

ME9 Engineering Graphics and Design

CAD Assignment 1

OBJECTIVES

- SolidWorks Installation
 - Students will verify that they have successfully installed SolidWorks on their computer as required for the course.
- Introduction to SolidWorks
 - o Students will become familiar with the SolidWorks interface.
 - Students will gain an understanding of the basic functionality of SolidWorks software and be able to create simple parts.

INTRODUCTION

Software Installation

All first-year and transfer students in the Bourns College of Engineering (BCOE) are required to have a personal laptop computer to run software needed for course work. Guidelines for minimum hardware configurations for laptops is available by clicking on the "Laptop Program" link on the <u>BCOE Systems</u> website.

If you need assistance with your computer account, click on the "Account Setup" link on the BCOE Systems website.

UCR and BCOE provide software used in many engineering courses, such as SolidWorks, MATLAB, the Microsoft Office Suite, and many others at no additional cost to students. For a complete list and installation instructions, click on the "Software" link on the BCOE Systems website.

You must install <u>SolidWorks</u> on your laptop before proceeding with this assignment. You must also install <u>GlobalProtect</u>, a program to establish a virtual private network (VPN) connection with UCR. This allows you to remotely access campus resources that are restricted to on-campus use. To access the Solidworks license server, *a VPN connection must be established to start Solidworks*.

Mac Users: Solidworks is only available for Windows. Mac versions are not available. To install Solidworks on a Mac, you must first install Windows using <u>Boot Camp</u>.

For assistance with installing software or any other hardware issues, please contact the BCOE Help Desk at helpdesk@engr.ucr.edu or in-person at WCH 107.

As stated in the Laptop Program website, BCOE Students are strongly encouraged to invest in some form of a backup solution. Students are responsible for ensuring that they have their own backups on an external hard drive, cloud storage, or archiving solution. Backups should be performed on a regular basis. Loss of data or files will not be accepted as an excuse for not turning in work, or for turning it in late.

All ME9 coursework shall be organized and saved in a folder designated for ME9 for at least the duration of the course.

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SolidWorks

SolidWorks is a 3D computer-aided design (CAD) software package used to model parts, assemblies and 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. 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 and is the foundation of the part. Additional features are created to shape the part by adding material (bosses) or removing material (cuts).

An assembly is a document in which parts are mated together. The parts exist in documents separate from the assembly. For example, a piston model can be mated to other parts, such as a connecting rod or cylinder to create 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.

The following exercises are intended to provide a general overview of solid modeling with SolidWorks. Solid modeling and the use of SolidWorks will be discussed extensively throughout the ME9 course. The methods and terminology introduced here will be discussed in much greater detail.

CAD ASSIGNMENTS

In the following two exercises, you will create SolidWorks part models. At the end of each lesson, you will save your model as a SolidWorks part file (.SLDPRT) and a screenshot image as PDF file (Portable Document Format). Specific instructions regarding how to save the files are provided in each lesson.

Begin the lessons by starting SolidWorks from the start menu or desktop icon.

After completing both lessons, submit the two PDF files (do not submit the SolidWorks files).

Submitting Your Work:

- 1. Access the ME9 course on Canvas.
- 2. In the Assignments folder, open **CAD Assignment 1** to submit your PDF files.

FOR THIS ASSIGNMENT SUBMIT PDF FILES ONLY

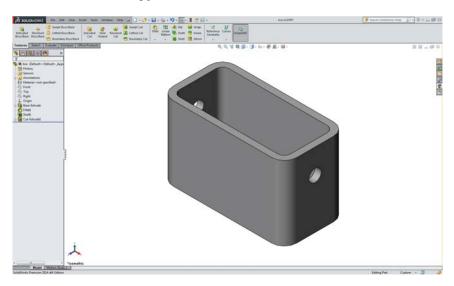
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PART A

Reference: SolidWorks CAD Student Guide Screenshots and menu layouts shown in this lesson may be different in various versions of Solidworks.

Goals of This Lesson

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



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.

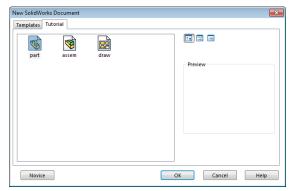


Create a New Part Document

Bar.

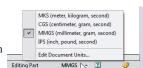
- 1 Create a new part. Click **New** on the Menu
 - The **New SolidWorks Document** dialog box appears.
- 2 Click the **Tutorial** tab.
- 3 Select the Part icon.
- 4 Click OK.

A new part document window appears.



Setting Units

The units for a document can be set by clicking **Tools**, **Options** and then under the **Document Properties** tab, selecting units from the menu on the left. In the status bar, units can also be changed by clicking **Unit System** and then selecting the new unit system.



Base Feature

The Base feature requires:

- □ Sketch plane Front
- □ 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 > Sketch 2.

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.

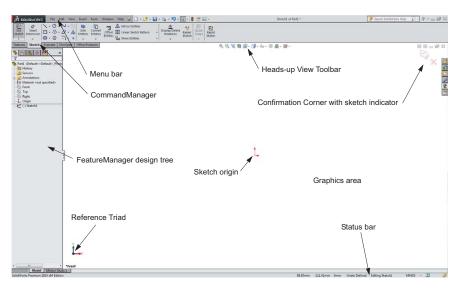


When other commands are active, the confirmation corner displays two symbols: a check mark and an X. The check mark executes the current command. The X cancels the command.



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

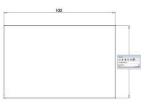
- 1 Click Sketch > Corner Rectangle .
- 2 Click the sketch origin to start the rectangle.
- **3** Move the pointer up and to the right, to create a rectangle.
- 4 Click the mouse button again to complete the rectangle.



Add Dimensions

- 1 Click **Sketch > Smart Dimension** 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 ✓.

Modify ✓ ★ 8 ₹ ±? D2@Sketch1 50mm

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.

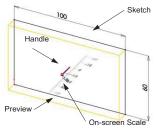
1 Click Features > Extruded Boss/Base The Boss-Extrude PropertyManager appears. The view of the sketch changes to trimetric.



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 On-screen Scale³ 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.

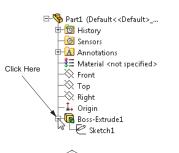
TIP:

The \mathbf{OK} button \checkmark 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.





View Display

Change the display mode. Click **Display Style > Hidden Lines Visible** on the Heads-up View toolbar.

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

Save the Part

- 1 Click **Save** on the Menu Bar, 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

Save in your ME9 folder!

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

- Click Features > Fillet
 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.

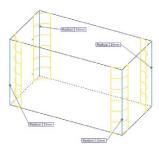
When you select the edge, a callout $_{\mbox{\scriptsize Radius:}}\mbox{\scriptsize $10mm$}$ appears.

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



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 Display Style > Shaded on the Heads-up View toolbar.

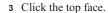


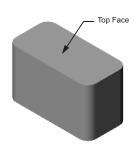
Hollow Out the Part

Remove the top face using the Shell feature.

- 1 Click Features > Shell .
 The Shell PropertyManager appