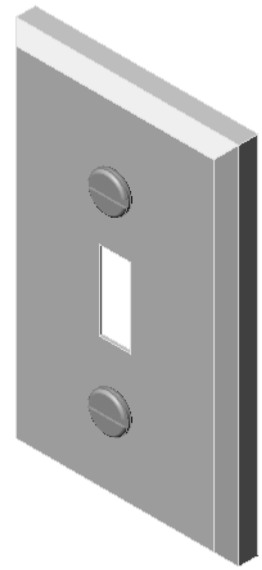
____

9 What mates are required for the `Tutor assembly`?

____

Exercises and Projects — Creating the Switchplate Assembly

Task 1 — Modifying Feature Size



The `switchplate` created in Lesson 3 requires two fasteners to complete the assembly.

Question:

How do you determine the size of the holes in the `switchplate`?

Answer:

By the size of the fasteners.

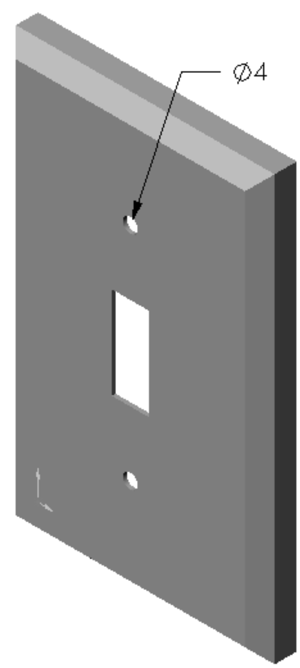
- Many aspects of a design are determined by the size, shape, and position of features in other components in an assembly.
- The `switchplate` is to be attached to an electrical switch.
- The electrical switch already has threaded holes in it for the screws.
- Those screws determine the size of the holes in the `switchplate`.
- The hole must be slightly larger than the fastener that goes into it.

Given:

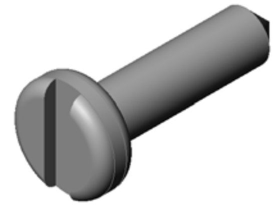
- The diameter of the fastener is **3.5mm**.
- The `switchplate` is **10mm** deep.

Procedure:

1. Open the `switchplate`.
2. Modify the diameter of the two holes to **4mm**.
3. Save the changes.

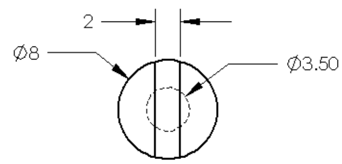


Task 2 — Designing a Fastener

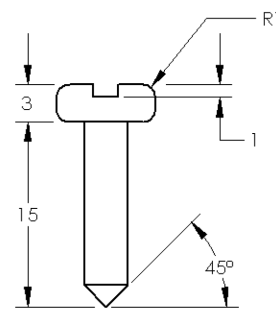


Design and model a fastener that is appropriate for the `switchplate`. Your fastener may (or may not) look like the one shown at the right.

Design Criteria:



- ❑ The fastener must be longer than the thickness of the switchplate.
- ❑ The `switchplate` is **10mm** thick.
- ❑ The fastener must be **3.5mm** in diameter.
- ❑ The head of the fastener must be larger than the hole in the `switchplate`.



Good Modeling Practice

Fasteners are almost always modeled in a simplified form. That is, although a real machine screw has threads on it, these are not included in the model.

Note to the Teacher

- ❑ A sample `fastener` part and its related drawing file are found in the `Lessons\Lesson04` folder located under `SolidWorks Teacher Tools`.
- ❑ The fasteners your students build do not have to exactly match the one illustrated on this page.
- ❑ This is a good opportunity for the students to develop independent solutions to the stated problem.
- ❑ It *is* important that the fasteners your students build meet the stated design criteria.

Task 3 — Creating an Assembly

Create the `switchplate-fastener` assembly.

Procedure:

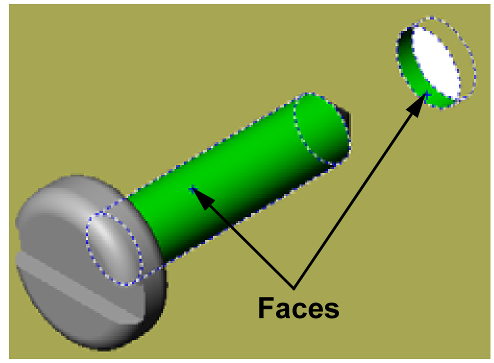
1 Create a new assembly.
The fixed component is the `switchplate`.

2 Drag the `switchplate` into the assembly window.

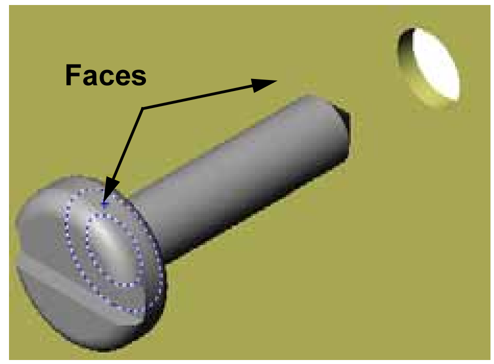
3 Drag the `fastener` into the assembly window.

The `switchplate-fastener` assembly requires three mates to fully define the assembly.

1 Create a **Concentric** mate between the cylindrical face of the `fastener` and the cylindrical face of the hole in the `switchplate`.

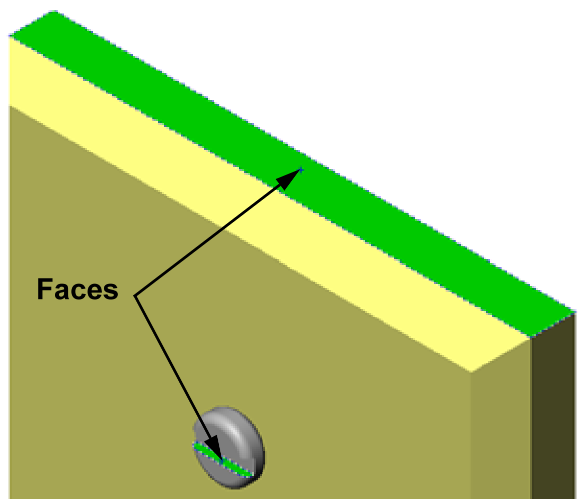


2 Create a **Coincident** mate between the back flat face of the `fastener` and the flat front face of the `switchplate`.



3 Create a **Parallel** mate between one of the flat faces on the slot of the `fastener` and the flat top face of the `switchplate`.

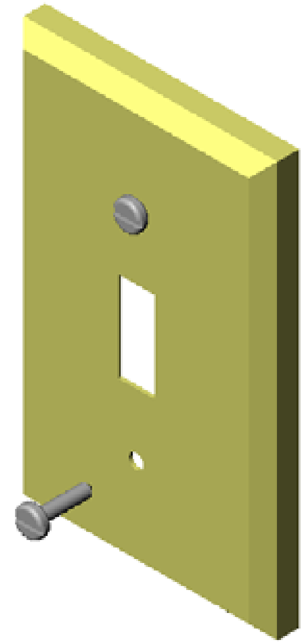
Note: If the necessary faces do not exist in the `fastener` or the `switchplate`, create the parallel mate using the appropriate reference planes in each component.



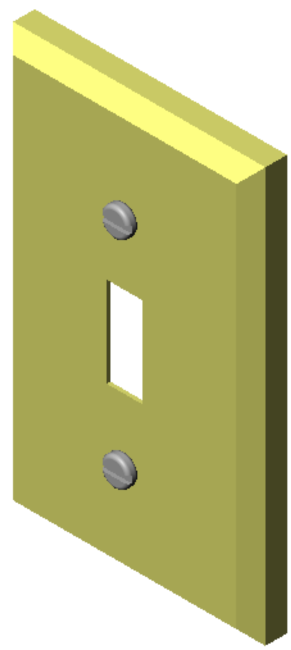
4 Add a second instance of the `fastener` to the assembly.

You can add components to an assembly by dragging and dropping:

- Hold the **Ctrl** key, and then drag the component either from the FeatureManager design tree, or from the graphics area.
- The pointer changes to .
- Drop the component in the graphics area by releasing the left mouse button and the **Ctrl** key.



5 Add three **mates** to fully define the second `fastener` to the `switchplate-fastener` assembly.



6 Save the `switchplate-fastener` assembly.

Note to the Teacher

The completed `switchplate-fastener` assembly is found in the `Lessons\Lesson04` folder in `SolidWorks Teacher Tools`.

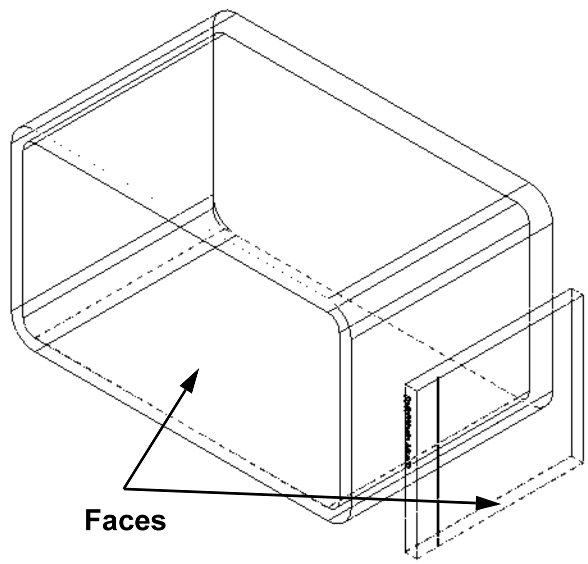
Exercises and Project — Creating CD Storage Box Assembly

Assemble the `cdcase` and `storagebox` that you created in Lesson 3.

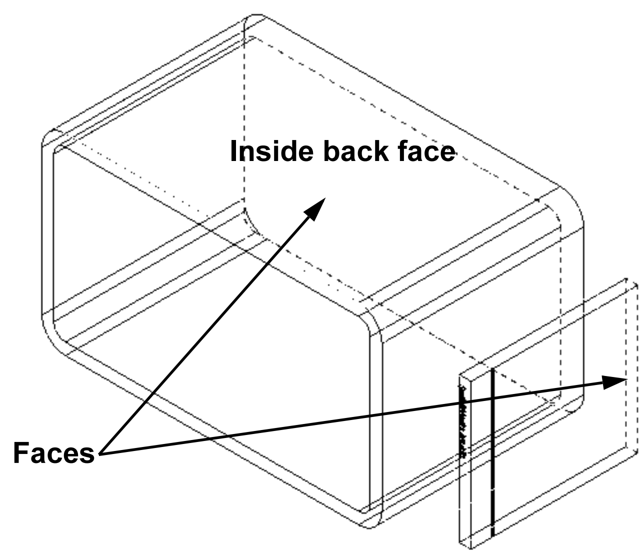
Note: The completed `cdcase-storagebox` assembly example is found in the `Lesson3` file folder.

Procedure:

1. Create a new assembly.
   The fixed component is the `storagebox`.
2. Drag the `storagebox` into the assembly window.
3. Drag the `cdcase` into the assembly window to the right of the `storagebox`.
4. Create a **Coincident** mate between the bottom face of the `cdcase` and the inside bottom face of the `storagebox`.

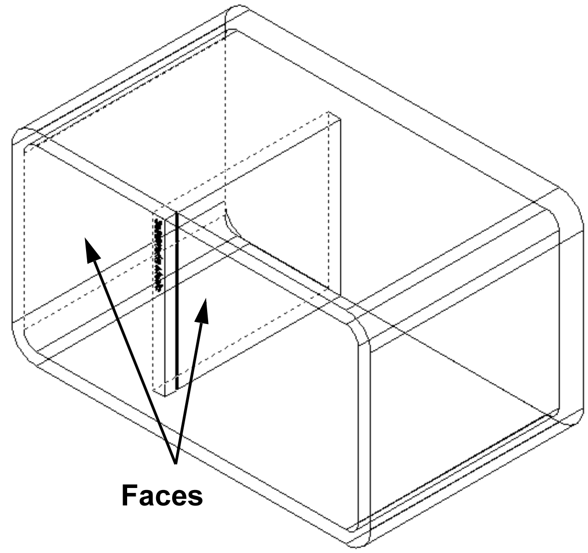


5. Create a **Coincident** mate between the back face of the `cdcase` and the inside back face of the `storagebox`.



6 Create a **Distance** mate between the *left* face of the `cdcase` and the inside left face of the `storagebox`.
Enter **1cm** for **Distance**.

7 Save the assembly.
Enter `cdcase-storagebox` for the filename.

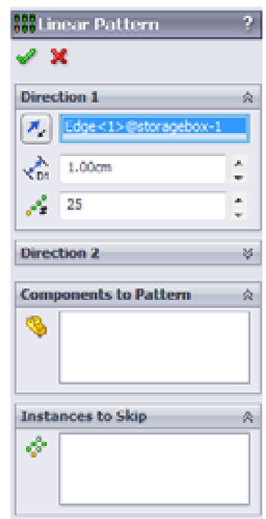


Component Patterns

Create a linear pattern of the `cdcase` component in the assembly.

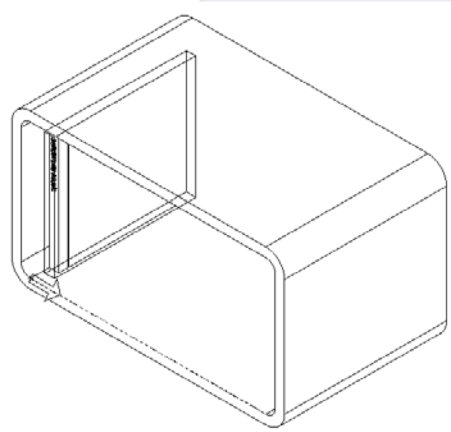
The `cdcase` is the seed component. The seed component is what gets copied in the pattern.

1 Click **Insert**, **Component Pattern**, **Linear Pattern.** The **Linear Pattern** PropertyManager appears.

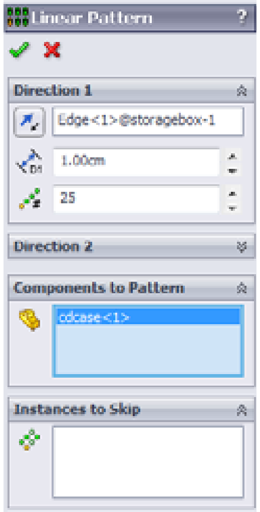


2 Define the direction for the pattern.
Click inside the **Pattern Direction** text box to make it active.
Click the bottom horizontal front edge of the `storagebox`.

3 Observe the direction arrow.
The preview arrow should point to the right. If it does not, click the **Reverse Direction** button.



4 Enter **1 cm** for **Spacing**. Enter **25** for **Instances**.

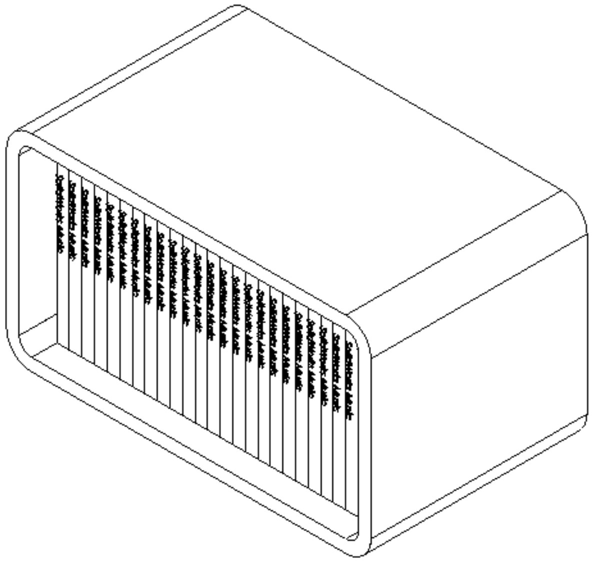


5 Select the component to be patterned.

Make sure the **Component to Pattern** field is active, and then select the `cdcase` component from the FeatureManager design tree or the graphics area.
Click **OK**.
The Local Component Pattern feature is added to the FeatureManager design tree.

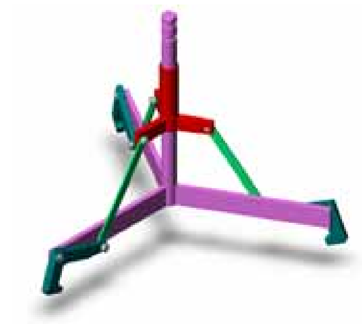
6 Save the assembly.

Click **Save**. Use the name `cdcase-storagebox`.



Exercises and Projects — Assembling a Mechanical Claw

Assemble the claw mechanism shown at the right. This assembly will be used later, in Lesson 11, to create a movie using the SolidWorks Animator software.

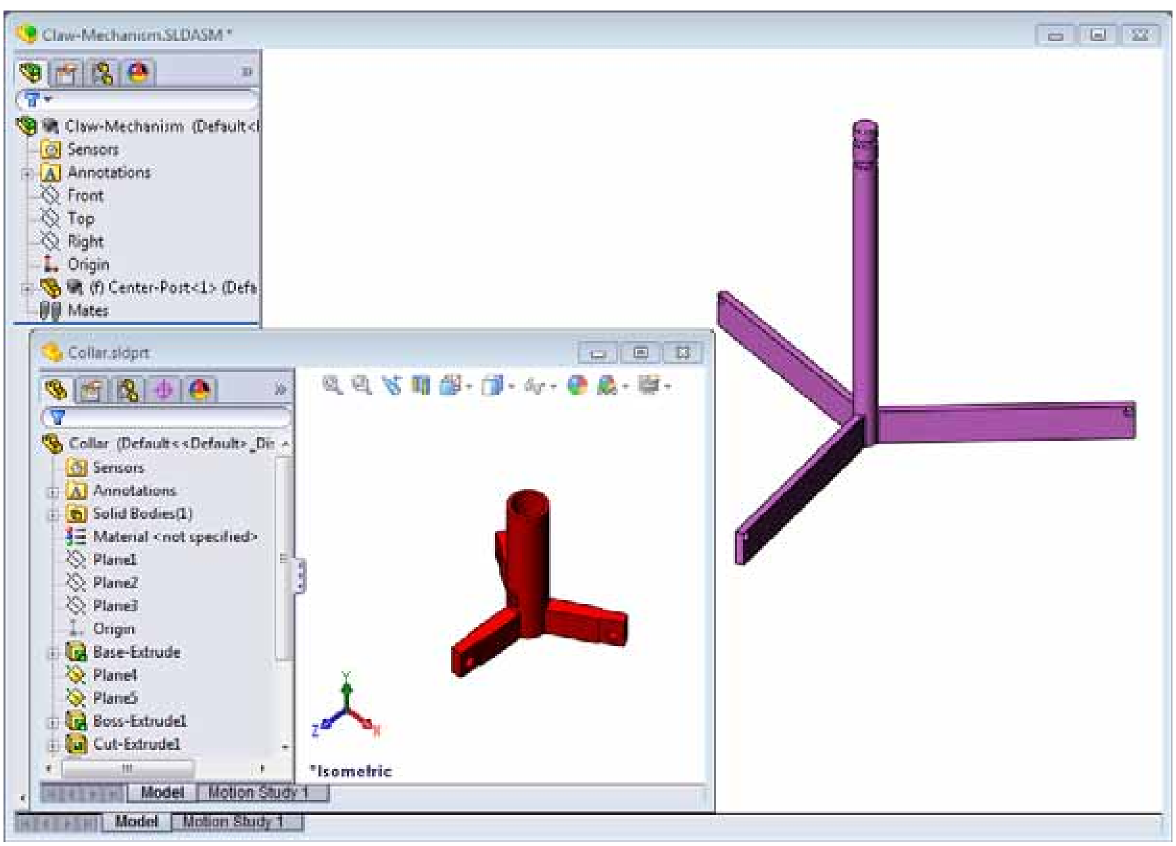


Procedure:

1. Create a new assembly.
2. Save the assembly. Name it `Claw-Mechanism`.
3. Insert the `Center-Post` component into the assembly. The files for this exercises are found in the `Claw` folder in the `Lesson04` folder.

4. Open the `Collar` part.

   Arrange the windows as shown below.



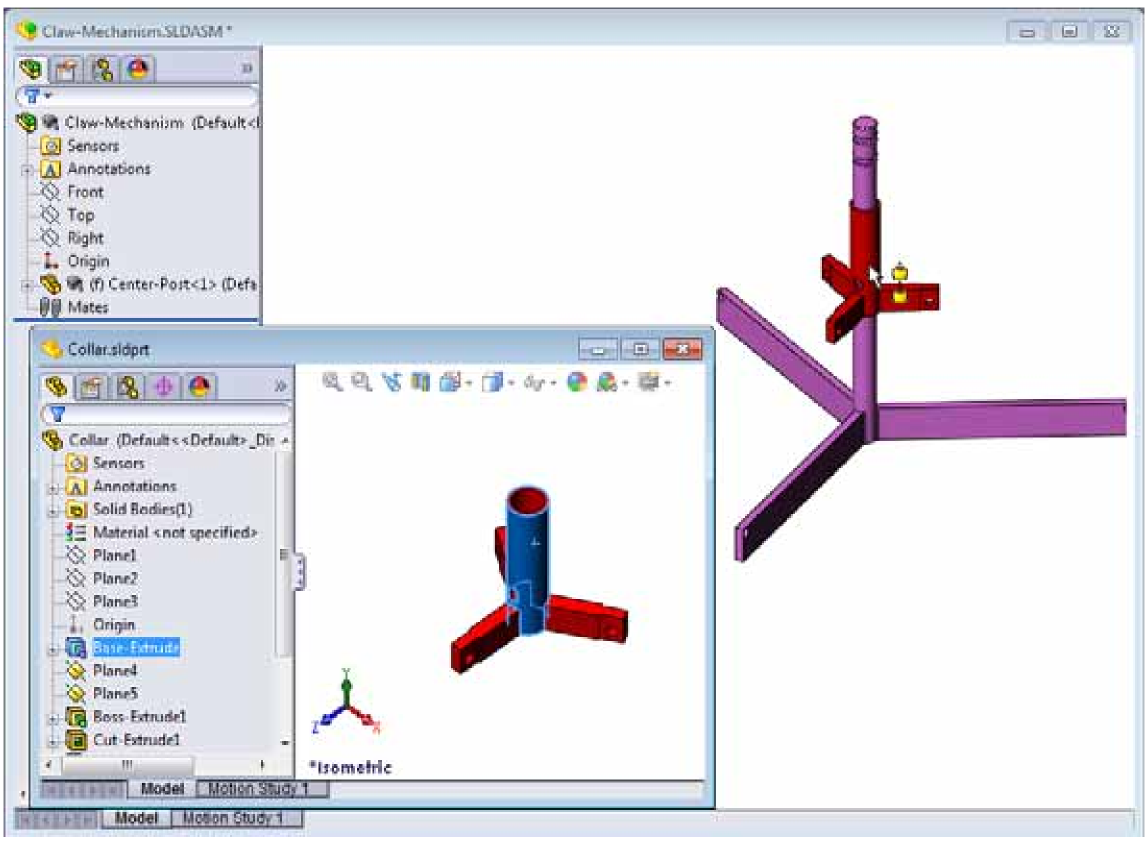
SmartMates

You can create some types of mating relationships automatically. Mates created with these methods are referred to as SmartMates.

You can create mates when you drag the part in specific ways from an open part window. The entity that you use to drag determines the types of mates that are added.

5 Select the cylindrical face of the `Collar`, and drag the `Collar` into the assembly. Point at the cylindrical face of the `Center-Post` in the assembly window.

When the pointer is over the `Center-Post`, the pointer changes to . This pointer indicates that a **Concentric** mate will result if the `Collar` is dropped at this location. A preview of the `Collar` snaps into place.

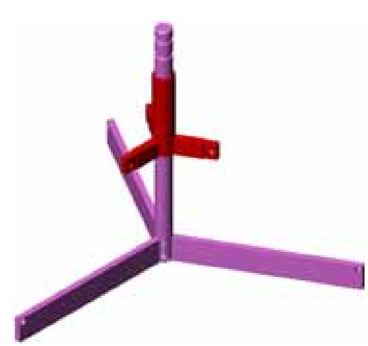


6 Drop the `Collar`.

A **Concentric** mate is added automatically.

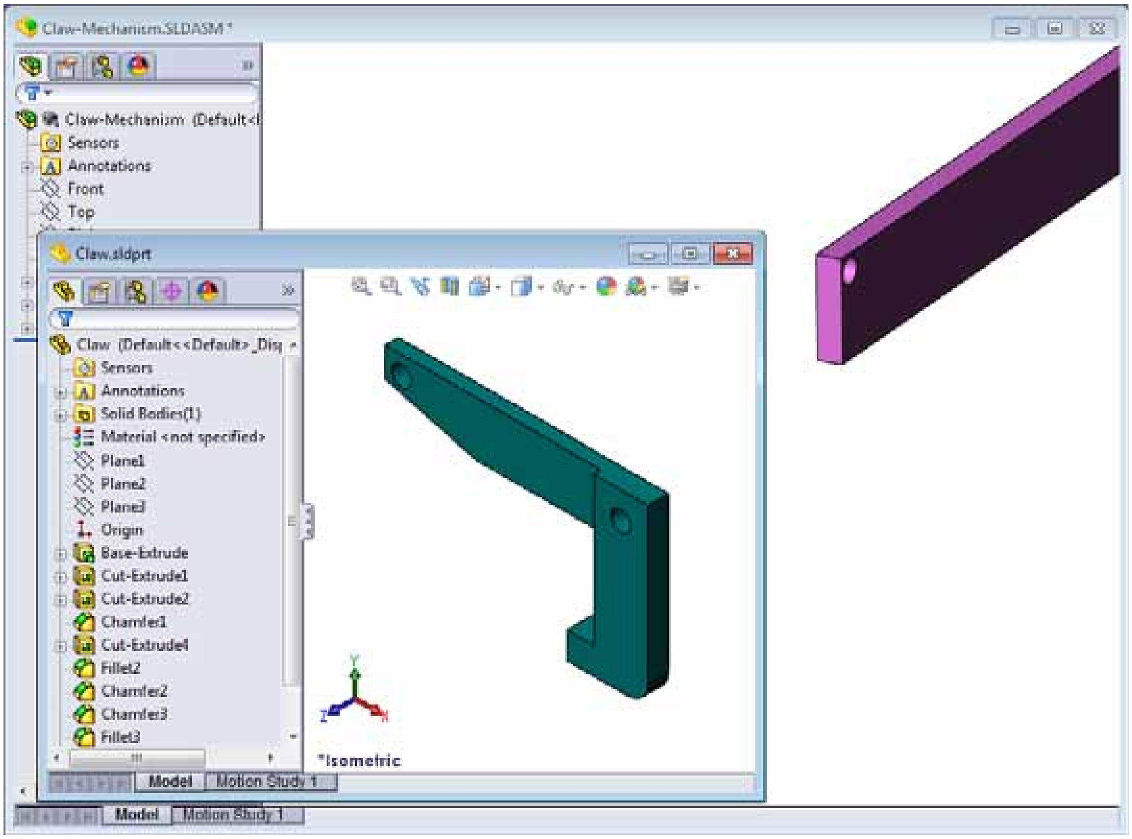
Click **Add/Finish Mate** .

7 Close the `Collar` part document.



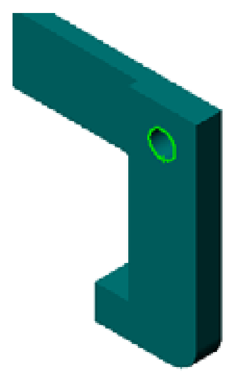
8 Open the `Claw`.

Arrange the windows as shown below.



9 Add the `Claw` to the assembly using SmartMates

- Select the *edge* of the hole in the `Claw`.

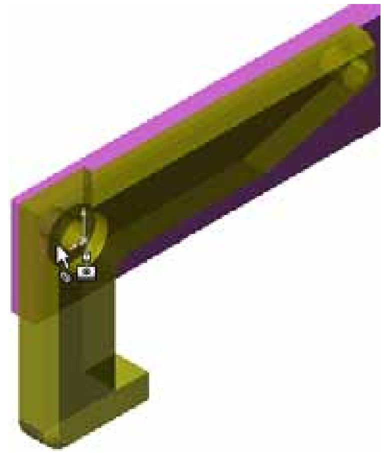


It is important to select the edge and not the cylindrical face. This is because this type of SmartMate will add two mates:

- A **Concentric** mate between the cylindrical faces of the two holes.
- A **Coincident** mate between the planar face of the `Claw` and the arm of the `Center-Post`.

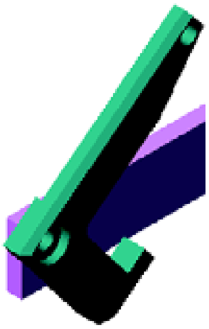
10 Drag and drop the `Claw` onto the *edge* of the hole in the arm.

The pointer looks like this  indicating that a **Concentric** and a **Coincident** mate will be added automatically. This SmartMate technique is ideal for putting fasteners into holes.



11 Close the `Claw` part document.

12 Drag the `Claw` as shown below. This makes it easier to select an edge in the next step.

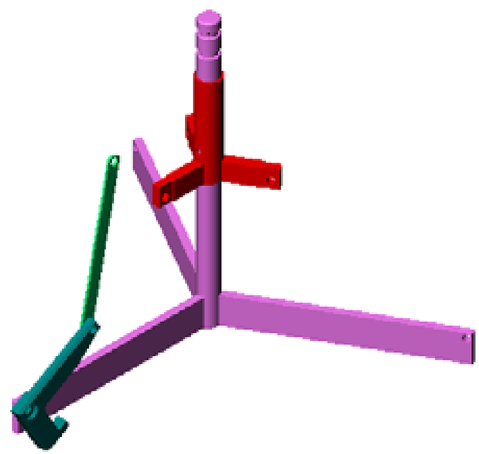


13 Add the `Connecting-Rod` to the assembly.

Use the same SmartMate technique you used in steps 9 and 10 to mate one end of the `Connecting-Rod` to the end of the `Claw`.

There should be two mates:

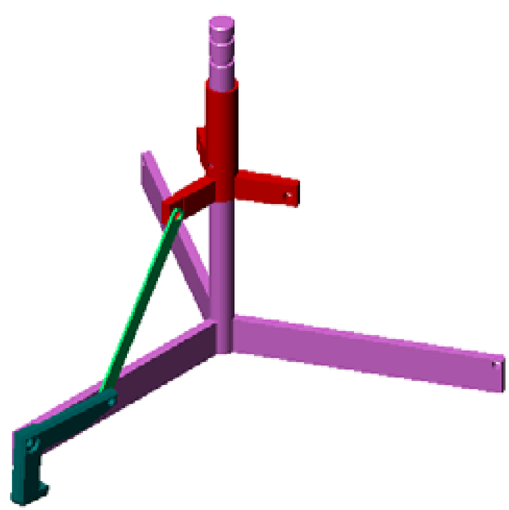
- **Concentric** between the cylindrical faces of the two holes.
- **Coincident** between the planar faces of the `Connecting-Rod` and the `Claw`.

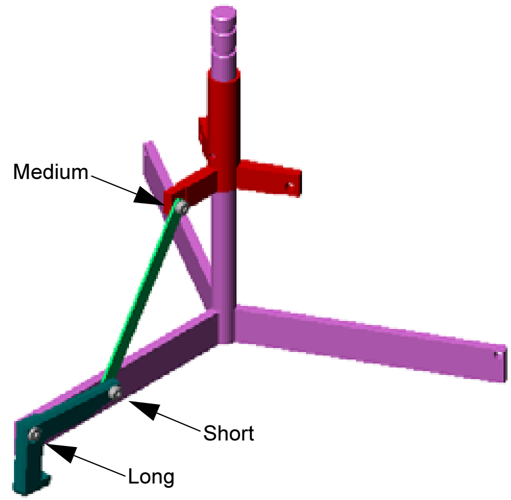


14 Mate the `Connecting-Rod` to the `Collar`.

Add a **Concentric** mate between the hole in the `Connecting-Rod` and the hole in the `Collar`.

Do not add a **Coincident** mate between the `Connecting-Rod` and the `Collar`.





15 Add the pins.

There are three different length pins:

- `Pin-Long` (1.745 cm)
- `Pin-Medium` (1.295 cm)
- `Pin-Short` (1.245 cm)

Students should use **Tools, Measure** to determine which pin goes in which hole.

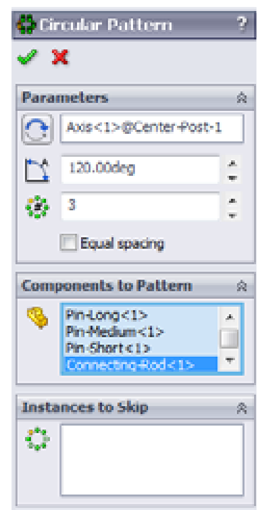
Add the pins using SmartMates.

Circular Component Pattern

Create a circular pattern of the `Claw`, `Connecting-Rod`, and pins.

1 Click **Insert**, **Component Pattern**, **Circular Pattern.**

The **Circular Pattern** PropertyManager appears.



2 Select the components to be patterned.

Make sure the **Components to Pattern** field is active, and then select the `Claw`, the `Connecting-Rod`, and the three pins.

3 Click **View, Temporary Axes**.

4 Click in the **Pattern Axis** field. Select the axis that runs down the center of the `Center-Post` for the center of rotation for the pattern.

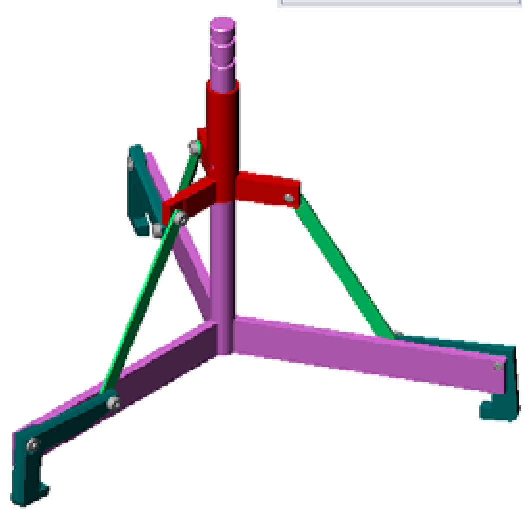
5 Set the **Angle** to 120°.

6 Set the **Instances** to 3.

7 Click **OK**.

8 Turn off the temporary axes.

Dynamic Assembly Motion



Moving under defined components simulates movement of a mechanism through dynamic assembly motion.

9 Drag the `Collar` up and down while observing the motion of the assembly.

10 Save and close the assembly.

Lesson 4 Vocabulary Worksheet — Answer Key

Name: ________________________________Class: _________ Date:_______________

Fill in the blanks with the words that are defined by the clues.

1. **Convert Entities** copies one or more curves into the active sketch by projecting them onto the sketch plane.
2. In an assembly, parts are referred to as: **Components**.
3. Relationships that align and fit components together in an assembly: **Mates**
4. The symbol `(f)` in the FeatureManager design tree indicates a component is: **Fixed**
5. The symbol `(-)` indicates a component is: **Underdefined**
6. When you make a component pattern, the component you are copying is called the **Seed** component.
7. A SolidWorks document that contains two or more parts: **Assembly**
8. You cannot move or rotate a fixed component unless you **Float** it first.

Lesson 4 Vocabulary Worksheet **REPRODUCIBLE**

Name: ________________________________Class: _________ Date:_______________

Fill in the blanks with the words that are defined by the clues.

1 ____________________ copies one or more curves into the active sketch by projecting them onto the sketch plane.

2 In an assembly, parts are referred to as: _______________________________________

3 Relationships that align and fit components together in an assembly:_______________

4 The symbol `(f)` in the FeatureManager design tree indicates a component is: _______
____

5 The symbol `(-)` indicates a component is: ___________________________________

6 When you make a component pattern, the component you are copying is called the __________________ component.

7 A SolidWorks document that contains two or more parts:________________________

8 You cannot move or rotate a fixed component unless you _________________ it first.

Lesson 4 Quiz — Answer Key

Name: ______________________________Class: _________ Date:______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you start a new Assembly document?

Answer: Click the **New** icon. Select a assembly template. Click **OK**.

2 What are components?

Answer: Components are parts or sub-assemblies contained in an assembly.

3 The **Convert Entities** sketch tool projects selected geometry onto the ________ plane?

Answer: Current sketch.

4 True or False. The **Offset Entities** sketch tool was used to copy the Cut-Extrude feature.

Answer: False.

5 How many mates were required to fully define the `Tutor` assembly?

Answer: The `Tutor` assembly required 3 **Coincident Mates**.

6 True or False. Edges and faces can be selected items for Mates in an assembly.

Answer: True.

7 A component in an assembly displays a (-) prefix in the FeatureManager design tree. Is the component fully defined?

Answer: No. A component that contains the (-) prefix is not fully defined. Additional mates are required.

8 When components are modified, describe the result to the assembly?

Answer: The assembly reflects the new component modifications.

9 What actions do you perform when an edge or face is too small to be selected by the pointer?

Answer:

- Use **Zoom** options from the View toolbar to increase the geometry size
- Use **Selection Filters**
- Right-click and choose **Select Other**

10 Name the mates required to fully define the `switchplate-fastener` assembly?

Answer: The `switchplate-fastener` assembly required 3 mates for each fastener: **Concentric Mate**, **Coincident Mate** and **Parallel Mate**.

Lesson 4 Quiz — REPRODUCIBLE

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you start a new Assembly document?

____

2 What are components?

____

3 The **Convert Entities** sketch tool projects selected geometry onto the _____________ _________________________ plane?

4 True or False. The **Offset Entities** sketch tool was used to copy the Cut-Extrude feature.

____

5 How many mates were required to fully define the `Tutor` assembly?

____

6 True or False. Edges and faces can be selected items for Mates in an assembly.

____

7 A component in an assembly displays a (-) prefix in the FeatureManager design tree. Is the component fully defined?

____

8 When components are modified, describe the result to the assembly?

____

9 What actions do you perform when an edge or face is too small to be selected by the pointer?

____

____

____

10 Name the mates required to fully define the `switchplate-fastener` assembly?

____

____

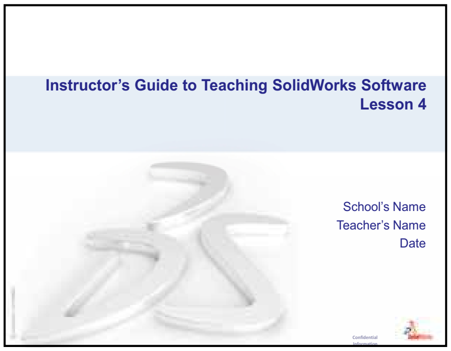
____

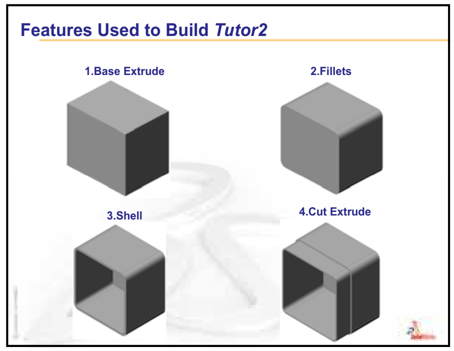
Lesson Summary

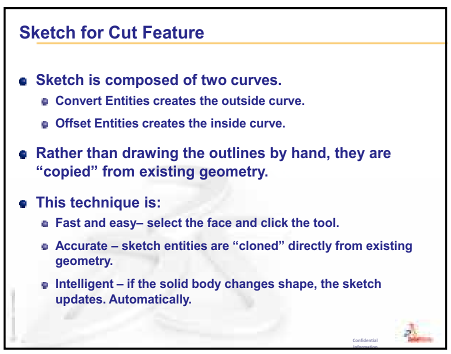
- An assembly contains two or more parts.
- In an assembly, parts are referred to as *components*.
- Mates are relationships that align and fit components together in an assembly.
- Components and their assembly are directly related through file linking.
- Changes in the components affect the assembly and changes in the assembly affect the components.
- The first component placed into an assembly is fixed.
- Under defined components can be moved using dynamic assembly motion. This simulates the movement of mechanisms.

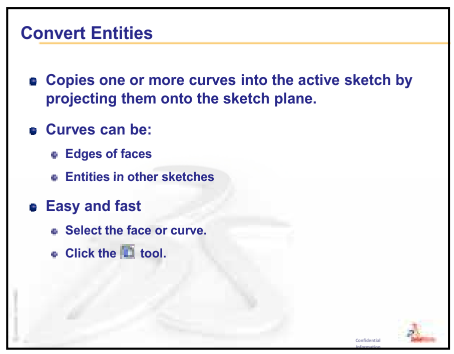
Thumbnail Images of PowerPoint Slides

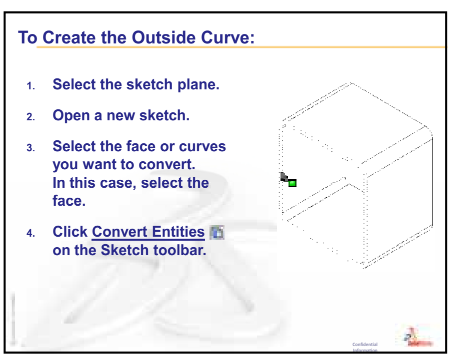
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

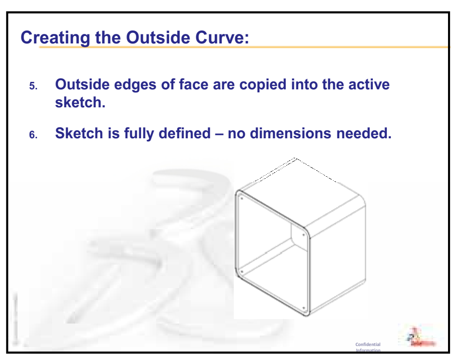










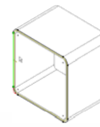


To Create the Inside Curve:

1. Click Offset Entities on the Sketch toolbar.
   The PropertyManager opens.
2. Enter the distance value of 2mm.
3. Select one of the converted entities.
4. The Select chain option causes the offset to go all the way around the contour.

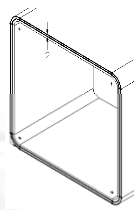
Creating the Inside Curve:

5. The system generates a preview of the resulting offset.
6. A small arrow points toward the cursor. If you move your cursor to the other side of the line, the arrow changes direction. This indicates on which side the offset will be created.
7. Move the cursor so it is *inside* the contour. Click the left mouse button to create the offset.



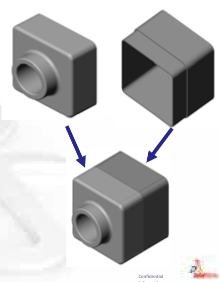
Creating the Inside Curve:

8. The resulting sketch is fully defined.
9. There is only one dimension. It controls the offset distance.



Tutor Assembly

- The *Tutor* assembly is comprised of two parts:
  - *Tutor1* (created in Lesson 2)
  - *Tutor2* (created in this lesson)

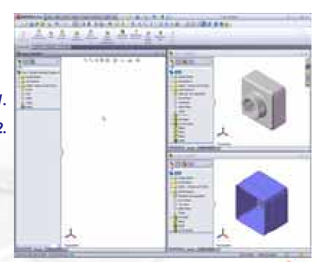


Assembly Basics

- An assembly contains two or more parts.
- In an assembly, parts are referred to as *components*.
- Mates are relationships that align and fit components together in an assembly.
- Components and their assembly are directly related through file linking.
- Changes in the components affect the assembly.
- Changes in the assembly affect the components.

To create the Tutor assembly:

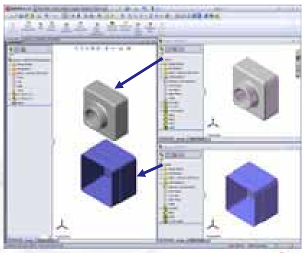
1. Open a new assembly document template.
2. Open *Tutor1*.
3. Open *Tutor2*.
4. Arrange the windows.



Creating the Tutor assembly:

5. Drag and drop the part icons into the assembly document.

   Save the assembly as Tutor.

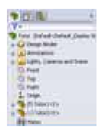


Assembly Basics

- The first component placed into an assembly is fixed.
- A fixed component cannot move.
- If you want to move a fixed component, you must Float (unfix) it first.
- Tutor1 is added to the FeatureManager design tree with the symbol (f).
- The symbol (f) indicates a fixed component.

Assembly Basics

- Tutor2 is added to the FeatureManager design tree with the symbol (-).
- The symbol (-) indicates an underdefined component.
- Tutor2 is free to move and rotate.

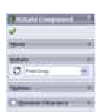


Manipulating Components

- Move components by dragging.
- Move components with a triad.
- Move Component – translates (moves) the selected component according to its available degrees of freedom.

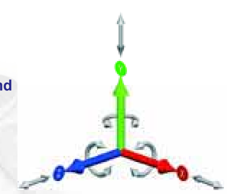
Manipulating Components

- Rotate components by dragging.
- Rotate components with a triad.
- Rotate Component – rotates the selected component according to its available degrees of freedom.



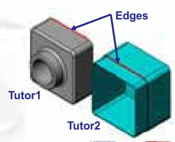
Degrees of Freedom: There are Six

- They describe how an object is free to move.
- Translation (movement) *along* X, Y, and Z axes.
- Rotation *around* X, Y, and Z axes.



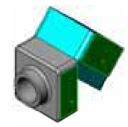
Mate Relationships

- Mate relationships align and fit together components in an assembly.
- The *Tutor* assembly requires three mates to fully define it.
  The three mates are:
- Coincident between the top back edge of *Tutor1* and the edge of the lip on *Tutor2.*



Mate Relationships

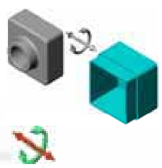
- Second Mate: Coincident mate between the right face of Tutor1 and the right face of Tutor2.



- Third Mate: Coincident mate between the top face of Tutor1 and the top face of Tutor2.

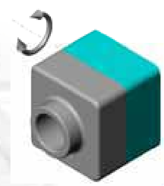
Mates and Degrees of Freedom

- The first mate removes all but two degrees of freedom.
- The remaining degrees of freedom are:
  - Movement *along* the edge.
  - Rotation *around* the edge.



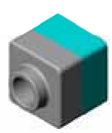
Mates and Degrees of Freedom

- The second mate removes one more degree of freedom.
- The remaining degree of freedom is:
  - Rotation *around* the edge.



Mates and Degrees of Freedom

- The third mate removes last degree of freedom.
- No remaining degrees of freedom.
- The assembly is fully defined.

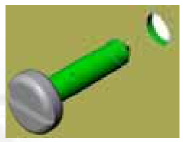


Additional Mate Relationships for Exercises and Projects

- The *switchplate* requires two fasteners.
- Create the *fastener*.
- Create the *switchplate-fastener* assembly.

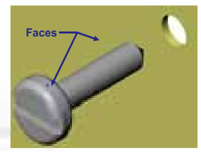
Additional Mate Relationships for Exercises and Projects

- The *switchplate-fastener* assembly requires three mates to be fully defined. The three mates are:
- First Mate:
  Concentric mate between the cylindrical face of the fastener and the cylindrical face of the *switchplate.*



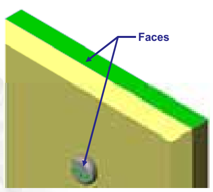
Additional Mate Relationships for Exercises and Projects

- Second Mate:
  Coincident mate between the flat circular back face of the fastener and the flat front face of the *switchplate.*



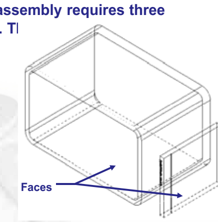
Additional Mate Relationships for Exercises and Projects

- Third Mate:
  Parallel mate between the flat cut face of the fastener and the flat top face of the *switchplate.*
- The *switchplate-fastener* assembly is fully defined.



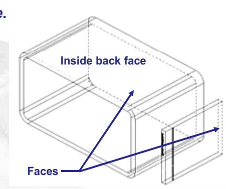
Additional Mate Relationships for Exercises and Projects

- The *cdcase-storagebox* assembly requires three mates to be fully defined. Tl
- First Mate:
  Coincident between the inside bottom face of the *storagebox* and the bottom face of the *cdcase.*



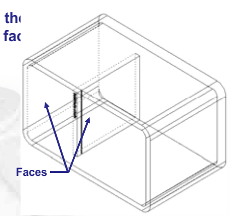
Additional Mate Relationships for Exercises and Projects

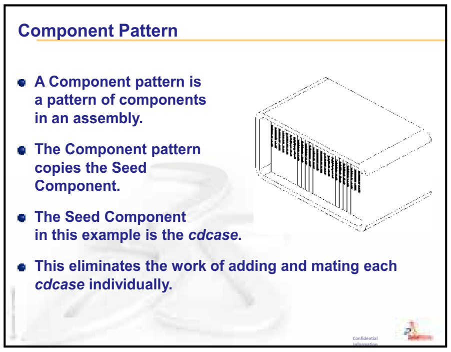
- Second Mate:
  Coincident mate between the inside back face of the *storagebox* and the back face of the *cdcase.*

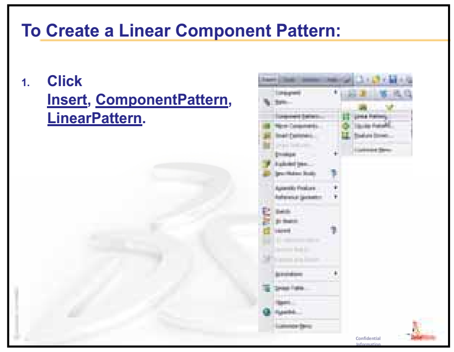


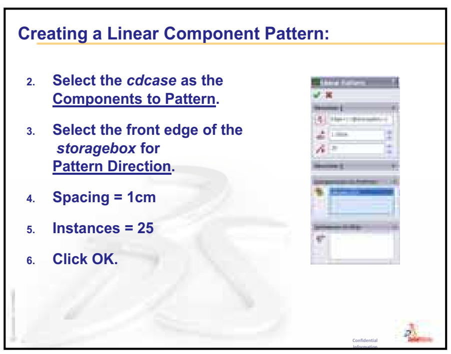
Additional Mate Relationships for Exercises and Projects

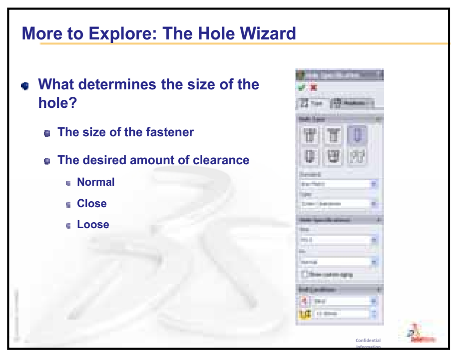
- Third Mate:
  Distance mate between thi *storagebox* and the left fac
- Distance = 1cm.
- Good job! Now, would you like to do this 24 more times?
- No!




Component Pattern
A Component pattern is a pattern of components in an assembly.
The Component pattern copies the Seed Component.
The Seed Component in this example is the cdcase.
This eliminates the work of adding and mating each cdcase individually.


To Create a Linear Component Pattern:
1. Click Insert, ComponentPattern, LinearPattern.


Creating a Linear Component Pattern:
2. Select the cdcase as the Components to Pattern.
3. Select the front edge of the storagebox for Pattern Direction.
4. Spacing = 1cm
5. Instances = 25
6. Click OK.


More to Explore: The Hole Wizard
What determines the size of the hole?
The size of the fastener
The desired amount of clearance
Normal
Close
Loose

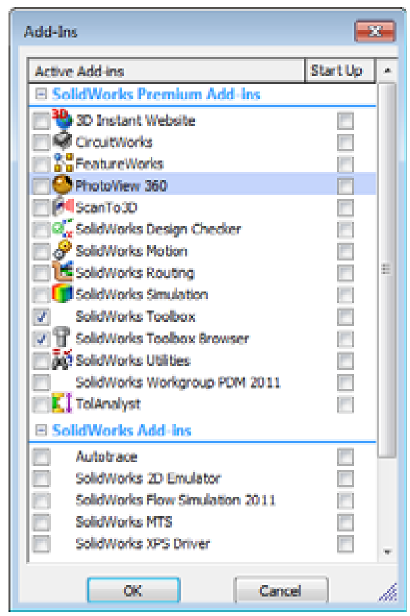
Lesson 5: SolidWorks Toolbox Basics

Goals of This Lesson

- ❑ Place standard SolidWorks Toolbox parts in assemblies.
- ❑ Modify Toolbox part definitions to customize standard Toolbox parts.

Before Beginning This Lesson

- ❑ Complete Lesson 4: Assembly Basics.
- ❑ Verify that **SolidWorks Toolbox** and **SolidWorks Toolbox Browser** are set up and running on your classroom/lab computers. Click **Tools**, **Add-Ins** to activate these add-ins. SolidWorks Toolbox and SolidWorks Toolbox Browser are SolidWorks add-ins which are not loaded automatically. These add-ins must be specifically added during installation.



Resources for This Lesson

This lesson plan corresponds to *Productivity Enhancements: Toolbox* in the SolidWorks Tutorials.

SolidWorks Toolbox contains 1000's of library parts including fasteners, bearings, and structural members.

Review of Lesson 4: Assembly Basics

Questions for Discussion

1 Describe an assembly.

Answer: An assembly combines two or more parts in a single document. In an assembly or sub-assembly, parts are referred to as components.

2 What does the command **Convert Entities** do?

Answer: **Convert Entities** projects one or more curves onto the active sketch plane. Curves can be edges of faces or entities in other sketches.

3 What does a selection filter do?

Answer: A selection filter enables you to more easily select the item you want in the Graphics Area by only allowing you to select a specified type of entity.

4 What does it mean when a component in an assembly is “fixed”?

Answer: A fixed component in an assembly cannot move. It is locked in place. By default, the first component added to an assembly is automatically fixed.

5 What are mates?

Answer: Mates are the relationships that align and position components in an assembly.

6 What are degrees of freedom?

Answer: Degrees of freedom describe how an object is free to move. There are six degrees of freedom. They are translation (movement) along the X, Y, or Z axes, and rotation around the X, Y, or Z axes.

7 How are degrees of freedom related to mates?

Answer: Mates eliminate degrees of freedom.

In Class Demonstration — Changing an Assembly

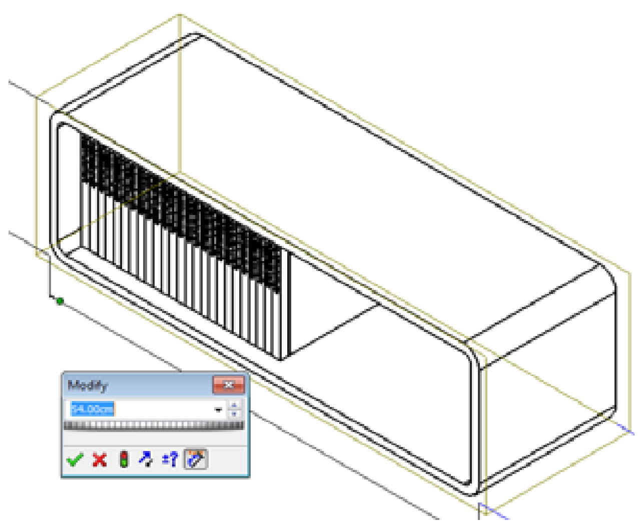
You receive a design change. The customer requires a storage box to hold 50 CD jewel cases.

1 Open the `cdcase-storagebox` assembly.

2 Double-click on the top face of the `storagebox` component.

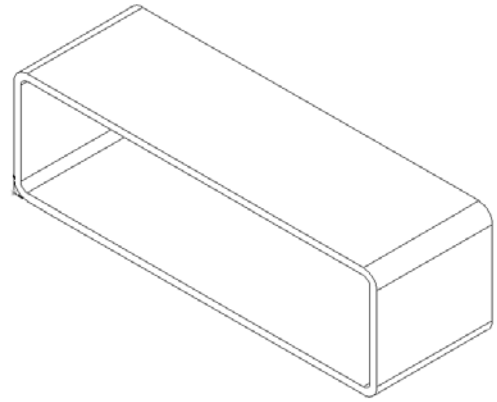
3 Double-click the width dimension. Enter a new value, **54 cm**.

4 Rebuild.



5 Open `storagebox`. Review the modified part.

Notice that when feature dimensions are modified in the assembly, the components change to reflect the modification.



Optional:

Change the number of instances in the assembly component pattern to 50.

Outline of Lesson 5

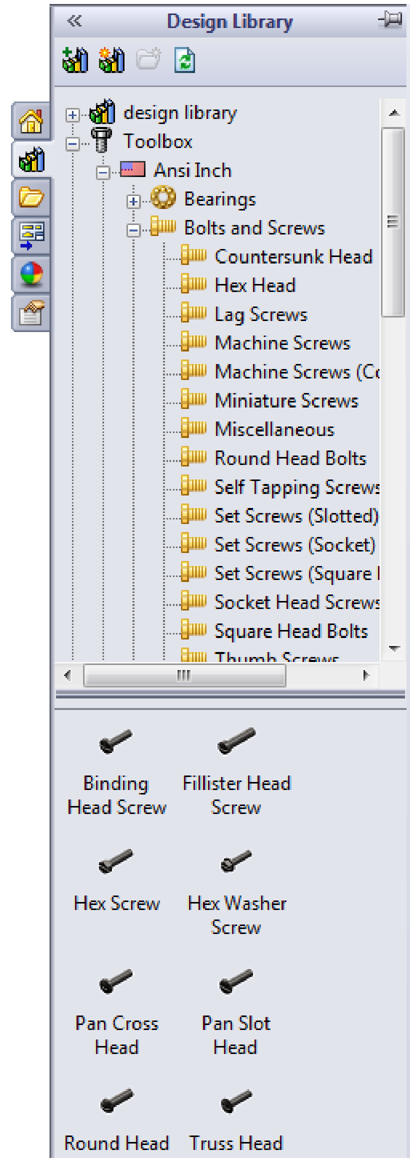
- ❑ In Class Discussion — What is Toolbox?
- ❑ Active Learning Exercises — Adding Toolbox Parts
  - Open the Switchplate Toolbox Assembly
  - Open Toolbox Browser, in the Design Library Task Pane
  - Selecting Appropriate Hardware
  - Placing Hardware
  - Specifying the Properties of the Toolbox Part
- ❑ Exercises and Projects — Bearing Block Assembly
  - Opening the Assembly
  - Placing Washers
  - Placing Screws
  - Thread Display
  - Making Sure the Screws Fit
  - Modifying Toolbox Parts
- ❑ More to Explore — Add Hardware to an Assembly
- ❑ Lesson Summary

Competencies for Lesson 5

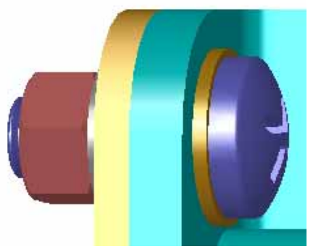
Students develop the following competencies in this lesson:

- ❑ **Engineering**: Select fasteners automatically based on hole diameter and depth. Utilize fastener vocabulary such as thread length, screw size, and diameter.
- ❑ **Technology**: Utilize the Toolbox Browser and display of thread style.
- ❑ **Math**: Relate diameter of screw to screw size.
- ❑ **Science**: Explore fasteners created from different materials.

In Class Discussion — What is Toolbox?



Toolbox includes a library of standard parts that are fully integrated with SolidWorks. These parts are ready-to-use components — such as bolts and screws.



To add these parts to an assembly, select the type of part you want to insert, then drag the Toolbox part into your assembly. As you drag Toolbox parts, they snap to the appropriate surfaces — automatically establishing a mate relationship. In other words, a screw recognizes that it belongs in a hole and snaps to it by default.

As you are placing the Toolbox parts, you can edit the property definitions to correctly size the Toolbox part to your needs. Holes created with the hole wizard are easy to match with properly-sized hardware from Toolbox.

The Toolbox Browser library of ready-to-use parts saves you the time that you would usually spend creating and adapting these parts if you built them yourself. With Toolbox, you have a complete catalog of parts.

Toolbox supports international standards such as ANSI, BSI, CISC, DIN, ISO, and JIS. In addition, Toolbox also includes standard parts libraries from leading manufacturers such as PEM®, Torrington®, Truarc®, SKF®, and Unistrut®.

Active Learning Exercises — Adding Toolbox Parts

Follow the instructions in *Productivity Enhancements: Toolbox* in the SolidWorks Tutorials. Then proceed with the exercise below.

Add screws to the switchplate using the predefined hardware in Toolbox.

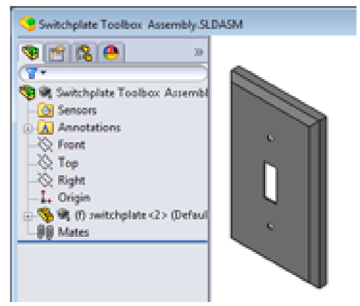
In the previous lesson, you added screws to the switchplate by modeling the screws and mating them to the switchplate in an assembly. As a general rule, hardware — such as screws — are standard components. Toolbox gives you the ability to apply standard hardware to assemblies without having to model it first.

Open the Switchplate Toolbox Assembly

Open the `Switchplate Toolbox Assembly`.

Notice that this assembly only has one part — or component — in it. `Switchplate` is the only part in the assembly.

An assembly is where you combine parts together. In this case, you are adding the screws to the switchplate.



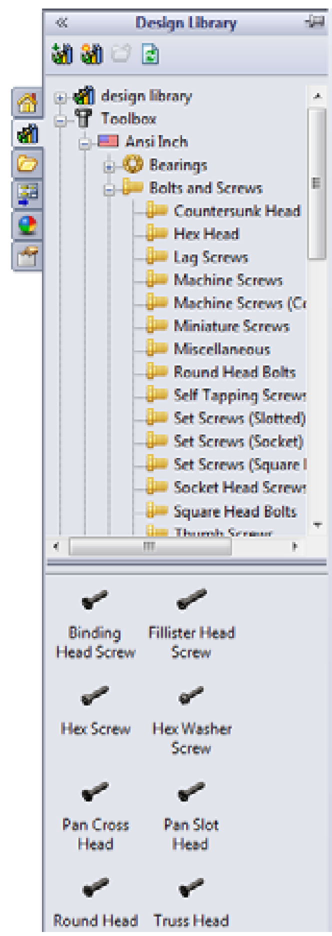
Open Toolbox Browser

Expand the Toolbox item 

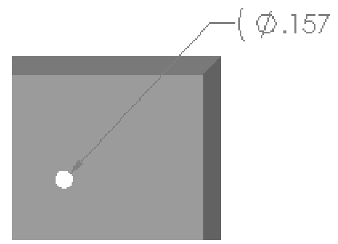
on the Design Library Task Pane. The Toolbox Browser appears.

The Toolbox Browser is an extension of the Design Library that contains all available Toolbox parts.

The Toolbox Browser is organized like a standard Windows Explorer folder view.



Selecting the Appropriate Hardware

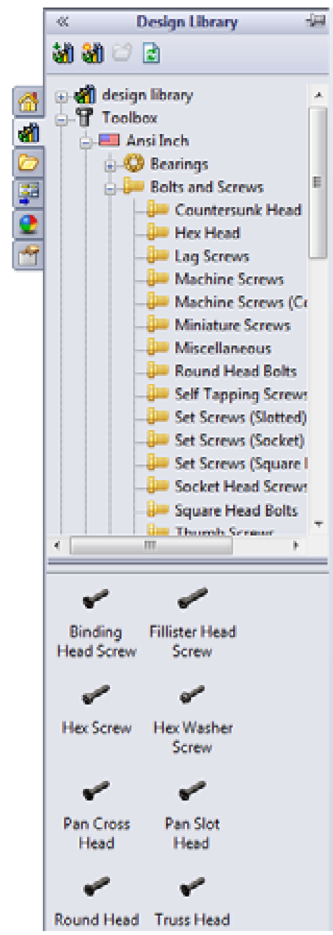


Toolbox contains a wide variety of hardware. Selecting the right hardware is often critical to the success of a model.

You must determine the size of the holes before selecting the hardware to use and match the hardware to the hole.

1 Click **Smart Dimension** on the Dimensions/ Relations toolbar or **Measure** on the Tools toolbar and select one of the holes on the switchplate to determine the hole size.

Note: The dimensions in this lesson are shown in inches.



2 In the Toolbox Browser, browse to **Ansi Inch**, **Bolts and Screws**, **Machine Screws** in the folder structure.

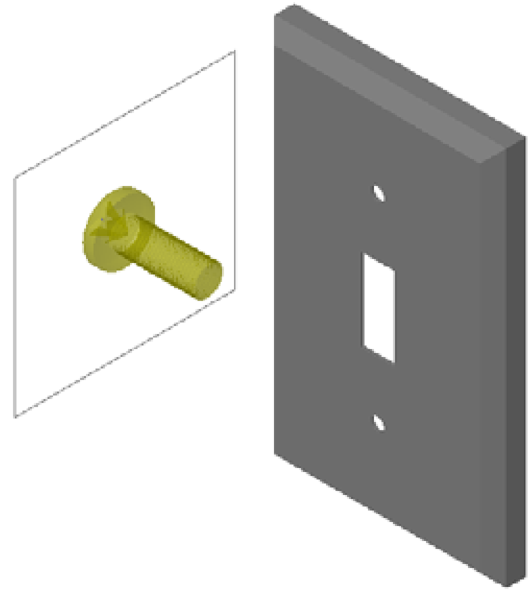
The valid types of machine screws display.

3 Click and hold **Pan Cross Head**.

Does this hardware selection make sense for this assembly? The switchplate was designed with the size of the fasteners in mind. The holes in the switchplate are specifically designed for a standard fastener size.

The fastener size is not the only consideration in selecting a part. The type of fastener is important too. For example, you would not use miniature screws or square head bolts for the switchplate. They are the wrong size. They would be either too small or too large. You also have to take into consideration the user of this product. This switchplate has to be attachable with the most common of household tools.

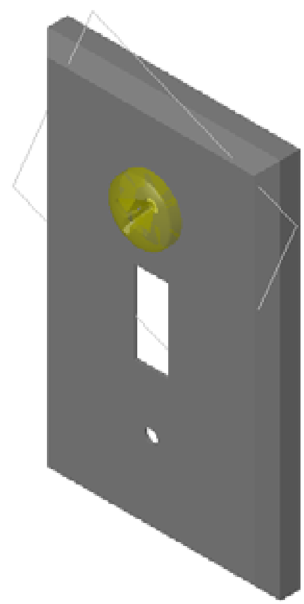
Placing Hardware



1 Drag the screw towards the switchplate.

As you begin to drag the screw, it may appear very large.

Note: Drag and drop parts by holding the left mouse button. Release the mouse button when the part is correctly oriented.



2 Slowly drag the screw towards one of the switchplate holes until the screw snaps into the hole.

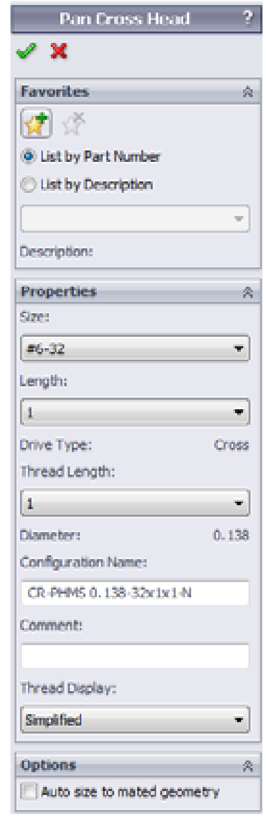
When the screw snaps into the hole, it is correctly oriented and properly mates with the surfaces of the part that it is combined with.

The screw still may appear too large for the hole.

3 When the screw is in the correct position, release the mouse button.

Specifying the Properties of the Toolbox Part

After you release the mouse button, a PropertyManager appears.



1 If necessary, change the properties of the screw to match the holes. In this case, a #6-32 screw with 1” length works with these holes.

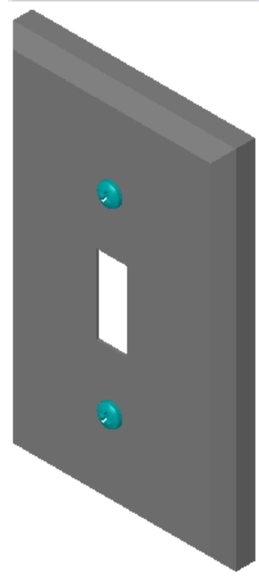
2 When you have completed the property changes, click **OK** .

The first screw is now placed in the first hole.

3 Repeat the process for the second hole.

You should not have to change any of the screw properties for the second screw. Toolbox remembers your last selection.

Both screws are now in the switchplate.



Lesson 5 — 5 Minute Assessment — Answer Key

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How would you determine the size of a screw to place in an assembly?

Answer: Measure the hole and the thickness of the material that the screw has to go through. The hole size determines the size of the screw. The thickness of the material determines the length of the screw.

2 In which window do you find ready-to-use hardware components?

Answer: Toolbox Browser.

3 True or False: Parts from Toolbox automatically size to the components they are being placed on.

Answer: False.

4 True or False: Toolbox parts can only be added to assemblies.

Answer: True

5 How can you resize components as you are placing them?

Answer: Use the window that pops up to change the part properties.

Lesson 5 — 5 Minute Assessment **REPRODUCIBLE**

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How would you determine the size of a screw to place in an assembly?

____

____

____

2 In which window do you find ready-to-use hardware components?

____

3 True or False: Parts from Toolbox automatically size to the components they are being placed on.

____

4 True or False: Toolbox parts can only be added to assemblies.

____

5 How can you resize components as you are placing them?

____

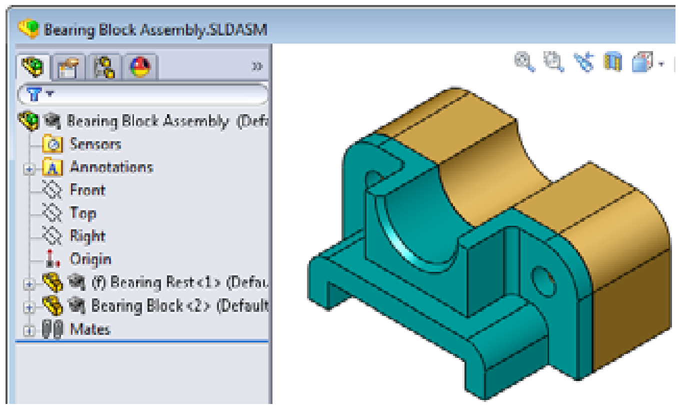
____

Exercises and Projects — Bearing Block Assembly

Add bolts and washers to fasten the bearing rest to the bearing block.

Opening the Assembly

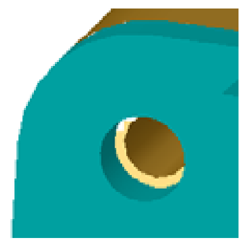
1 Open `Bearing Block Assembly`.



`Bearing Block Assembly` has `Bearing Rest` and `Bearing Block` as components.

In this exercise, you are going to bolt the bearing rest to the bearing block. The through holes in the bearing rest are designed to allow the bolts to pass through but not be loose. The holes in the bearing block are tapped holes. Tapped holes are threaded and specifically designed to act like nuts do. In other words, the bolt screws directly into the bearing block.

If you take a close look at the holes, you see that the holes in the bearing rest are larger than those of the bearing block. That is because the holes in the bearing block are represented with the amount of material needed for the creation of the screw threads. The screw threads are not visible. Threads are rarely shown in models.

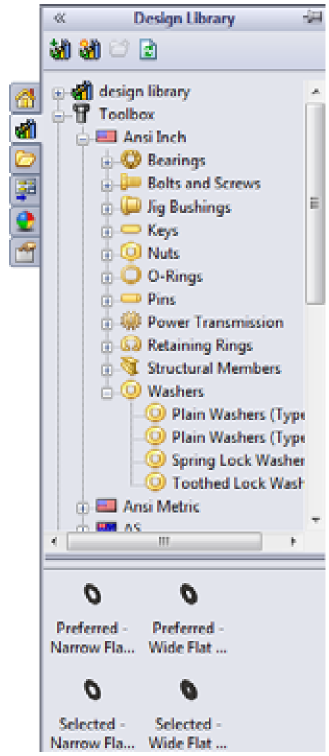


Placing Washers

Washers have to be placed before the screws or bolts. You do not have to use washers every time you place screws. However, when you do intend to use washers, they must be placed before screws, bolts, or nuts so that the correct relationships can be established.

The washers mate with the surface of the part and the screw or bolt mates with the washer. Nuts also mate with washers.

2 Expand the Toolbox Browser icon  in the Design Library Task Pane.



3 In the Toolbox Browser, browse for **Ansi Inch**, **Washers**, **Plain Washers (Type A)**.

The valid types of Type A Washers display.

4 Click and hold **Preferred - Narrow Flat Washer Type A** washer.

5 Slowly drag the washer towards one of the bearing rest through holes until the washer seems to snap onto the hole.

When the washer snaps onto the hole, it is correctly oriented and properly mates with the surfaces of the part that it is combined with.

The washer still may appear too large for the hole.

6 When the washer is in the correct position, release the mouse button.

After you release the mouse button, a pop-up window appears. This window enables you to edit the properties of the washer.

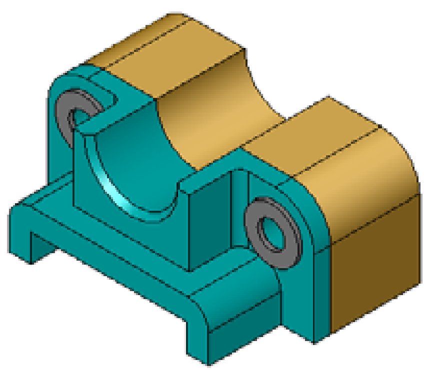
7 Edit the washer properties for a 3/8th hole and click **OK**.

The washer is placed.

Notice that the inside diameter is slightly larger than 3/8th. In general, the size of the washer indicates the size of the bolt or screw that must pass through it — not the actual size of the washer.

8 Place a washer on the other hole.

9 Close the **Insert Components** PropertyManager



Placing Screws

1 Select **Ansi Inch**, **Bolts and Screws**, and **Machine Screws** from Toolbox Browser.

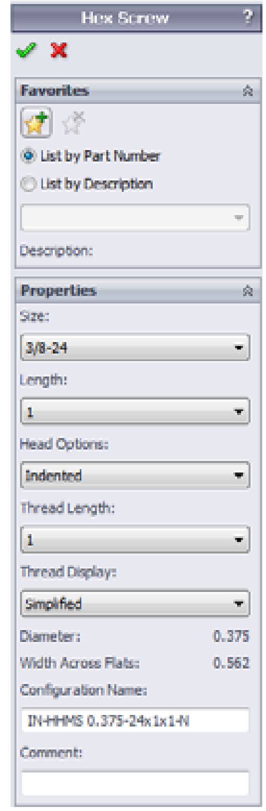
2 Drag a **Hex Screw** to one of the washers that you placed earlier.

3 Snap the screw into place and release the mouse button.

A window appears with the properties for the hex screw.

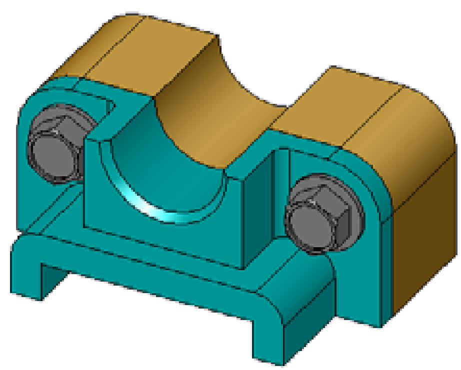
4 Select a 3/8-24 screw of the appropriate length and click **OK**.

The first screw is placed. The screw establishes a mate relationship with the washer.



5 Place the second screw in the same way.

6 Close the **Insert Components** PropertyManager.



Thread Display

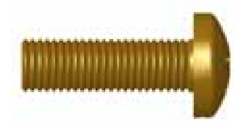
While fasteners such as bolts and screws are fairly detailed parts, they also very common ones. In general, bolts and screws are not the parts that you design. Instead you will use off-the-shelf hardware components. It is a well-established design practice to not draw all of the details of fasteners, but to specify their properties and show only an outline — or simplified — view of them.

The three display modes for bolts and screws are:

- Simplified — Represents the hardware with few details. This is the most common display. Simplified display shows the bolt or screw as if it were unthreaded.

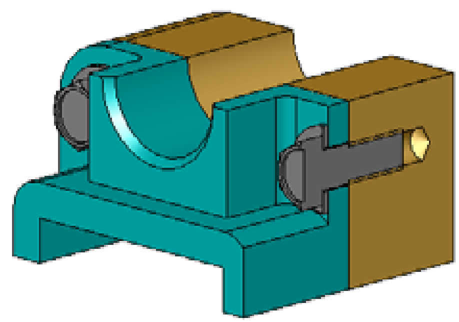
- ❑ Cosmetic — Represents some details of the hardware. Cosmetic display shows the barrel of the bolt or screw and represents the size of the threads as dashed lines.

- ❑ Schematic — Very detailed display which is rarely used. Schematic shows the bolt or screw as it really appears. This display is best used when designing a unique fastener or when specifying an uncommon one.



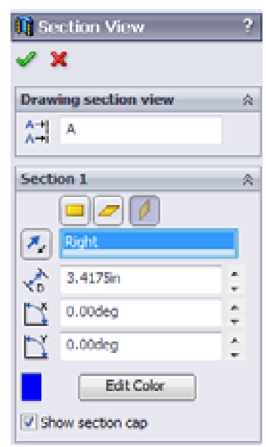
Making Sure That the Screws Fit

Before you placed the washers and screws, you should have measured the depth of the holes and the thickness of the washer as well as the diameter of the holes.



Even if you measured before placing the hardware, it is a good practice to verify that the screw fits as you intended it to. Viewing the assembly in wireframe, viewing it from different angles, using **Measure**, or creating a section view are some ways to do this.

A section view lets you look at the assembly as if you took a saw and cut it open.



1 Click **Section View** on the View toolbar.

The **Section View** PropertyManager appears.

2 Select **Right** as the **Reference Section Plane**.

3 Specify **3.4175** as the **Offset Distance**.

4 Click **OK**.

Now you see the cut away of the assembly right down the center of one of the screws. Is the screw long enough? Is it too long?

5 Click **Section View** again to turn off of the section view.

Modifying Toolbox Parts

If the screws — or other parts placed from Toolbox — are not the correct size you can modify their properties.

1 Select the part to modify, right-click, and select **Edit Toolbox Definition**.

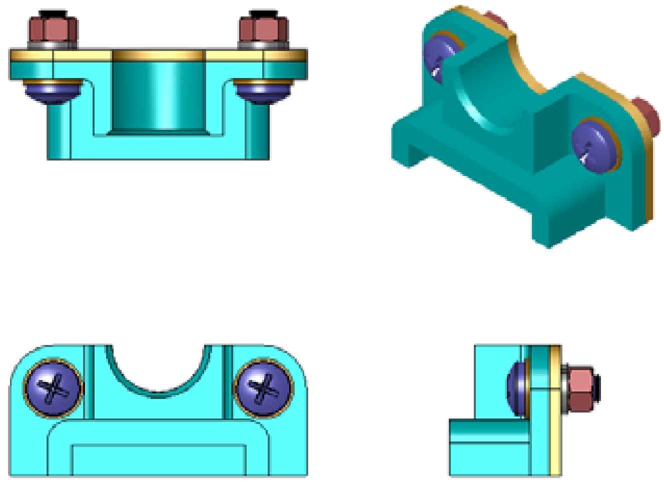
A PropertyManager appears with the name of the Toolbox part. It is the window that you used to specify the properties of Toolbox parts as you were placing them.

2 Modify the part properties and click **OK**.

The Toolbox part changes.

Note: After modifying parts, you should rebuild the assembly.

More to Explore — Add Hardware to an Assembly



In the previous exercise you used Toolbox to add washers and screws to an assembly. In that assembly, the screws went into blind holes. In this exercise, add washers, lock washers, screws, and nuts to an assembly.

1 Open `Bearing Plate Assembly`.
2 Add the washers (**Preferred - Narrow Flat Washer Type A** parts) to the through holes on the bearing rest first. The holes are 3/8th diameter.
3 Add the lock washers (**Regular Spring Lock Washer** parts) to the far side of the plate next.
4 Add 1-inch machine screws with a pan cross head. Snap these to the washers on the bearing rest.
5 Add hex nuts (**Hex Nut** parts). Snap these to the lock washers.
6 Use the techniques that you have learned to verify that the hardware is the correct size for this assembly.

Lesson 5 Vocabulary Worksheet — Answer Key

Name: ______________________________Class: _________ Date:_______________

Fill in the blanks with the words that are defined by the clues.

1 View that lets you look at the assembly as if you took a saw and cut it open: **Section view**

2 Type of hole that allows a screw or bolt to be screwed directly into it: **Tapped hole**

3 Common design practice that represents the screws and bolts showing outlines and few details: **Simplified**

4 Method for moving a Toolbox part from the Toolbox Browser to the assembly: **Drag and drop**

5 Area of Design Library Task Pane that contains all available Toolbox parts: **Toolbox Browser**

6 A file where you where you combine parts together: **Assembly**

7 Hardware — such as screws, nuts, washers, and lock washers — that you can select from the Toolbox Browser: **Toolbox parts**

8 Type of hole that allows a screw or bolt into it, but is not tapped: **Through hole**

9 Properties — such as size, length, thread length, display type — that describe a Toolbox part: **Toolbox definition**

Lesson 5 Vocabulary Worksheet **REPRODUCIBLE**

Name: ______________________________Class: _________ Date:______________

Fill in the blanks with the words that are defined by the clues.

1 View that lets you look at the assembly as if you took a saw and cut it open:_________
____

2 Type of hole that allows a screw or bolt to be screwed directly into it:_____________
____

3 Common design practice that represents the screws and bolts showing outlines and few details:____

4 Method for moving a Toolbox part from the Toolbox Browser to the assembly: ______
____

5 Area of Design Library Task Pane that contains all available Toolbox parts: _________
____

6 A file where you where you combine parts together:___________________________

7 Hardware — such as screws, nuts, washers, and lock washers — that you can select from the Toolbox Browser: ____

8 Type of hole that allows a screw or bolt into it, but is not tapped:_________________
____

9 Properties — such as size, length, thread length, display type — that describe a Toolbox part:____

Lesson 5 Quiz — Answer Key

Name: ______________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you establish a mate relationship between a Toolbox part and the part it is being placed on?

Answer: The mate relationship is established when the Toolbox part snaps to the other part. You do not have to explicitly define the relationship.

2 What does **Edit Toolbox Definition** enable you to change?

Answer: Toolbox part properties such as size, thread display, and length.

3 If you need a washer for a 3/8th diameter screw or bolt, is the inside dimension of the washer also 3/8th? If not, why not?

Answer: The inside diameter of washers is slightly larger than the outside dimension of the screw or bolt that it is combined with. This allows the screw or bolt to pass through it.

4 How would you determine the correct length of a machine screw that fastens two parts using a washer, lock washer, and nut?

Answer: Measure the thickness of both parts, the washer, the lock washer, and nut. Use a screw that is the next size longer so that the threads of the screw engage all of the threads of the nut.

5 How do you select a lock washer from Toolbox?

Answer: In the Toolbox Browser, select **Ansi Inch** (or other standard), **Washers**, and **Spring Lock Washers**.

6 True or False. To place a Toolbox part you have to specify the exact X, Y, Z coordinates.

Answer: False.

7 How do you specify the location of a Toolbox part?

Answer: You place Toolbox parts by dragging them and dropping them in the assembly.

8 How would you measure hole size?

Answer: Use either the **Measure** or **Dimension** commands.

9 True or False. Screw threads are always displayed in Schematic mode — showing all details.

Answer: True

Lesson 5 Quiz REPRODUCIBLE

Name: ________________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you establish a mate relationship between a Toolbox part and the part it is being placed on? ____

____

2 What does **Edit Toolbox Definition** enable you to change? ______________________

____

3 If you need a washer for a 3/8th diameter screw or bolt, is the inside dimension of the washer also 3/8th? If not, why not? ____

____

4 How would you determine the correct length of a machine screw that fastens two parts using a washer, lock washer, and nut? ____

____

5 How do you select a lock washer from Toolbox? ______________________________

____

6 True or False. To place a Toolbox part you have to specify the exact X, Y, Z coordinates. ____

7 How do you specify the location of a Toolbox part? ____________________________

____

8 How would you measure hole size? ____

____

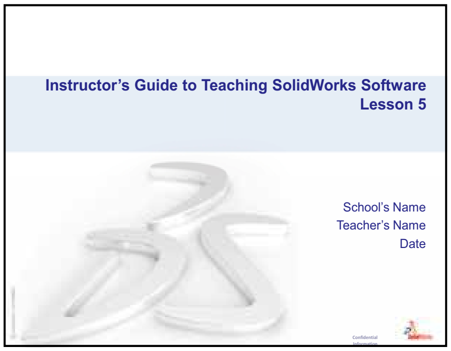
9 True or False. Screw threads are always displayed in Schematic mode — showing all details. ____

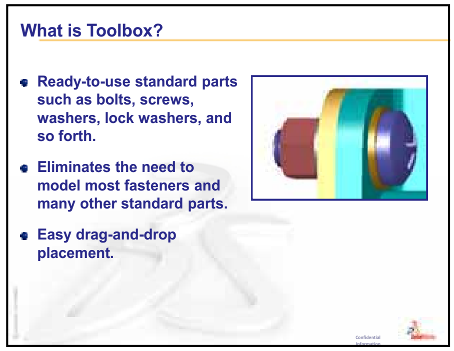
Lesson Summary

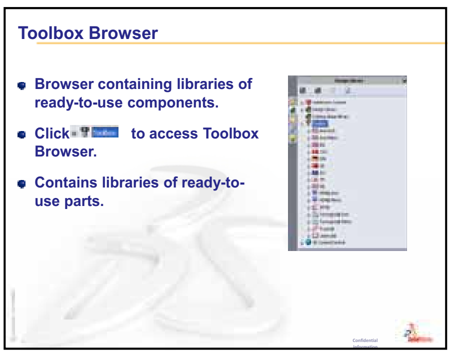
- Toolbox provides ready-to-use parts — such as bolts and screws.
- Toolbox parts are placed by dragging and dropping them in assemblies.
- You can edit the property definitions of Toolbox parts.
- Holes created with the hole wizard are easy to match with properly-sized hardware from Toolbox.

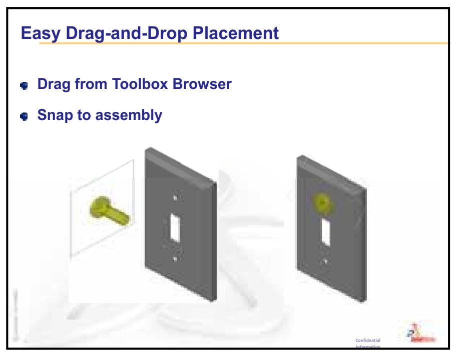
Thumbnail Images of PowerPoint Slides

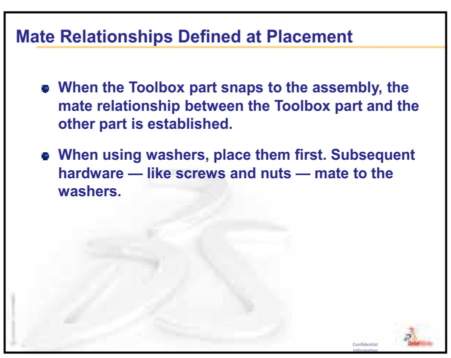
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

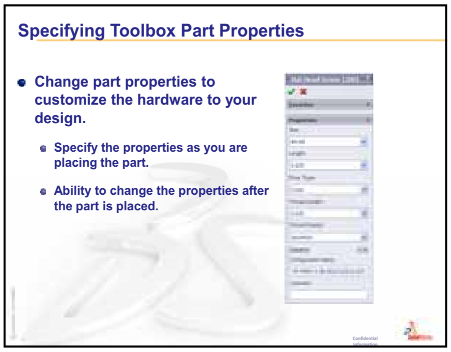


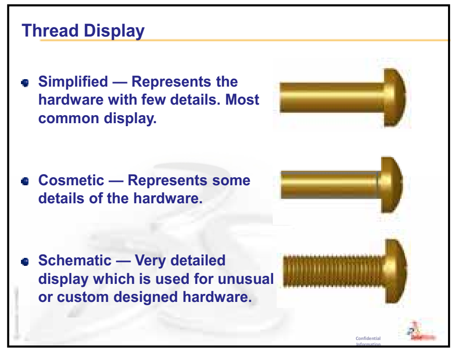




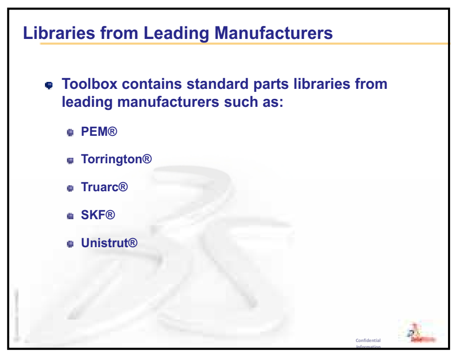





Specifying Toolbox Part Properties
Change part properties to customize the hardware to your design.
Specify the properties as you are placing the part.
Ability to change the properties after the part is placed.


Thread Display
Simplified — Represents the hardware with few details. Most common display.
Cosmetic — Represents some details of the hardware.
Schematic — Very detailed display which is used for unusual or custom designed hardware.


Supported Standards
Toolbox supports international standards
ANSI
BSI
CISC
DIN
ISO
JIS

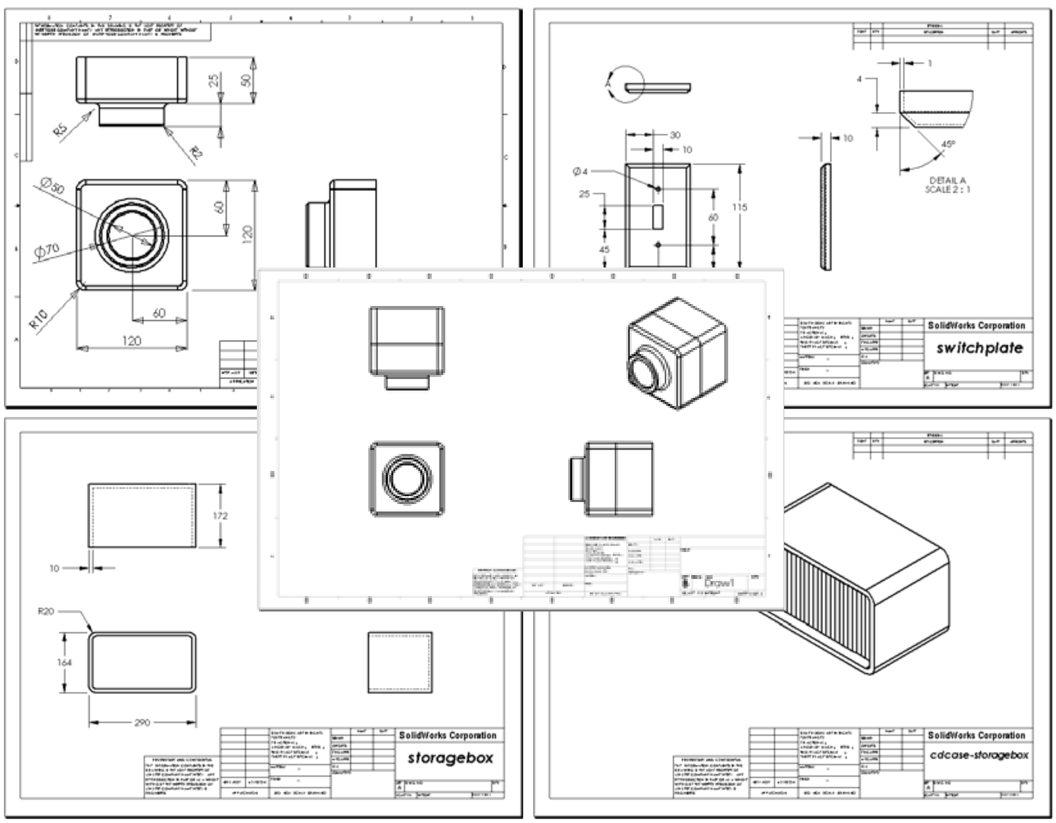

Libraries from Leading Manufacturers
Toolbox contains standard parts libraries from leading manufacturers such as:
PEM®
Torrington®
Truarc®
SKF®
Unistrut®

6

Lesson 6: Drawing Basics

Goals of This Lesson

- ❑ Understand basic drawing concepts.
- ❑ Create detailed drawings of parts and assemblies.



Before Beginning This Lesson

- ❑ Create `Tutor1` part from Lesson 3: The 40-Minute Running Start.
- ❑ Create `Tutor2` part and the `Tutor` assembly from Lesson 4: Assembly Basics.

Drawing skills are required by industry. Review industry examples, case studies and white papers at www.solidworks.com.

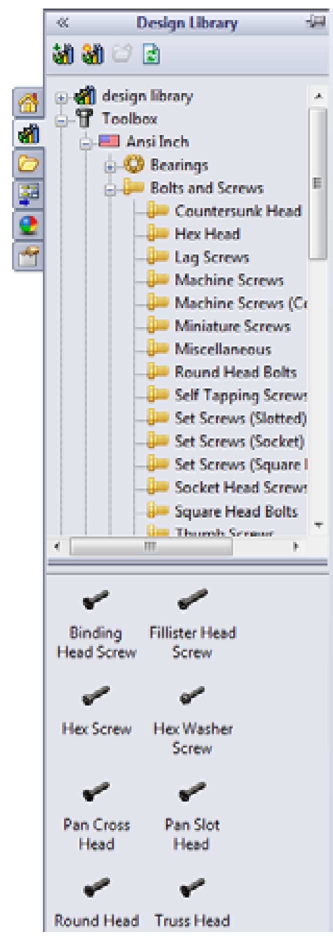
Resources for This Lesson

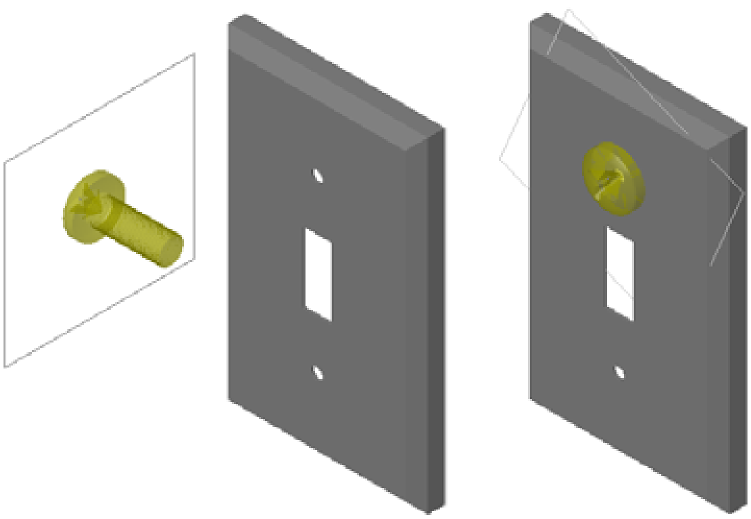
This lesson plan corresponds to *Getting Started: Lesson 3 – Drawings* in the SolidWorks Tutorials.

Additional information about drawings can be found in the *Working with Models: Advanced Drawings* lesson in the SolidWorks Tutorials.

Review of Lesson 5: SolidWorks Toolbox Basics

- ❑ Toolbox includes ready-to-use standard parts such as bolts, screws, washers, lock washers, and so forth.
- ❑ Eliminates the need to model most fasteners and many other standard parts.
- ❑ Toolbox Browser contains libraries of ready-to-use components.
- ❑ Easy drag-and-drop placement.
- ❑ Toolbox parts snap to assemblies.
- ❑ When the Toolbox part snaps to the assembly, the mate relationship between the Toolbox part and the other part is established.





Outline of Lesson 6

- ❑ In Class Discussion — Understanding Engineering Drawings
  - Engineering Drawings
  - General Drawing Rules – Views
  - General Drawing Rules – Dimensions
  - Editing the Title Block
- ❑ Active Learning Exercises — Creating Drawings
- ❑ Exercises and Projects — Creating a Drawing
  - Create a Drawing Template
  - Create a Drawing for Tutor2
  - Add a Sheet to an Existing Drawing
  - Add a Sheet to an Existing Assembly Drawing
- ❑ More to Explore — Creating a Parametric Note
- ❑ More to Explore — Add a Sheet to Switchplate Drawing
- ❑ Lesson Summary

Competencies for Lesson 6

Students develop the following competencies in this lesson:

- ❑ **Engineering**: Apply engineering drawing standards to part and assembly drawings. Apply concepts of orthographic projection to 2D standard views and isometric views.
- ❑ **Technology**: Explore associativity between different, but related file formats that change during the design process.
- ❑ **Math**: Explore how numeric values describe overall size and features of a part.

In Class Discussion — Understanding Engineering Drawings

Note to the Teacher

These course materials about SolidWorks are not intended to replace courses in mechanical drafting, or engineering drawing. However, we recognize that in many cases, the students will not have a background in drafting. Therefore, we have provided some *basic* background information about drafting that you may wish to use in your course. This material is not intended to be a complete discussion of mechanical drafting. It is intended only as a brief introduction to some of the principals of view definition and dimensioning practices.

The overhead masters for this lesson include illustrations of the concepts below. You can duplicate these and hand them out to your students if you wish.

Engineering Drawings

Drawings communicate three things about the objects they represent:

- Their shape – *views* are used to communicate the *shape* of an object.
- Their size – *dimensions* are used to communicate the size of an object.
- Other information – *notes* communicate non-graphic information about manufacturing processes such as drill, ream, bore, paint, grind, heat treat, remove burrs, and so forth.

General Drawing Rules – Views

- The general characteristics of an object will determine what views are required to describe its shape.
- Most objects can be described using three properly selected views. Sometimes you can use fewer. However, sometimes more are needed.
- Sometimes specialized views such as auxiliary views or section views are needed to fully and accurately describe an object.

General Drawing Rules – Dimensions

- There are two kinds of dimensions:
  - Size dimensions – how big is the feature?
  - Location dimensions – where is the feature located?
- For flat pieces, give the thickness dimension in the edge view, and all other dimensions in the outline view.
- Dimension features in the view where they can be seen true size and shape.
- Use diameter dimensions for circles. Use radial dimensions for arcs.
- Omit unnecessary dimensions.
- Place dimensions away from the profile lines.
- Allow space between individual dimensions.

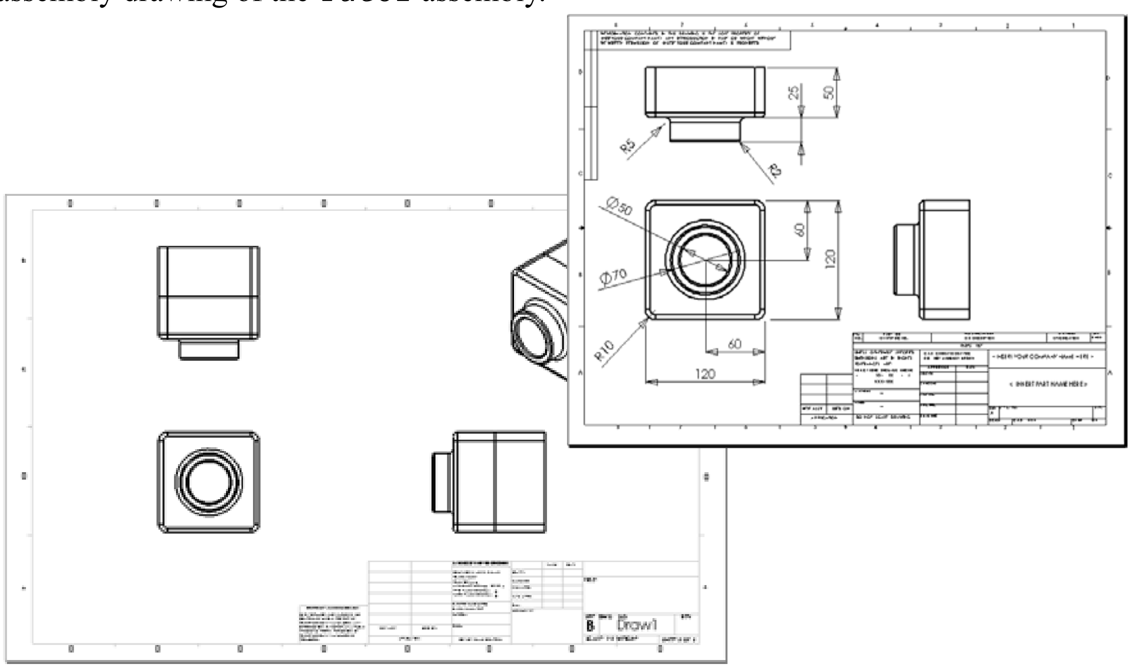
- ❑ A gap must exist between the profile lines and the extension lines.
- ❑ The size and style of leader line, text, and arrows should be consistent throughout the drawing.

Editing the Title Block

The masters for the overhead transparencies include a step-by-step procedure for customizing the part name in the title block so that the name of the referenced part or assembly is automatically filled in. This material is an *advanced topic* that may not be suitable for all classes. Use it at your discretion. Additional information about linking text notes to file properties can be found in the SolidWorks On-line Help. Click **Help, SolidWorks Help**, and locate the topic **Link to Property**.

Active Learning Exercises — Creating Drawings

Follow the instructions in *Getting Started: Lesson 3 – Drawings* in the SolidWorks Tutorials. In this lesson you will create two drawings. First, you will create the drawing for the part named `Tutor1` which you built in a previous lesson. Then you will create an assembly drawing of the `Tutor` assembly.



Lesson 6 — 5 Minute Assessment — Answer Key

Name: ______________________________Class: _________ Date:______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you open a drawing template?

Answer: Click **File**, **New**. Click the **Draw** icon.

2 What is the difference between **Edit Sheet Format** and **Edit Sheet**?

Answer: **Edit Sheet Format** provides the ability to change the title block size and text headings. **Edit Sheet** provides the ability to add or modify views, dimensions and or text. 99+% of the time you will work in **Edit Sheet** mode.

3 A title block contains information about the part and/or assembly. Name five pieces of information that can be contained in a title block.

Answer: Answers will vary but may include company name, part number, part name, drawing number, revision number, sheet number, material & finish, tolerance, drawing scale, sheet size, revision block and drawn by.

4 True or False. Right-click **Edit Sheet Format** to modify title block information.

Answer: True.

5 What three views are inserted into a drawing when you click **Standard 3 View**?

Answer: Front, Top, and Right. ***Note:*** *This answer applies when the type of view projection is third angle (as is almost universally the case in the United States). Most European countries use first angle projection which creates Front, Top, and Left views*.

6 How do you move a drawing view?

Answer: Click inside the view boundary. Drag the view by its border.

7 What command is used to import part dimensions into the drawing?

Answer: The command used to import part dimensions into a drawing is **Insert**, **Model Items.**

8 True or False. Dimensions must be clearly positioned on the drawing.

Answer: True.

9 Give four rules for good dimensioning practice.

Answer: Answers will vary but may include:
- For flat pieces, give the thickness dimension in the edge view, and all other dimensions in the outline view.
- Dimension features in the view where they can be seen true size and shape.
- Use diameter dimensions for circles.
- Use radial dimensions for arcs.
- Omit unnecessary dimensions.
- Place dimensions away from the profile lines.
- Allow space between individual dimensions.
- A gap must exist between the profile lines and the extension lines.
- The size and style of leader line, text, and arrows should be consistent.

Lesson 6 — 5 Minute Assessment **REPRODUCIBLE**

Name: ______________________________ Class: _________ Date: ______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you open a drawing template?

____

2 What is the difference between **Edit Sheet Format** and **Edit Sheet**?

____

____

____

3 A title block contains information about the part and/or assembly. Name five pieces of information that can be contained in a title block.

____

____

____

4 True or False. Right-click **Edit Sheet Format** to modify title block information.

____

5 What three views are inserted into a drawing when you click **Standard 3 View**?

____

6 How do you move a drawing view?

____

7 What command is used to import part dimensions into the drawing?

____

8 True or False. Dimensions must be clearly positioned on the drawing.

____

9 Give four rules for good dimensioning practice.

____

____

____

____

Exercises and Projects — Creating a Drawing

Task 1 — Create a Drawing Template

Create a new A-size ANSI standard drawing template.

For **Units** use millimeters.

Name the template `ANSI-MM-SIZEA`.

Procedure:

1. Create a new drawing using the Tutorial drawing template.
   This is an A-size sheet that uses the ISO drafting standard.
2. Click **Tools, Options** and then click the **Document Properties** tab.
3. Set the **Overall drafting standard** to **ANSI**.
4. Make any other desired changes to the document properties, such as the dimension text font and size.
5. Click **Units** and verify that the **Length** units are set to **millimeters**.
6. Click **OK** to apply the changes and close the dialog.
7. Click **File, Save As...**
8. From the **Save as type:** list, click **Drawing Templates (*.drwdot)**.
   The system automatically jumps to the directory where the templates are installed.
9. Click  to create a new folder.
10. Name the new folder `Custom`.
11. Browse to the `Custom` folder.
12. Enter `ANSI-MM-SIZEA` for the name.
13. Click **Save**.
    Drawing templates have the suffix `*.drwdot`

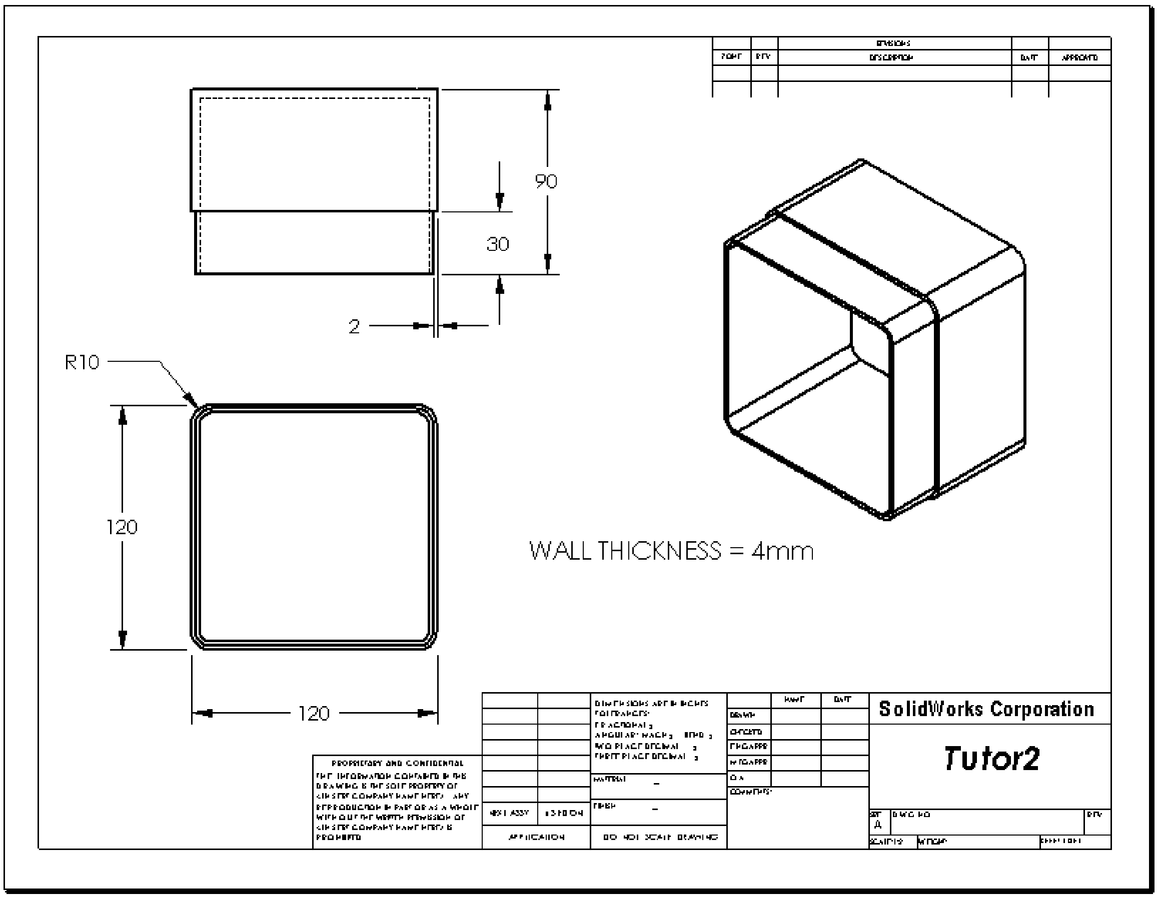
Task 2 — Create a Drawing for Tutor2

1 Create a drawing for `Tutor2`. Use the drawing template you created in Task 1.
Review the guidelines for determining which views are necessary. Since `Tutor2` is square, the top and right views communicate the same information. Only two views are necessary to fully describe the shape of `Tutor2`.

2 Create Front and Top views. Add an Isometric view.

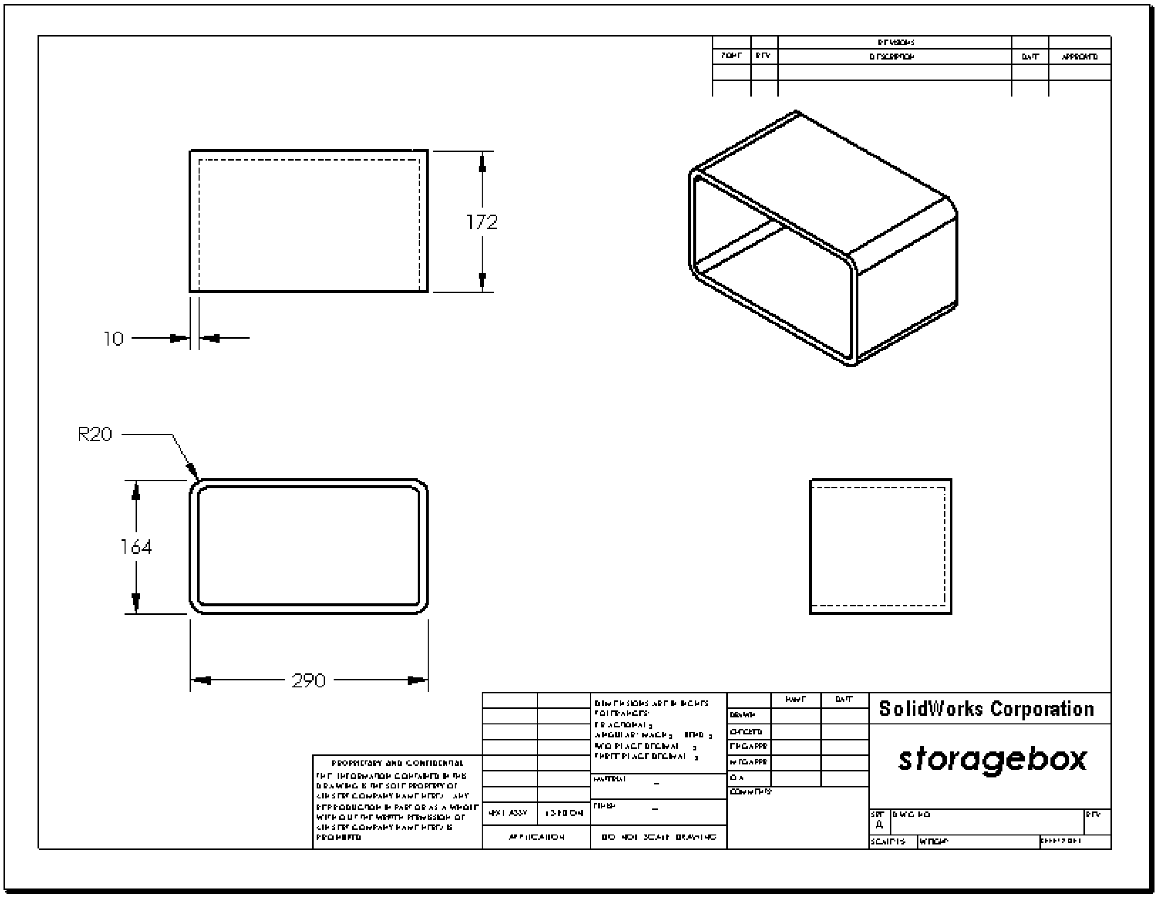
3 Import the dimensions from the part.

4 Create a note on the drawing to label the wall thickness.
Click **Insert, Annotations, Note**. Enter **WALL THICKNESS = 4mm**.



Task 3 — Add a Sheet to an Existing Drawing

1. Add a new sheet to the existing drawing you created in Task 2. Use the drawing template you created in Task 1.
2. Create three standard views for the `storagebox`.
3. Import the dimensions from the model.
4. Create an Isometric view in a drawing for the `storagebox`.



Note to the Teacher

Your students' designs and dimensions may vary from the ones illustrated here.

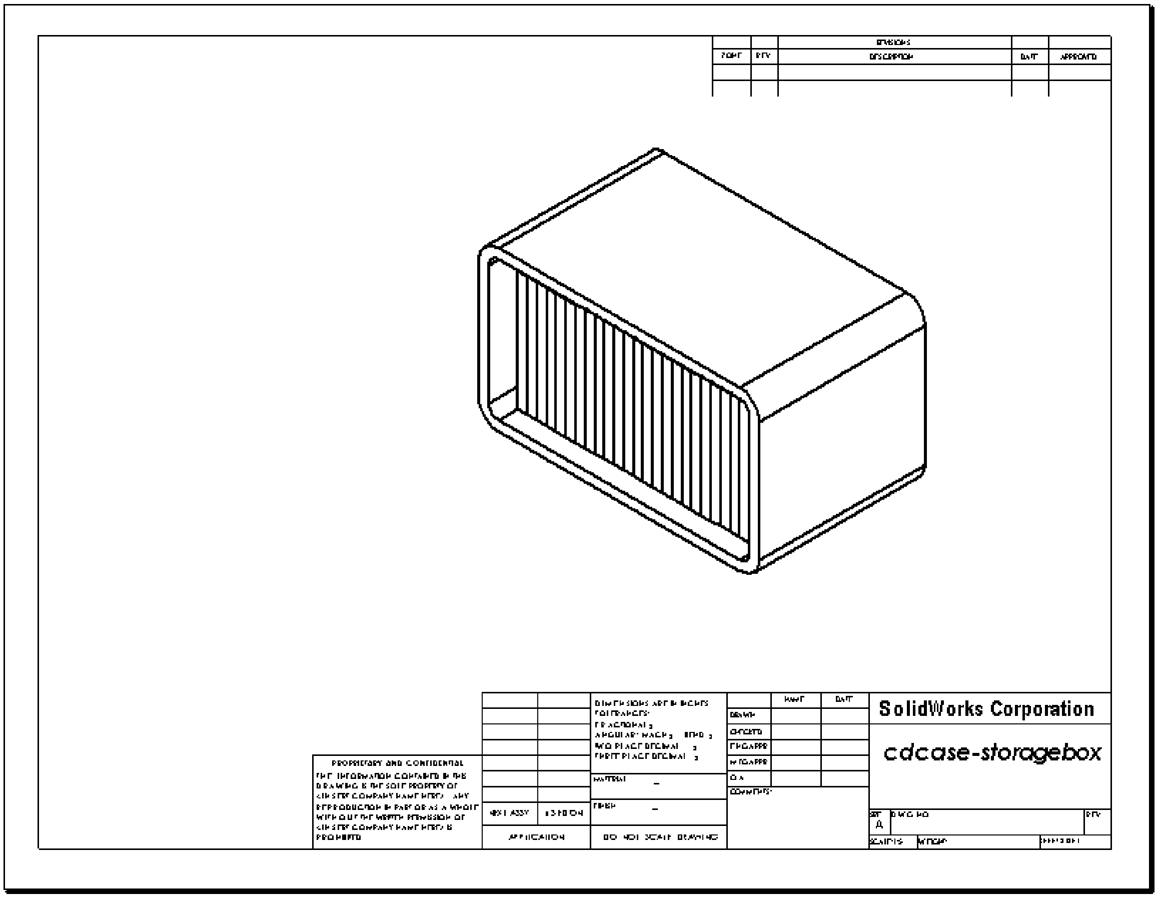
The drawing file is located in the `Lessons\Lesson06` folder in `SolidWorks Teacher Tools`. This file is named `Lesson6.SLDDRW`. The drawing file contains four sheets:

- Sheet 1 is the drawing for Task 2.
- Sheet 2 is the drawing for Task 3.
- Sheet 3 is the drawing for Task 4.
- Sheet 4 is the drawing for More to Explore - Add a Sheet to Switchplate Drawing.

Task 4 — Add a Sheet to an Existing Assembly Drawing

1 Add a new sheet to the existing drawing you created in Task 2. Use the drawing template you created in Task 1.

2 Create an Isometric view in a drawing for the `cdcase-storagebox` assembly.



More to Explore — Create a Parametric Note

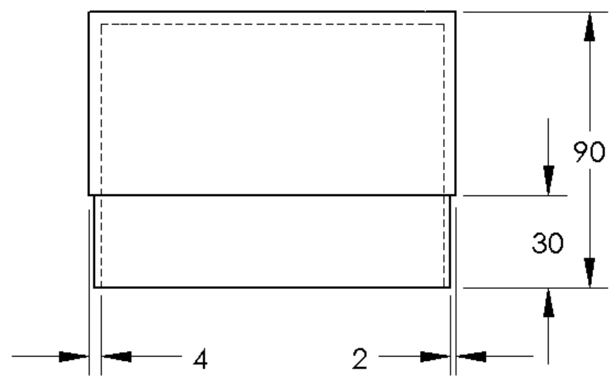
Investigate the on-line documentation to learn how to create a *parametric* note. In a parametric note, text, such as the numeric value of the wall thickness, is replaced with a dimension. This causes the note to update whenever the thickness of the shell is changed.

Once a dimension is linked to a parametric note, the dimension should *not* be deleted. That would break the link. However, the dimension can be hidden by right-clicking the dimension, and selecting **Hide** from the shortcut menu.

Note to the Teacher

The topic of creating a parametric note is an optional activity you might want to use as an independent study or enrichment activity with some of your more advanced students. To assist you in providing guidance to your students, the following is the procedure for creating a parametric note:

1 Import the model dimensions into the drawing.
When you import the dimensions from the model, the 4mm thickness dimension of the Shell feature will also be imported. This dimension is needed for the parametric note.



2 Click **Note**  on the Annotations toolbar or **Insert, Annotations, Note**.

3 Click to place the note on the drawing.
A text insertion box appears . Enter the note text. For example: **WALL THICKNESS =**

4 Select the dimension of the Shell feature.
Instead of typing the value, click the dimension. The system will enter the dimension into the text note.

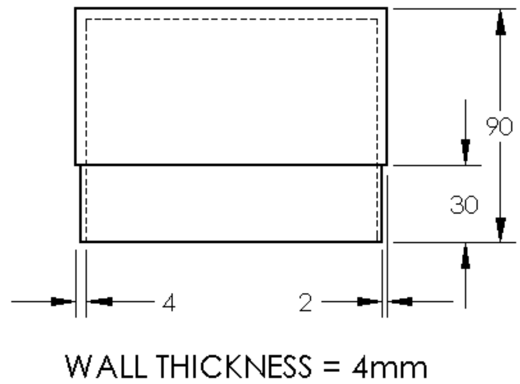
5 Type the rest of the note.
Make sure the text insertion cursor is at the end of the text string and type **mm**.

6 Click **OK** to close the **Note** PropertyManager.

Position the note on the drawing by dragging it.

7 Hide the dimension.

Right-click the dimension, and select **Hide** from the shortcut menu.



More to Explore — Add a Sheet to Switchplate Drawing

1 Add a new sheet to the existing drawing you created in Task 2. Use the drawing template you created in Task 1.

2 Create a drawing of the `switchplate`.

The chamfer is too small to be clearly seen and dimensioned in either the Top or Right views. A detail view is required. Detail views are views that usually show only a portion of the model, at a larger scale. To make a detail view:

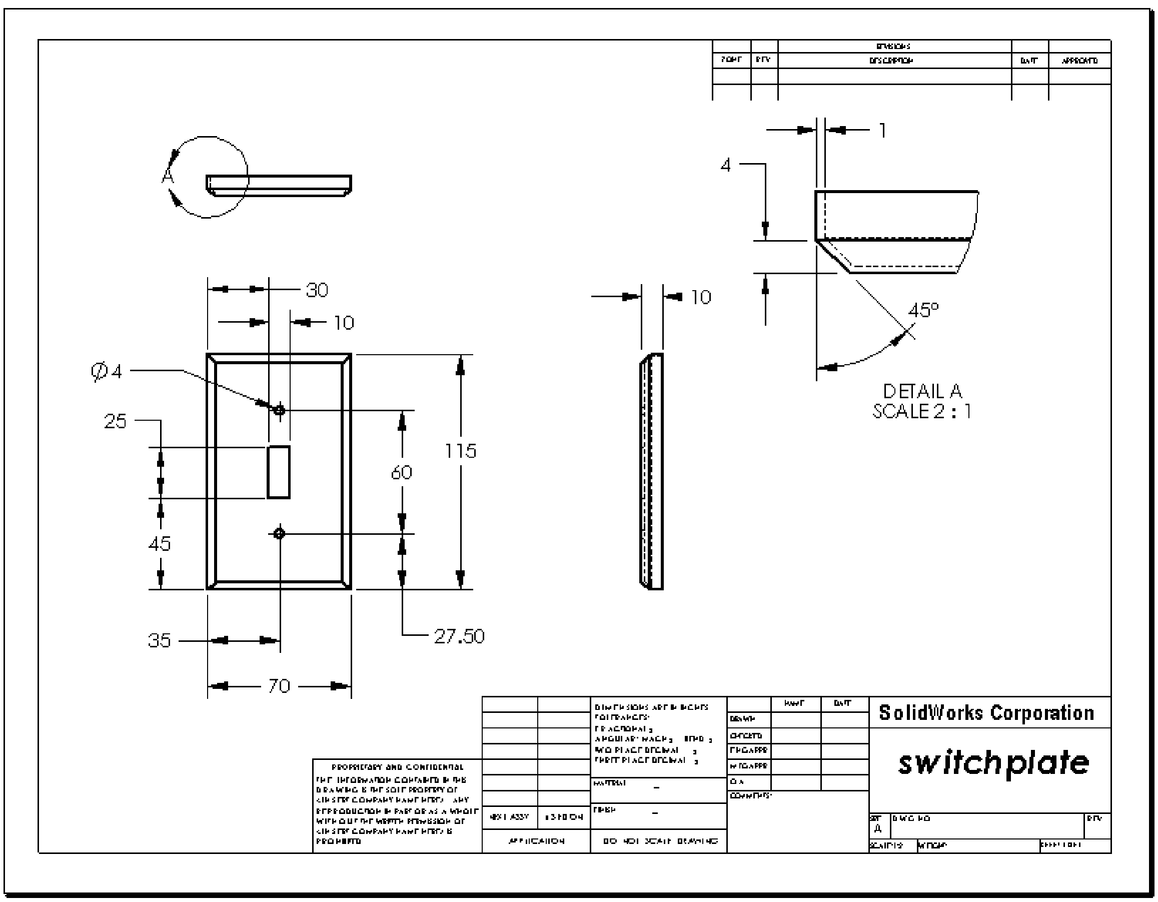
3 Select the view from which the detail view will be derived.

4 Click **Detail View** on the Drawing toolbar, or **Insert, Drawing View, Detail**.
This turns on the Circle sketch tool.

5 Sketch a circle around the area you want to show.
When you finish sketching the circle, a preview of the detail view appears.

6 Position the detail view on the drawing sheet.
The system automatically adds a label to the detail circle and the view itself. To change the scale of the detail view, edit the label's text.

7 You can import dimensions directly into a detail view, or drag them from other views.



Lesson 6 Quiz — Answer Key

Name: ______________________________ Class: _________ Date: ______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you begin a new drawing document?

Answer: To begin a new drawing document, click **File**, **New**. Select a drawing template.

2 What is the difference between **Edit Sheet Format** and **Edit Sheet**?

Answer: **Edit Sheet Format** provides the ability to change the title block size and text headings, incorporate a company logo and add drawing text. **Edit Sheet** provides the ability to add or modify views, dimensions and or text. **Edit Sheet** is used 99+% of the time.

3 Where on the drawing document would you find the name of the person who created the drawing?

Answer: The name of the person who created the drawing is located in the title block under Drawn by.

4 How do you modify the text size and text font of the part name in the title block?

Answer: To modify the title block part name, click **Edit Sheet Format**. Right-click **Properties**. Click **Font**.

5 How do you change the drawing standard from ISO to ANSI?

Answer: To change the drawing standard from ISO to ANSI, click **Tools**, **Options**. On the **Document Properties** tab, click **ANSI** for the **Overall drafting standard**.

6 Name the three standard drawing views.

Answer: The three standard drawing views are Front, Top, Right.

7 True or False. Dimensions used to detail the `Tutor2` drawing were created in the part.

Answer: True.

8 How do you move dimensions that have been placed on a drawing?

Answer: To move a dimension, click on the dimension text and drag to a new location.

9 When you modify an imported dimension on a drawing, what happens to the part?

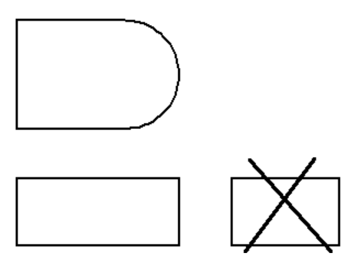
Answer: The part is also modified to reflect the changes.

10 What three types of information are found on engineering drawings?

Answer: *Views*, which communicate the *shape* of an object; *dimensions* which communicate the *size* of an object, and *notes*, which communicate *non-graphic information* about an object.

11 Good engineering drawings should have all the views necessary to describe the object, but no unnecessary views. In the illustration at the right, cross out the unnecessary view.

Answer: The right side view is not necessary.



Lesson 6 Quiz REPRODUCIBLE

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you begin a new drawing document?

____

____

2 What is the difference between **Edit Sheet Format** and **Edit Sheet**?

____

____

____

3 Where on the drawing document would you find the name of the person who created the drawing?

____

4 How do you modify the text size and text font of the part name in the title block?

____

____

5 How do you change the drawing standard from ISO to ANSI?

____

____

6 Name the three standard drawing views.

____

7 True or False. Dimensions used to detail the `Tutor2` drawing were created in the part.

____

8 How do you move dimensions that have been placed on a drawing?

____

9 When you modify an imported dimension on a drawing, what happens to the part?

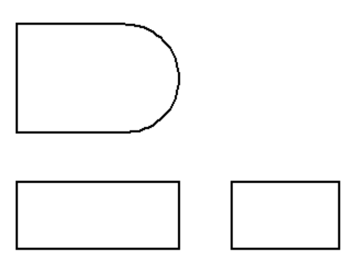
____

10 What three types of information are found on engineering drawings?

____

____

11 Good engineering drawings should have all the views necessary to describe the object, but no unnecessary views. In the illustration at the right, cross out the unnecessary view.

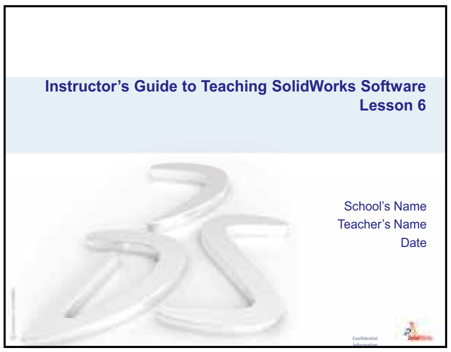


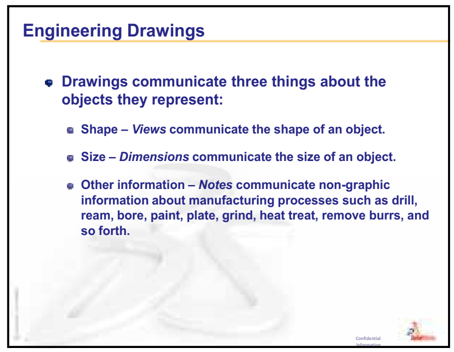
Lesson Summary

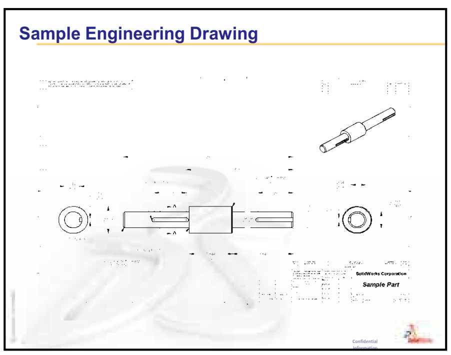
- Engineering Drawings communicate three things about the objects they represent:
  - Shape – *Views* communicate the shape of an object.
  - Size – *Dimensions* communicate the size of an object.
  - Other information – *Notes* communicate non-graphic information about manufacturing processes such as drill, ream, bore, paint, plate, grind, heat treat, remove burrs, and so forth.
- The general characteristics of an object will determine what views are required to describe its shape.
- Most objects can be described using three properly selected views.
- There are two kinds of dimensions:
  - Size dimensions – how big is the feature?
  - Location dimensions – where is the feature?
- A drawing template specifies:
  - Sheet (paper) size
  - Orientation - Landscape or Portrait
  - Sheet Format

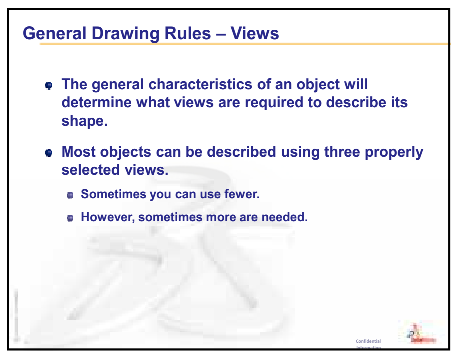
Thumbnail Images of PowerPoint Slides

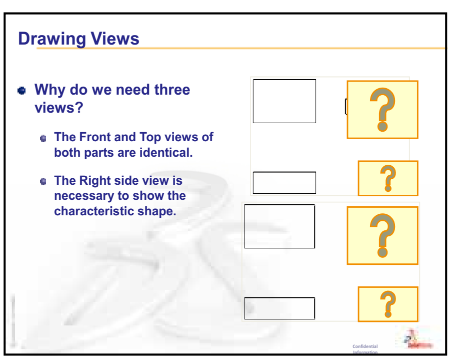
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

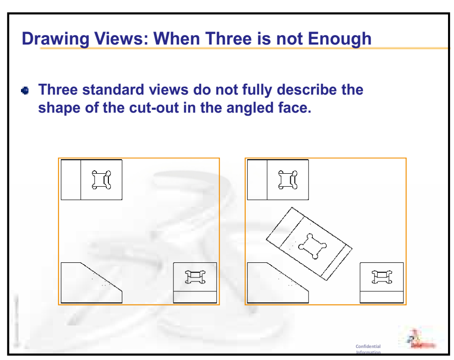






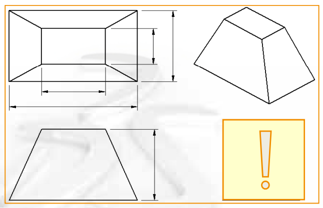






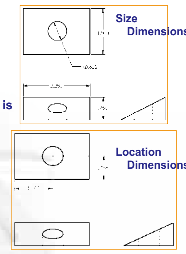
Drawing Views: When Three is too Many

- The Right side view is unnecessary.



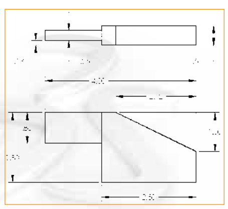
Dimensions

- There are two kinds of dimensions:
  - Size dimensions – how big is the feature?
  - Location dimensions – where is the feature?



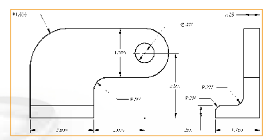
General Drawing Rules – Dimensions

- For flat pieces, give the thickness dimensions in the edge view, and all other dimensions in the outline view.



General Drawing Rules – Dimensions

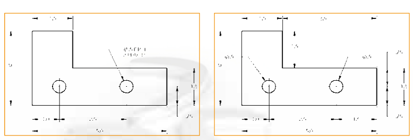
- Dimension features in the view where they can be seen true size and shape.



- Use diameter dimensions for circles.
- Use radial dimensions for arcs.

General Drawing Rules – Dimensions

- Omit unnecessary dimensions.

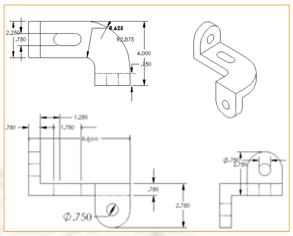


This Not This

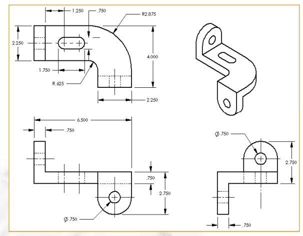
Dimension Guidelines – Appearance

- Place dimensions away from the profile lines.
- Allow space between individual dimensions.
- A gap must exist between the profile lines and the extension lines.
- The size and style of leader line, text, and arrows should be consistent throughout the drawing.
- Display only the number of decimal places required for manufacturing precision.
- Neatness counts!

Drawing Appearance – Not Good



Drawing Appearance – Much Better



What is a Drawing Template?

- A Drawing Template is the foundation for drawing information.

A drawing template specifies:

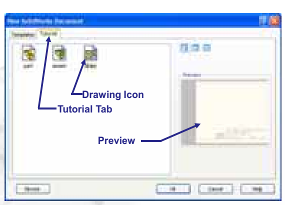
- Sheet (paper) size
- Orientation - Landscape or Portrait
- Sheet Format
  - Borders
  - Title block
  - Data forms and tables such as bill of materials or revision history

Drawing Templates Choices in SolidWorks

- Standard SolidWorks drawing template
- Tutorial drawing template
- Custom template
- No template

To Create a New Drawing Using a Document Template:

1. Click New on the Standard toolbar.
2. Click the Tutorial tab.
3. Double-click the drawing icon.



Sample Drawing Template

Edit Sheet vs. Edit Sheet Format

There are two modes in the drawing:

- Edit Sheet
  - This is the mode you use to make detailed drawings
  - Used 99+% of the time
  - Add or modify views
  - Add or modify dimensions
  - Add or modify text notes
- Edit Sheet Format
  - Change the title block size and text headings
  - Change the border
  - Incorporate a company logo
  - Add standard text that appears on every drawing

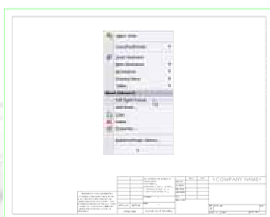
Title Block

- Contains vital part and/or assembly information.
- Each company can have a unique version of a title block.
- Typical title block information includes:

Company name	Material & Finish
Part number	Tolerance
Part name	Drawing scale
Drawing number	Sheet size
Revision number	Revision block
Sheet number	Drawn By/Checked By

To Edit the Title Block:

1. Right-click in the graphics area, and select Edit Sheet Format from the shortcut menu.

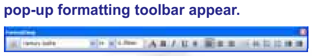


Editing the Title Block:

2. Zoom in on the title block.

Editing the Title Block:

3. Double-click the note that says <COMPANY NAME>. The PropertyManager and the pop-up formatting toolbar appear.



4. Enter your school name in the text insertion box.

Editing the Title Block:

5. Set the text justification to Align Left and change the size and style of the text font.

6. Click OK to apply the changes and close the PropertyManager.

Editing the Title Block:

7. Position the note so it is centered in the space.

Customizing the Part Name

Advanced Topic

- The name of the part or assembly shown on the drawing changes with every new drawing.
- It is not very efficient to have to edit the sheet format and the title block each time you make a new drawing.
- It would be nice if the title block would automatically be filled in with the name of the part or assembly that is shown on the drawing.
- This can be done.

Editing the Part Name:

Advanced Topic

1. Click Note on the Annotation toolbar, or click Insert, Annotations, Note.

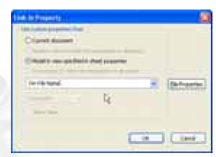
   The PropertyManager appears.

2. Click the Link to Property button.

Editing the Part Name:

Advanced Topic

3. Click Model in view specified in sheet properties, and choose SW-File Name from the list of properties.
4. Click OK to add the property.



Editing the Part Name:

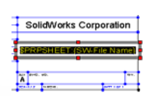
Advanced Topic

5. In the PropertyManager, set any other text properties such as justification, or font.

Editing the Part Name:

Advanced Topic

6. Click OK to apply the changes and close the PropertyManager.



Advanced Topic

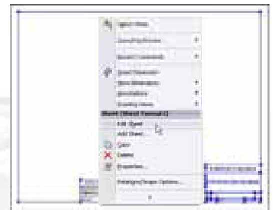
Advanced Topic

7. Results.

Currently the title block shows the text of the property. However, when the first view is added to the drawing, that text will change to become the file name of the referenced part or assembly.

Switching to Edit Sheet Mode:

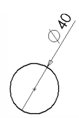
1. Right-click in the graphics area, and select Edit Sheet from the shortcut menu.
2. This is the mode you must be in when you make drawings.



Detailing Options

Dimensioning Standards

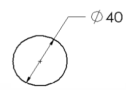
- Dimensioning standards determine things such as arrowhead style and dimension text position.
- The Tutorial drawing template uses the ISO standard.
- ISO stands for International Organization for Standardization.
- ISO is widely used in European countries.



Detailing Options

Dimensioning Standards

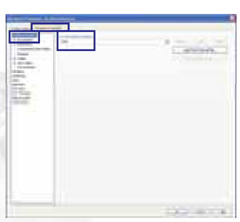
- ANSI is widely used in the United States.
- ANSI stands for American National Standards Institute.
- Other standards include BSI (British Standards Institution) and DIN (Deutsche Industries-Normen).
- Customize the drawing template to use the ANSI standard.



Detailing Options

Setting the dimensioning standard:

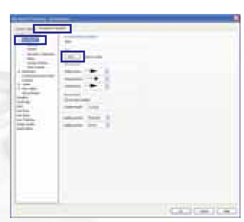
1. Click Tools, Options.
2. Click the Document Properties tab
3. Click Drafting Standard.
4. Select ANSI from the Overall drafting standard list.
5. Click OK.



Detailing Options

Setting text fonts:

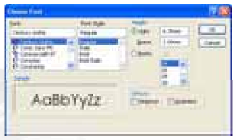
1. Click Tools, Options.
2. Click the Document Properties tab
3. Click Annotations.
4. Click Font.



Detailing Options

Setting text fonts continued:

5. The Choose Font dialog box opens.
6. Make the desired changes and click OK.



Saving a Custom Drawing Template:

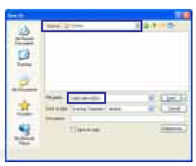
1. Click File, Save As...
2. From the Save as type: list, click Drawing Templates.

   The system automatically jumps to the directory where the templates are installed.
3. Click to create a new folder.

Saving a Custom Drawing Template:

4. Name the new folder *Custom*.
5. Browse to the *Custom* folder.
6. Enter *ANSI-MM-SIZEA* for the file name.
7. Click Save.

   Drawing templates have the suffix *.drwdot

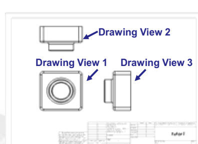


Creating a Drawing – General Procedure

1. Open the part or assembly you wish to detail.
2. Open a new drawing of the desired size.
3. Add views: usually three standard views plus any specialized views such as detail, auxiliary, or section views.
4. Insert the dimensions and arrange the dimensions on the drawing.
5. Add additional sheets, views and/or notes if required.

To Create Three Standard Views:

1. Click Standard 3 View .
2. Select Tutor1 from the Window menu.
3. Click OK.



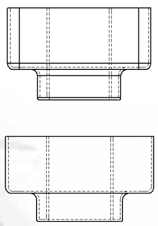
The drawing window reappears with the three views of the selected part.

Working with Drawing Views

- To select a view, click the view boundary. The view boundary is displayed in green.
- Drawing views 2 and 3 are aligned with view 1.
- Drag Drawing View1 (Front). Drawing View 2 (Top) and Drawing View 3 (Right) move, staying aligned to Drawing View1.
- Drawing View 3 can only be dragged left or right.
- Drawing View 2 can only be dragged up or down.

Working with Drawing Views

- Hidden line representation.
  - Hidden Lines Visible is usually used in orthographic views.
  - Hidden Lines Removed is usually used in isometric views.
- Tangent edge display.
  - Right-click inside the view border.
  - Select Tangent Edge, Tangent Edges Removed from the shortcut menu.



Dimensioning Drawings

- The dimensions used to create the part can be imported into the drawing.
- Dimensions can be added manually using the Smart Dimension tool .

Associativity

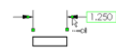
- Changing the values of imported dimensions will change the part.
- You cannot change the values of manually inserted dimensions.

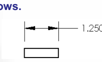
To Import Dimensions into the Drawing:

1. Click Model Items on the Annotation toolbar, or click Insert, Model Items.
2. Click the Import items into all views check box.
3. Click the option for Marked for drawing and Eliminate duplicates check box.
4. Click OK.

Manipulating Dimensions

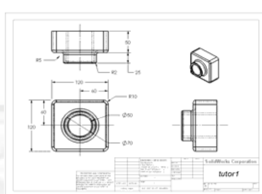
- Moving dimensions:
  - Click the dimension text.
  - Drag the dimension to the desired location.
  - To move a dimension into a different view, press and hold the Shift key while you drag it.
- Deleting dimensions:
  - Click the dimension text, and then press the Delete key.
- Flipping the arrows:
  - Click the dimension text.
  - A green dot appears on the dimension arrows.
  - Click the dot to flip the arrows in or out.





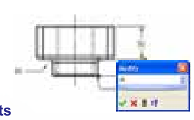
Finish the Drawing

- Position the views.
- Arrange the dimensions by dragging them.
- Set hidden line removal and tangent edge display.



Associativity

- Changing a dimension on the drawing changes the model.
  - Double-click the dimension text.
  - Enter a new value.
  - Rebuild.
- Open the part. The part reflects the new value.
- Open the assembly. The assembly also reflects the new value.

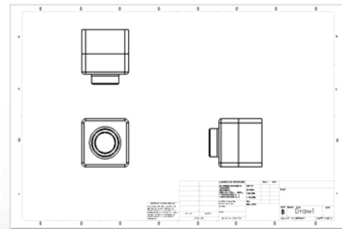


Multi-sheet Drawings

Drawings can contain more than one sheet.

- The first drawing sheet contains Tutor1.
- The second drawing sheet contains the Tutor assembly.
- Use the B-size landscape (11” x 17”) drawing Sheet Format.
- Add 3 standard views.
- Add an Isometric view of the assembly. The Isometric view is a model view.

Three View Drawing of Assembly

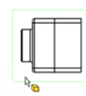


Model Views

- A model view shows the part or assembly in a specific orientation.
- Examples of model views are:
  - Standard Views such as Front, Top or Isometric view.
  - User-defined view orientations that were created in the part or assembly.
  - The current view in a part or assembly.

To Insert a model View:

1. Click Model View, or click Insert, Drawing view, Model.
2. Click inside the border of an existing view.



Important: Do not click directly on one of the parts in the assembly. Doing so will create a named view of that specific part.

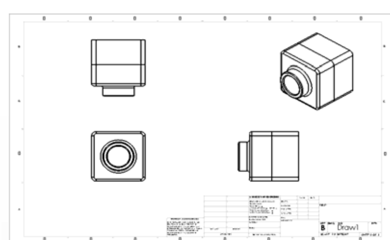
Inserting a Model View:

3. A selection of model view icons appears in the PropertyManager.

   Select the desired view, in this case, Isometric, from the selection.

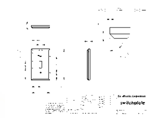
4. Place the view in the desired location on the drawing.

Isometric View Added to Drawing



Specialized Views

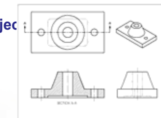
Detail View – used to show enlarged view of something.



1. Click Detail View, or click Insert, Drawing View, Detail.
2. Sketch a circle in the “source” view.
3. Position the view on drawing.
4. Edit the label to change scale.
5. Import dimensions or drag them into view.

Specialized Views

Section View – used to show internal aspects of objec



1. Click Section View, or click Insert, Drawing View, Section.
2. Sketch line in the “source” view.
3. Position the view on drawing.
4. Section view is automatically crosshatched.
5. Double-click section line to reverse arrows.

7

Lesson 7: SolidWorks eDrawings Basics

Goals of This Lesson

- ❑ Create eDrawings® files from existing SolidWorks files.
- ❑ View and manipulate eDrawings.
- ❑ Email eDrawings.

Before Beginning This Lesson

- ❑ Complete Lesson 6: Drawing Basics.
- ❑ An email application has to be loaded on the student's computer. If email is not present on the student's computer, you will not be able to complete *More to Explore- Emailing an eDrawings File*.
- ❑ Verify that eDrawings is set up and running on your classroom/lab computers. eDrawings is a SolidWorks add-in which is not loaded automatically. This add-in must be specifically added during installation.

Resources for This Lesson

This lesson plan corresponds to *Working with Models: SolidWorks eDrawings* in the SolidWorks Tutorials.

Save paper. To record your grades, use eDrawings and email.

Review of Lesson 6: Drawing Basics

Questions for Discussion

1 Name the three standard drawing views.

Answer: Front, Top and Right.

2 How do you move dimensions that have been placed in a drawing view?

Answer: Click on the dimension text. Drag the text to a new location.

3 How do you move a dimension from one view to another?

Answer: Hold down the **Shift** key while you drag the dimension.

4 You already have three standard views of a part on the drawing. How do you add an Isometric view?

Answer: Click **Model View** on the Drawing toolbar, or click **Insert, Drawing View, Model**. Click inside one of the existing views. Select **Isometric** from the **Orientation** list in the **Model View** PropertyManager. Position the view on the drawing.

Outline of Lesson 7

- In Class Discussion — eDrawings Files
- Active Learning Exercises — Creating an eDrawings File
  - Creating an eDrawings File
  - Viewing an Animated eDrawings File
  - Viewing Shaded and Wireframe eDrawings Files
  - Saving an eDrawings File
  - Markup and Measure
- Exercises and Projects — Exploring eDrawings Files
  - eDrawings of Parts
  - eDrawings of Assemblies
  - eDrawings of Drawings
  - Using the eDrawings Manager
  - The 3D Pointer
  - Overview Window
- More to Explore — Emailing an eDrawings File
- Lesson Summary

Competencies for Lesson 7

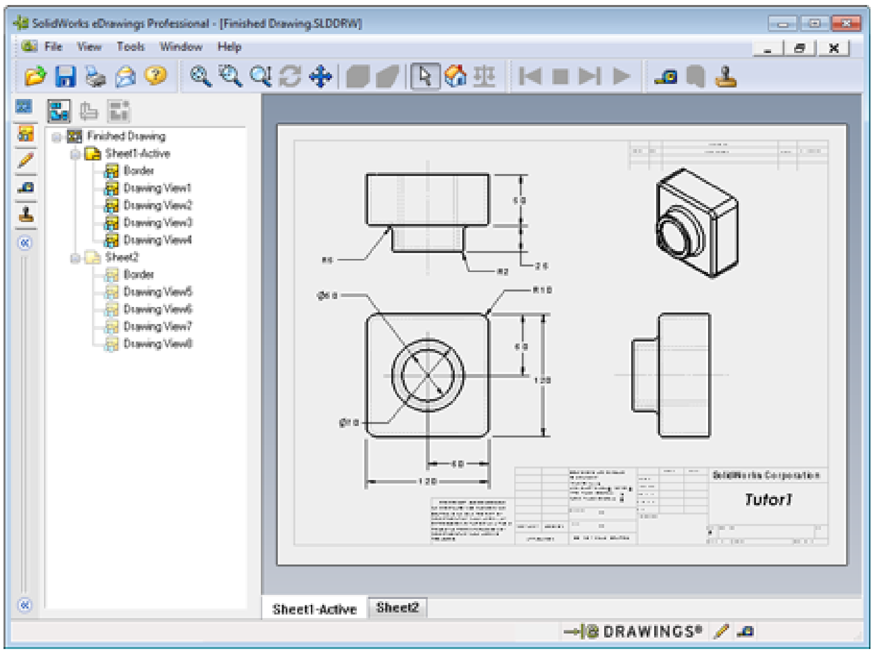
Students develop the following competencies in this lesson:

- **Engineering**: Mark up engineering drawings utilizing eDrawings comments. Understand how to communicate with manufacturing vendors.
- **Technology**: Work with different file formats including animations. Understand attachments for email.

In Class Discussion — eDrawings Files

SolidWorks eDrawings gives you the power to create, view, and share your 3D models and 2D drawings. You can create the following types of eDrawing files:

- ❑ 3D part files (*.eprt)
- ❑ 3D assembly files (*.easm)
- ❑ 2D drawing files (*.edrw)

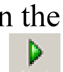


eDrawing files are small enough that you can share eDrawings with others by email. You can even send these files to others who do not have SolidWorks. eDrawings is an effective communication tool that enables you to work remotely from those reviewing your work. With eDrawings, they can easily look at your work and give you feedback.

eDrawings are not just static snapshots of parts, assemblies, and drawings. eDrawings can be viewed dynamically. This dynamic presentation is called animation.

Animation lets the recipient of an eDrawing view it from all angles, in all views, and at different scales. Graphic aids like the Overview Window, 3D Pointer, and Shaded mode help the eDrawing to clearly communicate.

eDrawing Toolbars

By default, when the eDrawings viewer starts, the toolbars are displayed with large buttons like this 

. This makes it easier to learn what the buttons do. However, you might want to use smaller buttons like this  to save screen space. To use small buttons, click **View, Toolbars, Large Buttons** in the eDrawings viewer. Clear the check mark in front of the menu listing. The remaining illustrations in this lesson are shown using small buttons.

Active Learning Exercises — Creating an eDrawings File

Follow the instructions in *Working with Models: SolidWorks eDrawings* in the SolidWorks Tutorials. Then proceed with the exercises below.

Create and explore an eDrawings file of the `switchplate` part created earlier.

Creating an eDrawings File

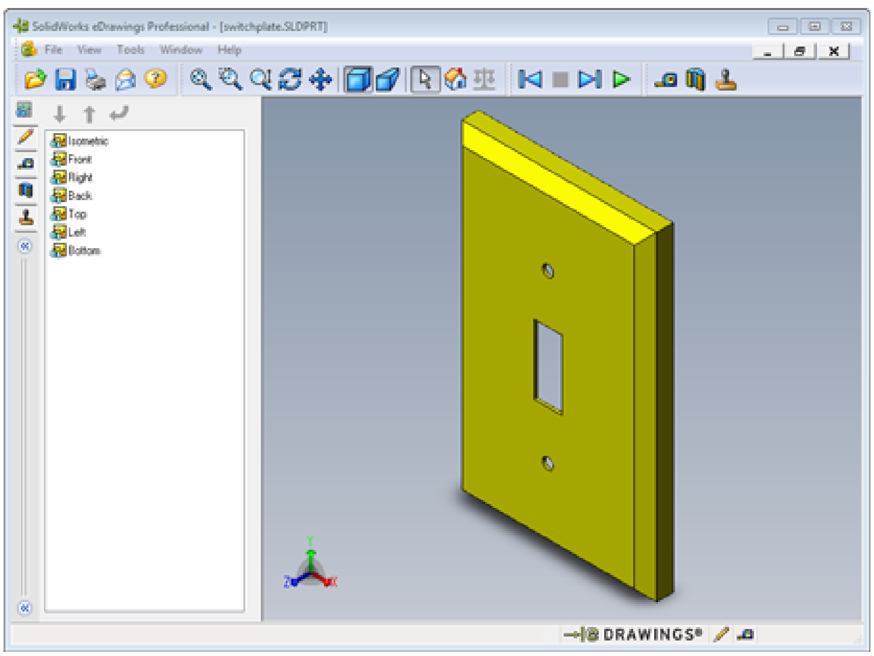
1 In SolidWorks, open the `switchplate` part.

Note: You created `switchplate` during Lesson 2.

2 Click **Publish an eDrawing**  on the eDrawings toolbar to publish an eDrawing of the part.

The eDrawing of `switchplate` appears in the eDrawings Viewer.

Note: You can create eDrawings from AutoCAD® drawings too. Refer to the topic *Creating SolidWorks eDrawing Files* in the eDrawings online help for more information.



Viewing an Animated eDrawings File

Animation enables you to dynamically view eDrawings.

1 Click **Next** .

The view changes to the Front view. You can click **Next** repeatedly to step through the views.

2 Click **Previous** .

The previous view is displayed.

3 Click **Continuous Play** .

Each view is displayed one by one in a continuous display.

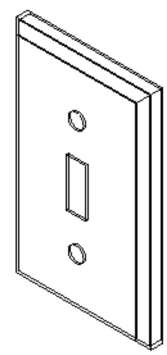
4 Click **Stop** .

The continuous display of views halts.

5 Click **Home** .

The default or home view is displayed.

Viewing Shaded and Wireframe eDrawings Files



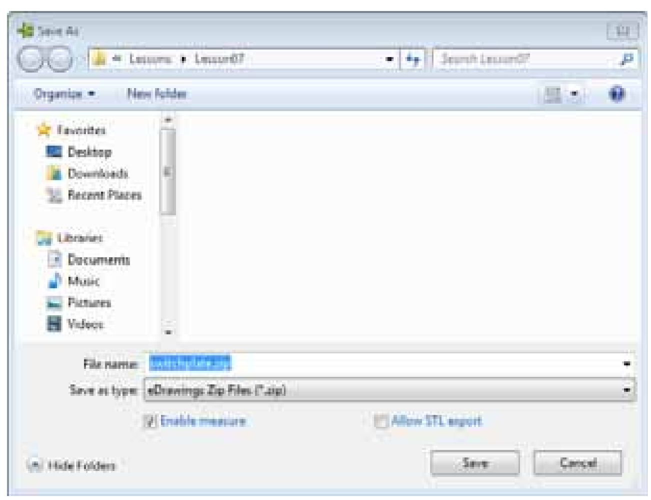
1 Click **Shaded** .

The display of the switch plate changes from shaded to wireframe.

2 Click **Shaded** again.

The display of the switch plate changes from wireframe to shaded.

Saving an eDrawings File



1 In the eDrawings Viewer click **File, Save As**.

2 Select **Enable measure**.

This option enables anyone viewing the eDrawing file to measure the geometry. This is called making the file "review-enabled".

3 Select **eDrawings Zip Files (*.zip)** from the **Save as type:** dropdown list.

This option saves the file as an eDrawings Zip file, which contains the eDrawings Viewer and the active eDrawings file.

4 Click **Save**.

Markup and Measure

You can markup eDrawings with tools from the Markup toolbar. Measure, if enabled (set at the time of eDrawing save in the save options dialog) allows for rudimentary dimension checking.

For tracking purposes markup comments appear as discussion threads on the Markup tab of the eDrawing Manager. In this example you will add a cloud with text and a leader.

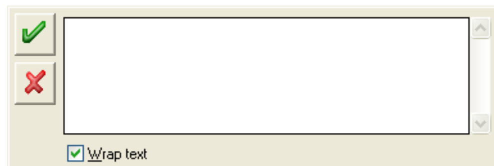
1 Click **Cloud with Leader** on the Markup toolbar.

Move the cursor into the graphics area. The pointer changes to .

2 Click the front face of the `switchplate`.

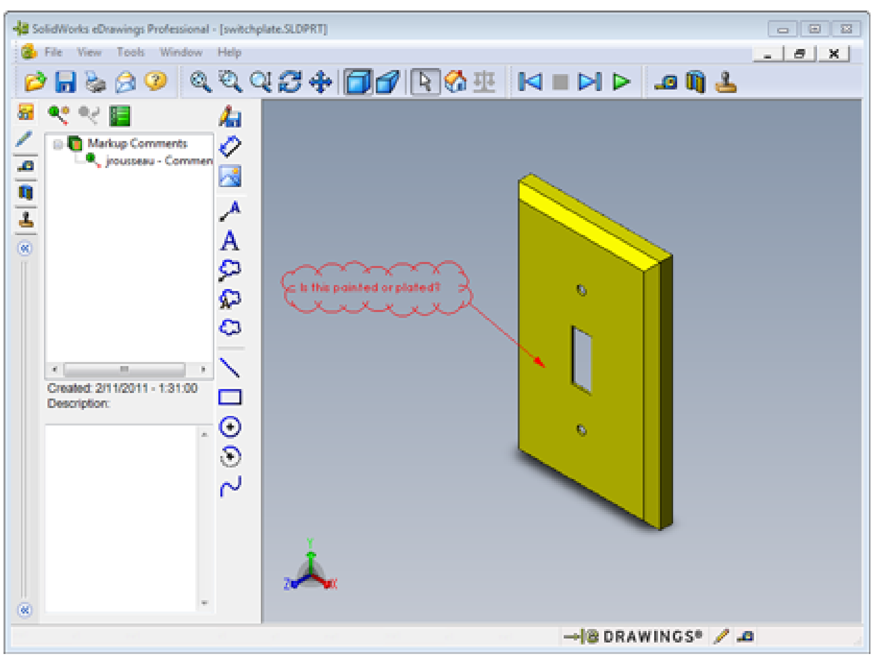
This is where the leader will begin.

3 Move the pointer to where you want to place the text and then click. A text box appears.



4 In the text box, type the text you want to appear in the cloud and then click **OK** .

The cloud with text appears attached to the leader. If necessary, click **Zoom to Fit** .



5 Close the eDrawing file, saving your changes.

Lesson 7 — 5 Minute Assessment — Answer Key

Name: ________________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you create an eDrawing?

Answer: There are two ways:

In SolidWorks, click **Publish an eDrawing** on the eDrawings toolbar.

Or, in SolidWorks click **File, Save As**. From the **Save as type** list, select eDrawing.

2 How do you send others eDrawings?

Answer: Email**.**

3 What is the quickest way to return to the default view?

Answer: Click **Home** .

4 True or False: You can make changes to a model in an eDrawing.

Answer: False. However if the eDrawing is review-enabled, you can measure geometry and add comments using markup tools.

5 True or False: You need to have the SolidWorks application in order to view eDrawings.

Answer: False.

6 What eDrawings feature enables you to dynamically view parts, drawings, and assemblies?

Answer: Animation.

Lesson 7 — 5 Minute Assessment **REPRODUCIBLE**

Name: ______________________________ Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you create an eDrawing?

____

____

____

2 How do you send others eDrawings?

____

3 What is the quickest way to return to the default view?

____

4 True or False: You can make changes to a model in an eDrawing.

____

____

5 True or False: You need to have the SolidWorks application in order to view eDrawings.

____

6 What eDrawings feature enables you to dynamically view parts, drawings, and assemblies?

____

Exercises and Projects — Exploring eDrawings Files

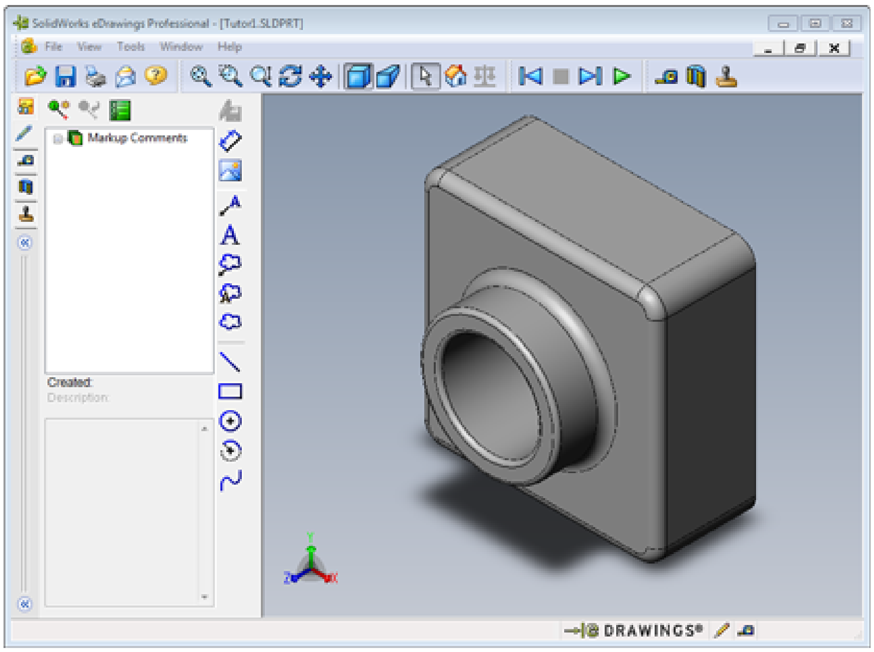
In this exercise, you explore eDrawings created from SolidWorks parts, assemblies, and drawings.

eDrawings of Parts

1 In SolidWorks, open the `Tutor1` part created in Lesson 3.

2 Click **Publish an eDrawing** .

An eDrawing of the part appears in the eDrawings Viewer.



3 Hold **Shift** and press one of the arrow keys.

The view of rotates 90° each time you press an arrow key.

4 Press an arrow key without holding **Shift**.

The view of rotates 15° each time you press an arrow key.

5 Click **Home** .

The default or home view is displayed.

6 Click **Continuous Play** .

Each view is displayed one by one in a continuous display. Observe this for a moment.

7 Click **Stop** .

The continuous display of views halts.

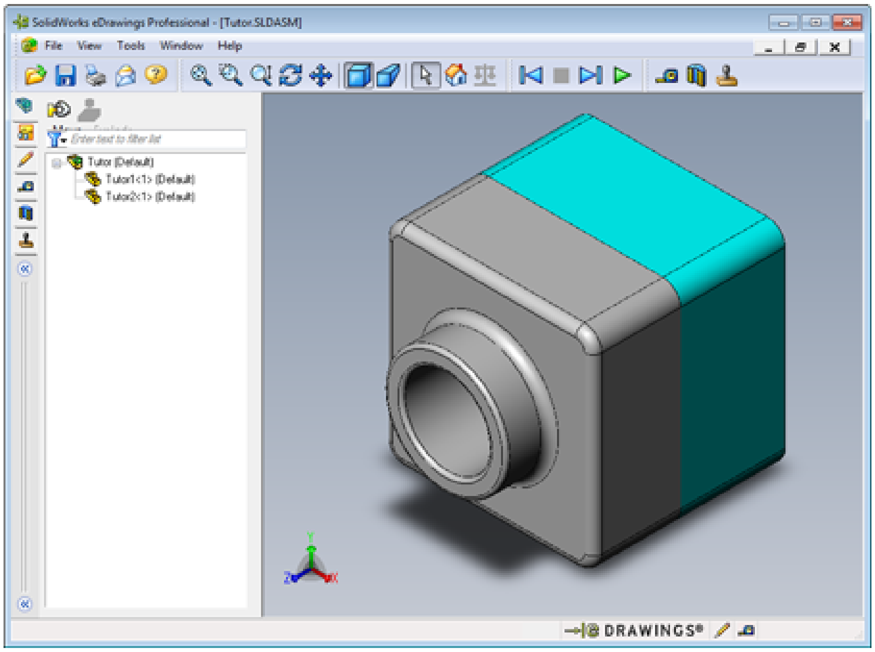
8 Close the eDrawing file without saving it.

eDrawings of Assemblies

1 In SolidWorks, open the `Tutor` assembly created in Lesson 4.

2 Click **Publish an eDrawing**.

An eDrawing of the assembly appears in the eDrawings Viewer.



3 Click **Continuous Play**.

Each view is displayed one by one. Observe this for a moment.

4 Click **Stop**.

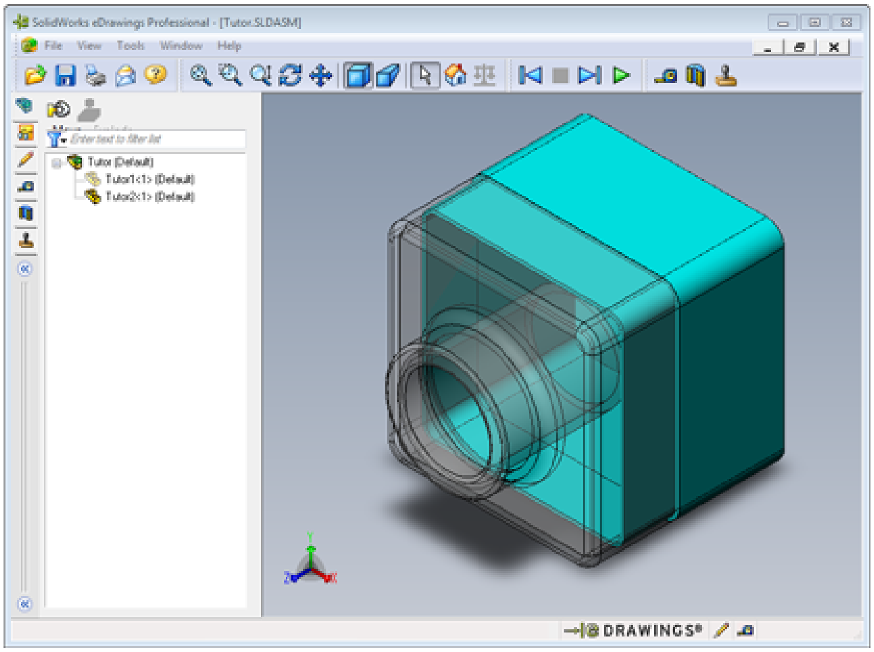
The continuous display of views halts.

5 Click **Home**.

The default or home view is displayed.

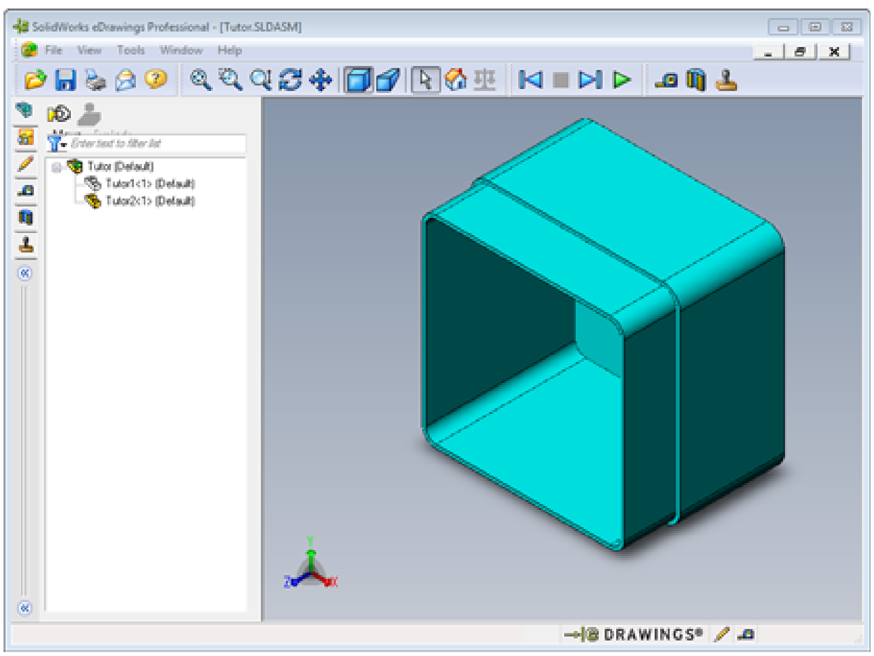
6 In the **Components** panel, right-click `Tutor1-1` and select **Make Transparent** from the shortcut menu.

The `Tutor1-1` part become transparent so you can see through it.



7 Right-click `Tutor1-1` and select **Hide** from the shortcut menu.

The `Tutor1-1` part no longer displays in the eDrawing. This part still exists in the eDrawing, it is just hidden.



8 Right-click `Tutor1-1` again and select **Show**.

The `Tutor1-1` part displays.

eDrawings of Drawings

1 Open the drawing you created in Lesson 6. This drawing has two sheets. Sheet 1 shows the part `Tutor1`. Sheet 2 shows the `Tutor` assembly. An example of this is in the `Lesson07` folder and is named `Finished Drawing.slddrw`.

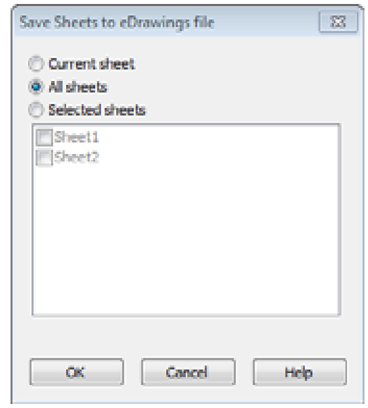
2 Click **Publish an eDrawing** .

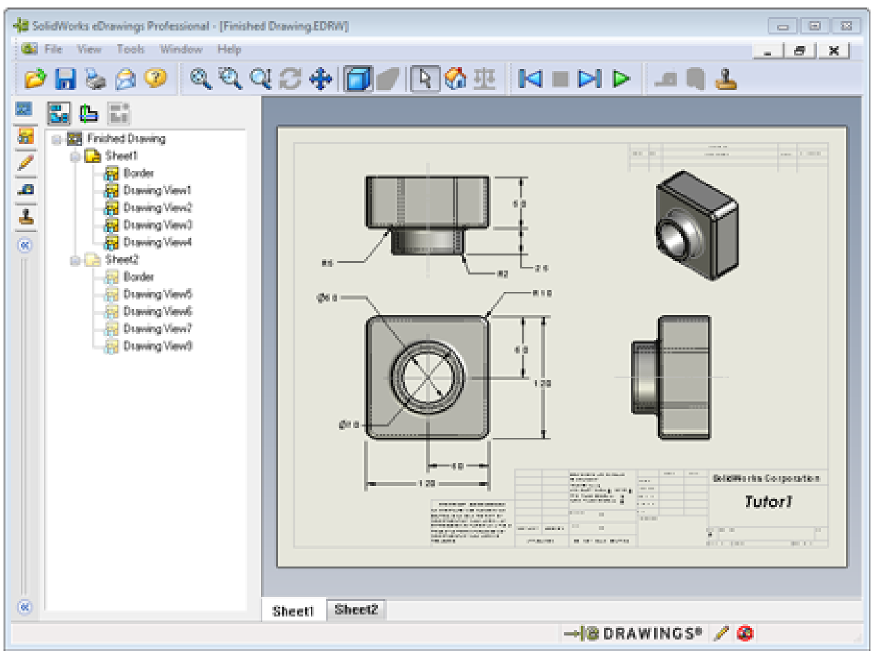
3 Select **All sheets**.

A window appears so you can select which sheets to include in the eDrawing.

Click **OK**.

An eDrawing of the drawing appears in the eDrawings Viewer.





4 Click **Continuous Play** .

Each view is displayed one by one. Observe this for a moment. Notice that the animation stepped through both sheets of the drawing.

5 Click **Stop** .

The continuous display of drawing views halts.

6 Click **Home** .

The default or home view is displayed.

Using the eDrawings Manager

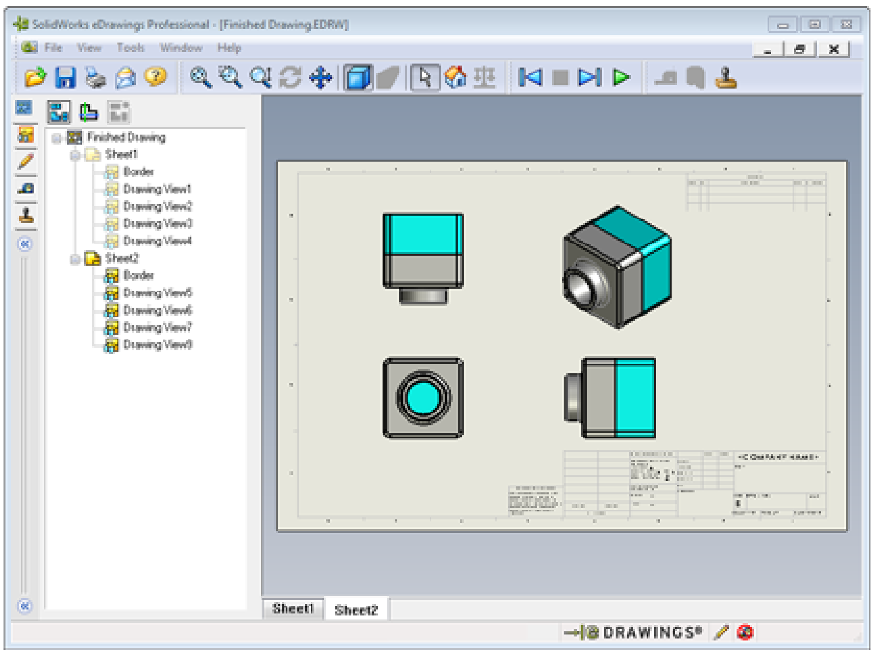
You can use the eDrawings Manager, located on the left side of the eDrawings Viewer, to display tabs that let you manage file information. When you open a file, the most appropriate tab is automatically active. For example, when you open a drawing file, the **Sheets** tab is active.

The **Sheets** tab makes it easy to navigate through a multi-sheet drawing.

1 In the **Sheets** tab of the eDrawings Manager, double-click `Sheet2`.

   `Sheet2` of the drawing is displayed in the eDrawings Viewer. Use this method to navigate a multi-sheet drawing.

Note: You can also switch between multiple sheets by clicking the tabs located below the graphics area.



2 In the **Sheets** tab of the eDrawings Manager, right-click one of the drawing views.

   The **Hide/Show** menu appears.

3 Click **Hide**.

   Notice how the eDrawings file changes.

4 Return to `Sheet1`.

The 3D Pointer

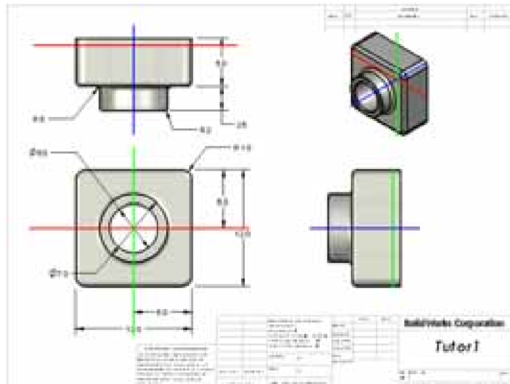
You can use the 3D Pointer to point to a location in all of the drawing views in drawing files. When you use the 3D Pointer, linked crosshairs appear in each of the drawing views. For example, you can place the crosshairs on an edge in one view and the crosshairs in the other views point to the same edge.

The crosshairs colors indicate the following:

Color	Axis
Red	X-Axis (perpendicular to YZ plane)
Blue	Y-Axis (perpendicular to XZ plane)
Green	Z-Axis (perpendicular to XY plane)

1 Click **3D Pointer** .

The eDrawing of the drawing displays the 3D pointer. The 3D pointer helps you to see the orientation of each view.



2 Move the 3D Pointer.

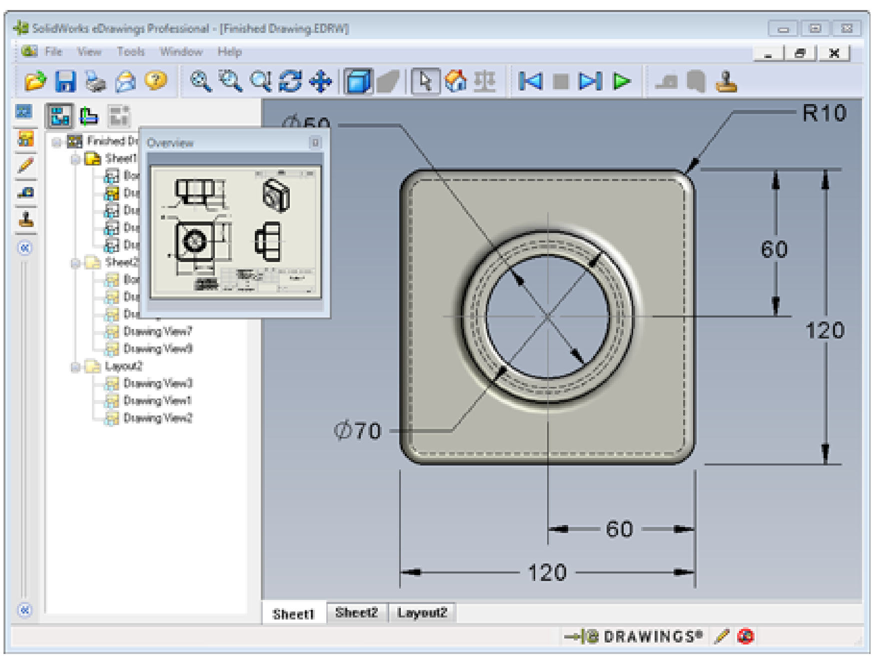
Notice how the pointer moves in each view.

Overview Window

The **Overview Window** gives you a thumbnail view of the entire drawing sheet. This is especially handy when working with large, complicated drawings. You can use the window to navigate among the views. In the **Overview Window**, click the view you want to look at.

1 Click **Overview Window** .

The **Overview Window** appears.



2 Click the Front view in the **Overview Window**.

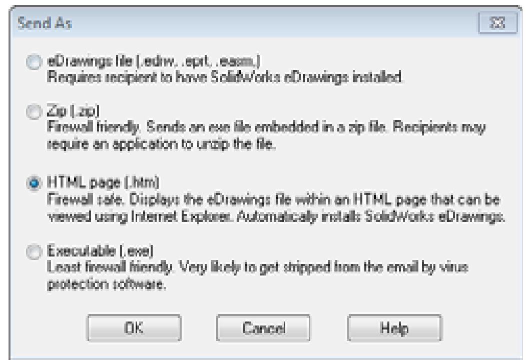
Notice how the eDrawings Viewer changes.

More to Explore — Emailing an eDrawings File

If your system is set up with an email application, you can see how easy it is to send an eDrawing to someone else.

1. Open one of the eDrawings that you created earlier in this lesson.
2. Click **Send** .

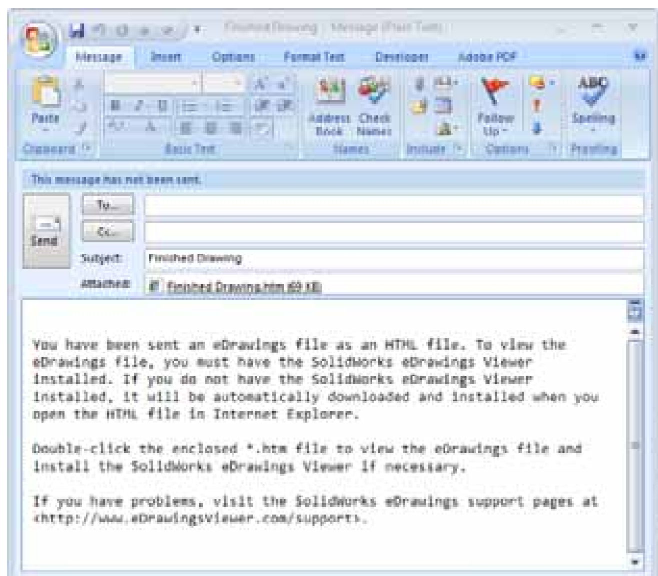
   The **Send As** menu appears.



3. Select the file type to send and click **OK**.

   An email message is created with the file attached.
4. Specify an email address to send the message to.
5. Add text to the email message if you would like to.
6. Click **Send**.

   The email is sent with the eDrawing attached. The person receiving it can view it, animate it, send it on to others, and so forth.



Teaching Suggestion

eDrawings Professional gives you the ability to measure and markup eDrawings. You may wish to use eDrawings Professional to review your student's work and give them feedback. eDrawings Professional is a communication tool that is well-suited for reviewing other's designs.

By using eDrawings Professional to evaluate and respond to student's work, you are closely simulating collaboration in the work world. Often, an engineer creates a design for someone located elsewhere. eDrawings Professional helps to bridge that gap.

Lesson 7 Vocabulary Worksheet — Answer Key

Name: ______________________________Class: _________ Date:_______________

Fill in the blanks with the words that are defined by the clues.

1 The ability to dynamically view an eDrawing: **Animate**
2 Halting a continuous play of an eDrawing animation: **Stop**
3 Command that enables you to step backwards one step at a time through an eDrawing animation: **Previous**
4 Non-stop replay of eDrawing animation: **Continuous Play**
5 Rendering of 3D parts with realistic colors and textures: **Shaded**
6 Go forward one step in an eDrawing animation: **Next**
7 Command used to create an eDrawing: **Publish**
8 Graphic aid that enables you to see the model orientation in an eDrawing created from a SolidWorks drawing: **3D Pointer**
9 Quickly return to the default view: **Home**
10 Command that enables you to use email to share eDrawings with others: **Send**

Lesson 7 Vocabulary Worksheet **REPRODUCIBLE**

Name: ______________________________Class: _________ Date:______________

Fill in the blanks with the words that are defined by the clues.

1 The ability to dynamically view an eDrawing: _______________________________

2 Halting a continuous play of an eDrawing animation:__________________________

3 Command that enables you to step backwards one step at a time through an eDrawing animation:____

4 Non-stop replay of eDrawing animation:___________________________________

5 Rendering of 3D parts with realistic colors and textures: _____________________

6 Go forward one step in an eDrawing animation: ____________________________

7 Command used to create an eDrawing: ____________________________________

8 Graphic aid that enables you to see the model orientation in an eDrawing created from a SolidWorks drawing: ____

9 Quickly return to the default view:_______________________________________

10 Command that enables you to use email to share eDrawings with others: __________

Lesson 7 Quiz — Answer Key

Name: ______________________________ Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What is the window that shows you a thumbnail view of the whole eDrawing?

Answer: Overview window.

2 Which command displays wireframe as solid surfaces with realistic colors and textures?

Answer: **Shaded**.

3 How do you create an eDrawing?

Answer: Click **Publish an eDrawing**  in the SolidWorks application.

4 What action does the **Home** command perform?

Answer: Returns to the default view.

5 Which command performs a non-stop replay of eDrawing animation?

Answer: **Continuous Play**.

6 True or False — eDrawings only displays part files, but not assemblies or drawings.

Answer: False.

7 True or False — You can hide assembly components or drawing views.

Answer: True.

8 In an eDrawing created from a SolidWorks drawing, how do you view a sheet other than the one currently displayed?

Answer: Answers will vary but may include:

- In the Sheets tab of the eDrawing Manager, double-click the sheet you want to view.
- Click the sheet tab located below the graphics area of the eDrawings viewer.

9 What visual aid helps you identify model orientation in a drawing?

Answer: 3D Pointer.

10 Holding **Shift** and pressing an arrow key rotates a view 90-degrees at a time. How would you rotate a view 15-degrees at a time?

Answer: Press an arrow key without holding **Shift**.

Lesson 7 Quiz REPRODUCIBLE

Name: ______________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What is the window that shows you a thumbnail view of the whole eDrawing?
____

2 Which command displays wireframe as solid surfaces with realistic colors and textures? ____

3 How do you create an eDrawing? ____

4 What action does the **Home** command perform? ___________________________

5 Which command performs a non-stop replay of eDrawing animation? ______________
____

6 True or False — eDrawings only displays part files, but not assemblies or drawings. __
____

7 True or False — You can hide assembly components or drawing views. ____________

8 In an eDrawing created from a SolidWorks drawing, how do you view a sheet other than the one currently displayed? ____

9 What visual aid helps you identify model orientation in a drawing? ________________
____

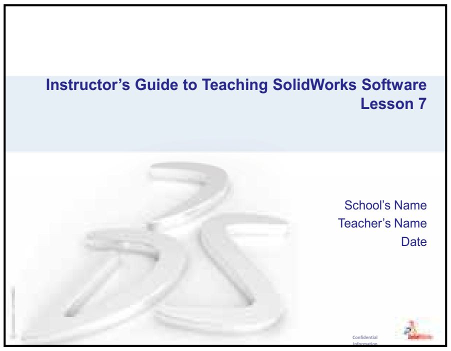
10 Holding **Shift** and pressing an arrow key rotates a view 90-degrees at a time. How would you rotate a view 15-degrees at a time? _________________________________
____

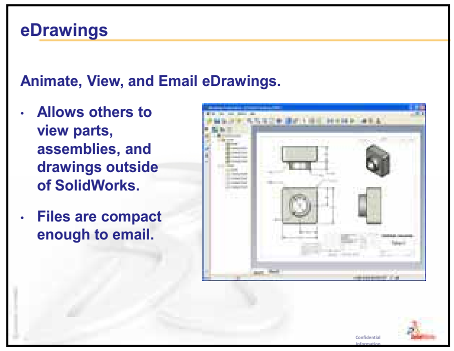
Lesson Summary

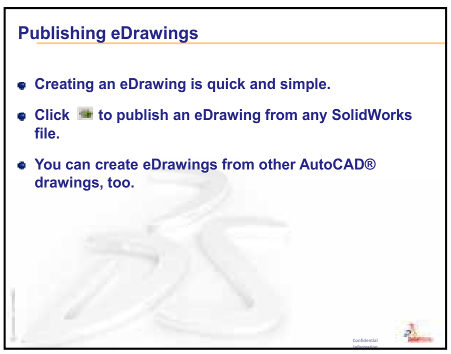
- eDrawings can be created quickly from part, assembly, and drawing files.
- You can share eDrawings with others — even if they don't have SolidWorks.
- Email is the easiest way to send an eDrawing to others.
- Animation enables you to see all views of a model.
- You can hide selected components of an assembly eDrawing and selected views of a drawing eDrawing.

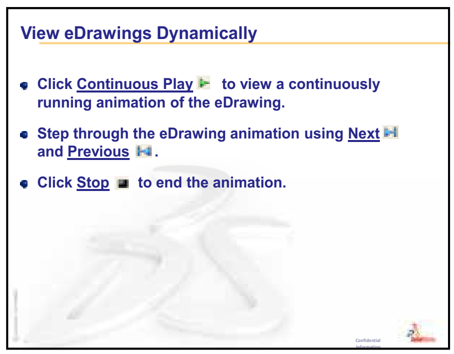
Thumbnail Images of PowerPoint Slides

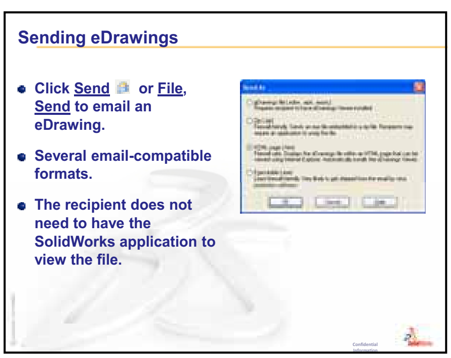
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

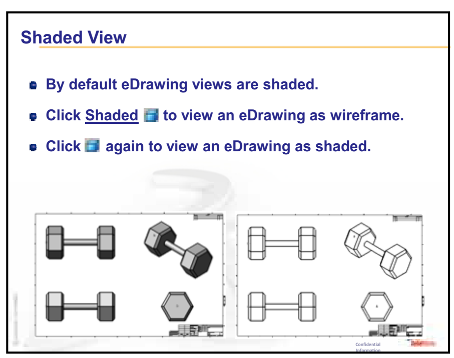


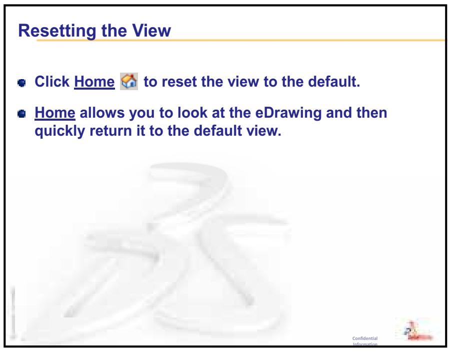


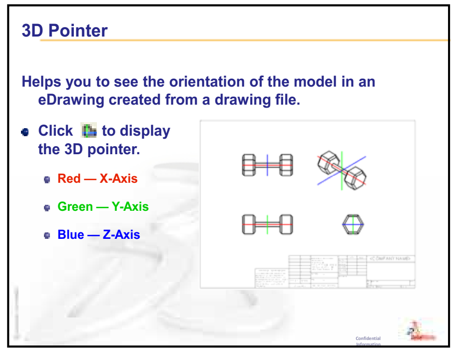









Resetting the View
Click Home to reset the view to the default.
Home allows you to look at the eDrawing and then quickly return it to the default view.


3D Pointer
Helps you to see the orientation of the model in an eDrawing created from a drawing file.
Click to display the 3D pointer.
Red — X-Axis
Green — Y-Axis
Blue — Z-Axis


Overview Window
Small thumbnail view of the eDrawing.
Click Overview Window to display the Overview window.

8

Lesson 8: Design Tables

Goals of This Lesson

Create a design table that generates the following configurations of `Tutor1`.

	A	B	C	D	E	F	G
1	Design Table for: Tutor3						
2		box_width @Sketch1	box_height @Sketch1	knob_dia@ Sketch2	hole_dia@ Sketch3	fillet_radiu s@Outside _corners	Depth@Kn ob
3	blk1	120	120	70	50	10	50
4	blk2	120	90	50	40	15	30
5	blk3	90	150	60	10	30	15
6	blk4	120	120	30	10	25	90

Before Beginning This Lesson

Design Tables requires the Microsoft Excel® application. Ensure that Microsoft Excel is loaded on your classroom/lab systems.

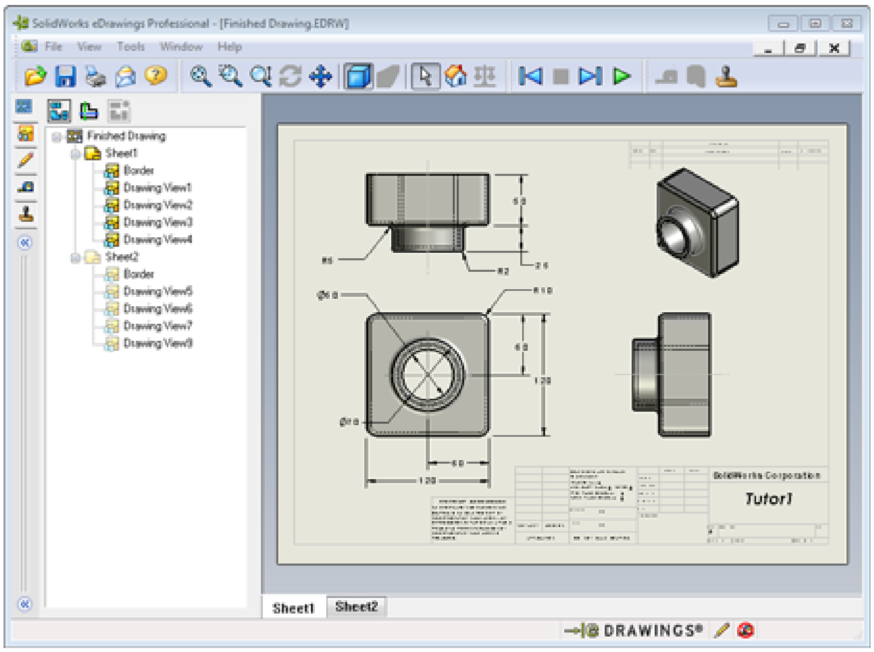
Resources for This Lesson

This lesson plan corresponds to *Productivity Enhancements: Design Tables* in the SolidWorks Tutorials.

The SolidWorks Teacher Blog, http://blogs.solidworks.com/teacher, SolidWorks Forums http://forums.solidworks.com and SolidWorks Users Groups http://www.swugn.org provide a great resource to instructors and students.

Review of Lesson 7: SolidWorks eDrawings Basics

- Animate, view, and email eDrawings.
- Enables others to view parts, assemblies, and drawings outside of SolidWorks.
- Files are compact enough to email.
- Publish an eDrawing from any SolidWorks file.
- You can create eDrawings from other CAD systems too.
- Animation enables you to view eDrawings dynamically.



Outline of Lesson 8

- ❑ In Class Discussion — Families of Parts
- ❑ Active Learning Exercises — Creating a Design Table
- ❑ Exercises and Projects — Creating a Design Table for Tutor2
  - Creating Four Configurations
  - Creating Three Configurations
  - Modifying Configurations
  - Determining Feasibility of Configurations
- ❑ Exercises and Projects - Creating Part Configurations Using Design Tables
- ❑ More to Explore — Configurations, Assemblies, and Design Tables
- ❑ Lesson Summary

Competencies for Lesson 8

Students develop the following competencies in this lesson:

- ❑ **Engineering**: Explore family of parts with a design table. Understand how design intent can be built into a part to allow for changes.
- ❑ **Technology**: Link an Excel spreadsheet with a part or an assembly. See how they relate a manufactured component.
- ❑ **Math**: Work with numerical values to change overall size and shape of a part and assembly. Develop width, height and depth values to determine volume of the CD Storage box modifications.

In Class Discussion — Families of Parts

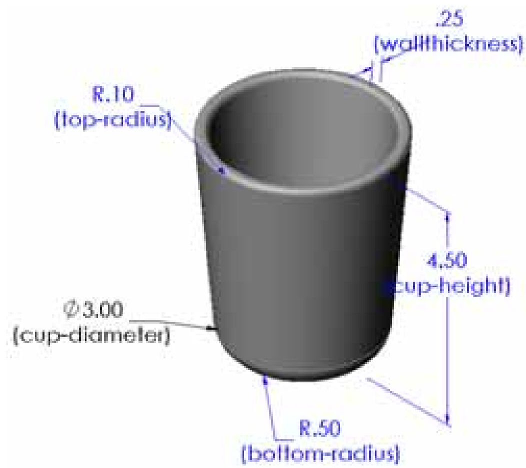
Many common objects come in a variety of sizes. Encourage discussion by having your students name examples. Some possibilities include:

- ❑ Nuts and bolts
- ❑ Paper clips
- ❑ Pipe fittings
- ❑ Bookends
- ❑ Sprockets on bicycles
- ❑ Wheels on cars
- ❑ Gears and pulleys
- ❑ Measuring spoons

Design tables make it easy to create a family of parts. Look around for examples.

Question:

Show the students a drinking cup. Ask the students to describe the features that make up the cup.



Answer:

- ❑ The Base feature is an extruded feature with a circular profile that was sketched on the Top plane.
- ❑ The taper was created by extruding the base feature with the **Draft** option. The **Draft** option creates a taper during the extrusion process. You can specify the amount of draft (the size of the angle) and whether it tapers outward or inward.
- ❑ The bottom of the cup was rounded with a fillet feature.
- ❑ The cup was hollowed out using a shell feature.
- ❑ The lip of the cup was rounded with a fillet feature.

Question:

What are the some of the dimensions that you would want to control if you were to make a series of different sized cups?

Answer:

Answers will vary but can include:

- ❑ The diameter of the cup
- ❑ The angle of the taper
- ❑ The radius of the fillet on the bottom
- ❑ The height of the cup
- ❑ The thickness of the wall
- ❑ The radius of the fillet on the lip

Question:

You work for a company that manufactures cups. Why should you use a design table?

Answer:

A design table saves design time. With a single part and a design table you can create numerous versions of the cup without having to model each one individually.

Question:

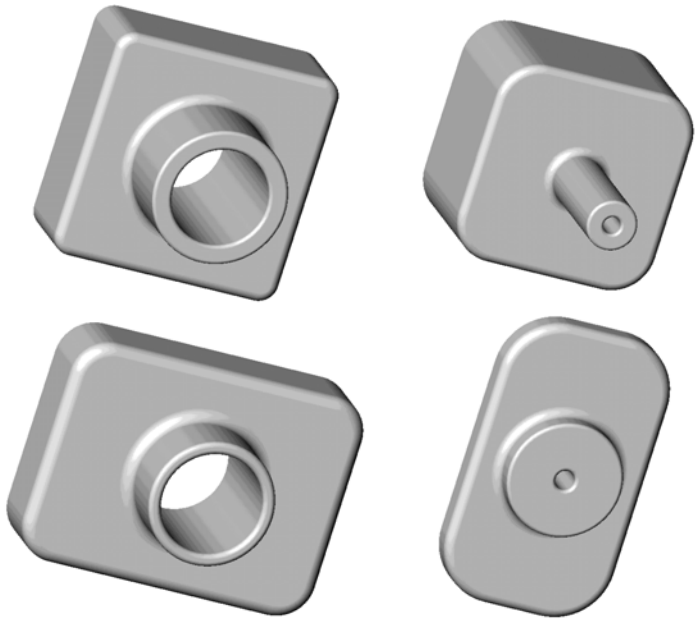
What are some other examples of products that lend themselves to design tables? You can bring in the actual objects or illustrations from magazines or catalogs.

Answer:

Answers will vary depending on the interests and resourcefulness of your students. Some ideas include hardware such as nuts and bolts, pipe fittings, wrenches, pulleys, or shelf brackets. If any of your students have an interest in bicycling, suggest looking at the chainring on a mountain bike. Is someone interested in cars? An automotive wheel (rim) would work well with a design table. Look around the classroom. Do you have different size paperclips? Collaborate with a teacher in another discipline. For example, a science teacher might have different sizes of glassware such as test tubes or beakers that they can loan you.

Active Learning Exercises — Creating a Design Table

Create the design table for `Tutor1`. Follow the instructions in *Productivity Enhancements: Design Tables* in the SolidWorks Tutorials.



	A	B	C	D	E	F	G
1	Design Table for: Tutor3						
2		box_width @Sketch1	box_height @Sketch1	knob_dia@ Sketch2	hole_dia@ Sketch3	fillet_radiu s@Outside _corners	Depth@Kn ob
3	blk1	120	120	70	50	10	50
4	blk2	120	90	50	40	15	30
5	blk3	90	150	60	10	30	15
6	blk4	120	120	30	10	25	90

Lesson 8 — 5 Minute Assessment — Answer Key

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What is a configuration?

Answer: A configuration is a way to create a family of similar parts within one file.

2 What is a design table?

Answer: A design table is a spreadsheet that lists the different values that are assigned to the various dimensions and features in a part. A design table is an easy way to create many configurations.

3 What additional Microsoft software application is required to create design tables in SolidWorks?

Answer: Microsoft Excel.

4 What are three key elements of a design table?

Answer: A design table requires configuration name, dimension name and dimension values.

5 True or False. **Link Values** equates a dimension value to a shared variable name.

Answer: True.

6 Describe the advantage of using geometric relations versus linear dimensions to position the `Knob` feature on the `Box` feature.

Answer: The advantage of using a geometric relation is that a midpoint relation ensures the `Knob` is always positioned in the center of the `Box`. If linear dimensions were used, the `Knob` would be located in various positions relative to the `Box`.

7 What is the advantage of creating a design table?

Answer: A design table saves design time, disk space and automatically drives the dimensions and features of an existing part to create multiple configurations.

Lesson 8 — 5 Minute Assessment **REPRODUCIBLE**

Name: ______________________________Class: _________ Date:______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What is a configuration?

____

2 What is a design table?

____

____

____

3 What additional Microsoft software application is required to create design tables in SolidWorks?

____

4 What are three key elements of a design table?

____

____

5 True or False. **Link Values** equates a dimension value to a shared variable name.

____

6 Describe the advantage of using geometric relations versus linear dimensions to position the `Knob` feature on the `Box` feature.

____

____

____

7 What is the advantage of creating a design table?

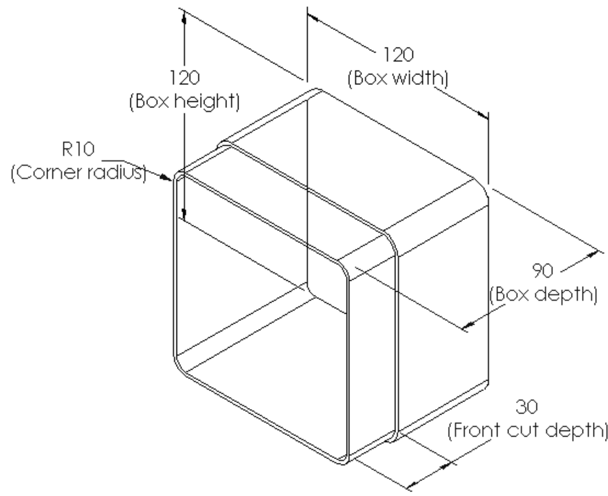
____

____

Exercises and Projects — Creating a Design Table for Tutor2

Task 1 — Creating Four Configurations

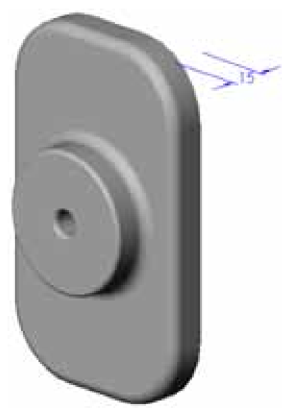
Create a design table for `Tutor2` that corresponds to the four configurations of `Tutor3`. Rename the features and the dimensions. Save the part as `Tutor4`.

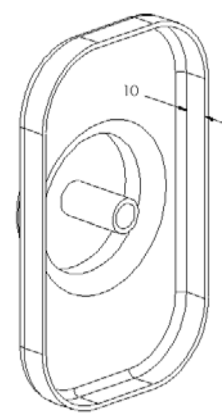


Answer:

- ❑ The height and width of `Tutor4` must equal the `box_width` and `box_height` dimension values in the `Tutor3` design table.
- ❑ The corner radii of `Tutor4` must match those of `Tutor3`.
- ❑ The depth of the front cut on `Tutor4` must be at least **5mm** *less* than the depth of `Tutor3`.

  This is important because some of the configurations of `Tutor3` (`blk3` for example) are not very deep.





If the front cut depth on `Tutor4` is not changed accordingly, the parts will not fit together correctly in the assembly.

If the depth of the front cut is set to a value less than the depth of `Tutor3`, the parts will fit correctly.

To explore this topic more fully with your students, see *More to Explore — Configurations, Assemblies, and Design Tables* on page 183 in this lesson.

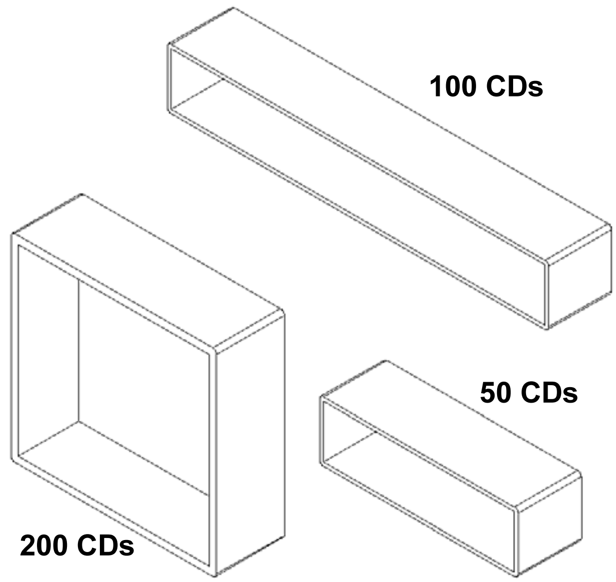
- One possible design table for the `Tutor4` is shown in the illustration at the right.

	A	B	C	D	E	F
1	Design Table for: Tutor4					
2		Box_width@Sketch1	Box_height@Sketch1	Box_depth@Base-Extrude	Corner_radius@Fillet1	Front-cut_depth@Cut-Extrude1
3	Version 1	120	120	90	10	30
4	Version 2	120	90	90	15	25
5	Version 3	90	150	90	30	10
6	Version 4	120	120	90	25	30

Book1 — Sheet1

Task 2 — Creating Three Configurations

Create three configurations of the `storagebox` to contain 50, 100 and 200 CDs. The maximum width dimension is 120cm.

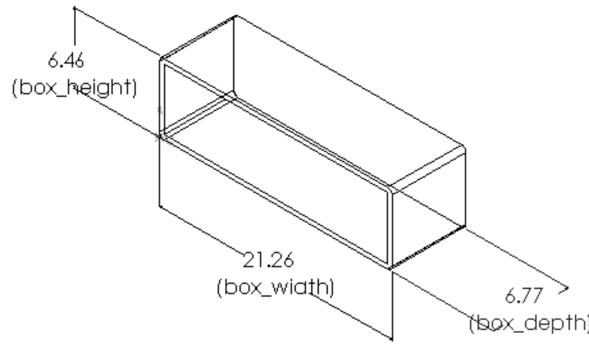


Answer:

- There are numerous answers to this question. The `storagebox` can have various widths and heights. Some examples are shown at the right. A sample file with suggested dimensions is found in the `Lessons\Lesson08` folder in `SolidWorks Teacher Tools`.

Task 3 — Modifying Configurations

Convert the overall dimensions of the 50 CD `storagebox` from centimeters to inches. The design for the CD `storagebox` was created overseas. The CD `storagebox` will be manufactured in the US.



Given:

- Conversion: 2.54cm = 1 inch
- `Box_width` = 54.0cm
- `Box_height` = 16.4cm
- `Box_depth` = 17.2cm

Answer:

- Overall dimensions = `box_width` **x** `box_height` **x** `box_depth`
- `Box_width` = 54.0 ÷ 2.54 = 21.26”
- `Box_height` = 16.4 ÷ 2.54 = 6.46”
- `Box_depth` = 17.2 ÷ 2.54 = 6.77”
- Use SolidWorks to confirm the conversion values.

Task 4 — Determining Feasibility of Configurations

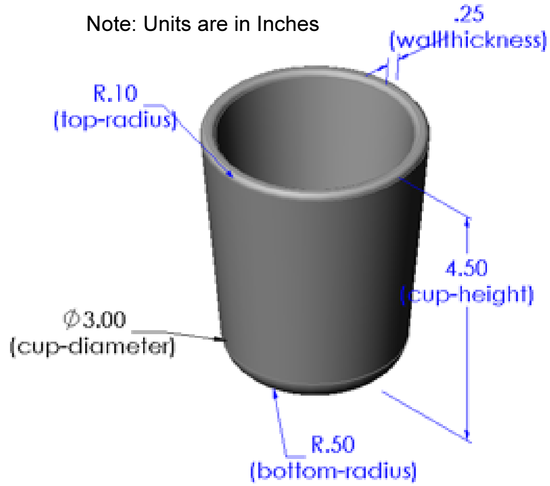
What CD `storagebox` configurations are feasible for use in your classroom?

Answer:

- Have the students work in groups to measure bookshelves, desks and tables in the classroom. Determine the CD `storagebox` size most feasible in each area. The answers will vary.

Exercises and Projects — Creating Part Configurations Using Design Tables

Create a cup. In the **Extrude Feature** dialog box, use a **5° Draft Angle**. Create four configurations using a design table. Experiment with different dimensions.



Answer:

Answers will vary. A sample design table for the cup is shown at the right.

Worksheet in Part1

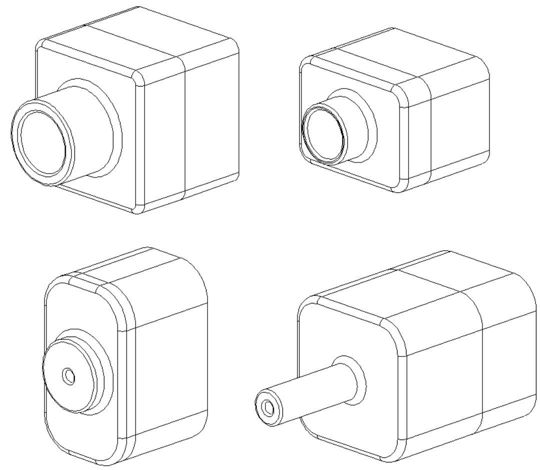
	A	B	C	D	E	F
1	Design Table for: Cup					
2		cup-diameter@Sketch1	cup-height@Base-Extrude	wallthickness@Shell1	top-radius@Fillet2	bottom-radius@Fillet1
3	2-5 inch diameter	2.50	4.00	0.25	0.100	0.50
4	3 inch diameter	3.00	4.50	0.25	0.100	0.50
5	2 inch diameter	2.00	3.00	0.20	0.050	0.25
6	4 inch diameter	4.00	6.00	0.25	0.125	0.75

Sheet1

More to Explore — Configurations, Assemblies, and Design Tables

When each component in an assembly has multiple configurations, it make sense that the assembly should have multiple configurations as well. There are two ways to accomplish this:

- ❑ Manually change the configuration being used by each component in the assembly.
- ❑ Create an *assembly* design table that specifies which configuration of each component is to be used for each version of the assembly.



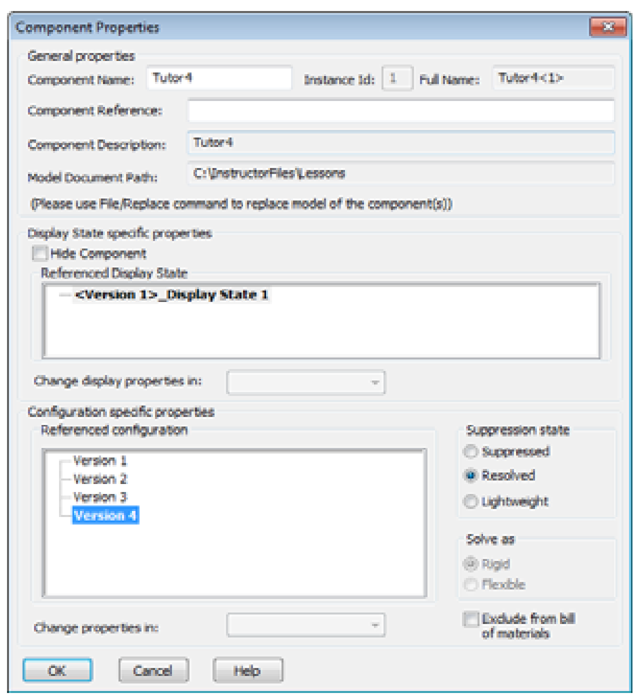
Note: If your students followed the directions in the tutorial, they saved `Tutor1` as `Tutor3` when they created the design table. Likewise in Task 1 of the exercises, `Tutor2` would have been saved as `Tutor4`. To explore assembly design tables, you will need an assembly that is made up of `Tutor3` and `Tutor4`. This assembly is located in the `Lessons\Lesson08` folder in `SolidWorks Teacher Tools`.

Changing the Configuration of a Component in an Assembly

To manually change the displayed configuration of a component in an assembly:

1. Open the assembly `Tutor Assembly` which is located in the `Lesson08` folder.
2. Right-click the component, either in the FeatureManager design tree or in the graphics area, and select **Properties** .
3. In the **Component Properties** dialog, select the desired configuration from the list in the **Referenced configuration** area.

   Click **OK**.
4. Repeat this procedure for each component in the assembly.



Assembly Design Tables

While manually changing the configuration of each component in an assembly works, it is neither efficient nor very flexible. Switching from one version of an assembly to another would be tedious. A better approach would be to create an assembly design table.

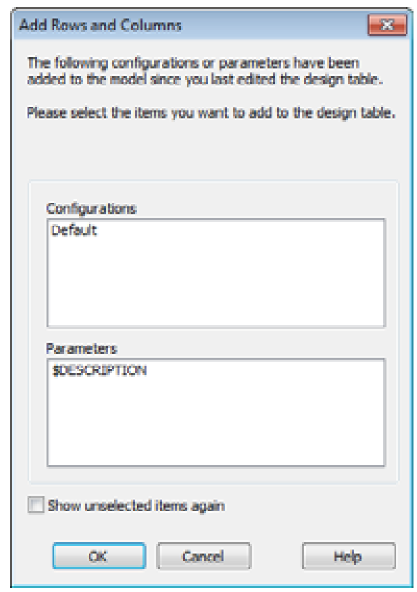
The procedure for creating an assembly design table is very similar to the procedure for creating a design table in an individual part. The most significant difference is the choice of different keywords for the column headers. The keyword we will explore here is `$CONFIGURATION@component<instance>`.

Procedure

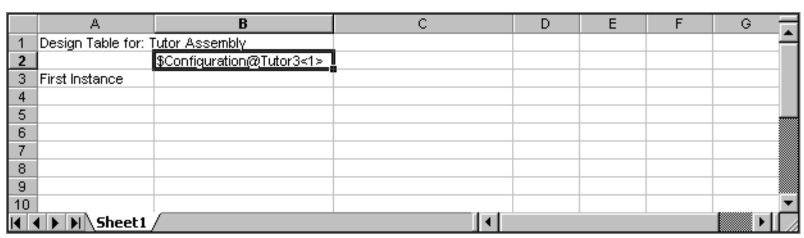
1. Click **Insert, Tables**, **Design Table**.

   The **Design Table** PropertyManager appears.
2. For **Source**, click **Blank** and then click **OK** .
3. The **Add Rows and Columns** dialog box appears.

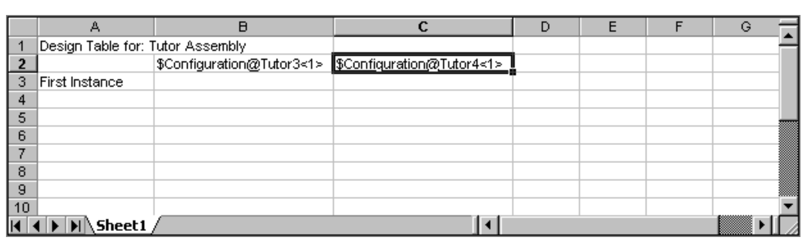
   If the assembly already contained configurations that were created manually they would be listed here. You could select them and they would automatically be added to the design table.



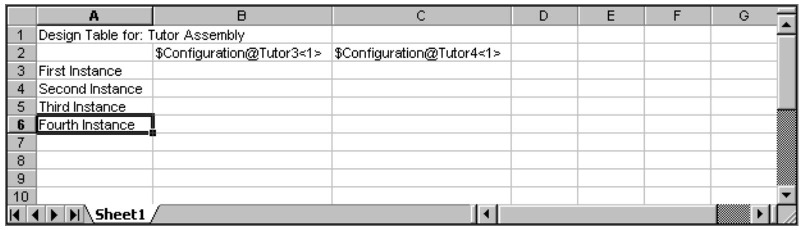
4. Click **Cancel**.
5. In cell B2, enter the keyword `$Configuration@` followed by the name of the component and its instance number. In this example, the component is `Tutor3` and the instance is `<1>`.



6. In cell C2, enter the keyword `$Configuration@Tutor4<1>`.

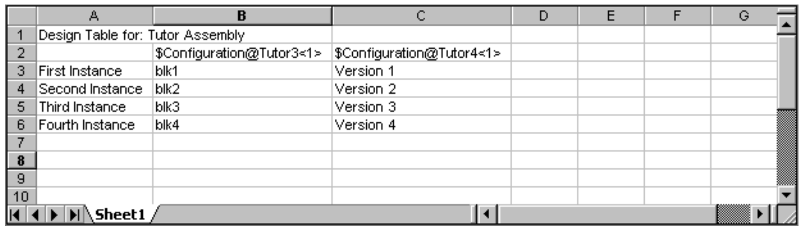


7 Add the configuration names in column A.



	A	B	C	D	E	F	G
1	Design Table for: Tutor Assembly						
2		$Configuration@Tutor3<1>	$Configuration@Tutor4<1>				
3	First Instance						
4	Second Instance						
5	Third Instance						
6	Fourth Instance						
7							
8							
9							
10							

8 Fill in the cells of columns B and C with the appropriate configurations for the two components.

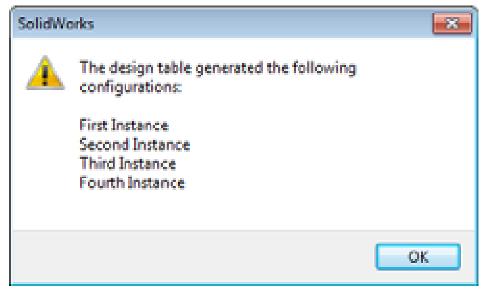


	A	B	C	D	E	F	G
1	Design Table for: Tutor Assembly						
2		$Configuration@Tutor3<1>	$Configuration@Tutor4<1>				
3	First Instance	blk1	Version 1				
4	Second Instance	blk2	Version 2				
5	Third Instance	blk3	Version 3				
6	Fourth Instance	blk4	Version 4				
7							
8							
9							
10							

9 Finish inserting the design table.

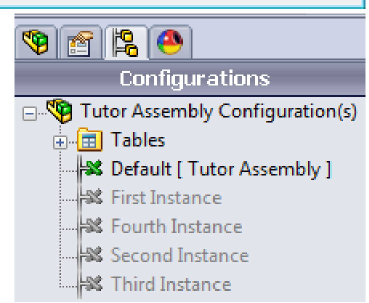
Click in the graphics area. The system reads the design table and generates the configurations.

Click **OK** to close the message dialog.



10 Switch to the ConfigurationManager.

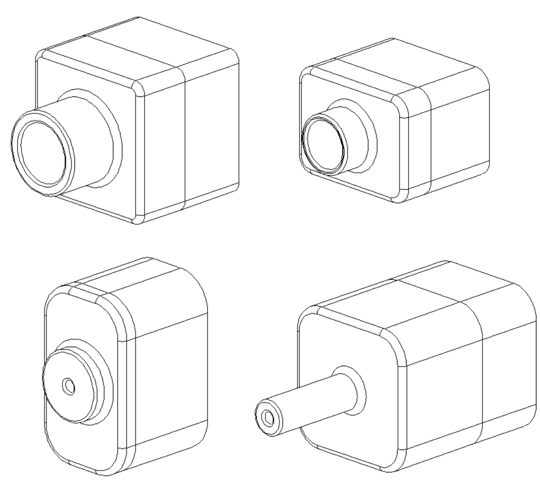
Each of the configurations specified in the design table should be listed.



Note: The configuration names are listed in the ConfigurationManager alphabetically, *not* in the order in which they appeared in the design table.

11 Test the configurations.

Double-click on each configuration to verify that they display correctly.



Lesson 8 Quiz — Answer Key

Name: ______________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What is a design table?

Answer: A design table is a spreadsheet that lists the different values that are assigned to the various dimensions and features in a part. A design table is an easy way to create many configurations.

2 List three elements of a design table.

Answer: Answers will vary but may include Configuration Name, Dimension Name and Dimension Values, Feature Name, Component Name (in assembly design tables).

3 Design tables are used to create different ____________________________ of a part.

Answer: Configurations

4 Why should you rename feature names and dimension names?

Answer: Renaming feature names and dimension names makes them more meaningful. Meaningful names make it easier to read the design table and to understand what dimensions and features are being controlled by it.

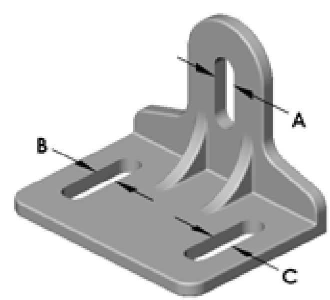
5 What Microsoft software application is required to create design tables in SolidWorks?

Answer: Microsoft Excel.

6 How do you display all feature dimensions?

Answer: Right-click the **Annotations Folder**. Click **Show Feature Dimensions**.

7 Examine the part shown at the right. The design intent is that the width of the three slots, A, B, and C must always be the same. To do this, should you use **Link Values** or the geometric relation **Equal**?



Answer: You should use **Link Values**. An **Equal** geometric relation will not work because **Equal** only works inside a sketch. Features A, B, and C cannot be in the same sketch.

8 How do you hide all dimensions of a feature?

Answer: Right-click the feature in the FeatureManager design tree, and select **Hide All Dimensions**.

9 How is the ConfigurationManager used in SolidWorks?

Answer: The ConfigurationManager is used to switch from one configuration to another.

10 What is the advantage of creating a design table?

Answer: A design table saves design time and disk space by automatically driving the dimensions and features of an existing part to create multiple versions of that part. This is more efficient than building many separate part files.

11 What type of parts lend themselves to using a design table?

Answer: Parts that have similar characteristics such as shape, but that have different values for their dimensions.

Lesson 8 Quiz **REPRODUCIBLE**

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What is a design table?____

____

____

2 List three elements of a design table. ______________________________________

____

____

3 Design tables are used to create different _____________________________ of a part.

4 Why should you rename feature names and dimension names? __________________

____

____

5 What Microsoft software application is required to create design tables in SolidWorks?

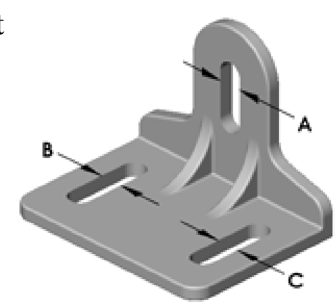
____

6 How do you display all feature dimensions? _______________________________

____

____

7 Examine the part shown at the right. The design intent is that the width of the three slots, A, B, and C must always be the same. To do this, should you use **Link Values** or the geometric relation **Equal**?



____

8 How do you hide all dimensions of a feature?

____

____

9 How is the ConfigurationManager used in SolidWorks?_________________________

____

____

10 What is the advantage of creating a design table? ____________________________

____

____

11 What type of parts lend themselves to using a design table? ____________________

____

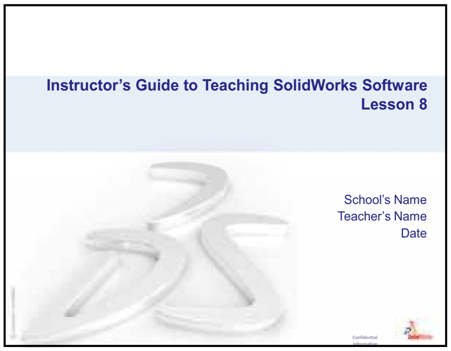
____

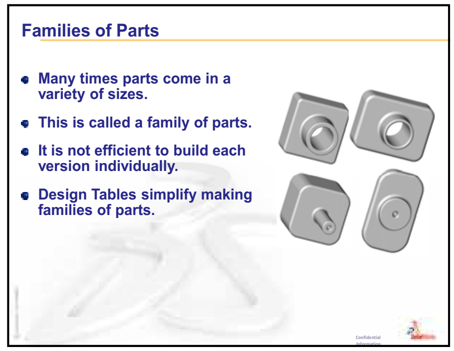
Lesson Summary

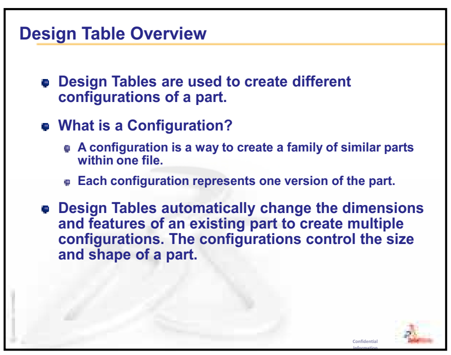
- Design Tables simplify making families of parts.
- Design Tables automatically change the dimensions and features of an existing part to create multiple configurations. The configurations control the size and shape of a part.
- Design Tables requires Microsoft Excel application.

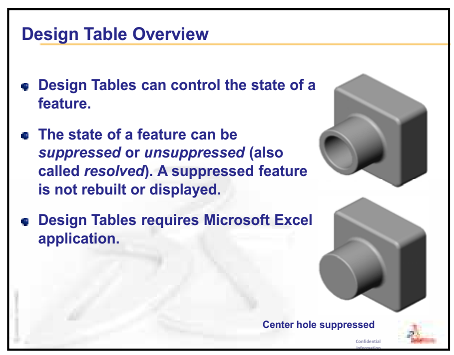
Thumbnail Images of PowerPoint Slides

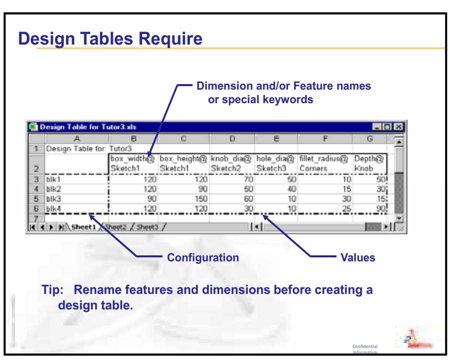
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

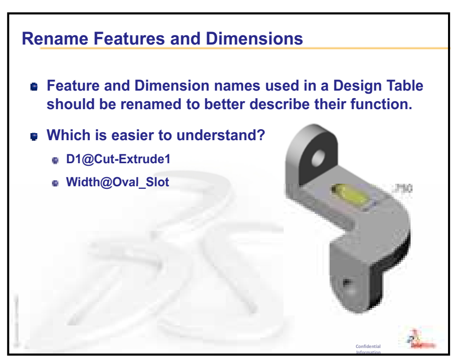












To Rename a Feature

- Click-pause-click on *Extrude1* in the FeatureManager design tree (do not double-click).

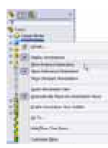
  Tip: Instead of the click-pause-click technique, you can select the feature, and then press the function key F2.
- The feature name is highlighted in blue, ready to be edited.
- Type the new name, *Box*, and press Enter.

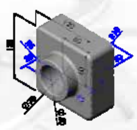
Rename the Other Features Used in the Design Table

- Rename *Extrude2* to *Knob.*
- Rename *Cut-Extrude1* to *Hole_in_knob*.
- Rename *Fillet1* to *Outside_corners.*

To Display Feature Dimensions

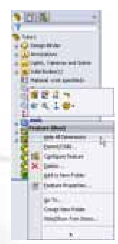
- Right-click the *Annotations* folder, and select Show Feature Dimensions from the shortcut menu.





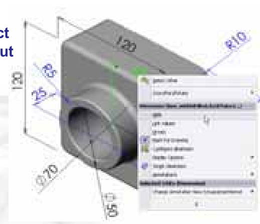
To Hide All the Feature Dimensions for a Selected Feature

- Right-click the feature in the FeatureManager design tree, and select Hide All Dimensions from the shortcut menu.



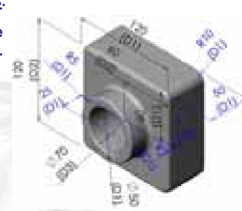
To Hide Individual Dimensions

- Right-click the dimension, and select Hide from the shortcut menu.



To Display Dimension Names

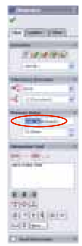
1. Click Tools, Options.
2. Click General on the System Options tab.
3. Click Show dimension names.
4. Click OK.



To Rename a Dimension

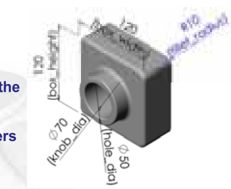
1. Display the dimension.
   - Either double-click the feature to display its dimensions.
   - Or, right-click the Annotations folder, and select Show Feature Dimensions.
2. Click the 70mm diameter dimension, and in the PropertyManager, rename the dimension to *knob_dia*, then click OK.

Note: "*@Sketch2*" *is automatically added to the dimension name.*



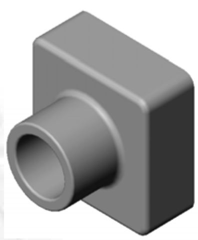
Rename these Dimensions

- Height of the box to *box_height.*
- Width of the box to *box_width.*
- Diameter of the hole in the knob to *hole_dia.*
- Radius of outside corners to *fillet_radius.*



Design Intent

- The depth of the *Knob* should always be equal to the depth of the *Box* (the base feature).
- The *Knob* should always be centered on the *Box*.
- Dimensions alone are not always the best way to capture design intent.

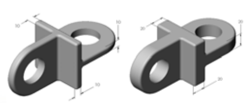


Linking Values

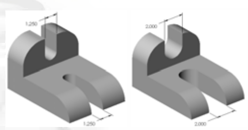
- The Link Values command relates dimensions to each other through shared variable names.
- If the value of one linked dimension is modified, then all of the linked dimensions are modified.
- Link Values is excellent for making feature dimensions equal to each other.
- This is an important tool for capturing design intent.

Examples of Uses for Link Values

- The thickness of the square and the two tabs is always equal.



- The width of both slots is always equal.



Link the Depth of the *Box* to the Depth of the *Knob*

1. Display the dimensions.
2. Right-click on the depth dimension for the *Box*, and select Link Values from the shortcut menu.

Linking the *Box* to the *Knob*

3. Type *Depth* in the Name text box and then click OK.

4. Right-click on the depth dimension for the *Knob*, and select Link Values from the shortcut menu.

Linking the *Box* to the *Knob*

5. Select *Depth* from the list, and click OK.
6. Both dimensions have the same name and value.
7. Rebuild the part to update the geometry.

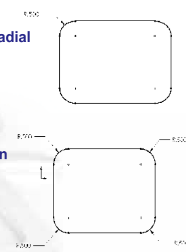
Tip: Use the CTRL key to select several dimensions at the same time and link them in one step.

Geometric Relations

- Relate geometry through physical relationships such as:
  - Concentric
  - Coradial
  - Midpoint
  - Equal
  - Collinear
  - Coincident

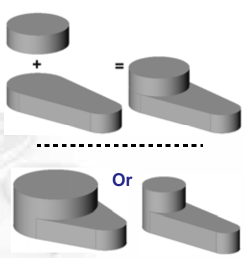
Examples of Geometric Relations

- The Sketch Fillet tool automatically creates one radial dimension and 3 Equal relations.
- Changing the dimension changes all 4 fillets.
- This technique is better than having 4 radial dimensions.



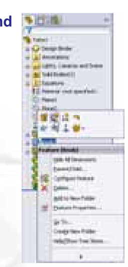
Examples of Geometric Relations

- Two features.
- Making the circle for the boss Coradial with the edge of the base ensures that the boss will always be the correct size regardless of how the base changes.



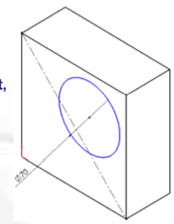
To Center the *Knob* on the *Box*

1. Right-click the *Knob* feature, and select Edit Sketch from the shortcut menu.



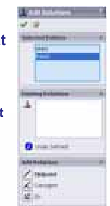
Centering the *Knob* on the *Box*

2. Delete the linear dimensions.
3. Notice the circle is blue, indicating it is under defined.
4. Drag the circle to one side. Without dimensions to locate it, it is free to move.
5. Click Centerline, and sketch a diagonal Centerline.



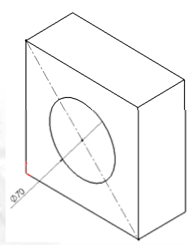
Centering the *Knob* on the *Box*

6. Click Add Relation .
7. Select the centerline and the point at the center of the circle.
   - Note: If the centerline is still highlighted when Add Relations opens, the line automatically appears in the Selected Entities list and you do not have to select it again.
   - If you select the wrong entity, right-click in the graphics area, and select Clear Selections.



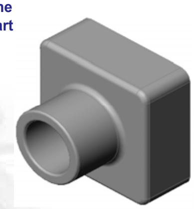
Centering the *Knob* on the *Box*

8. Click Midpoint, and then click Apply and Close.
9. The circle will now stay centered on the *Box* feature.



Centering the *Knob* on the *Box*

10. Click Rebuild to exit the sketch and rebuild the part

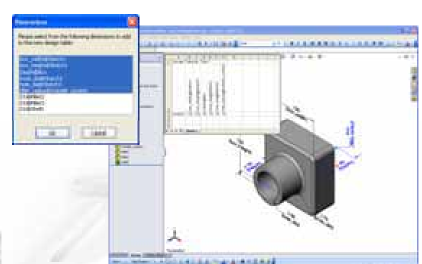


To Insert a New Design Table

1. Position the part in the lower right hand corner of the graphics area.
2. Click Insert, Design Table.

   The PropertyManager appears.
3. Select the Auto-create option to create a new design table automatically .

Inserting a New Design Table

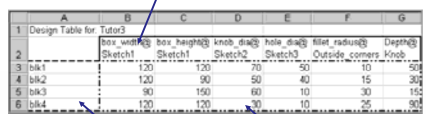


Inserting a New Design Table

- An Excel worksheet is displayed in the part document window.
- Excel toolbars replace the SolidWorks toolbars.
- By default, the first configuration is named *Default*. You can (and should) change this to something more meaningful.

Review of a Design Table's Format

Dimension and/or Feature names or special keywords go in this row.



	A	B	C	D	E	F	G
1	Design Table for: Tutor3						
2		box_width@ Sketch1	box_height@ Sketch1	knob_dia@ Sketch2	hole_dia@ Sketch3	fillet_radius@ Outside_corners	Depth@ Knob
3	blk1	120	120	70	50	10	50
4	blk2	120	90	50	40	15	30
5	blk3	90	150	60	10	30	15
6	blk4	120	120	30	10	25	90

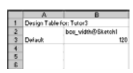
Configuration names go in this column.

Values go here.

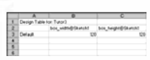
Inserting a New Design Table

1. Double-click the *box_width* dimension.

   The full dimension name is inserted into cell B2. The dimension value is inserted into cell B3.
   The next cell, C2, is automatically selected.

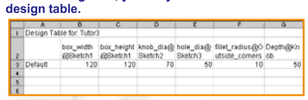


2. Double-click the *box_height* dimension.



Inserting a New Design Table

3. Repeat this process for *knob_dia*, *hole_dia*, *fillet_radius*, and *Depth*.
   - Note: Since the depth dimensions of the Knob and the Box are linked together, you only need one of them in the design table.

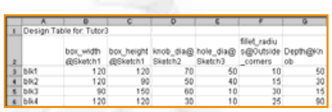


	A	B	C	D	E	F	G
1	Design Table for: Tutor3						
2		box_width @Sketch1	box_height @Sketch1	knob_dia@ Sketch2	hole_dia@ Sketch3	fillet_radius@O utside_corners	Depth@Kn ob
3	Default	120	120	70	50	10	50

Excel tip: Dimension names tend to be very long. Use the Excel command Format, Cells, and click Wrap Text on the Alignment tab.

Inserting a New Design Table

1. Enter new configuration names in column A:
   - Replace Default with blk1.
   - Fill cells A4 through A6 with blk2, blk3, and blk4.
2. Fill in the dimension values as shown below.



	A	B	C	D	E	F	G
1	Design Table for: Tutor3						
2		box_width @Sketch1	box_height @Sketch1	knob_dia@ Sketch2	hole_dia@ Sketch3	fillet_radiu s@Outside _corners	Depth@Kn ob
3	blk1	120	120	70	50	10	50
4	blk2	120	90	50	40	15	30
5	blk3	90	150	60	10	30	15
6	blk4	120	120	30	10	25	90

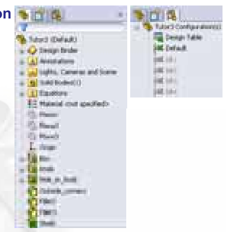
To Close the Excel Worksheet

1. Click in the graphics area outside the worksheet.
2. The system builds the configurations.

3. Click OK.
   The Design Table is embedded and stored in the part document.
   The design table icon appears in the FeatureManager.
4. Save the part document.

To View Part Configurations

1. Click the ConfigurationManager tab at the top of the FeatureManager window.
   The list of configurations is displayed.
2. Double-click each configuration.



Viewing Part Configurations

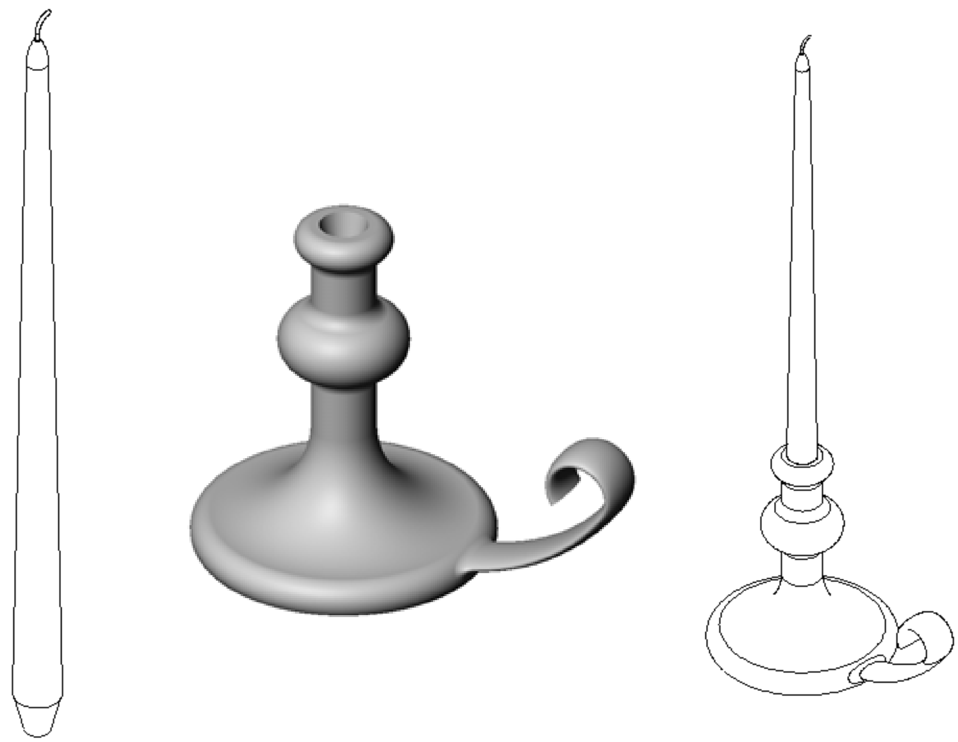
3. The part is automatically rebuilt using the dimension values from the design table.

9

Lesson 9: Revolve and Sweep Features

Goals of This Lesson

Create and modify the following parts and assembly.



Resources for This Lesson

This lesson plan corresponds to *Building Models: Revolves and Sweeps* in the SolidWorks Tutorials.

The Certified SolidWorks Associate Exam (CSWA) proves to employers that students have the fundamental design competencies www.solidworks.com/cswa.

Review of Lesson 8: Design Tables

Questions for Discussion

1. What is a configuration?

   Answer: A configuration is a way to create a family of similar parts within one file.

2. What is a design table?

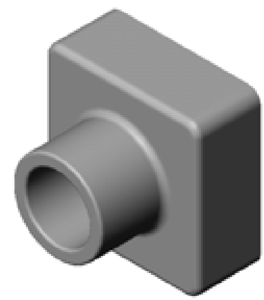
   Answer: A design table is a spreadsheet that lists the different values that are assigned to the various dimensions and features in a part. A design table is an easy way to create many configurations.

3. What are three key elements of a design table?

   Answer: Configuration names, dimension and/or feature names, and their values.

4. What features in `Tutor3` were used to create the design table?

   Answer: The features used to create the design table are: `Box`, `Knob`, `Hole_in_Knob`, and `Outside_corners`.



5. What additional features in `Tutor3` could be added to the design table?

   Answer: The additional features that could be added to the design table are: `Fillet2`, `Fillet3`, and `Shell1`.

Outline of Lesson 9

- ❑ In Class Discussion — Describing a Swept Feature
- ❑ Active Learning Exercises — Creating a Candlestick
- ❑ Exercises and Projects — Creating a Candle to Fit the Candlestick
  - Revolve Feature
  - Create an Assembly
  - Create a Design Table
- ❑ Exercises and Projects — Modify the Outlet Plate
  - Sketch the Sweep Section
  - Create the Sweep Path
- ❑ More to Explore — Design and Model a Mug
- ❑ More to Explore — Use Revolve Feature to Design a Top
- ❑ Lesson Summary

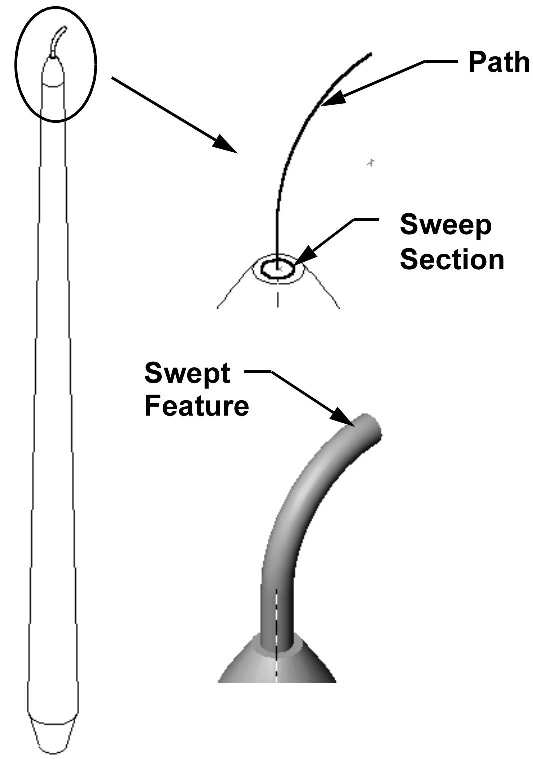
Competencies for Lesson 9

Students develop the following competencies in this lesson:

- ❑ **Engineering**: Explore different modeling techniques that are utilized for parts molded or machined in a lathe process. Modify the design to accept a candle of different sizes.
- ❑ **Technology**: Explore the difference in plastic design for cups and travel mugs.
- ❑ **Math**: Create axes and a profile of revolution to create a solid, 2D ellipse, and arcs.
- ❑ **Science**: Calculate the volume and unit conversion for a container.

In Class Discussion — Describing a Swept Feature

- ❑ Show your students a candle.
- ❑ Ask them to describe the swept feature of the candle wick.



Answer

The swept feature is created with a sketched 2D path and a circular cross section.

The path is sketched on the `Right` plane.

The sweep section is sketched on the top circular face. The top face is parallel to the `Top` plane.

Active Learning Exercises — Creating a Candlestick

Create the candlestick. Follow the instructions in *Building Models: Revolves and Sweeps* in the SolidWorks Tutorials.

The part name is `Cstick.sldprt`. However, throughout this lesson, we will refer to it as "candlestick" because that makes more sense.

Lesson 9 — 5 Minute Assessment — Answer Key

Name: ______________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What features did you use to create the candlestick?

Answer: Revolved boss, swept boss, and extruded cut features.

2 What special piece of sketch geometry is useful, but *not required* for a revolve feature?

Answer: A centerline.

3 Unlike an extruded feature, a swept feature requires a minimum of two sketches. What are these two sketches?

Answer: The sweep section and the sweep path.

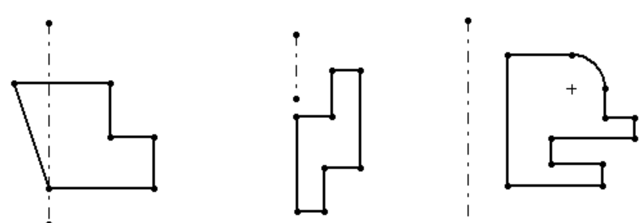
4 What information does the pointer provide while sketching an arc?

Answer: The pointer displays: arc angle in degrees, arc radius and inferences to model or sketch geometry.

5 Examine the three illustrations at the right. Which one is not a valid sketch for a revolve feature?

Why?

Answer: Sketch **A** is not a valid sketch for a revolve feature because the profile crosses the centerline.



A **B** **C**

Lesson 9 — 5 Minute Assessment **REPRODUCIBLE**

Name: ______________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1. What features did you use to create the candlestick?

   ____

2. What special piece of sketch geometry is useful, but *not required* for a revolve feature?

   ____

3. Unlike an extruded feature, a swept feature requires a minimum of two sketches. What are these two sketches?

   ____

4. What information does the pointer provide while sketching an arc?

   ____

   ____

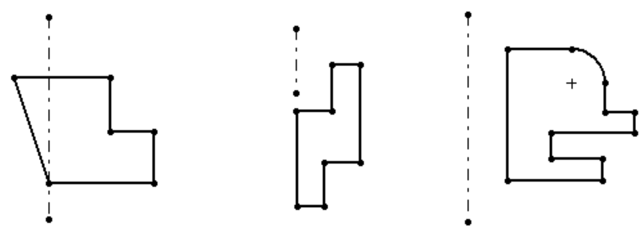
5. Examine the three illustrations at the right. Which one is not a valid sketch for a revolve feature?

   Why?

   ____

   ____

   ____



A B C

   ____

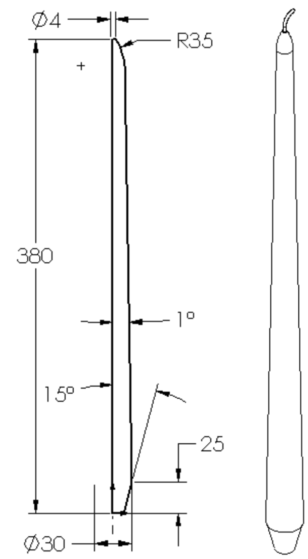
   ____

Exercises and Projects — Creating a Candle to Fit the Candlestick

Task 1 — Revolve Feature

Design a candle to fit the candlestick.

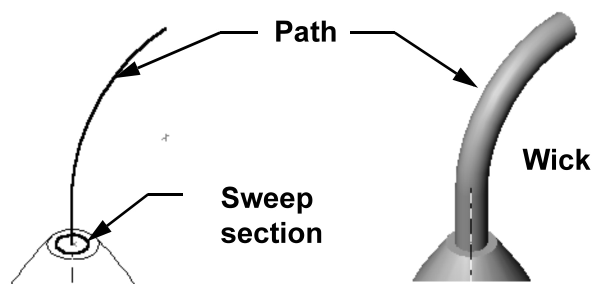
- ❑ Use a revolve feature as the base feature.
- ❑ Taper the bottom of the candle to fit into the candlestick.
- ❑ Use a sweep feature for the wick.



Answer:

There are numerous answers to this question. One possible solution is shown at the right. Below are the key design issues:

- ❑ Review the dimensions of the extruded cut on the candlestick.
  - The diameter of the extruded cut is 30mm.
  - The depth of the extruded cut is 25mm.
  - The draft angle is 15°.
- ❑ The dimensions of the taper on the end of the candle must equal the dimensions of the extruded cut on the top of the candlestick. Otherwise the candle will not properly fit into the candlestick.
- ❑ The swept feature for the wick is created with a sketched 2D path and a circular sweep section.
  - The path is sketched on the `Right` plane.
  - The cross section is sketched on the top circular face. The top face is parallel to the `Top` plane.



Swept Feature

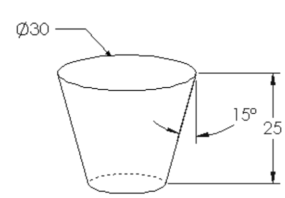
Question:

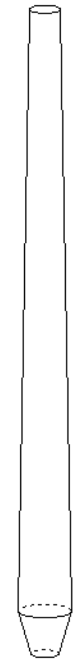
What other features could you use to create the candle? Use a sketch to illustrate your answer if necessary.

Answer:

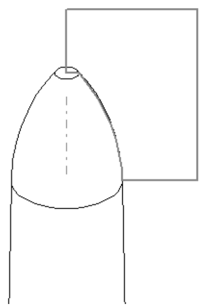
Answers may vary. One solution is shown in the illustrations below.

Sketch a **30mm** diameter circle on the `Top` plane and extrude it a depth of **25mm** with a draft angle of **15°**. This forms the taper at the base of the candle.



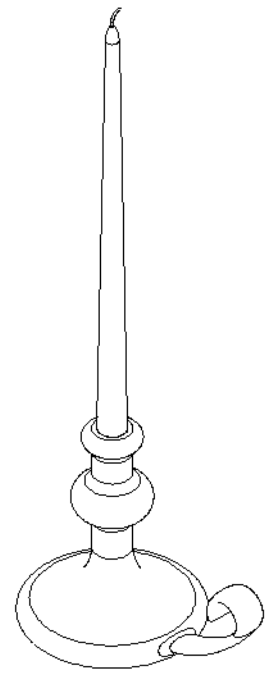


- ❑ Open a sketch on the top face of the taper. Use **Convert Entities** to copy the edge, and extrude a boss the desired height of the candle with a draft angle of **1°**.
- ❑ Make a revolved *cut* feature to round off the top of the candle.



Task 2 — Create an Assembly

Create a candlestick assembly.



Answer:

The appearance of the completed assembly will depend on the design of the student's candle.

- A sample candlestick assembly is found in the `Lessons\Lesson09` folder in `SolidWorks Teacher Tools`.
- Two mates are required to fully define the assembly:
  - **Concentric** mate between the two conical faces.

Note: The conical faces are the cone-shaped faces, one on the tapered hole in the candlestick, and one on the taper at the bottom of the candle.

  - **Coincident** mate between the `Front` planes of the candle and the candlestick. This keeps the candle from rotating.

Task 3 — Create a Design Table

You work for a candle manufacturer. Use a design table to create 380 mm, 350 mm, 300 mm, and 250 mm candles.

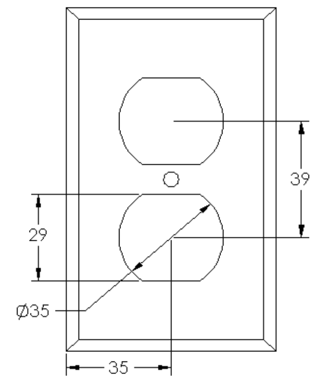
Answer:

	A	B
1	Design Table for: candle	
2		Length@Sketch1
3	380 mm candle	380
4	350 mm candle	350
5	300 mmh candle	300
6	250 mm candle	250

Sheet1

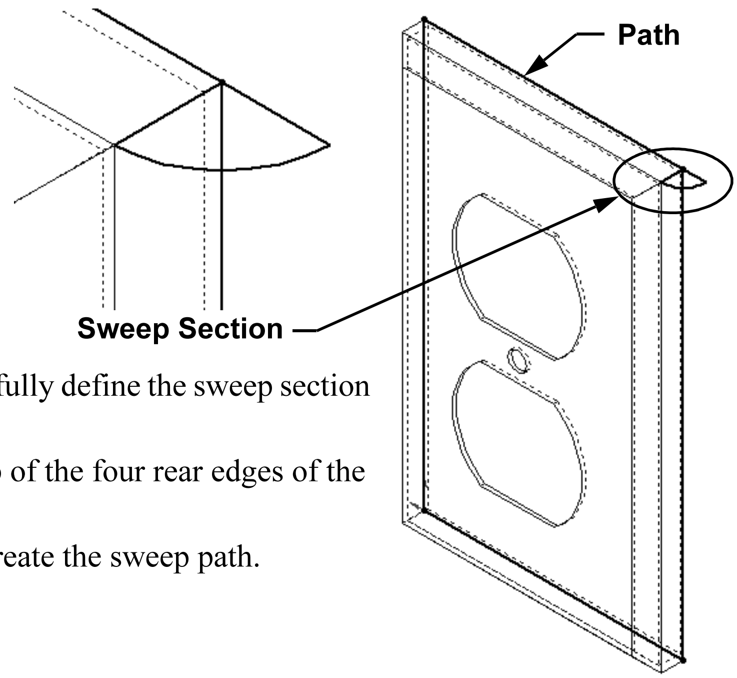
- A design table requires configuration names, dimension and/or feature names, and their values.
- The configuration names are:
  - `380 mm candle`
  - `350 mm candle`
  - `300 mm candle`
  - `250 mm candle`
- The dimension name is `Length`.
- The four dimension values are 380, 350, 300 and 250 mm.
- Change the default configuration name from `First Instance` to `380 mm candle`.

Exercises and Projects — Modify the Outlet Plate



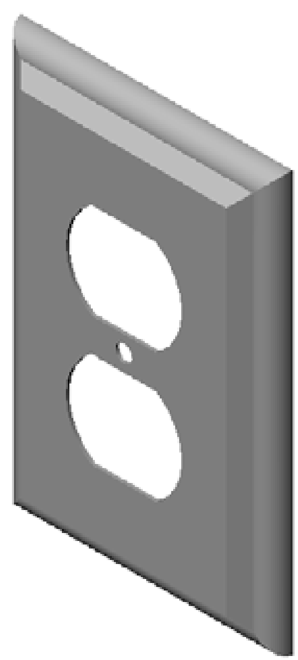
Modify the `outletplate` that you created earlier in Lesson 2.

- ❑ Edit the sketch for the circular cuts that form the openings for the outlet. Create new cuts using the sketch tools. Apply what you have learned about **Link Values** and geometric relations to properly dimension and constrain the sketch.



- ❑ Add a swept boss feature to the back edge.
  - The sweep section includes a 90° arc.
  - The radius of the arc is equal to the length of the model edge as shown in the accompanying illustration.
  - Use geometric relations to fully define the sweep section sketch.
  - The sweep path is made up of the four rear edges of the part.
  - Use **Convert Entities** to create the sweep path.

- ❑ The desired result is shown in the illustration at the right.

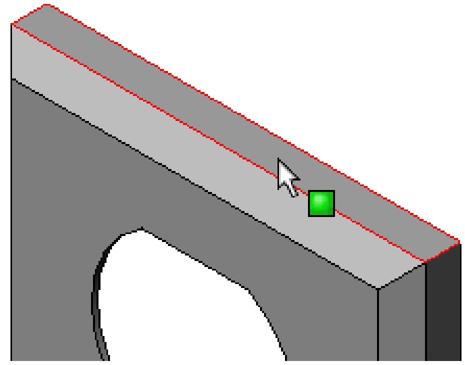


Answer:

- ❑ The `modified outletplate` is found in the `Lesson09` folder.
- ❑ If your students need assistance creating the swept feature, here is the procedure:

Sketch the Sweep Section

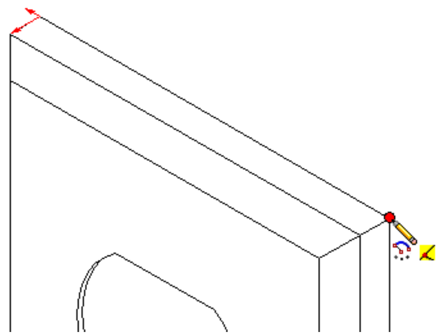
1 Select the upper face of the `outletplate`, and click **Insert, Sketch**, or click **Sketch** on the Sketch toolbar. This will be the sketch plane for the sweep section.



2 Click **Centerpoint Arc** on the Sketch toolbar.

3 Position the pointer at the end of the model edge.

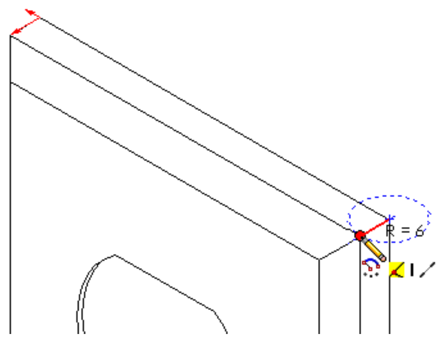
Look for the coincident relation in the pointer indicating that you are snapping coincident to the end of the model edge. This establishes the center of the arc.



4 Define the radius.

Click the left mouse button. Move the pointer to the other end of the edge. Again, look for the coincident relation in the pointer.

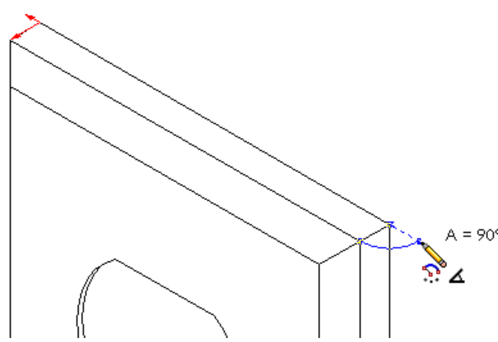
5 Click the left mouse button. This establishes the radius of the arc.



6 Define the circumference.

As you move the pointer to define the circumference, look for the inference line that indicates the endpoint of the arc is lined up with the back edge of the model.

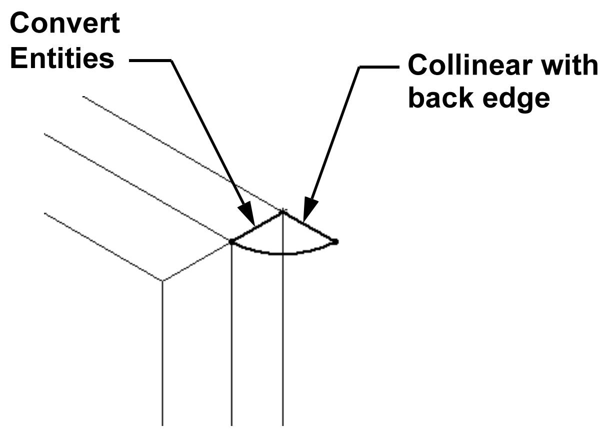
When you see the inference line indicating a 90° arc, click the left mouse button.



7 Finish the profile.

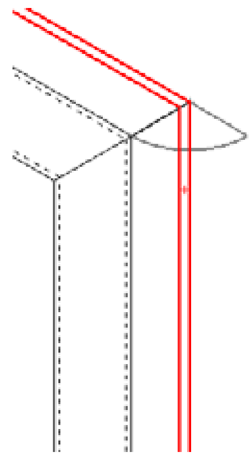
Two lines are needed to close the profile. One line can be created by using **Convert Entities** on the model edge. The second line should be Collinear with the back edge of the model.

8 Exit the sketch.



Create the Sweep Path

1 Select the rear face of the model and insert a new sketch.

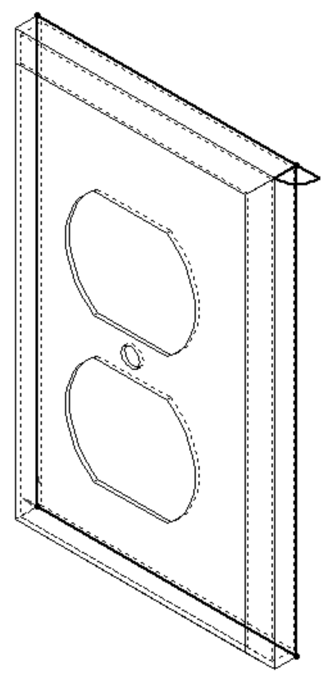


2 Convert the edges.

Use **Convert Entities** to copy the edges of the rear face into the active sketch.

3 Exit the sketch.

4 Sweep the feature.



More to Explore — Design and Model a Mug

Design and model a mug. This is a rather open-ended assignment. You have an opportunity to express your creativity and ingenuity. The design of a mug can vary from the simple to the complex. A couple of examples are shown at the right.

Simple design

More complex design – a commuter's spill-proof travel mug

There are two specific requirements:

- ❑ Use a revolve feature for the body of the mug.
- ❑ Use a swept feature for the handle.

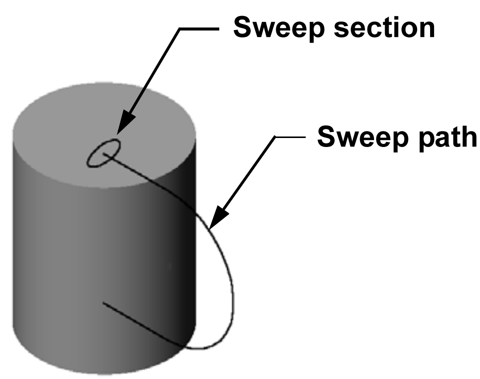
Note: This task can present some interesting challenges for your students. Some of these challenges arise from the lack of knowledge about more advanced modeling techniques.

Here are some representative examples of situations that may arise. They are illustrated using a simple mug design:

- ❑ How to make the handle:

  The handle is a swept feature. Assuming that the typical way of looking at a mug is from the front, the sweep path would be sketched on the `Front` reference plane.

  The sweep section would be sketched on the `Right` reference plane. It should be related to the end of the path with a geometric relation.



Note: The sweep section does *not* have to be an ellipse.

- ❑ The handle sticks through into the inside of the mug.

  This is caused by sweeping the handle *after* the mug is hollowed out.

  Solution: Sweep the handle *before* hollowing out the mug.

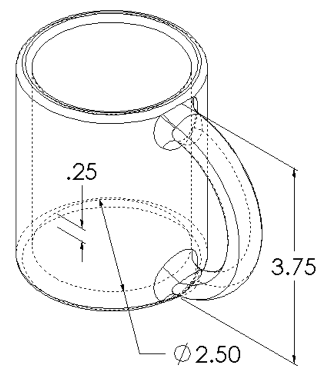
- Ending up with a hollow handle.

  This is caused by hollowing out the mug with a shell feature. When you use the shell feature, you identify the face to be removed, hollowing out the part. Depending on the wall thickness, this can result in a hollow handle, too. If the wall thickness is too great for the size of the handle cross section, the shell feature may fail, also.

  Solution: Use a cut feature to hollow out the mug.

Task 4 — Determine Volume of Mug

How much coffee does the mug shown at the right hold?



Given:

- Inside Diameter = 2.50”
- Overall height of the mug = 3.75”
- Thickness of the bottom = 0.25”
- Coffee cups are not filled to the brim. Allow 0.5” space at the top.

Answer:

- Volume of a cylinder = π * Radius^2 * Height
- “Height” of coffee = 3.75” - 0.25” - 0.5” = 3.0”
- Radius = Diameter ÷ 2
- Volume = $3.14 * 1.25^2 * 3.0 = 14.72\ in^3$

Conversion:

A cup of coffee in the US is sold by the fluid ounce, not by the cubic inch. How many ounces does the mug hold?

Given:

1 gallon = 231 in^3

128 ounces = 1 gallon

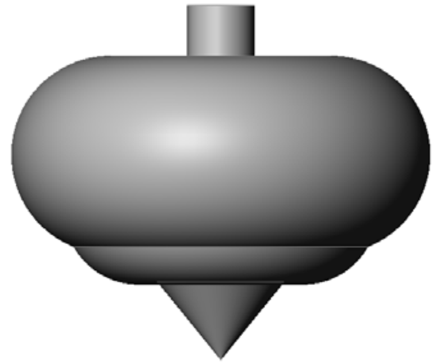
Answer:

- 1 ounce = 231 in^3/gallon ÷ 128 ounces/gallon = 1.80 in^3/ounce.
- Volume = 14.72 in^3 ÷ 1.80 in^3/ounce = 8.18 ounces.

The `mug` conveniently holds 8 ounces of coffee.

More to Explore — Use Revolve Feature to Design a Top

Use a revolve feature to create a toy top of your own design.



Answer:

There are numerous answers to this question. An example is found in the `Lesson9` file folder.

Lesson 9 Quiz — Answer Key

Name: ______________________________ Class: ________ Date:______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

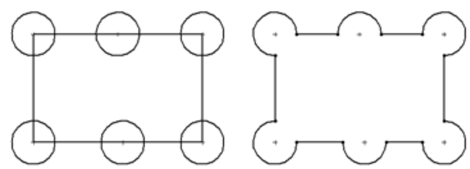
1 How do you create a Revolve feature?

Answer: A Revolve feature is created by rotating a 2D profile around an axis of revolution. Sketch a profile on a 2D plane. Optionally sketch a centerline to be used as an axis. The profile must not cross the axis of revolution. Click the **Revolved Boss/ Base** tool. Enter a rotation angle.

2 What two sketches are required to create a Sweep feature?

Answer: The sweep feature requires a Sweep Path sketch and a Sweep Section sketch.

3 Examine the *Before* and *After* pictures at the right. What sketch tool should you use to delete the unwanted portions of the lines and circles?



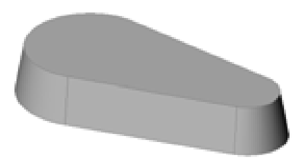
Before *After*

Answer: The **Trim** tool.

4 Where can you find additional sketch tools that are not located on the Sketch Tools toolbar?

Answer: Click **Tools**, **Sketch Entities** from the main menu.

5 Multiple choice. Examine the illustration at the right. How should you create this object?



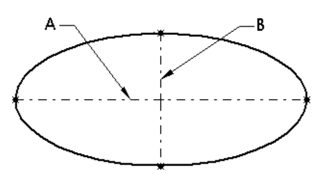
a. Use a **Revolve** feature

b. Use a **Sweep** feature

c. Use an **Extrude** feature with the option **Draft while extruding**.

Answer: c.

6 Examine the illustration of the ellipse at the right. The two axes are labeled **A** and **B**. Identify the two axes.



Answer: **A** is the major axis and **B** is the minor axis.

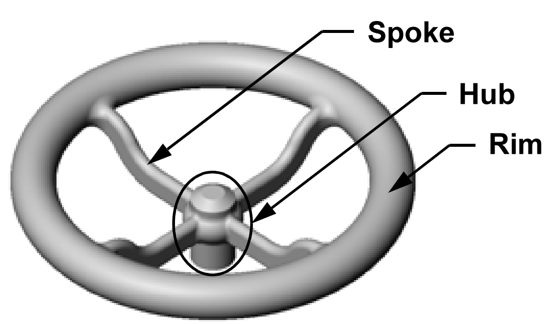
7 True or False. A Base feature is always an Extrude feature.

Answer: False

8 True or False. A sketch must be fully defined in order to create a Revolve feature.

Answer: False

9 Study the illustration at the right. In the space provided, indicate what SolidWorks feature would be *best* to use for each part of the hand wheel.



Answer:

The **Hub**: **Revolve feature**______________

The **Spoke**: **Sweep feature** ______________

The **Rim**: **Revolve feature**______________

Lesson 9 Quiz — REPRODUCIBLE

Name: ______________________ Class: ________ Date: ____________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you create a Revolve feature?

____

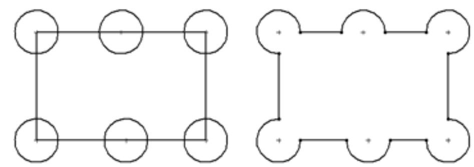
____

2 What two sketches are required to create a Sweep feature?

____

3 Examine the *Before* and *After* pictures at the right. What sketch tool should you use to delete the unwanted portions of the lines and circles?

____



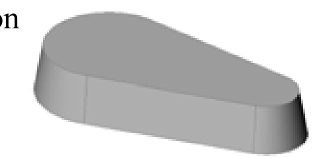
Before *After*

4 Where can you find additional sketch tools that are not located on the Sketch Tools toolbar?

____

____

5 Multiple choice. Circle the best answer. Examine the illustration at the right. How should you create this object?

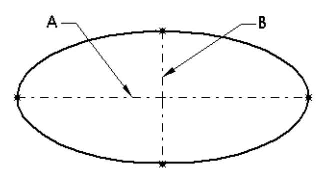


a. Use a **Revolve** feature

b. Use a **Sweep** feature

c. Use an **Extrude** feature with the option **Draft while extruding**.

6 Examine the illustration of the ellipse at the right. The two axes are labeled **A** and **B**. Identify the two axes.



____

____

7 True or False. A Base feature is always an Extrude feature.

____

8 True or False. A sketch must be fully defined in order to create a Revolve feature.

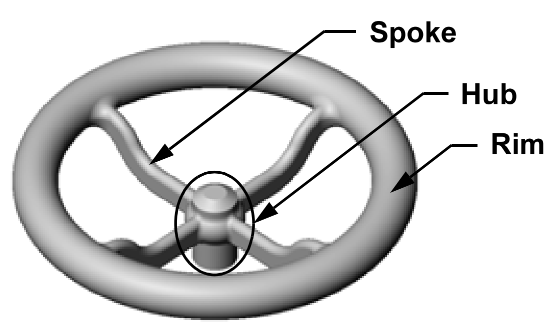
____

9 Study the illustration at the right. In the space provided, indicate what SolidWorks feature would be *best* to use for each part of the hand wheel.

The **Hub**: ____________________

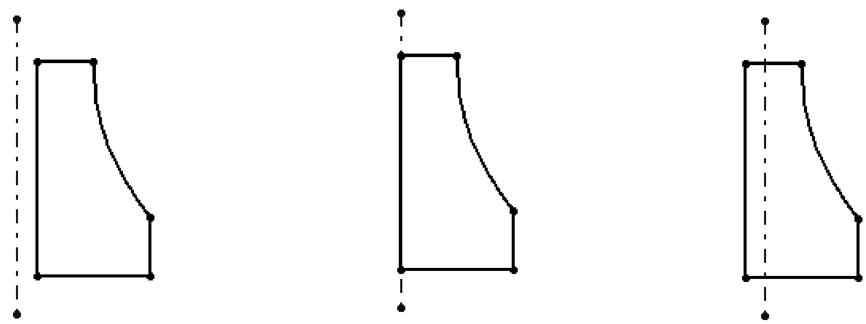
The **Spoke**: __________________

The **Rim**: ____________________



Lesson Summary

- A Revolve feature is created by rotating a 2D profile sketch around an axis of revolution.
- The profile sketch can use a sketch line (that is part of the profile) or a centerline as the axis of revolution.
- The profile sketch *cannot* cross the axis of revolution.

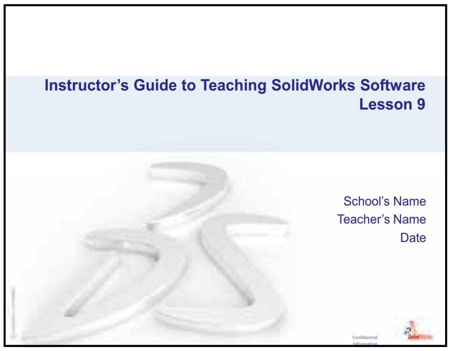


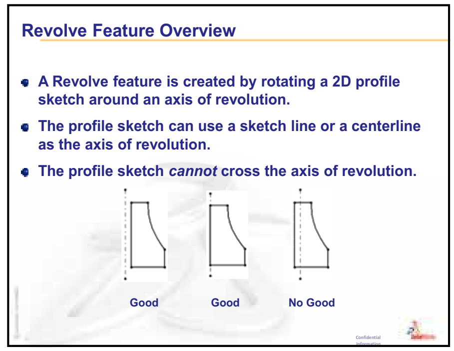
Good Good No Good

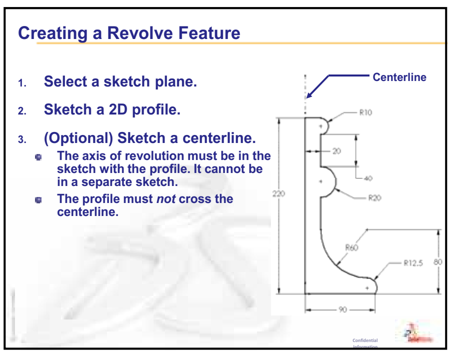
- The Sweep feature is created by moving a 2D profile along a path.
- The Sweep feature requires two sketches:
  - Sweep Path
  - Sweep Section
- Draft tapers the shape. Draft is important in molded, cast, or forged parts.
- Fillets are used to smooth edges.

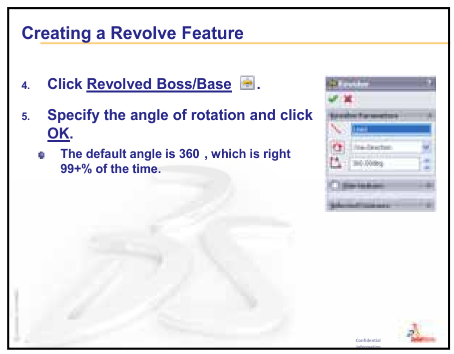
Thumbnail Images of PowerPoint Slides

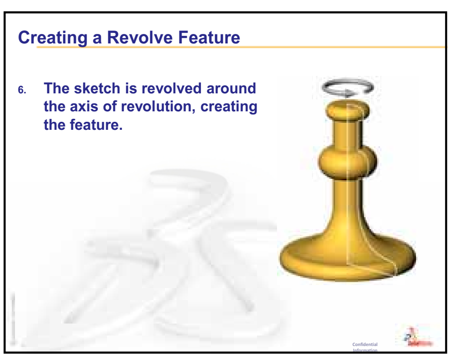
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

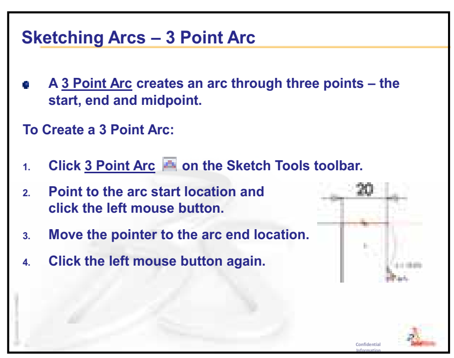






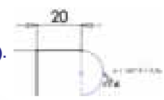






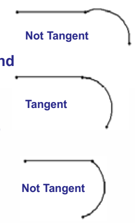
Creating a 3 Point Arc:

5. Drag the arc midpoint to establish the radius and direction (convex vs. concave).
6. Click the left mouse button a third time.



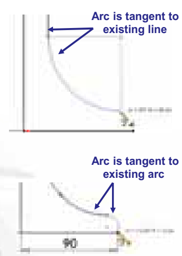
Sketching Arcs – Tangent Arc

- The Tangent Arc tool creates an arc that has a smooth transition to an existing sketch entity.
- Saves the work of sketching an arc and then manually adding a geometric relation to make it tangent.
- Start point of the arc *must* connect to an existing sketch entity.



To Create a Tangent Arc:

1. Click Tangent Arc on the Sketch Tools toolbar.
2. Point to the arc start location, and click the left mouse button.
3. Drag to create the arc.
   - The arc angle and radius values are displayed on the pointer when creating arcs.
4. Click the left mouse button.



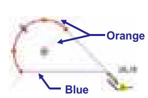
Pointer Feedback

- As you sketch, the pointer provides feedback and information about alignment to sketch entities and model geometry.

	Horizontal		Midpoint
	Vertical		Intersection
	Parallel		Endpoint, Vertex or Centerpoint
	Perpendicular		On
	Tangent		

Inferencing

- Dotted lines appear when you sketch, showing alignment with other geometry.
- This alignment information is called *inferencing.*

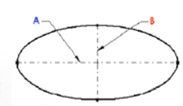


- Inference lines are two different colors: orange and blue.
  - Orange inference lines capture and add a geometric relation such as Tangent.
  - Blue lines show alignment and serve as an aid to sketching, but do not actually capture and add a geometric relation.

  (Note: Orange inference lines may appear as yellow in the SolidWorks graphics view. Orange is used here to aid visibility.)

Ellipse Sketch Tool

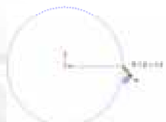
- Used to create the sweep section for the handle of the candlestick.
- An Ellipse has two axes:
  - Major axis, labeled A at the right.
  - Minor axis labeled B at the right.



- Sketching an ellipse is a two-step operation, similar to sketching a 3 Point Arc.

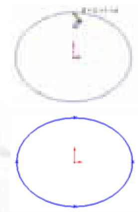
To Sketch an Ellipse:

1. Click Tools, Sketch Entity, Ellipse.
   - Tip: You can use Tools, Customize to add the Ellipse tool to the Sketch Tools toolbar.
2. Position the pointer at the center of the ellipse.
3. Click the left mouse button, and then move the pointer horizontally to define the major axis.
4. Click the left mouse button a second time.



Sketching an Ellipse:

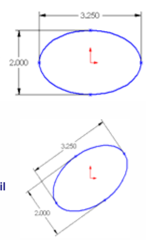
5. Move the pointer vertically to define the minor axis.
6. Click the left mouse button a third time. This completes sketching the ellipse.



Fully Defining an Ellipse

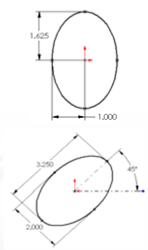
Requires 4 pieces of information:

- Location of the center:
  - Either dimension the center or locate it with a geometric relation such as Coincident.
- Length of the major axis.
- Length of the minor axis.
- Orientation of the major axis.
  - Even though the ellipse at the right is dimensioned, and its center is located coincident to the origin, it is free to rotate until the orientation of the major axis is defined.



More About Ellipses

- The major axis does not have to be horizontal.
- You can dimension half the major and/ or minor axis.
  - It is like dimensioning the radius of a circle instead of the diameter.
- You do not have to use a geometric relation to orient the major axis.
  - A dimension works fine.

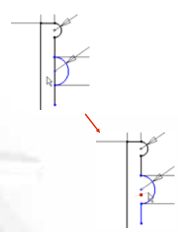


Trimming Sketch Geometry

- The Trim tool is used to delete a sketch segment.
- Power trim is the quickest and most intuitive method. Other methods are useful in certain circumstances.
- With Power trim, segments are deleted up to their intersection with another sketch entity.
- The entire sketch segment is deleted if it does not intersect any other sketch entity.
- To use Power trim, click and drag the pointer over the segment(s) to be removed. Multiple segments can be deleted in one operation.

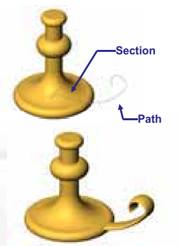
To Trim a Sketch Entity:

1. Click Trim on the Sketch Tools toolbar.
2. Select Power trim .
3. Position the pointer adjacent to the segment to be trimmed, and click and hold the left mouse button.
4. Drag the cursor across the segment, and release the mouse button.
5. The segment is deleted.



Sweep Overview

- The Sweep feature is created by moving a 2D profile along a patl
- A Sweep feature is used to crea the handle on the candlestick.
- The Sweep feature requires two sketches:
  - Sweep Path
  - Sweep Section



Sweep Overview – Rules

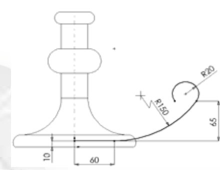
- The sweep path is a set of sketched curves contained in a sketch, a curve, or a set of model edges.
- The sweep section must be a closed contour.
- The start point of the path must lie on the plane of the sweep section.
- The section, path or the resulting solid cannot be self-intersecting.

Sweep Overview – Tips

- Make the sweep path first. Then make the section.
- Create small cross sections away from other part geometry.
- Then move the sweep section into position by adding a <u>Coincident</u> or <u>Pierce</u> relation to the end of the sweep path.

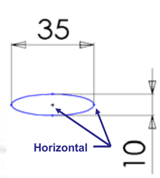
To Create the Sweep Path:

1. Open a sketch on the Front plane.
2. Sketch the Sweep path using the <u>Line</u> and <u>Tangent Arc</u> sketch tools.
3. Dimension as shown.
4. Close the sketch.



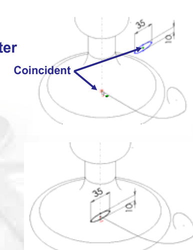
To Create the Sweep Section:

1. Open a sketch on the Right plane.
2. Sketch the Sweep section using the <u>Ellipse</u> sketch tool.
3. Add a <u>Horizontal</u> relation between the center of the ellipse and one end of the major axis.
4. Dimension the major and minor axes of the ellipse.



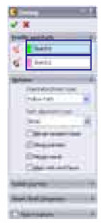
Creating the Sweep Section:

5. Add a <u>Coincident</u> relation between the center of the ellipse and the endpoint of the path.
6. Close the sketch.



To Sweep the Handle:

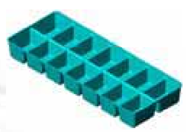
1. Click Swept Boss/Base on the Features toolbar.
2. Select the Sweep path sketch.
3. Select the Sweep section sketch.
4. Click OK.



Sweeping the Handle – Results

Extruded Cut with Draft Angle

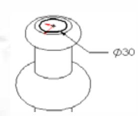
- Creates the opening for a candle in the top of the candlestick.
- Same process as extruding a boss except it removes material instead of adding it.
- Draft tapers the shape.
- Draft is important in molded, cast, or forged parts.
  - Example: Ice cube tray – without draft it would be very hard to get the ice cubes out of the tray.
  - Find other examples.



To Create the Cut:

1. Open a sketch on the top face of the candlestick.

2. Sketch a circular profile Concentric to the circular face.
3. Dimension the circle.

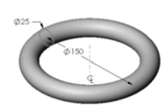


Creating the Cut:

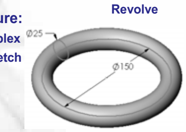
4. Click Extruded Cut on the Features toolbar.
5. End Conditions:
   - Type = Blind
   - Depth = 25mm
   - Draft = On
   - Angle = 15
6. Click OK.

Best Practice – Keep it Simple

- Do not use a sweep feature when a revolve or extrude will work.
- Sweeping a circle along a circular path appears to give the same result as a revolve feature.
- However, the revolve feature:
  - Is mathematically less complex
  - Is easier to sketch – one sketch vs. two



Revolve



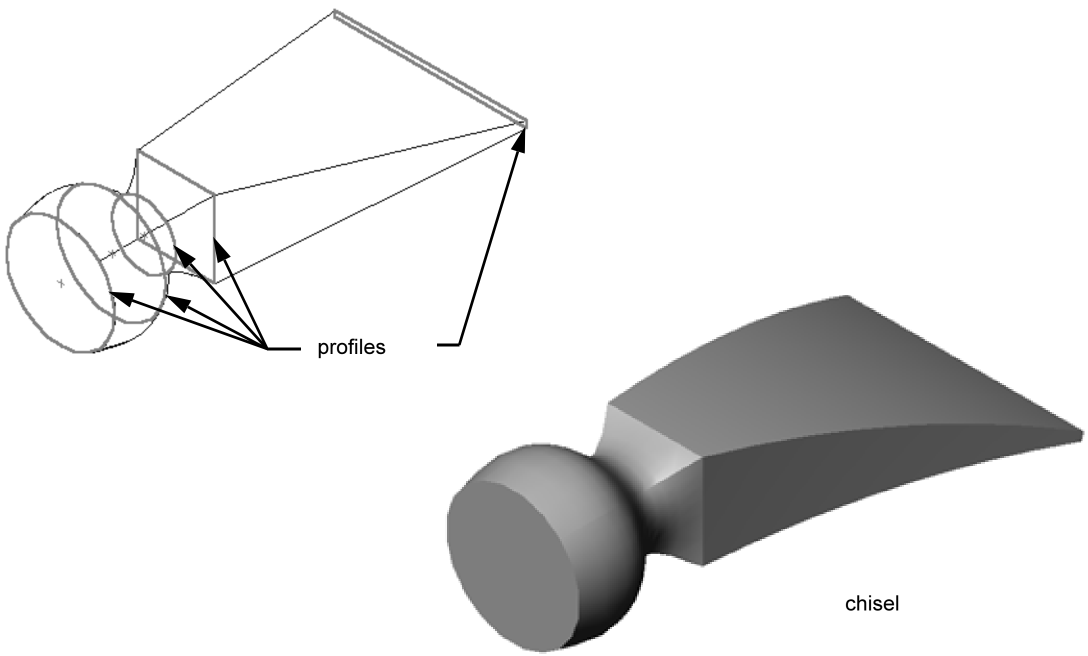
Sweep

10

Lesson 10: Loft Features

Goals of This Lesson

Create the following part.



Resources for This Lesson

This lesson plan corresponds to *Building Models: Lofts* in the SolidWorks Tutorials.

Additional SolidWorks tutorials provide knowledge into sheet metal, plastic and machine parts.

Review of Lesson 9: Revolve and Sweep Features

Questions for Discussion

1 Describe the steps required to create a revolved feature.

<u>**Answer:**</u> To create a revolved feature:

- Sketch a profile on a 2D plane.
- The profile sketch may optionally include a centerline as the axis of revolution. The centerline (or sketch line as axis of revolution) must not cross the profile.
- Click **Revolved Boss/Base**  on the Features toolbar.
- Enter a rotation angle. The default angle is 360°.

2 Describe the steps required to create a swept feature.

<u>**Answer:**</u> To create a swept feature:

- Sketch the Sweep path. The path must not be self-intersecting.
- Sketch the Sweep section.
- Add a Geometric Relation between the sweep section and the path.
- Click **Swept Boss/Base**  on the Features toolbar.
- Select the Sweep path.
- Select the Sweep cross section.

3 Each of the following parts was created with *one* feature.

- Name the Base feature for each part.
- Describe the 2D geometry used to create the Base feature of the part.
- Name the sketch plane or planes required to create the Base feature.

Part 1

Part 2

Part 3

<u>**Answer:**</u>

- Part 1: Extrude – created with an L-shaped profile sketched on the `Right` plane.
- Part 2: Revolve – created with 3 tangent arcs and 3 lines and a centerline sketched on the `Top` plane. The angle of rotation is 270°. **Note:** The 2D profile could also be sketched on the `Right` plane.
- Part 3: Sweep – created with an ellipse cross section sketched on the `Right` plane and an S-shaped path composed of 2 lines and 2 tangent arcs sketched on the `Front` plane.

Outline of Lesson 10

- In Class Discussion — Identifying Features
- Active Learning Exercises — Creating the Chisel
- Exercises and Projects — Creating the Bottle
- Exercises and Projects — Creating a Bottle with Elliptical Base
- Exercises and Projects — Creating a Screwdriver
- More to Explore — Designing a Sports Drink Bottle
  - Design a Bottle
  - Calculate Costs
- Lesson Summary

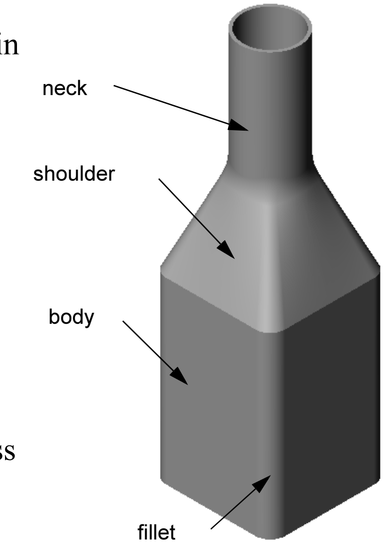
Competencies for Lesson 10

Students develop the following competencies in this lesson:

- **Engineering**: Explore different design changes to modify the function of a product.
- **Technology**: Knowledge of how thin wall plastic parts are developed from lofts.
- **Math**: Understand tangency effects on surfaces.
- **Science**: Estimate volume for different containers.

In Class Discussion — Identifying Features

Show the students the finished bottle they will build in Task 1. The completed bottle is in the Leson10 folder in the SolidWorks Teacher Tools directory. Ask the students to describe the features that make up the bottle.



- ❑ What feature would be used to create the body of the bottle?
- ❑ How do you create the shoulder of the bottle?
- ❑ Describe the other features used to create the bottle.

Answer:

- ❑ The body of the bottle is created with an extruded boss feature. Sketch a square profile on the Top plane. Use a Fillet feature to round the edges of the body.
- ❑ The shoulder of the bottle is created with a Loft feature. The Loft feature is composed of two profiles. The first is the top face of the extruded boss feature. The second profile is a circle sketched on a plane parallel to the Top plane.
- ❑ The neck of the bottle is created with a extrude boss feature. The sketch is a circle converted from the top face of the shoulder.
- ❑ The shell feature is used to hollow out the bottle.
- ❑ A fillet feature is used to remove the sharp edge between the shoulder and the neck.

Question

What would be the result if the body and shoulder were created as a single feature by lofting through three profiles?

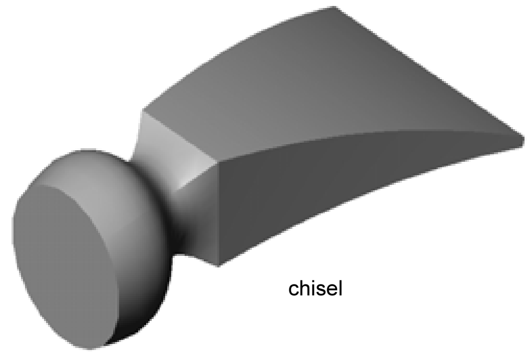
Answer:

The result is shown at the right.

- ❑ A 5mm fillet is added to the four edges of the body/shoulder after the loft is complete.
- ❑ The neck is extruded as before.
- ❑ A 15mm fillet is created around the joint where the neck meets the shoulder.
- ❑ A 1mm shell is used to hollow out the bottle.

Active Learning Exercises — Creating the Chisel

Create the `chisel`. Follow the instructions in *Building Models: Lofts* in the SolidWorks Tutorials.



chisel

Lesson 10 — 5 Minute Assessment — Answer Key

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What features were used to create the `chisel`?
Answer: Two Loft features and a Flex feature.

2 Describe the steps required to create the first Loft feature for the `chisel`.
Answer: To create a first Loft feature:
- Create the planes required for the profile sketches.
- Sketch a profile on the first plane.
- Sketch the remaining profiles on the corresponding planes.
- Click **Loft** on the Features toolbar.
- Select the profiles.
- Review the connecting curve.
- Click **OK**.

3 What is the minimum number of profiles required for a Loft feature?
Answer: The minimum number of profiles for a Loft feature is two.

4 Describe the steps to copy a Sketch onto another plane.
Answer: To copy a Sketch to an existing reference plane:
- Select the sketch in the FeatureManager design tree.
- Click **Copy** on the Standard toolbar.
- Select the new plane in the FeatureManager design tree.
- Click **Paste** on the Standard toolbar.

Lesson 10 — 5 Minute Assessment **REPRODUCIBLE**

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What features were used to create the `chisel`?

____

2 Describe the steps required to create the first Loft feature for the `chisel`.

____

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3 What is the minimum number of profiles required for a Loft feature?

____

4 Describe the steps to copy a Sketch onto another plane.

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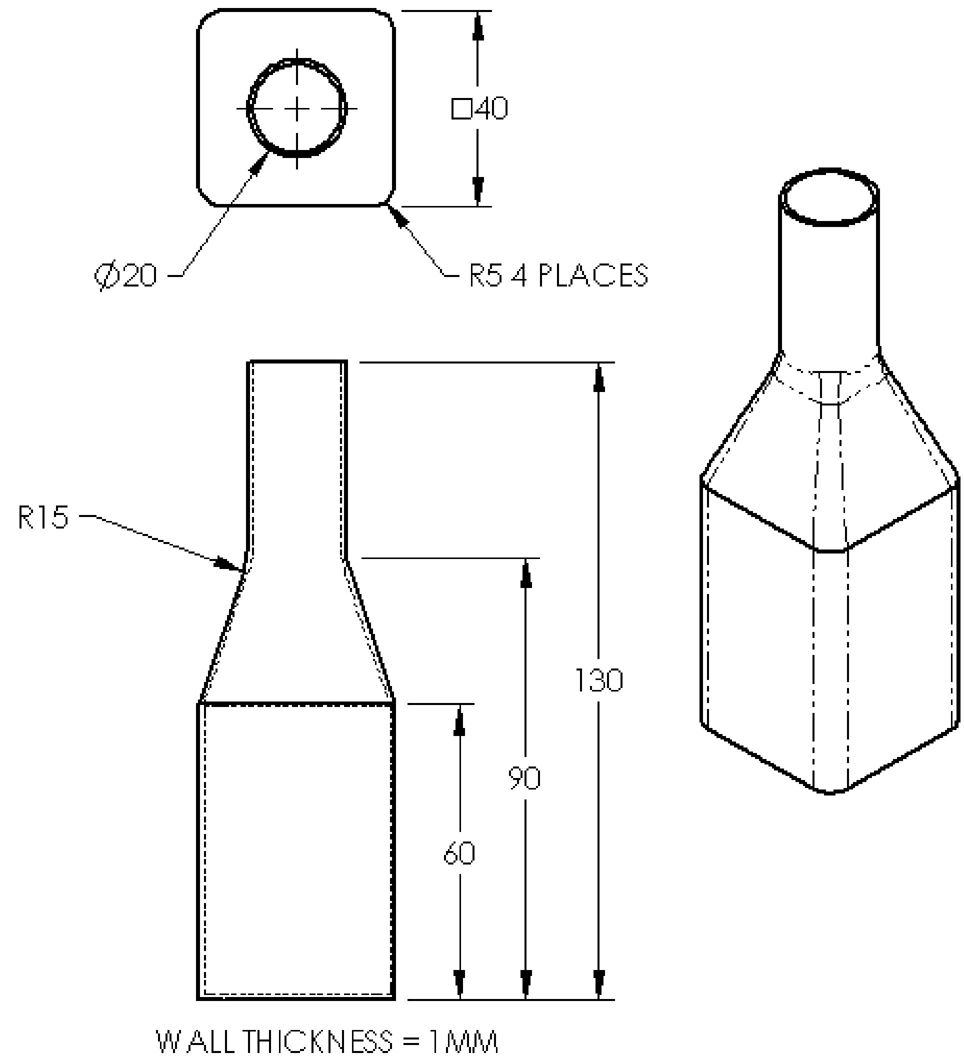
Exercises and Projects — Creating the Bottle

Create the `bottle` as shown in the drawing.



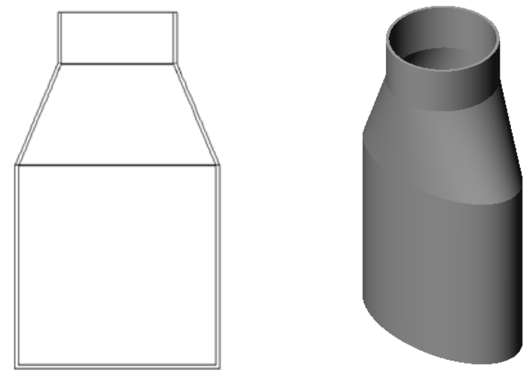
Note: All dimensions in the Bottle exercise are in millimeters.

A completed example of the `Bottle` is found in the `Lesson10` file folder.

Exercises and Projects — Creating a Bottle with Elliptical Base

Create `bottle2` with an elliptical extruded boss feature. The top of the bottle is circular. Design `bottle2` with your own dimensions.

Note: `Bottle2` is found in the `Lesson10` file folder.

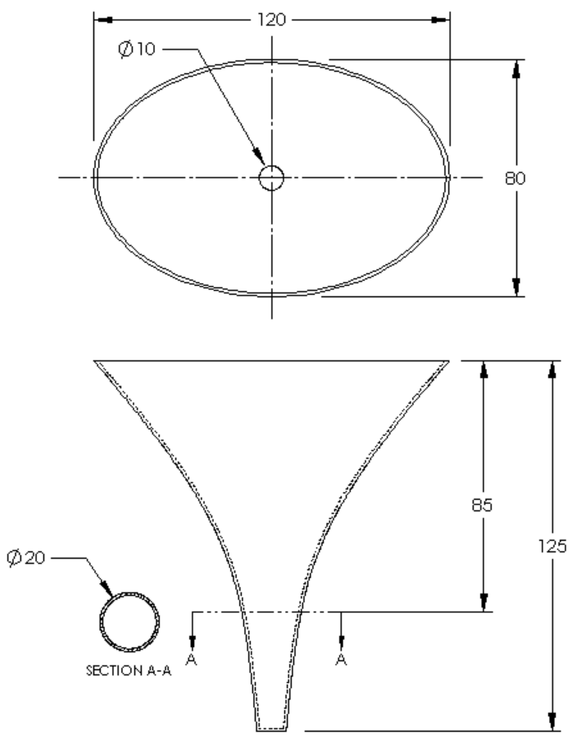


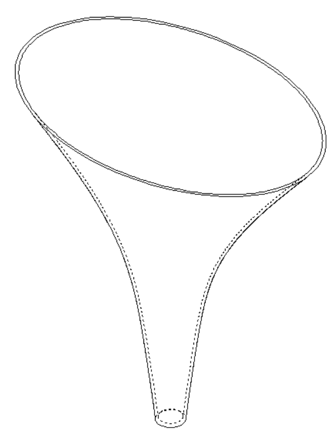
`bottle2`

Exercises and Projects — Creating a Funnel

Create the `funnel` as shown in the drawing below.

- ❑ Use **1mm** for the wall thickness.



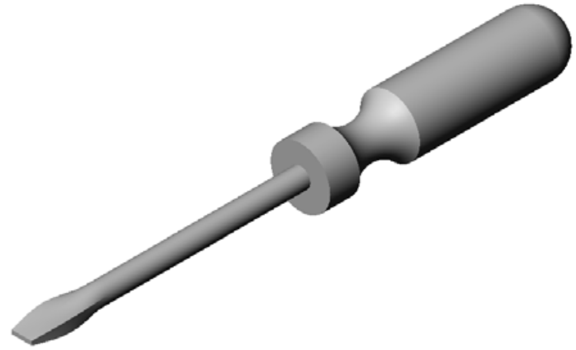


The completed `funnel` is found in the `Lesson10` file folder.

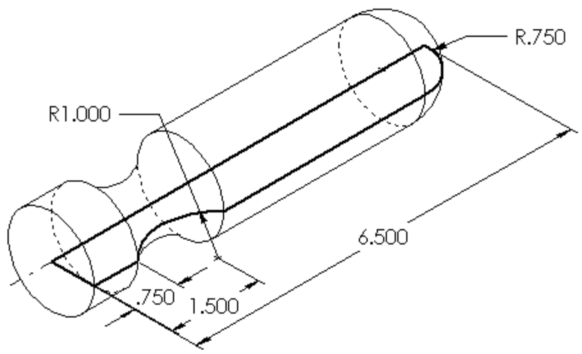
Exercises and Projects — Creating a Screwdriver

Create the `screwdriver`.

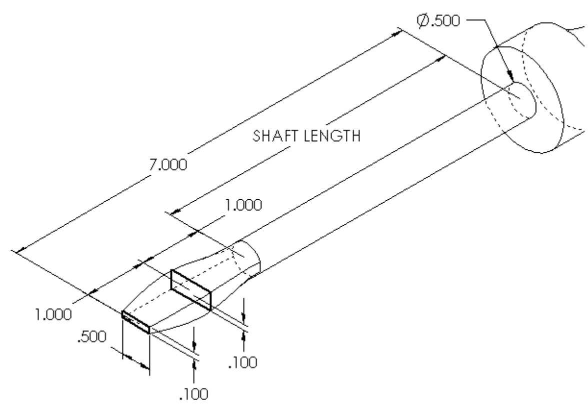
- ❑ Use **inches** for the units.



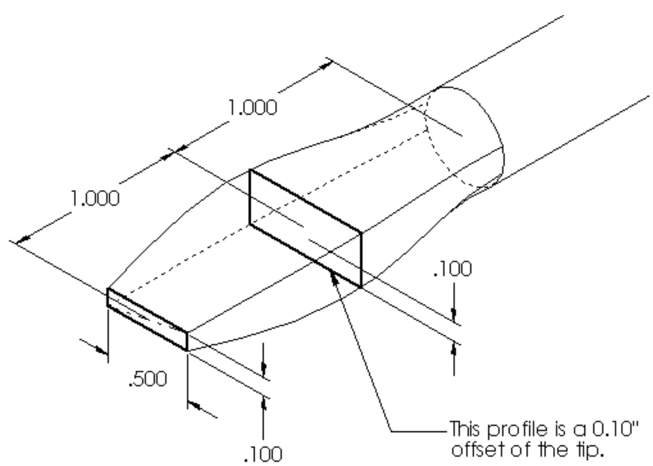
- ❑ Create the handle as the first feature. Use a revolved feature.



- ❑ Create the shaft as the second feature. Use an extruded feature.
- ❑ The overall length of the blade (shaft and tip together) is **7 inches**. The tip is **2 inches** long. Compute the length of the shaft.



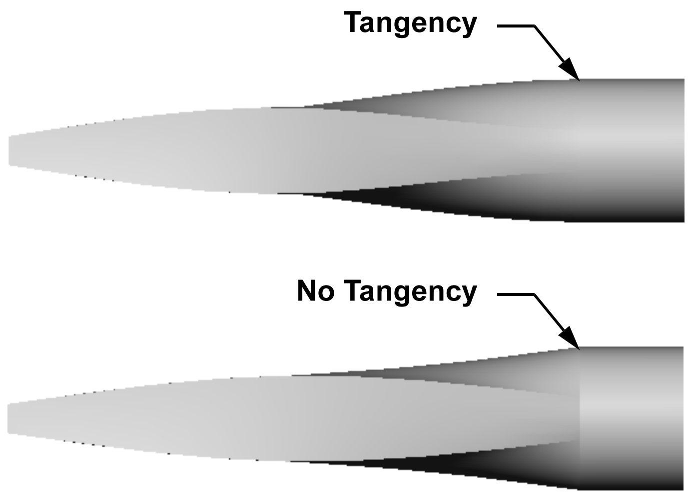
- ❑ Create the tip as the third feature. Use a loft feature.
- ❑ Create the sketch for the end of the tip first. This is a rectangle **0.50”** by **0.10”**.
- ❑ The middle — or second profile — is sketched using a **0.10”** offset (to the outside) of the tip.
- ❑ The third profile is the circular face on the end of the shaft.



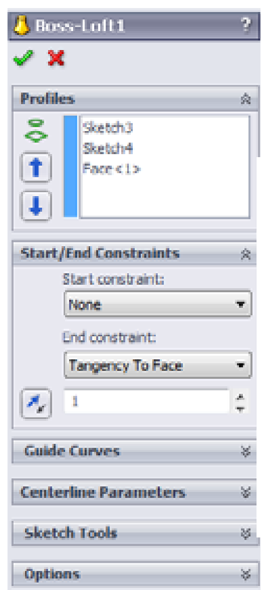
Matching Tangency

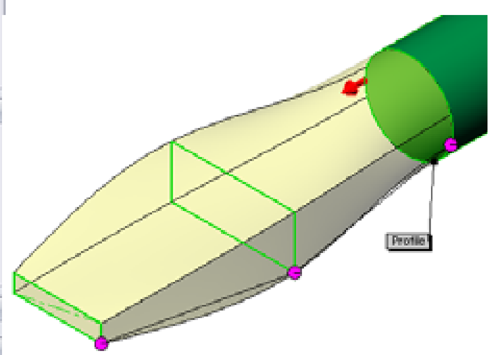
When you want to blend a loft feature into an existing feature such as the shaft, it is desirable to have the face blend smoothly.

Look at the illustrations at the right. In the upper one, the tip was lofted with tangency matching to the shaft. The lower example was not.



In the **Start/End Constraints** box of the PropertyManager, there are some tangency options. **End constraint** applies to the last profile, which in this case, is the face on the end of the shaft.

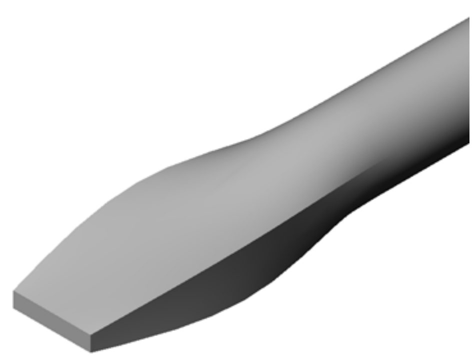




Note: If you picked the face of the shaft as the *first* profile, you would use the **Start constraint** option.

Select **Tangency to Face** for one end and **None** for the other end. The option **Tangency To Face** will make the lofted feature tangent to the sides of the shaft.

The result is shown at the right.



Note: The completed `screwdriver` is found in the `Lesson10` file folder.

More to Explore — Designing a Sports Drink Bottle

Task 1 — Design a Bottle

- ❑ Design a 16 ounce `sportsbottle`. How would you calculate the capacity of the bottle?
- ❑ Create a `cap` for the `sportsbottle`.
- ❑ Create a `sportsbottle` assembly.

sportsbottle assembly

Question

How many liters are contained in the `sportsbottle`?

Conversion

- ❑ 1 fluid ounce = 29.57ml

Answer:

- ❑ Volume = 16 fluid ounces * (29.57ml/fluid ounce) = 473.12ml
- ❑ Volume = 0.473 liters

There are numerous answers to this question. Students should be directed to develop their own solutions. Creativity, ingenuity, and imagination should be encouraged.

An example of the `sportsbottle` assembly is found in the `Lesson10` file folder.

Task 2 — Calculate Costs

A designer for your company receives the following cost information:

- ❑ Sports Drink = $0.32 per gallon based on 10,000 gallons
- ❑ 16 ounce sport bottle = $0.11 each based on 50,000 units

Question

How much does it cost to produce a filled 16 oz. sportsbottle to the nearest cent?

Answer:

- ❑ 1 gallon = 128 ounces
- ❑ Sports Drink Cost = 16 ounce * ($0.32/128 ounces) = $0.04
- ❑ Container Cost (sports bottle) = $0.11
- ❑ Total Cost = Sports Drink Cost + Container Cost
- ❑ Total Cost = $0.04 + $0.11 = $0.15

Lesson 10 Quiz — Answer Key

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What are two methods for creating an offset plane?

<u>**Answer:**</u>

- Use the command **Insert, Reference Geometry, Plane**
- Hold down the **Ctrl** key and drag a copy of an existing plane.

2 Describe the steps required to create a Loft feature.

<u>**Answer:**</u>

- Create the planes required for the profile sketches.
- Sketch a profile on the first plane.
- Sketch the remaining profiles on the corresponding planes.
- Click **Loft** on the Features toolbar.
- Select the profiles.
- Review the connecting curve.
- Click OK.

3 What is the minimum number of profiles for a Loft feature?

<u>**Answer:**</u> The minimum number of profiles for a Loft feature is two.

4 Describe the steps to *copy* a sketch onto a different plane.

<u>**Answer:**</u>

- Select the sketch in the FeatureManager design tree or the graphics area.
- Click **Copy** on the Standard toolbar. (Or use **Ctrl+C**.)
- Select the new plane in the FeatureManager design tree or the graphics area.
- Click **Paste** on the Standard toolbar. (Or use **Ctrl+V**.)

5 What is the command to view all reference planes?

<u>**Answer:**</u> **View, Planes**

6 You have an offset plane. How do you change its **Offset** distance?

<u>**Answer:**</u> There are two acceptable answers:

- Right-click the plane, and select **Edit Feature** from the shortcut menu. Set the **Distance** to a new value. Click **OK**.
- Double-click the plane to display its dimension. Double-click the dimension and enter a new value in the **Modify** box. Click **Rebuild**.

7 True or False. The location where you select each profile determines how the Loft feature is created.

<u>**Answer:**</u> True.

8 What is the command used to *move* a sketch onto a different plane?

<u>**Answer:**</u> **Edit Sketch Plane**

Lesson 10 Quiz **REPRODUCIBLE**

Name: ______________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What are two methods for creating an offset plane?

____

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2 Describe the steps required to create a Loft feature.

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3 What is the minimum number of profiles for a Loft feature?

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4 Describe the steps to *copy* a sketch onto a different plane.

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5 What is the command to view all reference planes?

____

6 You have an offset plane. How do you change its **Offset** distance?

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7 True or False. The location where you select each profile determines how the Loft feature is created.

____

8 What is the command used to *move* a sketch onto a different plane?

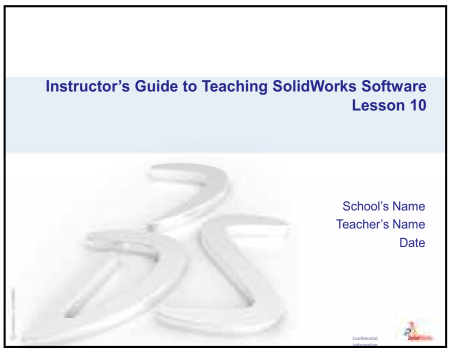
____

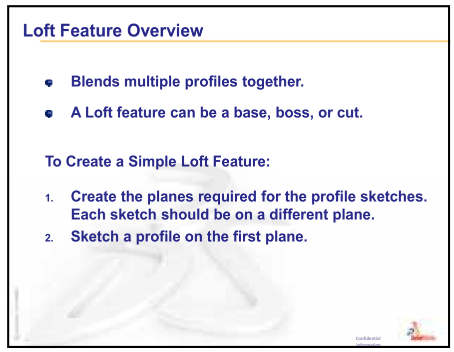
Lesson Summary

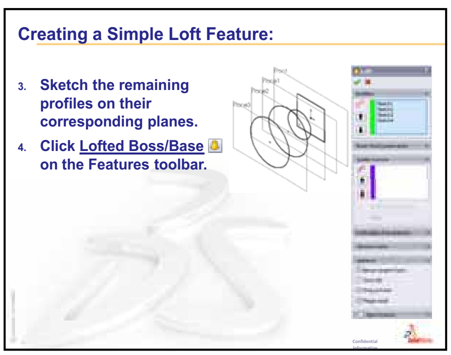
- ❑ A Loft blends multiple profiles together.
- ❑ A Loft feature can be a base, boss, or cut.
- ❑ Neatness counts!
  - Select the profiles in order.
  - Click corresponding points on each profile.
  - The vertex closest to the selection point is used.

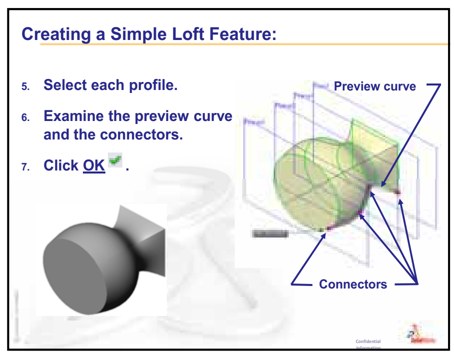
Thumbnail Images of PowerPoint Slides

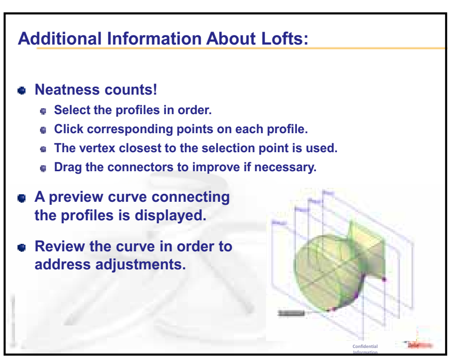
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson.

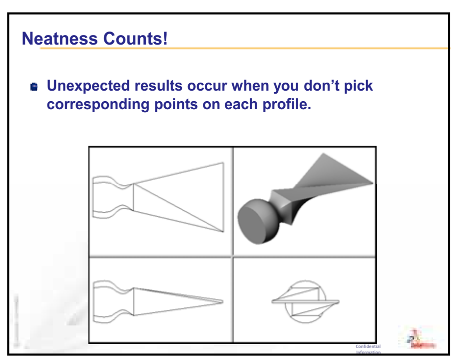






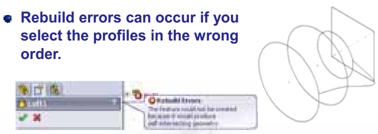






Neatness Counts!

- Rebuild errors can occur if you select the profiles in the wrong order.

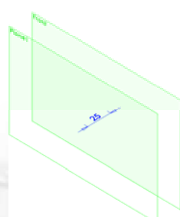


To Create an Offset Plane:

1. Hold down Ctrl and drag the Front plane in the direction you want the offset to go.

   NOTE: Ctrl-drag is a common Windows technique for copying objects.
2. The Plane PropertyManager appears.
3. Enter 25mm for Distance.
4. Click OK.

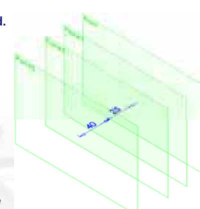
Creating an Offset Plane – Results



Setting up the Planes

Additional offset planes are required.

- *Plane2* is offset 25mm from *Plane1*.
- *Plane3* is offset 40mm from *Plane2*.
- Verify the positions of the planes.
  - Click View, Planes.
  - Double-click the planes to see their offset dimensions.

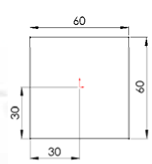


Sketch the Profiles

- The Loft feature is created with 4 profiles.
- Each profile is on a separate plane.

To Create the First Profile:

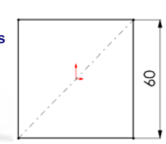
1. Open a sketch on the Front plane.
2. Sketch a square.
3. Exit the sketch.



Best Practice

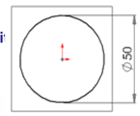
There is a better way to sketch a centered square:

1. Sketch a Center Rectangle starting at the origin. This keeps the rectangle centered.
2. Add an Equal relation to one horizontal and one vertical line. This makes the rectangle a square.
3. Dimension one side of the square.



Sketch the Remaining Profiles:

1. Open a sketch on *Plane1*.
2. Sketch a circle and dimension i
3. Exit the sketch.
4. Open sketch on *Plane2*.
5. Sketch a circle whose circumference is coincident with the corners of the square.
6. Exit the sketch.



To Copy a Sketch:

1. Select *Sketch3* in the FeatureManager design tree or graphics area.
2. Click Edit, Copy or click Copy on the Standard toolbar.
3. Select *Plane3* in the FeatureManager design tree or graphics area.
4. Click Edit, Paste or click Paste on the Standard toolba

   A new sketch, *Sketch4*, is created on *Plane3*.

More About Copying Sketches

- External relations are deleted.
- For example, when you copied *Sketch3*, the geometric relations locating the center and defining the circumference were deleted.
- Therefore, *Sketch4* is underdefined.
- To fully define *Sketch4*, add a Coradial relation between the copied circle and the original.
- If you sketch a profile on the wrong plane, move it to the correct plane using Edit Sketch Plane. Do not copy it.

To Move a Sketch to a Different Plane:

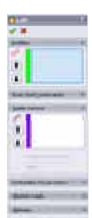
1. Right-click the sketch in the FeatureManager design tree.
2. Select Edit Sketch Plane from the shortcut menu.

3. Select a different plane.
4. Click OK.

Loft Feature

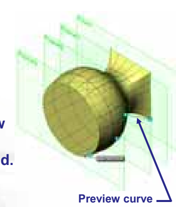
- The Loft feature blends the 4 profiles to create the handle of the *chisel.*

1. Click Lofted Boss / Base on the Features toolbar.



Creating the Loft Feature:

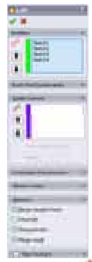
2. Select each profile.
   Click on each sketch in the same relative location – the right side.
3. Examine the preview curve.
   The preview curve shows how the profiles will be connected when the loft feature is created.



Creating the Loft Feature:

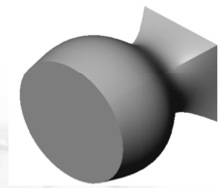
4. The sketches are listed in the Profiles box.

   The Up/Down arrows are used to rearrange the order of the profiles.



Creating the Loft Feature:

5. Click OK .

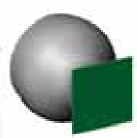


A Second Loft Feature Creates the Bit of the *Chisel*:

- The second Loft feature is composed of two profiles: Sketch5 and Sketch6.

To Create Sketch5:

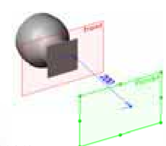
1. Select the square face.
2. Open a sketch.
3. Click Convert Entities .
4. Exit the sketch.



To Create *Sketch6*:

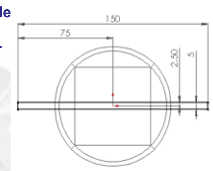
1. Offset *Plane4 behind* the *Front* plane.

   Hold down Ctrl and drag the *Front* plane in the direction you want the offset to go.
2. The Plane PropertyManager appears.
3. Enter 200mm for Distance.
4. Click OK .



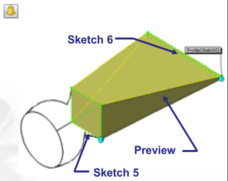
To Create *Sketch6*:

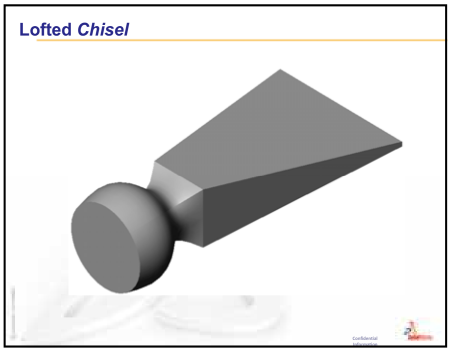
5. Open a sketch on *Plane4*.
6. Sketch a narrow rectangle
7. Dimension the rectangle.
8. Exit the sketch.

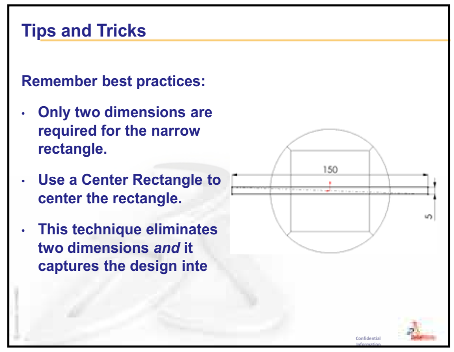


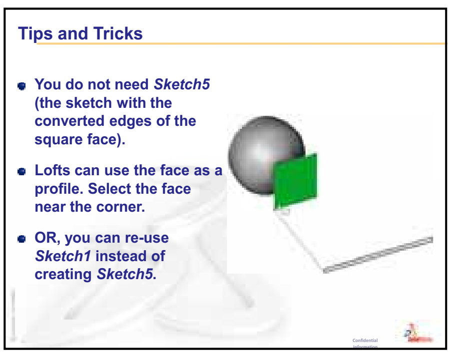
To Create the Second Loft Feature:

1. Click Lofted Boss / Base on the Features toolbar.
2. Select *Sketch5* in the lower right corner of the square.
3. Select *Sketch6* in the lower right corner of the rectangle.
4. Examine the preview curve.
5. Click OK.




Lofted *Chisel*


Tips and Tricks
Remember best practices:
Only two dimensions are required for the narrow rectangle.
Use a Center Rectangle to center the rectangle.
This technique eliminates two dimensions *and* it captures the design inte
150
5

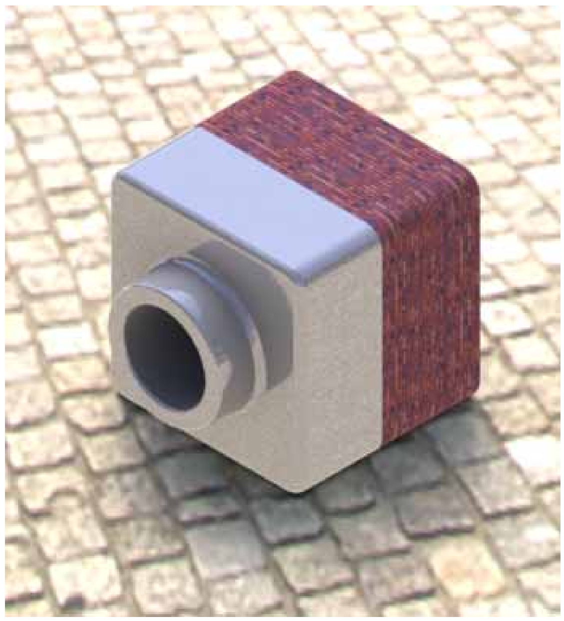

Tips and Tricks
You do not need *Sketch5* (the sketch with the converted edges of the square face).
Lofts can use the face as a profile. Select the face near the corner.
OR, you can re-use *Sketch1* instead of creating *Sketch5.*

11

Lesson 11: Visualization

Goals of This Lesson

- Create an image with the PhotoView 360 application.
- Create an animation using SolidWorks MotionManager.



Before Beginning This Lesson

- This lesson requires copies of `Tutor1`, `Tutor2` and the `Tutor` assembly that are found in the `Lessons\Lesson11` folder in the `SolidWorks Teacher Tools` folder. `Tutor1`, `Tutor2` and the `Tutor` assembly were built earlier in the course.
- This lesson also requires the `Claw-Mechanism` that was built in Lesson 4: Assembly Basics. A copy of this assembly is located in the `Lessons\Lesson11\Claw` folder in the `SolidWorks Teacher Tools` folder.
- Verify that PhotoView 360 is set up and running on your classroom/lab computers.

Resources for This Lesson

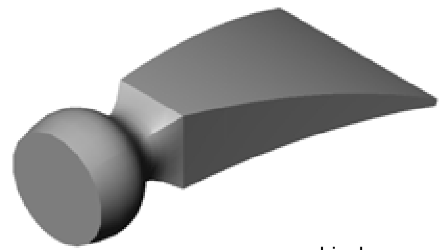
This lesson plan corresponds to *Working with Models: Animation* in the SolidWorks Tutorials.

Combine photorealistic images and animations to create professional presentations.

Review of Lesson 10: Loft Features

Questions for Discussion

1 Describe the *general* steps required to create a Loft feature such as was used in the `chisel`.



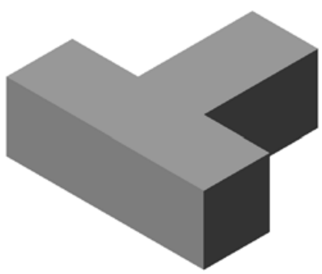
chisel

Answer: To create a Loft feature:

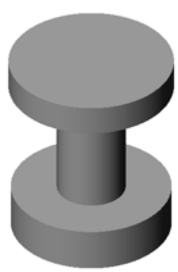
- Create the planes required for the profile sketches.
- Create the profile sketches, each on the appropriate plane.
- Click **Loft**  on the Features toolbar.
- Select the profiles exercising care to select them in the correct order and selecting them in corresponding locations to prevent twisting.
- Review the connecting curve.
- Click **OK**.

2 Each of the following parts were created with *one* feature.

- Name the Base feature for each part.
- Describe the 2D geometry used to create the Base feature of each part.
- Name the sketch plane or planes required to create the Base feature.

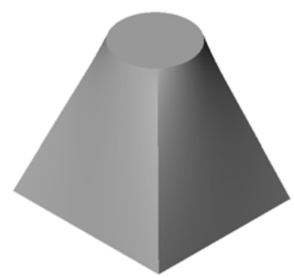


Part 1



Part 2

Part 3



Part 4

Answer:

- Part 1: Extruded boss feature is created with an T-shaped profile sketched on the `Top` plane.
- Part 2: Revolved boss feature is created with C-shaped profile and a centerline sketched on the `Front` plane. The angle of rotation is 360°. **Note:** The C-shaped profile could also be sketched on the `Right` plane.
- Part 3: Swept boss feature is created with a circular cross section sketched on a plane that is perpendicular to the end of the path. The path is a series of tangent lines and arcs. A number of different combinations of planes could have been used. For example, the path could be sketched on the `Top` plane and the sweep section on the `Front` plane. There must be a slight gap between the loops of the paper clip because a sweep feature must not self-intersect.
- Part 4: Lofted boss feature is created with a square profile on the `Top` plane and a circular sketch created on a plane that is offset from the `Top` plane.

Outline of Lesson 11

- ❑ In Class Discussion — Using PhotoView 360 and MotionManager
- ❑ Active Learning Exercises — Using PhotoView 360
  - Applying an Appearance
  - Setting the Background Scene
  - Rendering and Saving the Image
- ❑ Active Learning Exercise — Creating an Animation
- ❑ Exercises and Projects — Creating an Exploded View of an Assembly
  - Using PhotoView 360 and MotionManager Together
  - Creating an Exploded View of an Assembly
- ❑ Exercises and Projects — Creating and Modifying Renderings
  - Creating a Rendering of a Part
  - Modifying a Rendering of a Part
  - Creating a Rendering of an Assembly
  - Rendering Additional Parts
- ❑ Exercises and Projects — Creating an Animation
- ❑ Exercises and Projects — Creating an Animation of the Claw-Mechanism
- ❑ More to Explore — Creating an Animation of Your Own Assembly
- ❑ Lesson Summary

Competencies for Lesson 11

Students develop the following competencies in this lesson:

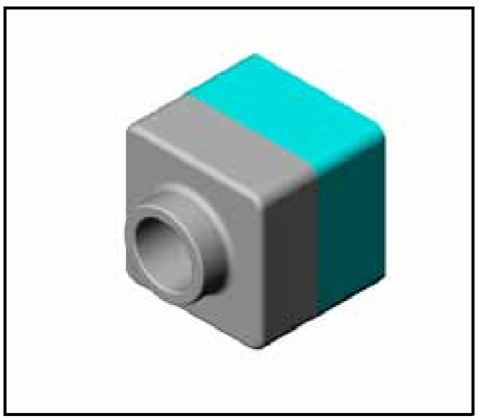
- ❑ **Engineering**: Enhance the appeal of a product with visualization and animation.
- ❑ **Technology**: Work with different file formats to enhance presentation skills.

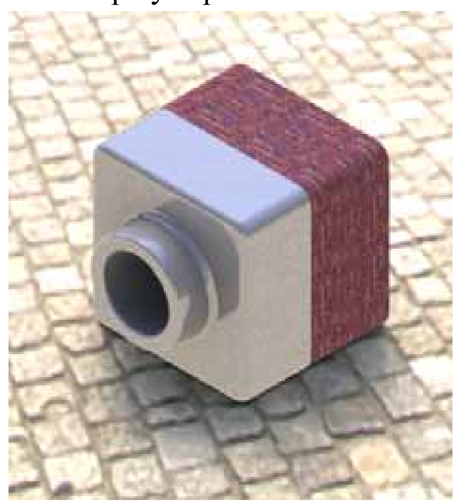
In Class Discussion — Using PhotoView 360 and MotionManager

Ideally, you want to view your designs in as realistic a manner as possible. Being able to view designs realistically reduces prototyping costs and speeds time to market. PhotoView 360 lets you use realistic surface appearances, lighting, and advanced visual effects to display your models. SolidWorks MotionManager lets you capture and replay motion. Together, PhotoView 360 and SolidWorks MotionManager display realistically.

PhotoView 360 uses advanced graphics to create photorealistic images of SolidWorks models. You can select appearances to display the model as the built part would appear — if it existed. For example, if a part is being designed to have a chrome finish, you can display it in chrome. If chrome does not look right, you can change the display to brass.

In addition to advanced appearances, PhotoView 360 also has advanced lighting, reflectance, texture, transparency, and roughness display capabilities.





SolidWorks MotionManager is effective in realistically communicating the basic design intent of a SolidWorks part or assembly. You can animate and capture motion of SolidWorks parts and assemblies that you can play back. This enables you to communicate design intentions — using SolidWorks MotionManager as a feedback tool. Often, an animation is a quicker and more effective communication tool than static drawings.

You can animate standard behaviors such as explode and collapse or other behaviors such as rotate.

SolidWorks MotionManager generates Windows-based animations (`*.avi` files). The `*.avi` file uses a Windows-based Media Player to playback the animation. You can use these animation files for product illustrations, design reviews, and so forth.

Active Learning Exercises — Using PhotoView 360

Watch the tutorial videos at http://www.solidworksgallery.com/index.php?p=tutorials_general..

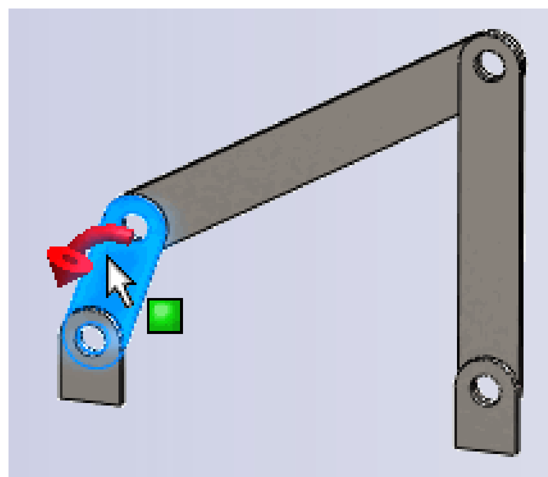
The videos show PhotoView 360 in a stand-alone window. You can access the commands for PhotoView 360 on the Render Tools tab of the CommandManager or the Render Tools toolbar in the SolidWorks window.

Create a PhotoView 360 rendering of `Tutor1` which you built in a previous lesson. Do the following:

- ❑ Apply the **Chromium plate** appearance from the **Metals\Chrome** class.
- ❑ Apply the **Factory** scene from the **Scenes\Basic Scenes** folder.
- ❑ Render and save the `Tutor Rendering.bmp` image.

Active Learning Exercises — Creating an Animation

Create an animation of the 4-bar linkage. Follow the instructions in *Working with Models: Animation* in the SolidWorks Tutorials.



Lesson 11 — 5 Minute Assessment — Answer Key

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What is PhotoView 360?

Answer: PhotoView 360 is a software application that creates realistic images from SolidWorks models.

2 List the rendering effects that are used in PhotoView 360?

Answer: Appearances, Backgrounds, Lights and Shadows.

3 The PhotoView 360_________ ___________ enables you to specify and preview appearances.

Answer: Appearance Editor

4 Where do you set the scene background?

Answer: Scene Editor - Background

5 What is SolidWorks MotionManager?

Answer: SolidWorks MotionManager is a software application that animates and captures motion of SolidWorks part and assemblies.

6 List the three types of animations that can be created using the AnimationWizard.

Answer: Rotate Model, Explode View, Collapse View.

Lesson 11 — 5 Minute Assessment **REPRODUCIBLE**

Name: ________________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What is PhotoView 360?

____

2 List the rendering effects that are used in PhotoView 360?

____

3 The PhotoView 360_________ ___________ enables you to specify and preview appearances.

4 Where do you set the scene background?

____

5 What is SolidWorks MotionManager?

____

____

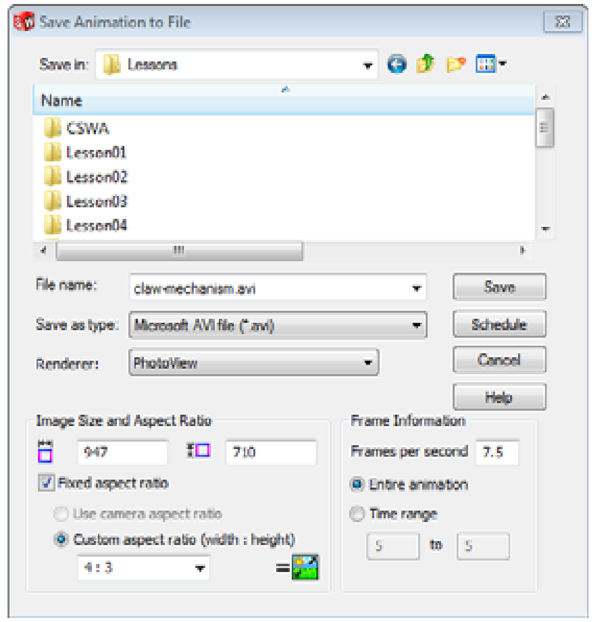
6 List the three types of animations that can be created using the AnimationWizard.

____

Exercises and Projects — Creating an Exploded View of an Assembly

Using PhotoView 360 and MotionManager Together

When you record an animation, the default rendering engine that is used is the SolidWorks shaded image software. This means the shaded images that make up the animation will look just like the shaded images you see in SolidWorks.



Earlier in this lesson you learned how to make photo-realistic images using the PhotoView 360 application. You can record animations that are rendered using the PhotoView 360 software. Since PhotoView 360 rendering is much slower than SolidWorks shading, recording an animation this way takes much more time.

To use the PhotoView 360 rendering software select **PhotoView** from the **Renderer:** list on the **Save Animation to File** dialog box.

Note: The file types `*.bmp` and `*.avi` increase in file size as more appearances and advanced rendering effects are applied. The larger the image size the more time is required to create the image and animation files.

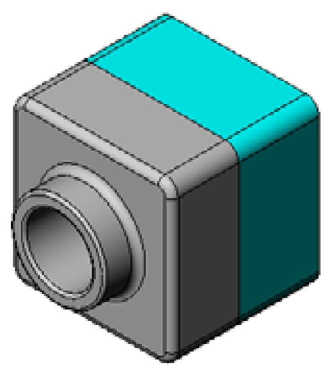
Creating an Exploded View of an Assembly

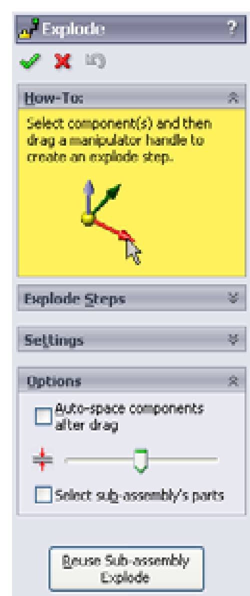
The `Claw-Mechanism` which you used earlier already had an exploded view. To add an exploded view to an assembly, the `Tutor` assembly for example, follow this procedure:

1 Click **Open** on the Standard toolbar, and open the assembly, `Tutor`, which you built earlier.

2 Click **Insert, Exploded View...** or click **Exploded View** on the Assembly toolbar.

The **Explode** PropertyManager appears.

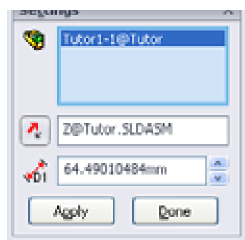


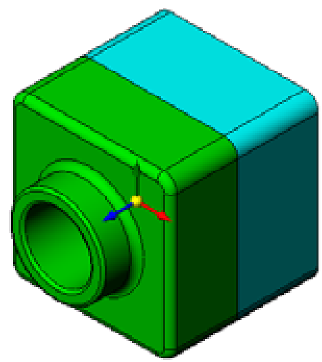


3 The **Explode Steps** section of the dialog displays the explode steps in sequence, and is used to edit, navigate through, or delete explode steps. Each movement of a component in a single direction is considered a step.

The **Settings** section of the dialog controls the details of each explode step, including what component(s), what direction, and how far to move each component. The simplest way is just to drag the component(s).

4 First select a component to begin a new explode step. Select `Tutor1`; a reference triad appears on the model. Next choose the other explode criteria:





- **Direction to explode along**

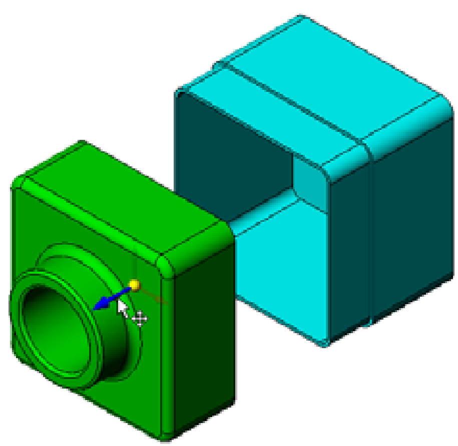
  The default is **Along Z** (`z@tutor.sldasm`), the blue triad pointer. A different direction can be specified by selecting a different arrow of the triad or a model edge.

- **Distance**

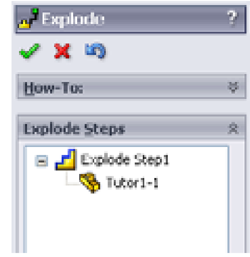
  The distance the component is exploded can be done by eye in the graphics area, or more exactly by manipulating the value in the dialog.

5 Click on the blue triad arrow, and drag the part to the left. It is constrained to this axis (**Along Z**).

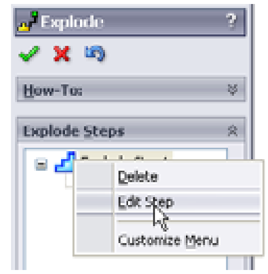
Drag the part to the left by clicking and holding the left mouse button.



6 When the part is released (release the left mouse button), the explode step is created. The part or parts are displayed under the step in the tree.



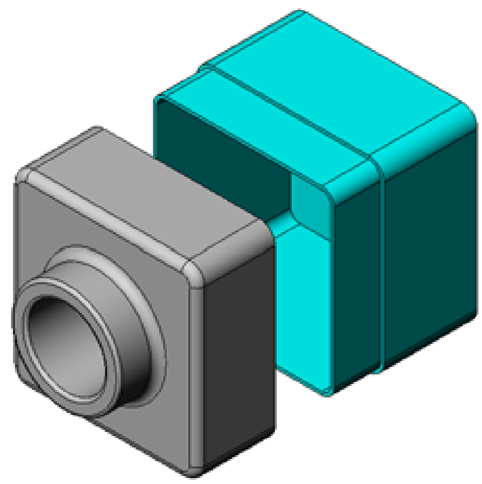
7 The explode distance can be changed by editing the step. Right-click on `Explode Step1`, and select **Edit Step**. Change the distance to **70mm**, and click **Apply**.

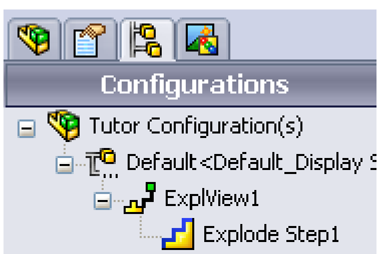


8 Since there is only one component to explode, this completes making the exploded view.

9 Click **OK** to close the **Explode** PropertyManager.

Note: Exploded views are related to and stored in configurations. You can have only one exploded view per configuration.





10 To collapse an exploded view, right-click the assembly icon at the top of the FeatureManager design tree, and select **Collapse** from the shortcut menu.

11 To explode an existing exploded view, right-click the assembly icon in the FeatureManager design tree, and select **Explode** from the shortcut menu.

Exercises and Projects — Creating and Modifying Renderings

Task 1 — Creating a Rendering of a Part

Create a PhotoView 360 rendering of `Tutor2`. Use the following settings:

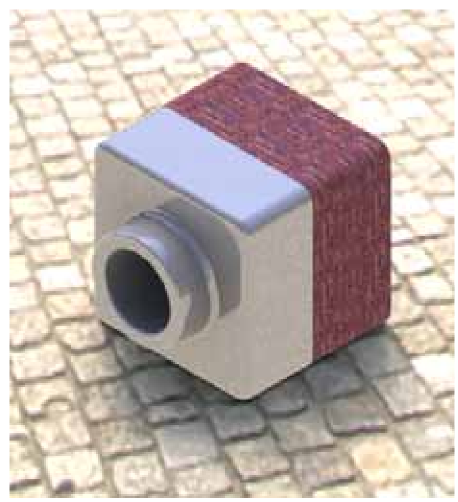
- ❑ Use **old english brick2** appearance from the **stone\brick** class. Adjust the scale to your liking.
- ❑ Set the background to **Plain White** from **Basic Scenes**.
- ❑ Render and save the image.

Task 2 — Modifying a Rendering of a Part

Modify the PhotoView 360 rendering of `Tutor1` that you created in the preceding Active Learning Exercise. Use the following settings:

- ❑ Change the appearance to **wet concrete2d** from the **Stone\Paving** class.
- ❑ Set the background to **Plain White** from **Basic Scenes**.
- ❑ Render and save the image.

Task 3 — Creating a Rendering of an Assembly



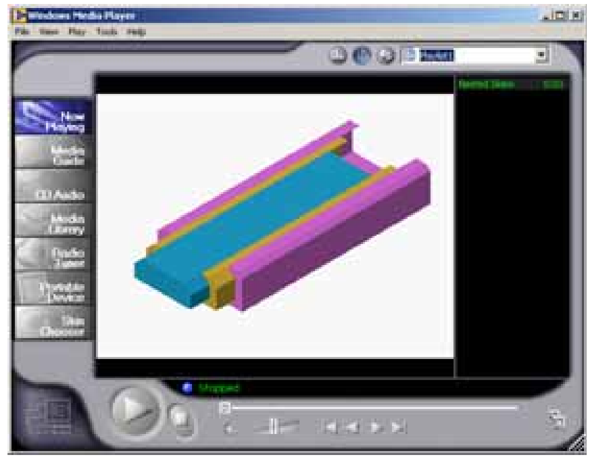
Create a PhotoView 360 rendering of the `Tutor` assembly. Use the following settings:

- ❑ Set the scene to **Courtyard Background** from **Presentation Scenes**.
- ❑ Render and save the image.

Task 4 — Rendering Additional Parts

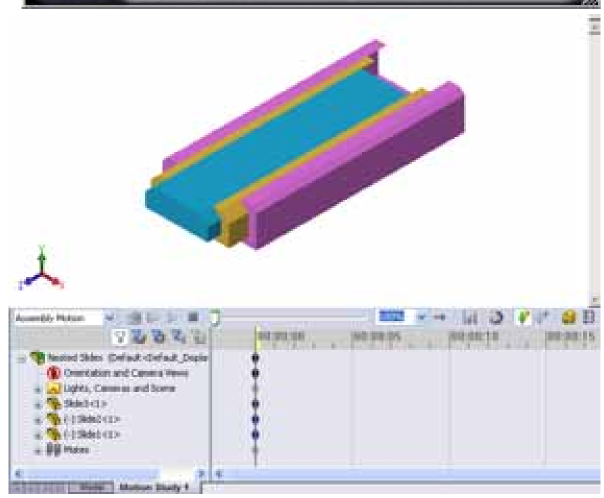
Create PhotoView 360 renderings of any of the parts and assemblies you built during class. For example, you might render the candlestick or the sports bottle you made created earlier. Experiment with different appearances and scenes. You can try to create as realistic an image as possible, or you can create some unusual visual effects. Use your imagination. Be creative. Have fun.

Exercises and Projects — Creating an Animation



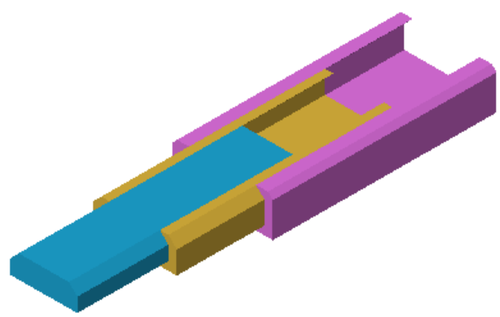
Create an animation that shows how the slides move relative to each other. In other words, create an animation where at least one of the slides moves. You cannot accomplish this task with the Animation Wizard.

1 Open the `Nested Slides` assembly. It is located in the `Lesson11` folder.



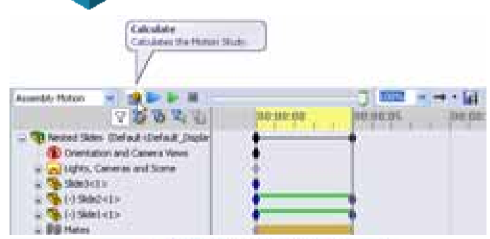
2 Select the `Motion Study1` tab at the bottom of the graphics area to access the MotionManager controls.

3 The parts are in their initial position. Move the time bar to 00:00:05.

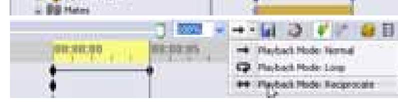


4 Select `Slide1`, the innermost slide. Drag `Slide1` so that it is almost completely out of `Slide2`.

5 Next drag `Slide2` about halfway out of `Slide3`. The MotionManager shows with green bars that the two slides are set to move in this time frame.



6 Click **Calculate** on the MotionManager Toolbar to process and preview the animation. Once calculated, use the **Play** and **Stop** controls.



7 If desired, you can cycle the animation by using the **Reciprocate** command.

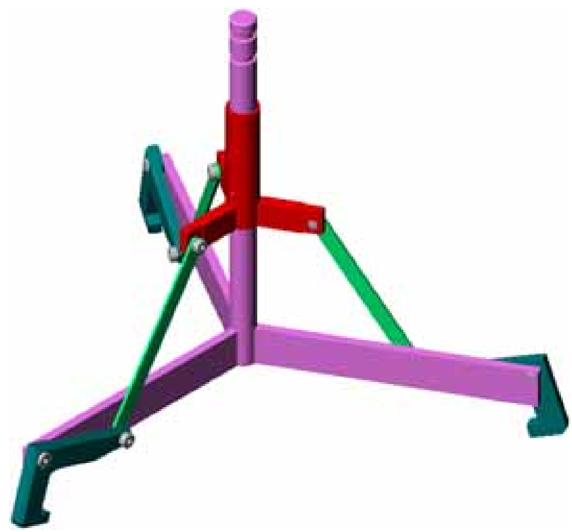
Or, to create an animation of the complete cycle, move the time bar forward (to 00:00:10), then return the components to their original positions.

8 Save the animation to an .avi file.

Exercises and Projects — Creating an Animation of the Claw-Mechanism

Create an animation of the `Claw-Mechanism`. Some suggestions include exploding and collapsing, and moving the `Collar` up and down to show assembly motion.

A completed copy of the `Claw-Mechanism` is located in the `Lesson11` folder. This version is slightly different than the one you built in Lesson 4. This one does not have a component pattern. Each component was assembled individually. This is so the assembly will explode better.



More to Explore — Creating an Animation of Your Own Assembly

Earlier you created an animation from an existing assembly. Now create an animation of the `Tutor` assembly that you built earlier, using the Animation Wizard . The animation should include the following:

- ❑ Explode the assembly for a duration of 3 seconds.
- ❑ Rotate the assembly around the Y axis for a duration of 8 seconds.
- ❑ Collapse the assembly for a duration of 3 seconds.
- ❑ Record the animation. **Optional:** Record the animation using the PhotoView 360 renderer.

Lesson 11 Quiz — Answer Key

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What is PhotoView 360?

Answer: PhotoView 360 is a software application that creates realistic images from SolidWorks models.

2 What is SolidWorks MotionManager?

Answer: SolidWorks MotionManager is a software application that animates and captures motion of SolidWorks part and assemblies.

3 List the two rendering effects that were used when rendering the `Tutor` assembly.

Answer: Appearances and Background.

4 ___________ ____________ is the basis for all images in PhotoView 360.

Answer: Shaded Rendering.

5 Where do you modify the scene background?

Answer: Scene Editor - Background.

6 True or False. You cannot modify the color of the **old english brick2** appearance.

Answer: True.

7 Image Background is the portion of the graphics area not covered by the __________.

Answer: Model.

8 True or False. PhotoView 360 output renders to graphics window or renders to a file.

Answer: True.

9 Identify the Renderer option that must be used to add PhotoView 360 appearances and scenes to an animation.

Answer: PhotoView buffer.

10 SolidWorks MotionManager produces what type of file?

Answer: `*.avi`.

11 List the three types of animations that can be created using the AnimationWizard.

Answer: Rotate Model, Explode View, Collapse View.

12 For a given animation, list three factors that affect the file size when the animation is recorded.

Answer: Possible answers include number of frames per second, type of renderer used, amount of video compression, number of key frames, and screen size. If the rendering is done with the PhotoView buffer, the appearance, scene, and lighting effects such as shadows all affect file size.

Lesson 11 Quiz REPRODUCIBLE

Name: ________________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What is PhotoView 360?

____

2 What is SolidWorks MotionManager?

____

3 List the two rendering effects that were used when rendering the `Tutor` assembly.

____

4 ___________ ____________ is the basis for all images in PhotoView 360.

____

5 Where do you modify the scene background?

____

6 True or False. You cannot modify the color of the **old english brick2** appearance.

____

7 Image Background is the portion of the graphics area not covered by the __________.

____

8 True or False. PhotoView 360 output renders to the graphics window or renders to a file.

____

9 Identify the Renderer option that must be used to add PhotoView 360 appearances and scenes to an animation.

____

10 SolidWorks MotionManager produces what type of file?

____

11 List the three types of animations that can be created using the AnimationWizard.

____

12 For a given animation, list three factors that affect the file size when the animation is recorded. ____

____

____

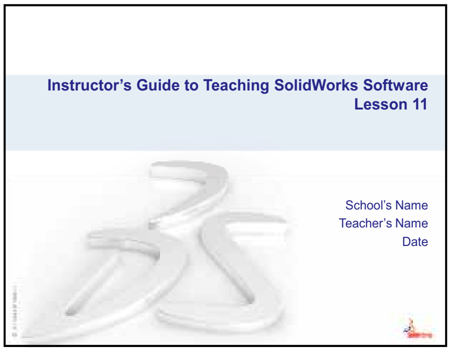
____

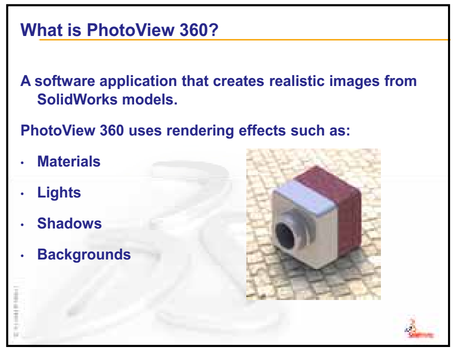
Lesson Summary

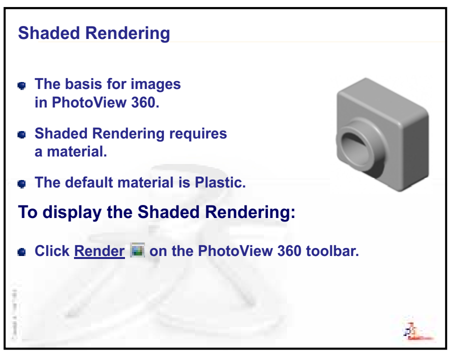
- PhotoView 360 and SolidWorks MotionManager create realistic representations of models.
- PhotoView 360 uses realistic textures, appearances, lighting, and other effects to produce true to life models.
- SolidWorks MotionManager animates and captures motion of SolidWorks parts and assemblies.
- SolidWorks MotionManager generates Windows-based animations (*.avi files). The *.avi file uses a Windows-based Media Player.

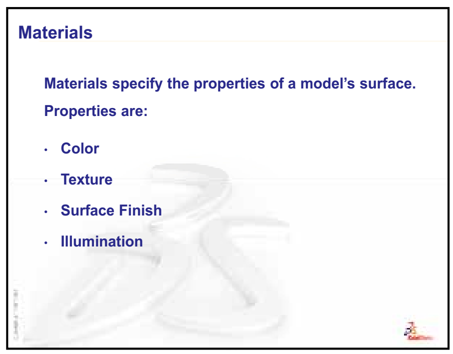
Thumbnail Images of PowerPoint Slides

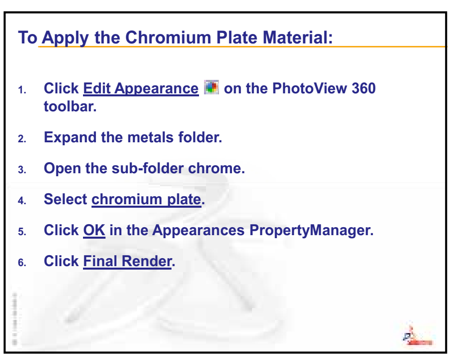
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson











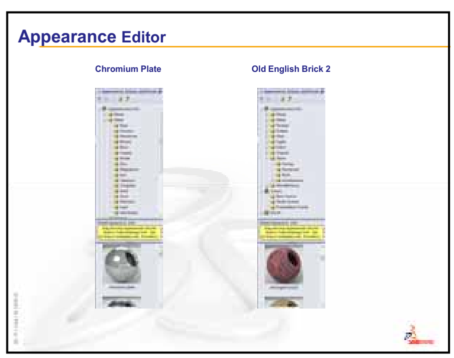


Image Background

The portion of the graphics area not covered by the model.

- Background styles vary in complexity and rendering speed.
- Background styles controlled by Scene Editor.
- Incorporate advanced rendering effects into a PhotoView 360 Scene.
  - Shadows
  - Reflections

To Change the Background Style:

1. Click Edit Scene on the PhotoView 360 toolbar.
2. Expand the *Presentation Scenes* folder.
3. Select Courtyard Background.
4. Click Apply.

To Save the Image File

1. Click Final Render on the PhotoView 360 toolbar.
2. Click Save Image.
3. Enter a file name.
4. Specify a file type.

SolidWorks MotionManager Application

What is SolidWorks MotionManager?

- SolidWorks MotionManager animates and captures motion of SolidWorks parts and assemblies.
- SolidWorks MotionManager generates Windows-based animations (*.avi files). The *.avi file uses a Windows-based Media Player.
- SolidWorks MotionManager can be combined with PhotoView 360.

Renderer Options

The Renderer affects the quality of the saved image. There are two options:

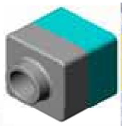
- SolidWorks screen
- PhotoView 360 buffer

Factors Affecting File Size

- Number of frames per second
- Renderer used
  - PhotoView 360 buffer creates a larger file than SolidWorks screen
- If using PhotoView 360 buffer:
  - Materials
  - Background
  - Shadows
  - Multiple-light sources
- Video compression
- Key frames

To Create an Exploded View:

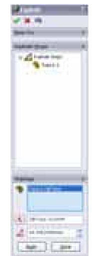
1. Click Open on the Standard toolbar, and open the assembly, *Tutor*.
2. Click Exploded View on the Assembly toolbar. The Explode PropertyManager appears.



Creating an Exploded View:

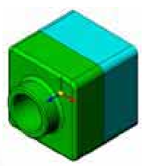
3. Click on the component to explode to begin a new explode step. Drag the component to the explode location. The dialog box contains selection lists for:

- Component(s) to explode
- Direction to explode along
- Distance



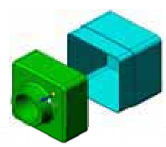
Creating an Exploded View:

4. Click the component to explode, in this case *Tutor1*. The component name appears in the dialog. Select the desired explode direction from the model triad. This selection is indicated in the Direction area of the dialog (Along Z, Z@Tutor.SLDASM by default).



Creating an Exploded View:

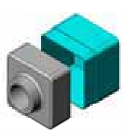
5. Drag the component to the desired distance. Release the mouse button to create the Explode step.
6. Edit the step (right-click on the new Explode step, and select Edit Step) to adjust the Distance to exactly 70mm and click Apply in the dialog.
7. Since there is only one component to explode, this completes making the exploded view. Click OK to close the Explode PropertyManager.



Creating an Exploded View:

8. Results.

Note: Exploded views are related to and stored in configurations. You can only have one exploded view per configuration.



Collapsing an Exploded View:

- Right-click the assembly icon in the FeatureManager design tree, and select Collapse from the shortcut menu.

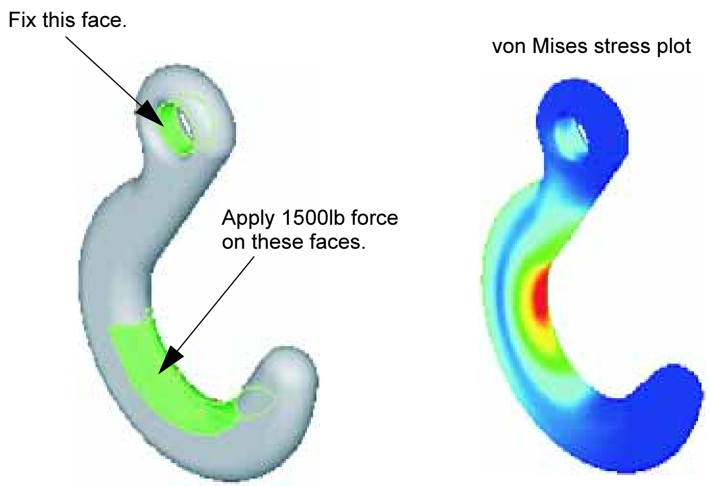
To Explode an Existing Exploded View:

- Right-click the assembly icon in the FeatureManager design tree, and select Explode from the shortcut menu.

Lesson 12: SolidWorks SimulationXpress

Goals of This Lesson

- ❑ Understand basic concepts of stress analysis.
- ❑ Calculate the stress and displacement in the following part subjected to a load.



Before Beginning This Lesson

- ❑ If SolidWorks Simulation is active, you must clear it from the Add-Ins list of compatible software products to access SolidWorks SimulationXpress. Click **Tools**, **Add-Ins** and clear the check mark in front of **SolidWorks Simulation**.

Resources for This Lesson

This lesson plan corresponds to *Design Analysis: SolidWorks SimulationXpress* in the SolidWorks Tutorials.

The Simulation Guides, Sustainability guide, Structural Bridge, Race Car, Mountain Board, and Trebuchet Design Projects apply concepts of engineering, math, and science.

Review of Lesson 11: Visualization

Questions for Discussion

1 What is PhotoView 360?

Answer: PhotoView 360 is a software application that creates realistic images from SolidWorks models.

2 What are the rendering effects used by PhotoView 360?

Answer: Appearances, Backgrounds, Lights and Shadows.

3 What is SolidWorks MotionManager?

Answer: SolidWorks MotionManager is a software application that animates and captures motion of SolidWorks parts and assemblies.

4 List the three types of animations that can be created using the Animation Wizard.

Answer: Rotate Model, Explode View, Collapse View.

5 What types of files are generated by SolidWorks MotionManager to playback the animation?

Answer: SolidWorks MotionManager generates Windows-based animations (`*.avi` files).

Outline of Lesson 12

- ❑ In Class Discussion — Stress Analysis
  - Stress on the Legs of a Chair
  - Stress on the Body of a Standing Student
- ❑ Active Learning Exercises — Analyze a Hook and a Control Arm
- ❑ Exercises and Projects — Analyze a CD Storage Box
  - Calculate the Weight of the CD Cases
  - Determine the Displacement in the Storage Box
  - Determine the Displacement in a Modified Storage Box
- ❑ More to Explore — Analysis Examples
  - Analyze the Anchor Plate
  - Analyze the Spider
  - Analyze the Link
  - Analyze the Faucet
- ❑ More to Explore — Other Guides and Projects
  - Introduction to Analysis Guides
  - Trebuchet Design Project
  - Structural Bridge Design Project
  - CO_2 Car Design Project
- ❑ Lesson Summary

Competencies for Lesson 12

Students develop the following competencies in this lesson:

- ❑ **Engineering**: Explore how material properties, forces, and restraints affect part behavior.
- ❑ **Technology**: Knowledge of the finite element process to analyze force and pressure on a part.
- ❑ **Math**: Understand units and apply matrices.
- ❑ **Science**: Investigate density, volume, force, and pressure.

In Class Discussion — Stress Analysis

SolidWorks SimulationXpress offers an easy-to-use first pass stress analysis tool for SolidWorks users. SolidWorks SimulationXpress can help you reduce cost and time-to-market by testing your designs on the computer instead of expensive and time-consuming field tests.

SolidWorks SimulationXpress uses the same design analysis technology that SolidWorks Simulation users to perform stress analysis. The wizard interface of SolidWorks SimulationXpress guides you through a five step process to specify material, restraints, loads, run the analysis, and view the results.

The purpose of this section is to encourage students to think about the applications of stress analysis. Ask the students to identify objects around them and what loads and restraints to specify.

Stress on the Legs of a Chair

Estimate the stress on the legs of a chair.

Stress is force per unit area or force divided by area. The legs support the weight of the student plus the weight of the chair. The chair design and how the student is sitting determine the share of each leg. The average stress is the weight of the student plus the weight of the chair divided by the area of the legs.

Stress on the Body of a Standing Student

Estimate the stress on the feet of a student when they stand up. Is the stress the same at all points? What happens if the student leans forward, backward, or to the side? How about the stress on the knee and ankle joints? Is this information useful in designing artificial joints?

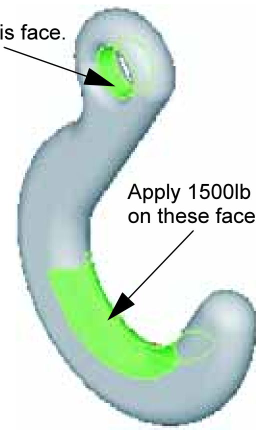
Stress is force per unit area or force divided by area. The force is the weight of the student. The area that supports the weight is the area of the foot in contact with the shoes. The shoes redistribute the load and transmit it to the floor. The reaction force from the floor should be equal to the student's weight.

When standing upright, each foot approximately takes half the weight. When walking, one foot supports the whole weight. The student could feel that the stress (pressure) is higher at some points. When standing upright, the students can move their toes indicating that there is little or no stress on the toes. As the students lean forward, the stress is redistributed with more stress on the toes and less on the heel. The average stress is the weight divided by the area of the feet in contact with the shoes.

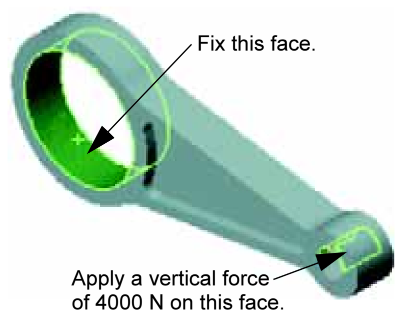
We can estimate the average stresses on the knee and ankle joints if we know the area that carry the weight. Detailed results require performing stress analysis. If we can build the knee or ankle joint assembly in SolidWorks with the proper dimensions and if we know the elastic properties of the various parts, then static analysis can give us the stresses at every point of the joint under different support and load scenarios. The results can help us improve designs for artificial joint replacements.

Active Learning Exercises — Analyze a Hook and a Control Arm

Follow the instructions in *Design Analysis: SolidWorks SimulationXpress: SimulationXpress Basic Functionality* in the SolidWorks Tutorials. In this lesson, you determine the maximum von Mises stress and displacement after subjecting the hook to a load.



Follow the instructions in *Design Analysis: SolidWorks SimulationXpress: Using Analysis to Save Material* in the SolidWorks Tutorials. In this lesson, you use the results from SolidWorks SimulationXpress to reduce the volume of a part.



Lesson 12 — 5 Minute Assessment — Answer Key

Name: ______________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you start SolidWorks SimulationXpress?

Answer: With a part open in SolidWorks, click **Tools**, **SimulationXpress**.

2 What is analysis?

Answer: Analysis is a process to simulate how your design performs in the field.

3 Why is analysis important?

Answer: Analysis can help you design better, safer, and cheaper products. It saves you time and money by reducing traditional, expensive design cycles.

4 What does static analysis calculate?

Answer: Static analysis calculates stresses, strains, displacements, and reaction forces in the part.

5 What is stress?

Answer: Stress is the intensity of force or force divided by area.

6 SolidWorks SimulationXpress reports that the factor of safety is 0.8 at some locations. Is the design safe?

Answer: No. The minimum factor of safety should not be less than 1.0 for a safe design.

Lesson 12 — 5 Minute Assessment REPRODUCIBLE

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 How do you start SolidWorks SimulationXpress?

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2 What is analysis?

____

3 Why is analysis important?

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4 What does static analysis calculate?

____

____

5 What is stress?

____

6 SolidWorks SimulationXpress reports that the factor of safety is 0.8 at some locations. Is the design safe?

____

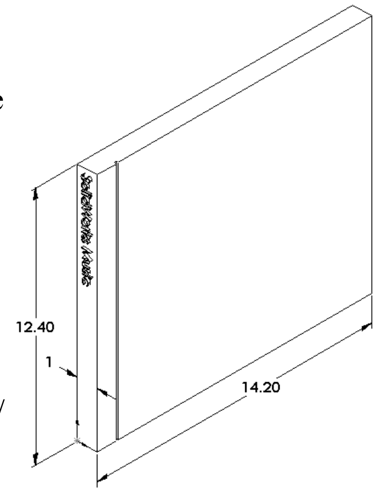
____

Exercises and Projects — Analyze a CD Storage Box

You are part of the design team that created the `storagebox` to hold the CD cases in an earlier lesson. In this lesson, you use SimulationXpress to analyze the `storagebox`. First, you determine the deflection of `storagebox` under the weight of 25 CD cases. Then, you modify the wall thickness of the `storagebox`, perform another analysis, and compare the deflection to the original value.

Task 1 — Calculate the Weight of the CD Cases

You are given the measurements of a single CD case as shown. `Storagebox` holds 25 CD cases. The density of the material used for the CD cases is 1.02 g/cm^3.



What is the weight of 25 CD cases in pounds?

Answer:

- ❑ Volume of 1 CD case = 14.2 cm x 12.4 cm x 1 cm = 176.1 cm^3
- ❑ Weight of 1 CD case = 176.1 cm^3 x 1.02 g/cm^3 x 1 kg/ 1000 g = 0.18 kg
- ❑ Weight of 25 CD cases = 0.18 kg x 25 x 2.2 lbs / kg = 9.9 lbs

The answer is that 25 CD cases weigh approximately 10 lbs.

Task 2 — Determine the Displacement in the Storage Box

Determine the maximum displacement of `storagebox` under the weight of 25 CD cases.

1 Open `storagebox.sldprt` in the `Lesson12` file folder.

2 Click **Tools**, **SimulationXpress** to start SolidWorks SimulationXpress.

Options

Set the units to English (IPS) to enter the force in pounds and see the deflection in inches.

1 In the **SolidWorks SimulationXpress** Task pane, click **Options**.

2 Select **English (IPS)** for **System of Units**.

3 Click **OK**.

4 Click **Next** in the Task pane.

Material

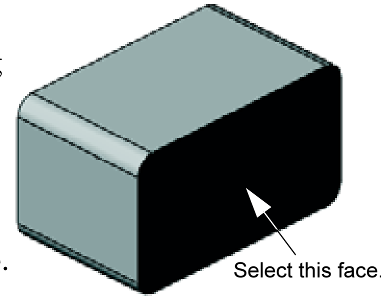
Choose a solid nylon material for `storagebox` from the library of standard materials.

1 Click **Material** in the Task pane, then click **Change material**.

2 In the **Plastics** folder, select **Nylon 101**, click **Apply**, then click **Close**.

3 Click **Next**.

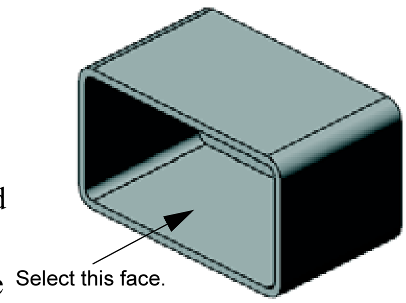
Fixtures/Restraints



Restrain the back face of `storagebox` to simulate hanging the box on a wall. Restrained faces are fixed; they do not move during the analysis. In reality, you would probably hang the box using a couple screws but we will restain the entire back face.

1. Click **Fixtures** in the Task pane, then click **Add a fixture**.
2. Select the back face of `storagebox` to restrain that face, then click **OK** in the PropertyManager.
3. Click **Next** in the Task pane.

Loads



Apply a load inside `storagebox` to simulate the weight of the 25 CD cases.

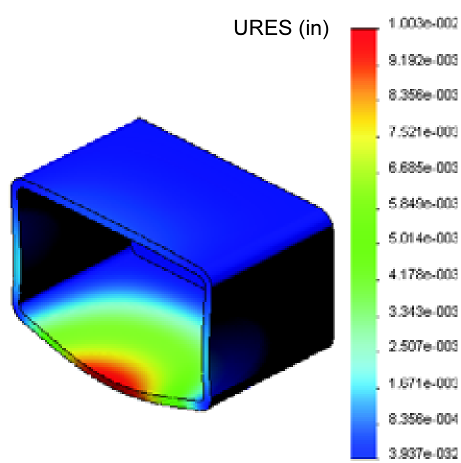
1. Click **Loads** in the Task pane, then click **Add a force**.
2. Select the inside face of `storagebox` to apply the load to that face.
3. Type **10** for the value of the force in pounds. Make sure the direction is set to **Normal**. Click **OK** in the PropertyManager.
4. Click **Next** in the Task pane.

Analyze

Perform the analysis to calculate displacements, strains, and stresses.

1. Click **Run** in the Task pane, then click **Run Simulation**.
2. After analysis is complete, click **Yes, continue** to display the Factor of Safety plot.

Results



View the results.

1. On the **Results** page of the Task pane, click **Show displacement**.

   A plot displaying the displacement of `storagebox` appears in the graphics area.

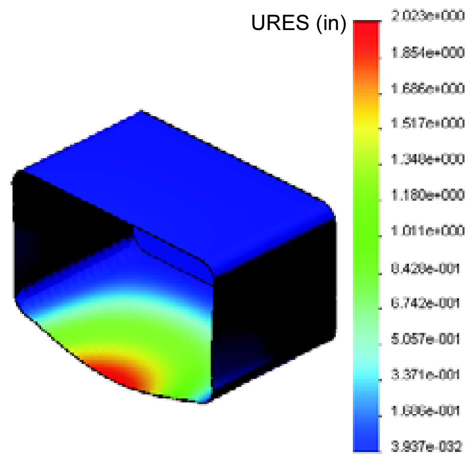
   The maximum displacement is 0.01 inches.
2. Close the Task pane and click **Yes** to save the SolidWorks SimulationXpress data.

Task 3 — Determine the Displacement in a Modified Storage Box

The current wall thickness is 1 centimeter. What if you changed the wall thickness to 1 millimeter? What would the maximum displacement be?

Answer:

- ❑ Edit the `Shell1` feature and change the thickness to **1 mm**.
- ❑ Re-open the **SolidWorks SimulationXpress** Task pane. Notice that **Fixtures**, **Loads**, and **Material** already have check marks. This is because you saved the results when you completed the previous task.
- ❑ Click **Run** in the Task pane, then click **Run simulation**.
- ❑ View the displacement results. Switch to the **Results** tab and display the displacement plot.



The maximum displacement is 2 inches when the wall thickness is 1 millimeter.

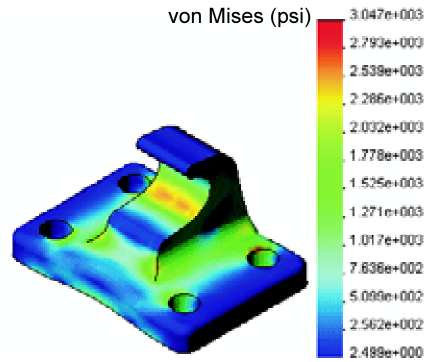
Note that the two displacement plots look similar. The red, yellow, and green areas of the two plots appear in the same place. You must use the legend on the right of the displacement plot to see that the values of displacement are quite different.

More to Explore — Analysis Examples

The *Design Analysis: SolidWorks SimulationXpress: Analysis Examples* section of the SolidWorks Tutorials contains four additional examples. This section does not provide a step-by-step procedural discussion that shows you how to perform each step of the analysis in detail. Rather the purpose of this section is to show examples of analysis, provide a description of the analysis, and outline the steps to complete the analysis.

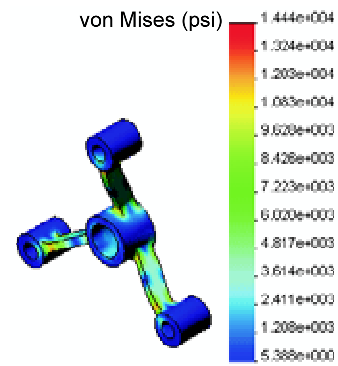
Task 1 — Analyze the Anchor Plate

Determine the maximum force that the anchor plate can support while maintaining a factor of safety of 3.0.



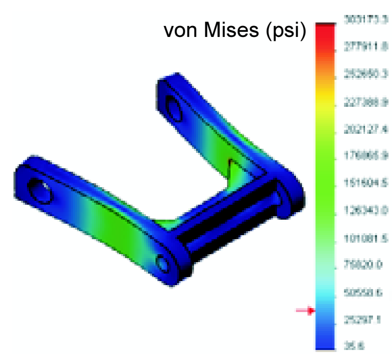
Task 2 — Analyze the Spider

Based on a factor of safety of 2.0, find out the maximum force that the spider can support when a) all outer holes are fixed, b) two outer holes are fixed, and c) only one outer hole is fixed.



Task 3 — Analyze the Link

Determine the maximum force that you can safely apply to each arm of the link.



Task 4 — Analyze the Faucet

Calculate the magnitudes of the front and sideways horizontal forces that will cause the faucet to yield.

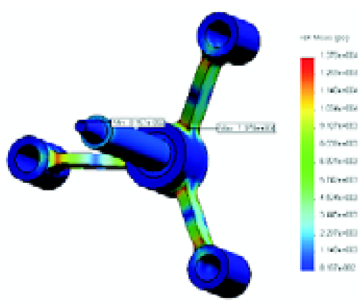
More to Explore — Other Guides and Projects

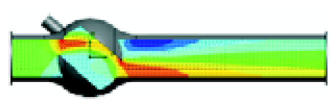
There are additional guides and projects that teach simulation and analysis.

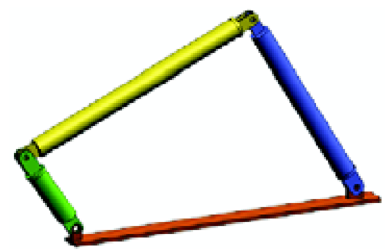
Introduction to Analysis Guides

These guides include:

- *An Introduction to Stress Analysis Applications with SolidWorks Simulation*. Features an introduction to the principles of stress analysis. Fully integrated with SolidWorks, design analysis is an essential part of completing a product. SolidWorks tools simulate the testing of your model's prototype working environment. It can help answer questions such as how safe, efficient, and economical is your design?
- *An Introduction to Flow Analysis Applications with SolidWorks Flow Simulation*. Features an introduction to SolidWorks Flow Simulation. This is an analysis tool for predicting the characteristics of various flows over and inside 3D objects modeled by SolidWorks, thereby solving various hydraulic and gas dynamic engineering problems.
- *An Introduction to Motion Analysis Applications with SolidWorks Motion*. Features an introduction to SolidWorks Motion with step-by-step examples to incorporate dynamic and kinematic theory through virtual simulation.


stress analysis


flow analysis


motion analysis

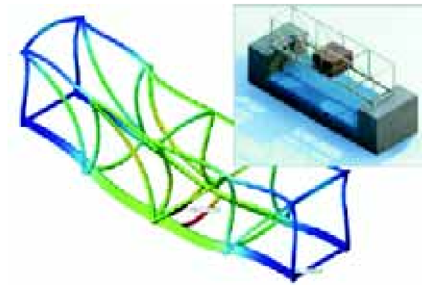
Trebuchet Design Project

The *Trebuchet Design Project* document steps a student through the parts, assemblies, and drawings used to construct a trebuchet. Utilizing SolidWorks SimulationXpress, students analyze structural members to determine material and thickness.

Mathematics and physics competency-based exercises explore algebra, geometry, weight, and gravity.

An optional hands-on construction with models is provided by Gears Education Systems, LLC.

Structural Bridge Design Project



The *Structural Bridge Design Project* document steps a student through the engineering method for constructing a trussed wooden bridge. Students utilize SolidWorks Simulation to analyze different loading conditions of the bridge.

An optional hands-on activity is provided by Pitsco, Inc., with classroom kits.

CO_2 Car Design Project

The *CO_2 Car Design Project* document leads students through the steps of designing and analyzing a CO_2-powered car, from the car body design in SolidWorks to the analysis of air flow in SolidWorks Flow Simulation. Students must make design changes in the car body to reduce drag.

They will also explore the design process through production drawings.

An optional hands-on activity is provided by Pitsco, Inc., with classroom kits.

SolidWorks Sustainability

From raw material extraction and manufacturing to product use and disposal, SolidWorks Sustainability shows designers how the choices they make can change the overall environmental impact of any product they create. SolidWorks Sustainability measures the environmental impact over the life cycle of your product in terms of four factors: carbon footprint, air acidification, water eutrophication, and total energy consumed.

There are tutorials for SolidWorks Sustainability and SustainabilityXpress. Visit *All SolidWorks Tutorials (Set 2)* in the SolidWorks Tutorials.

The *SolidWorks Sustainability* document leads students through the environmental impact of a brake assembly. Students analyze the entire brake assembly and take a closer look at a single part, the rotor.

Lesson 12 Quiz — Answer Key

Name: ______________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1. What are the steps used when performing an analysis with SolidWorks SimulationXpress?

   Answer: Assign material, specify restraints, apply loads, run the analysis, and view the results.

2. True or False. You can use SolidWorks SimulationXpress to perform thermal, frequency, and buckling analyses.

   Answer: False. You need SolidWorks Simulation to perform those analysis types.

3. After completing an analysis, you change the geometry. Do you need to run analysis again?

   Answer: Yes. You must run the analysis again to obtain updated results. It may also be necessary to update the restraints and loads depending on the nature of the geometry changes.

4. What does it mean when the Factor of Safety is less than one?

   Answer: When the Factor of Safety is less than one, the part has exceeded its Yield Strength.

5. Can SolidWorks SimulationXpress be used to analyze parts where the sum of the forces do not add up to zero?

   Answer: No, SolidWorks SimulationXpress can only analyze parts that are static (the sum of the forces and moments must equal zero.)

6. Where can you apply a material to a part so that it can be used in SolidWorks SimulationXpress?

   Answer: You can either apply the material in the part, or you can apply the material in the SolidWorks SimulationXpress Task pane.

7. Name at least three of the result plots you can generate using SolidWorks SimulationXpress.

   Answer: Factor of safety, stress distribution (von Mises), displacement distribution (URES), and deformation.

8. True or False. You can create a SolidWorks eDrawings file containing the result plots.

   Answer: True

Lesson 12 Quiz **REPRODUCIBLE**

Name: ______________________________ Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided or circle the answer as directed.

1 What are the steps used when performing an analysis with SolidWorks SimulationXpress?

____

____

2 True or False. You can use SolidWorks SimulationXpress to perform thermal, frequency, and buckling analyses.

____

3 After completing an analysis, you change the geometry. Do you need to run analysis again?

____

____

4 What does it mean when the Factor of Safety is less than one?

____

____

5 Can SolidWorks SimulationXpress be used to analyze parts where the sum of the forces do not add up to zero?

____

____

6 Where can you apply a material to a part so that it can be used in SolidWorks SimulationXpress?

____

____

7 Name at least three of the result plots you can generate using SolidWorks SimulationXpress.

____

____

8 True or False. You can create a SolidWorks eDrawings file containing the result plots.

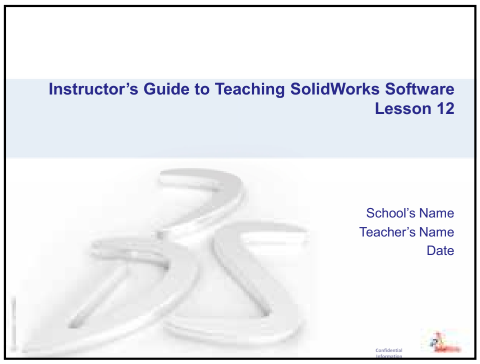
____

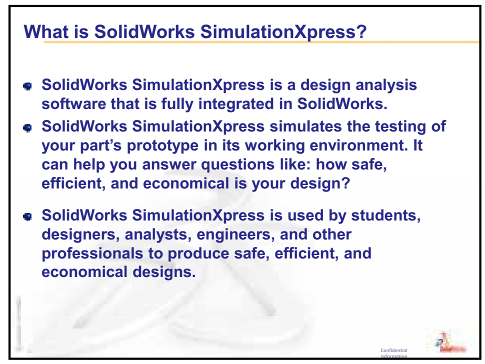
Lesson Summary

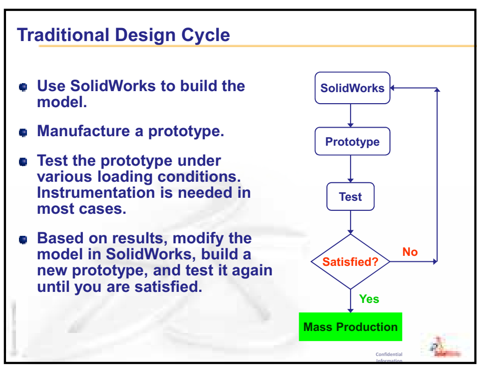
- SolidWorks SimulationXpress is fully integrated in SolidWorks.
- Design analysis can help you design better, safer, and cheaper products.
- Static analysis calculates displacements, strains, stresses, and reaction forces.
- Materials start to fail when stress reach a certain limit.
- von Mises stress is a number that gives an overall idea about the state of stresses at a location.
- SolidWorks SimulationXpress calculates the factor of safety at a point by dividing the yield strength of the material by the von Mises stress at that point. A factor of safety of less than 1.0 indicates that the material at that location has yielded and the design is not safe.

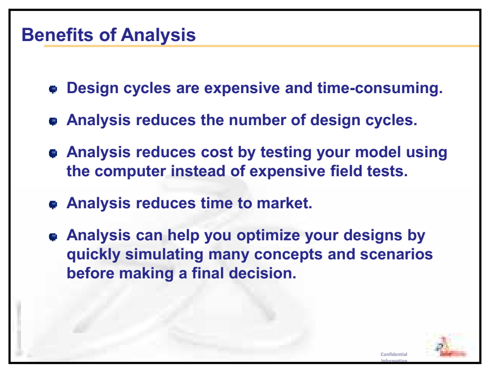
Thumbnail Images of PowerPoint Slides

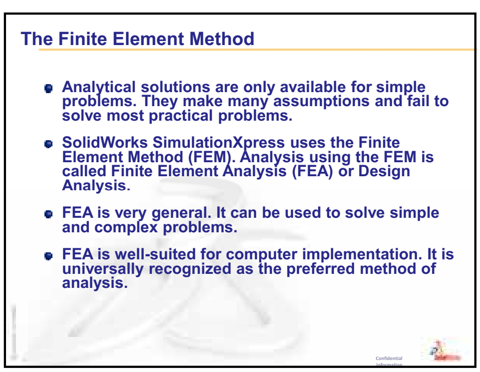
The following thumbnail images, arranged left to right, show the PowerPoint slides provided with this lesson

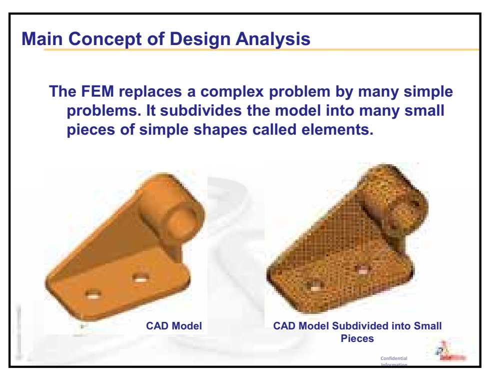






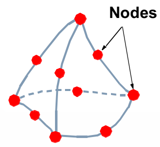






Main Concept of Design Analysis

- The elements share common points called nodes. The behavior of these elements is well-known under all possible support and load scenarios.
- The motion of each node is fully described by translations in the X, Y, and Z directions. These are called degrees of freedom (DOF). Each node has 3 DOF.



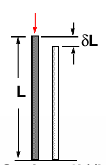
Tetrahedral Element

Main Concept of Design Analysis

- SolidWorks SimulationXpress writes the equations governing the behavior of each element taking into consideration its connectivity to other elements.
- These equations relate the unknowns, for example displacements in stress analysis, to known material properties, restraints and loads.
- Next, the program assembles the equations into a large set of simultaneous algebraic equations. There could be hundreds of thousands or even millions of these equations.

Main Concept of Design Analysis

- In static analysis, the solver finds the displacements in the X, Y, and Z directions at each node.
- Now that the displacements are known at every node of each element, the program calculates the strains in various directions. Strain is the change in length divided by the original length.
- Finally, the program uses mathematical expressions to calculate stresses from the strains.



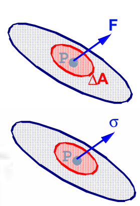
Strain = $(\delta L)/L$

Static or Stress Analysis

- This is the most common type of analysis. It assumes linear material behavior and neglects inertia forces. The body returns to its original position when loads are removed.
- It calculates displacements, strains, stresses, and reaction forces.
- A material fails when the stress reaches a certain level. Different materials fail at different stress levels. With static analysis, we can test the failure of many materials.

What is Stress?

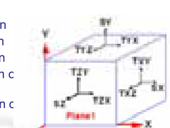
- When a load is applied to a body, the body tries to absorb the effect by generating internal forces that vary from one point to another.
- The intensity of these forces is called stress. Stress is force per unit area.
- Stress at a point is the intensity of force on a small area around that point.



$\sigma = \lim_{\Delta A \to 0} F/\Delta A$

What is Stress?

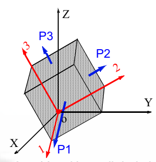
- Stress is a tensor quantity described by magnitude and direction in reference to a certain plane. Stress is fully described by six components:
  - SX: Normal stress in the X-direction
  - SY: Normal stress in the Y-direction
  - SZ: Normal stress in the Z-direction
  - TXY: Shear stress in the Y-direction c YZ-plane
  - TXZ: Shear stress in the Z-direction c YZ-plane
  - TYZ: Shear stress in the Z-direction o XZ-plane
- Positive stress indicates tension and negative stress indicates compression.



Stress Components

Principal Stresses

- **Shear stresses vanish for some orientations. Normal stresses at these orientations are called principal stresses.**
  - P1: Normal stress in the first principal direction (largest).
  - P2: Normal stress in the second principal direction (intermediate).
  - P3: Normal stress in the third principal direction (smallest).



Axes 1,2, and 3 are called principal directions and the normal stresses P1, P2, and P3 are called principal stresses.

Confidential Information

von Mises Stress

- **von Mises stress is a positive scalar number that has no direction. It describes the stress state by one number.**
- **Many materials fail when the von Mises stress exceeds a certain level.**
- **In terms of normal and shear stresses, von Mises stress is given by:**

$$VON = \left\{\left(\frac{1}{2}\right)[(SX-SY)^2+(SX-SZ)^2+(SY-SZ)^2]+3(TXY^2+TXZ^2+TYZ^2)\right\}^{(1/2)}$$

- **In terms of principal stresses, von Mises stress is given by:**

$$VON = \left\{\left(\frac{1}{2}\right)[(P1-P2)^2+(P1-P3)^2+(P2-P3)^2]\right\}^{(1/2)}$$

Confidential Information

Analysis Steps

1. **Assign materials. What is the part made of?**
2. **Specify restraints. Which faces are fixed and do not move?**
3. **Apply loads. Where are the forces or pressures acting on the part?**
4. **Run the analysis.**
5. **View the results. What is the factor of safety? What are the resultant displacements or stresses?**

Confidential Information

Additional Analysis Types

- **SolidWorks SimulationXpress performs linear, static stress analysis on parts. Other software tools provide additional means of analyzing parts and assemblies.**
- **SolidWorks Simulation includes:**
  - Linear, static stress analysis on assemblies.
  - Non-linear static analysis
  - Buckling analysis
  - Frequency analysis
  - Thermal and Thermal stress analysis
  - Optimization analysis
  - Dynamic analysis
  - Fatigue analysis
  - Drop test analysis

Confidential Information

Additional Analysis Types

- **SolidWorks Flow Simulation includes:**
  - Flow simulation of liquids and gases over and inside 3D objects
- **SolidWorks Motion Simulation includes:**
  - Dynamic and kinematic simulation

Confidential Information

G

Glossary

animate View a model or eDrawing in a dynamic manner. Animation simulates motion or displays different views.

assembly An assembly is a document in which parts, features, and other assemblies (sub-assemblies) are mated together. The parts and sub-assemblies exist in documents separate from the assembly. For example, in an assembly, a piston can be mated to other parts, such as a connecting rod or cylinder. This new assembly can then be used as a sub-assembly in an assembly of an engine. The extension for a SolidWorks assembly file name is.SLDASM. See also sub-assembly and mate.

axis An axis is a straight line that can be used to create model geometry, features, or patterns. An axis can be made in a number of different ways, including using the intersection of two planes. See also temporary axis, reference geometry

block A block is a user-defined annotation for drawings only. A block can contain text, sketch entities (except points), and area hatch, and it can be saved in a file for later use as, for example, a custom callout or a company logo.

boss/base A base is the first solid feature of a part, created by a boss. A boss is a feature that creates the base of a part, or adds material to a part, by extruding, revolving, sweeping, or lofting a sketch, or by thickening a surface.

broken-out section A broken-out section exposes inner details of a drawing view by removing material from a closed profile, usually a spline.

chamfer A chamfer bevels a selected edge or vertex.

click-click As you sketch, if you click and then release the pointer, you are in click-click mode. Move the pointer and click again to define the next point in the sketch sequence.

click-drag As you sketch, if you click and drag the pointer, you are in click-drag mode. When you release the pointer, the sketch entity is complete.

closed profile A closed profile (or closed contour) is a sketch or sketch entity with no exposed endpoints; for example, a circle or polygon.

collapse Collapse is the opposite of explode. The collapse action returns an exploded assembly's parts to their normal positions.

component A component is any part or sub-assembly within an assembly.

configuration A configuration is a variation of a part or assembly within a single document. Variations can include different dimensions, features, and properties. For example, a single part such as a bolt can contain different configurations that vary the diameter and length. See design table.

Configuration Manager The ConfigurationManager on the left side of the SolidWorks window is a means to create, select, and view the configurations of parts and assemblies.

cut A feature that removes material from a part.

coordinate system A coordinate system is a system of planes used to assign Cartesian coordinates to features, parts, and assemblies. Part and assembly documents contain default coordinate systems; other coordinate systems can be defined with reference geometry. Coordinate systems can be used with measurement tools and for exporting documents to other file formats.

degrees of freedom Geometry that is not defined by dimensions or relations is free to move. In 2D sketches, there are three degrees of freedom: movement along the X and Y axes, and rotation about the Z axis (the axis normal to the sketch plane). In 3D sketches and in assemblies, there are six degrees of freedom: movement along the X, Y, and Z axes, and rotation about the X, Y, and Z axes. See under defined.

design table A design table is an Excel spreadsheet that is used to create multiple configurations in a part or assembly document. See configurations.

document A SolidWorks document is a file containing a part, assembly, or drawing.

drawing A drawing is a 2D representation of a 3D part or assembly. The extension for a SolidWorks drawing file name is.SLDDRW.

drawing sheet A drawing sheet is a page in a drawing document.

edge The boundary of a face.

eDrawing Compact representation of a part, assembly, or drawing. eDrawings are compact enough to email and can be created for a number of CAD file types including SolidWorks.

face A face is a selectable area (planar or otherwise) of a model or surface with boundaries that help define the shape of the model or surface. For example, a rectangular solid has six faces. See also surface.

feature A feature is an individual shape that, combined with other features, makes 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. However, not all features have associated geometry. Features are always listed in the FeatureManager design tree. See also surface, out-of-context feature.

FeatureManager design tree The FeatureManager design tree on the left side of the SolidWorks window provides an outline view of the active part, assembly, or drawing.

fillet A fillet is an internal rounding of a corner or edge in a sketch, or an edge on a surface or solid.

graphics area The graphics area is the area in the SolidWorks window where the part, assembly, or drawing appears.

helix A helix is defined by pitch, revolutions, and height. A helix can be used, for example, as a path for a swept feature cutting threads in a bolt.

instance An instance is an item in a pattern or a component that occurs more than once in an assembly.

layer A layer in a drawing can contain dimensions, annotations, geometry, and components. You can toggle the visibility of individual layers to simplify a drawing or assign properties to all entities in a given layer.

line A line is a straight sketch entity with two endpoints. A line can be created by projecting an external entity such as an edge, plane, axis, or sketch curve into the sketch.

loft A loft is a base, boss, cut, or surface feature created by transitions between profiles.

mate A mate is a geometric relationship, such as coincident, perpendicular, tangent, and so on, between parts in an assembly. See also SmartMates.

mategroup A mategroup is a collection of mates that are solved together. The order in which the mates appear within the mategroup does not matter.

mirror (1) A mirror feature is a copy of a selected feature, mirrored about a plane or planar face. (2) A mirror sketch entity is a copy of a selected sketch entity that is mirrored about a centerline. If the original feature or sketch is modified, the mirrored copy is updated to reflect the change.

model A model is the 3D solid geometry in a part or assembly document. If a part or assembly document contains multiple configurations, each configuration is a separate model.

mold A mold cavity design requires (1) a designed part, (2) a mold base that holds the cavity for the part, (3) an interim assembly in which the cavity is created, and (4) derived component parts that become the halves of the mold.

named view A named view is a specific view of a part or assembly (isometric, top, and so on) or a user-defined name for a specific view. Named views from the view orientation list can be inserted into drawings.

open profile An open profile (or open contour) is a sketch or sketch entity with endpoints exposed. For example, a U-shaped profile is open.

origin The model origin is the point of intersection of the three default reference planes. The model origin appears as three gray arrows and represents the (0,0,0) coordinate of the model. When a sketch is active, a sketch origin appears in red and represents the (0,0,0) coordinate of the sketch. Dimensions and relations can be added to the model origin, but not to a sketch origin.

over defined A sketch is over defined when dimensions or relations are either in conflict or redundant.

parameter A parameter is a value used to define a sketch or feature (often a dimension).

part A part is a single 3D object made up of features. A part can become a component in an assembly, and it can be represented in 2D in a drawing. Examples of parts are bolt, pin, plate, and so on. The extension for a SolidWorks part file name is .SLDPRT.

pattern A pattern repeats selected sketch entities, features, or components in an array, which can be linear, circular, or sketch-driven. If the seed entity is changed, the other instances in the pattern update.

planar An entity is planar if it can lie on one plane. For example, a circle is planar, but a helix is not.

plane Planes are flat construction geometry. Planes can be used for a 2D sketch, section view of a model, a neutral plane in a draft feature, and others.

point A point is a singular location in a sketch, or a projection into a sketch at a single location of an external entity (origin, vertex, axis, or point in an external sketch). See also vertex.

profile A profile is a sketch entity used to create a feature (such as a loft) or a drawing view (such as a detail view). A profile can be open (such as a U shape or open spline) or closed (such as a circle or closed spline).

Property Manager The PropertyManager is on the left side of the SolidWorks window for dynamic editing of sketch entities and most features.

rebuild The rebuild tool updates (or regenerates) the document with any changes made since the last time the model was rebuilt. Rebuild is typically used after changing a model dimension.

relation A relation is a geometric constraint between sketch entities or between a sketch entity and a plane, axis, edge, or vertex. Relations can be added automatically or manually.

revolve Revolve is a feature tool that creates a base or boss, a revolved cut, or revolved surface by revolving one or more sketched profiles around a centerline.

section A section is another term for profile in sweeps.

section view A section view (or section cut) is (1) a part or assembly view cut by a plane, or (2) a drawing view created by cutting another drawing view with a section line.

shaded A shaded view displays a model as a colored solid. See also HLR, HLG, and wireframe.

sheet format A sheet format typically includes page size and orientation, standard text, borders, title blocks, and so on. Sheet formats can be customized and saved for future use. Each sheet of a drawing document can have a different format.

shell Shell is a feature tool that hollows out a part, leaving open the selected faces and thin walls on the remaining faces. A hollow part is created when no faces are selected to be open.

sketch A 2D sketch is a collection of lines and other 2D objects on a plane or face that forms the basis for a feature such as a base or a boss. A 3D sketch is non-planar and can be used to guide a sweep or loft, for example.

SmartMates A SmartMate is an assembly mating relation that is created automatically. See mate.

sub-assembly A sub-assembly is an assembly document that is part of a larger assembly. For example, the steering mechanism of a car is a sub-assembly of the car.

surface A surface is a zero-thickness planar or 3D entity with edge boundaries. Surfaces are often used to create solid features. Reference surfaces can be used to modify solid features. See also face.

sweep A sweep creates a base, boss, cut, or surface feature by moving a profile (section) along a path.

template A template is a document (part, assembly, or drawing) that forms the basis of a new document. It can include user-defined parameters, annotations, or geometry.

Toolbox A library of standard parts that are fully integrated with SolidWorks. These parts are ready-to-use components — such as bolts and screws.

under defined A sketch is under defined when there are not enough dimensions and relations to prevent entities from moving or changing size. See degrees of freedom.

vertex A vertex is a point at which two or more lines or edges intersect. Vertices can be selected for sketching, dimensioning, and many other operations.

wireframe Wireframe is a view mode in which all edges of the part or assembly are displayed. See also HLR, HLG, shaded.

A

Appendix A: Certified SolidWorks Associate Program

Certified SolidWorks Associate (CSWA)

The Certified SolidWorks Associate (CSWA) Certification Program provides the skills students need to work in the design and engineering fields. Successfully passing the CSWA Exam assessment proves competency in 3D CAD modeling technology, application of engineering principles, and recognition of global industry practices.

Learn more at http://www.solidworks.com/cswa.

Exam Information

DISCLAIMER: This sample exam is provided to show you the format and approximate difficulty level of the real exam. It is not meant to give away the whole CSWA exam.

These questions are an example of what to expect in the CSWA exam.

How to take this sample exam:

1. To best simulate the conditions of the real test, it is best NOT to print this exam. Since the Virtual Tester client window runs concurrently with SolidWorks you must switch back and forth between the two applications. Keeping this document open and consulting it on your computer while running SolidWorks is the best method to simulate the real test conditions.
2. The multiple choice answers should serve as a check for you to ensure that your model is on the right track while completing this exam. If you do not find your answer in the selections offered then most likely there is something wrong with your model at that point.
3. Answers to the questions are on the last pages of this sample test document. There are also hints that can help save time during the exam.
4. If you can complete this exam and get at least 6 out of the 8 questions correctly in 90 minutes or less then you should be ready to take the real CSWA exam.

What you will need for the real CSWA exam:

1. A computer that is running SolidWorks 2007 or higher.
2. The computer must have a connection to the Internet.
3. A double-monitor is recommended but not necessary.
4. If you will be running the Virtual Tester client on a separate computer from the one that is running SolidWorks, make sure there is a way to transfer files from one computer to the other. You will be required to download SolidWorks files during the real test to be able to correctly answer some of the questions.

The following is the topic and question breakdown of the CSWA exam:

- ❑ Drafting Competencies (3 Questions of 5 Points Each):
  - Miscellaneous questions on drafting functionality
- ❑ Basic Part Creation and Modification (2 Questions of 15 Points Each):
  - Sketching
  - Extrude Boss
  - Extrude Cut
  - Modification of Key Dimensions
- ❑ Intermediate Part Creation and Modification (2 Questions of 15 Points Each):
  - Sketching
  - Revolve Boss
  - Extrude Cut
  - Circular Pattern
- ❑ Advanced Part Creation and Modification (3 Questions of 15 Points Each):
  - Sketching
  - Sketch Offset
  - Extrude Boss
  - Extrude Cut
  - Modification of Key Dimensions
  - More Difficult Geometry Modifications
- ❑ Assembly Creation (4 Questions of 30 Points Each):
  - Placing of Base Part
  - Mates
  - Modification of Key Parameters in Assembly

Total Questions: 14

Total Points: 240

165 out of 240 points needed to pass the CSWA.

The sample test below will show the basic format of the CSWA exam in three sections:

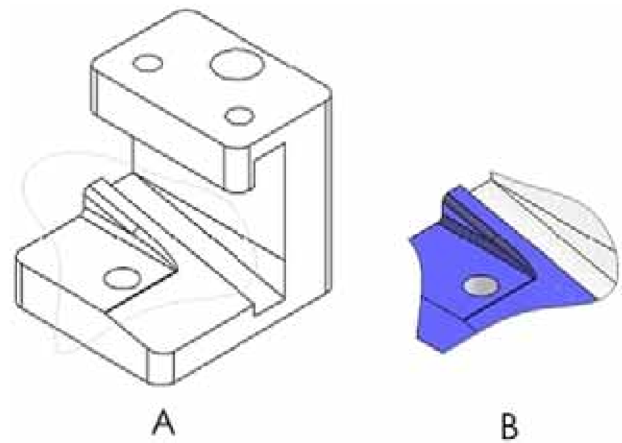
- Drafting Competencies
- Part Modeling
- Assembly Creation

Sample Exam

Drafting Competencies

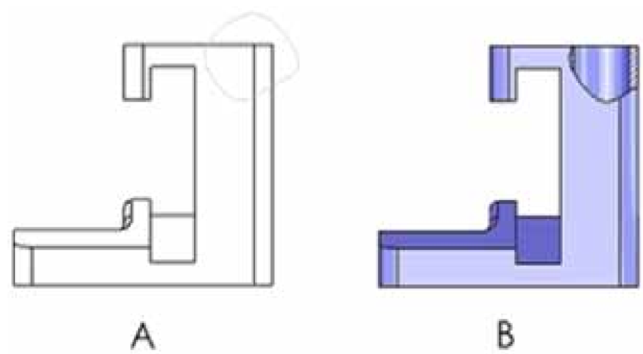
1 To create drawing view ‘B’ it is necessary to sketch a spline (as shown) on drawing view ‘A’ and insert which SolidWorks view type?

a) Section
b) Crop
c) Projected
d) Detail



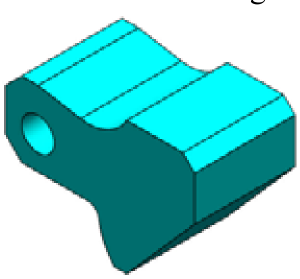
2 To create drawing view ‘B’ it is necessary to sketch a spline (as shown) on drawing view ‘A’ and insert which SolidWorks view type?

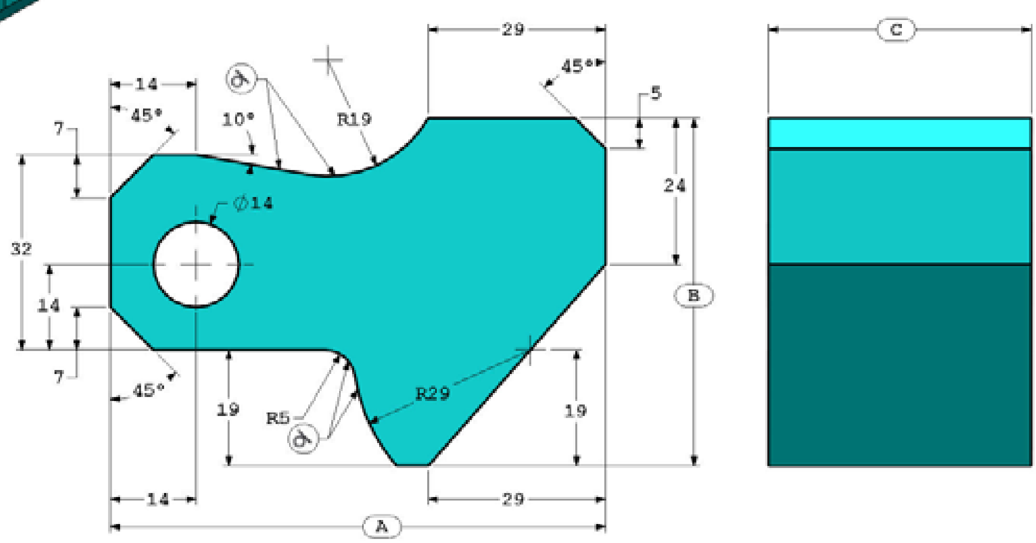
a) Aligned Section
b) Detail
c) Broken-out Section
d) Section



Part Modeling

The following images are to be used to answer Questions #3-4.





3 Part (Tool Block) - Step 1
Build this part in SolidWorks.
(Save part after each question in a different file in case it must be reviewed)

Unit system: MMGS (millimeter, gram, second)
Decimal places: 2
Part origin: Arbitrary
All holes through all unless shown otherwise.
Material: AISI 1020 Steel
Density = 0.0079 g/mm^3

A = 81.00
B = 57.00
C = 43.00

What is the overall mass of the part (grams)?

Hint: If you don't find an option within 1% of your answer please re-check your solid model.

a) 1028.33
b) 118.93
c) 577.64
d) 939.54

4 Part (Tool Block) - Step 2
Modify the part in SolidWorks.

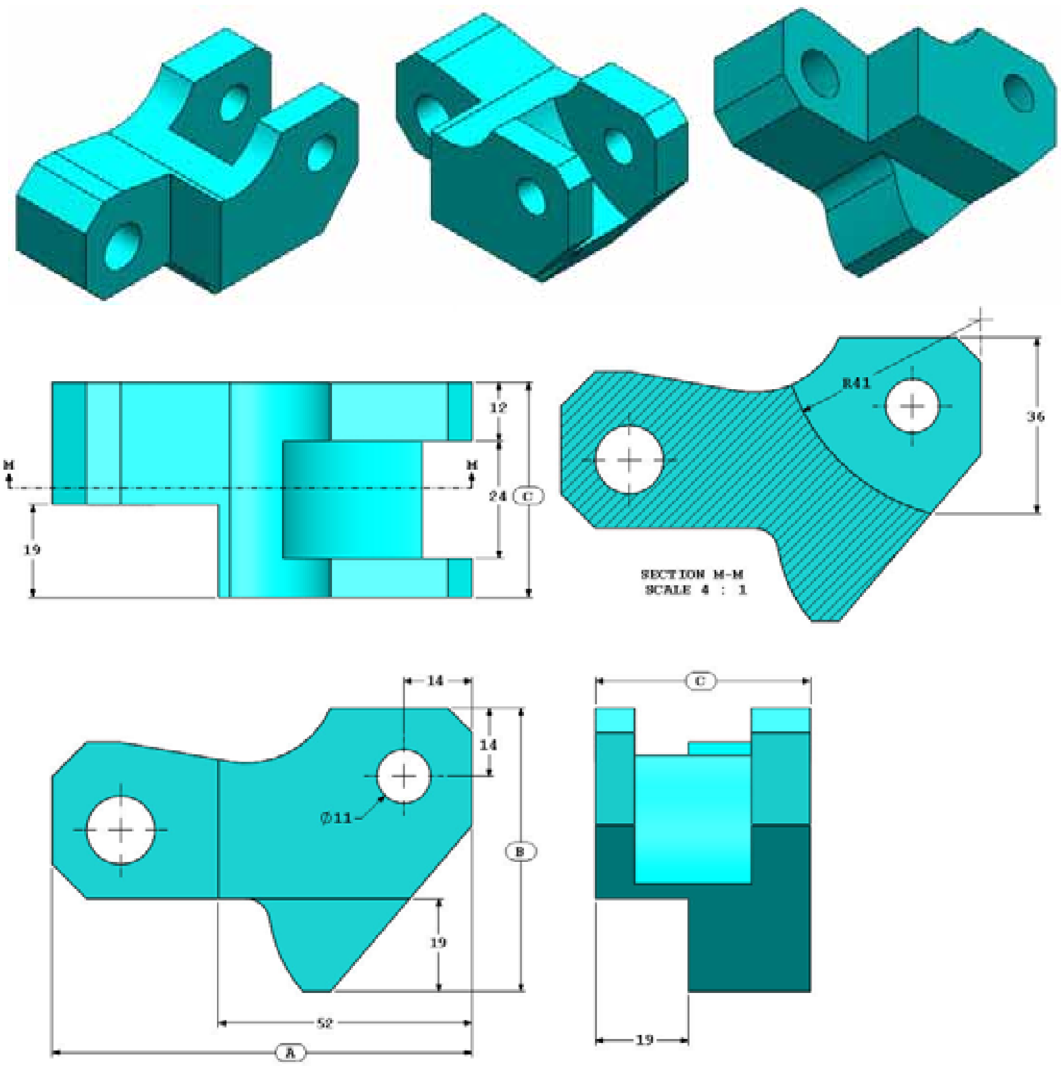
Unit system: MMGS (millimeter, gram, second)
Decimal places: 2
Part origin: Arbitrary
All holes through all unless shown otherwise.
Material: AISI 1020 Steel
Density = 0.0079 g/mm^3

Use the part created in the previous question and modify it by changing the following parameters:
A = 84.00
B = 59.00
C = 45.00
Note: Assume all other dimensions are the same as in the previous question.

What is the overall mass of the part (grams)?

Part Modeling

The following images are to be used to answer Question #5.



5 Part (Tool Block) - Step 3
Modify this part in SolidWorks.

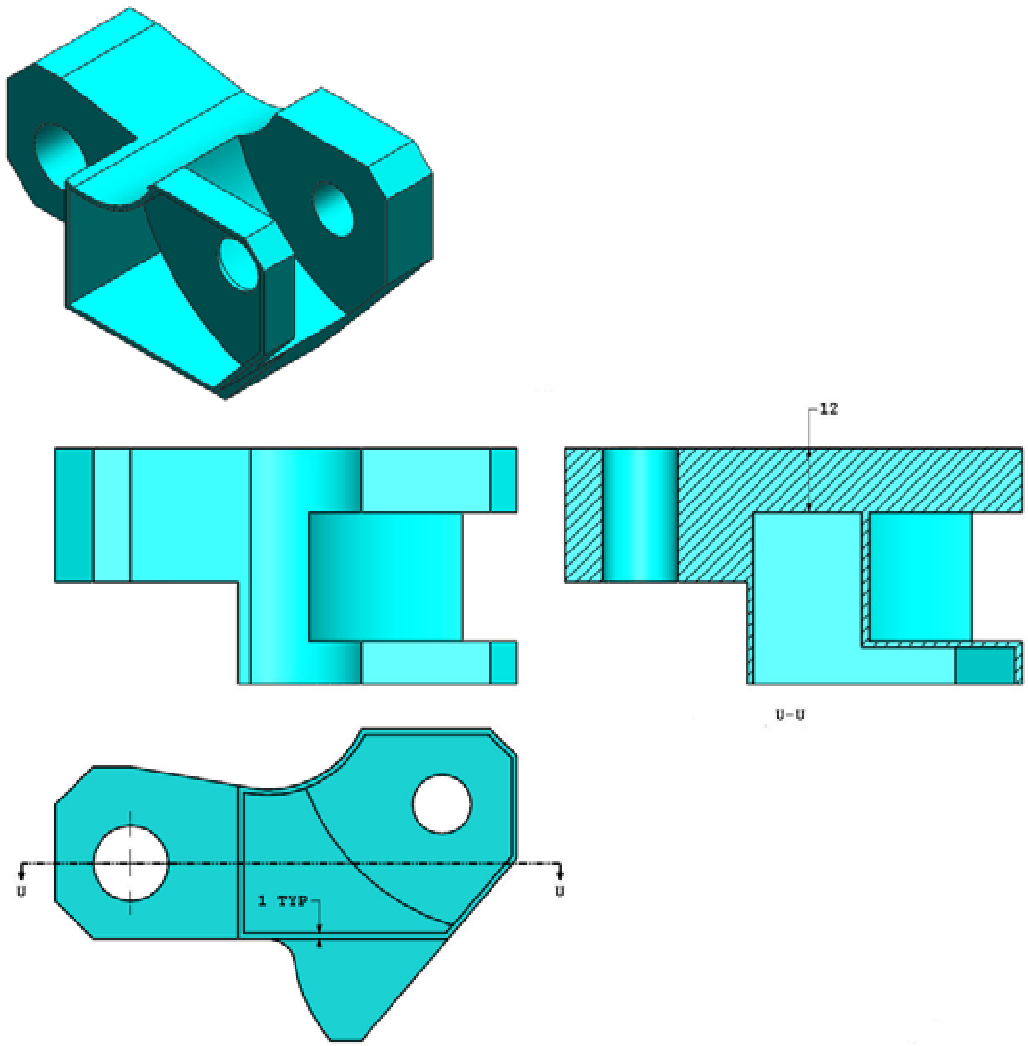
Unit system: MMGS (millimeter, gram, second)
Decimal places: 2
Part origin: Arbitrary
All holes through all unless shown otherwise.
Material: AISI 1020 Steel
Density = 0.0079 g/mm^3

Use the part created in the previous question and modify it by removing material and also by changing the following parameters:
A = 86.00
B = 58.00
C = 44.00

What is the overall mass of the part (grams)?

Part Modeling

The following images are to be used to answer Question #6.



6 Part (Tool Block) - Step 4
Modify this part in SolidWorks.

Unit system: MMGS (millimeter, gram, second)
Decimal places: 2
Part origin: Arbitrary
All holes through all unless shown otherwise.
Material: AISI 1020 Steel
Density = 0.0079 g/mm^3

Use the part created in the previous question and modify it by adding a pocket.

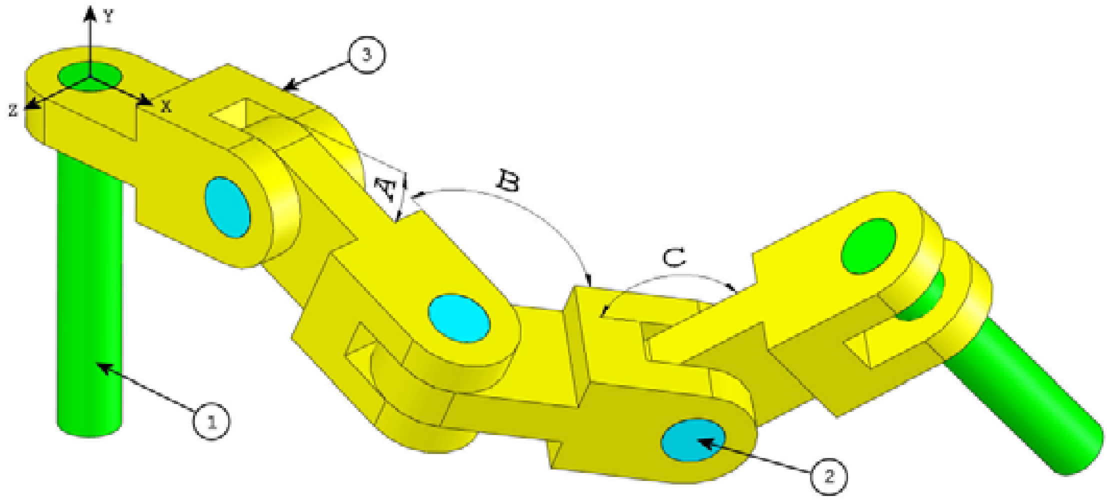
Note 1: Only one pocket on one side is to be added. This modified part is not symmetrical.

Note 2: Assume all unshown dimensions are the same as in the previous question #5.

What is the overall mass of the part (grams)?

Assembly Creation

The following image is to be used to answer Question #7-8.



7 Build this assembly in SolidWorks (Chain Link Assembly)
It contains 2 long_pins (1), 3 short_pins (2), and 4 chain_links (3).

Unit system: MMGS (millimeter, gram, second)

Decimal places: 2

Assembly origin: Arbitrary

Use the files in the `Lessons\CSWA` folder.

- Save the contained parts and open those parts in SolidWorks. (Note: If SolidWorks prompts "Do you want to proceed with feature recognition?" please click "No".)
- IMPORTANT: Create the Assembly with respect to the Origin as shown in isometric view. (This is important for calculating the proper Center of Mass)

Create the assembly using the following conditions:

- Pins are mated concentric to chain link holes (no clearance).
- Pin end faces are coincident to chain link side faces.
- A = 25 degrees
- B = 125 degrees
- C = 130 degrees

What is the center of mass of the assembly (millimeters)?

Hint: If you don't find an option within 1% of your answer please re-check your assembly.

a) X = 348.66, Y = -88.48, Z = -91.40
b) X = 308.53, Y = -109.89, Z = -61.40
c) X = 298.66, Y = -17.48, Z = -89.22
d) X = 448.66, Y = -208.48, Z = -34.64

8 Modify this assembly in SolidWorks (Chain Link Assembly)
Unit system: MMGS (millimeter, gram, second)
Decimal places: 2
Assembly origin: Arbitrary

Using the same assembly created in the previous question modify the following parameters:

- A = 30 degrees
- B = 115 degrees
- C = 135 degrees

What is the center of mass of the assembly (millimeters)?

More Information and Answers

For further preparation, please complete the SolidWorks tutorials, located in SolidWorks under the Help Menu, before taking the CSWA Exam. Review the information on the CSWA exam located at http://www.solidworks.com/cswa.

Good Luck,

Certification Program Manager, SolidWorks Corporation

Answers:

1 b) Crop

2 c) Broken-out Section

3 d) 939.54 g

4 1032.32 g

5 628.18 g

6 432.58 g

7 a) X = 348.66, Y = -88.48, Z = -91.40

8 X = 327.67, Y = -98.39, Z = -102.91

Hints and Tips:

- Hint #1: To prepare for the Drafting Competencies section of the CSWA, review all the drawing views that can be created. These commands can be found by opening any drawing and going to the View Layout command manager toolbar or in the menu Insert > Drawing View.
- Hint #2: For a detailed explanation of each View type, access the individual feature Help section by selecting the Help icon in the PropertyManager for that View Feature.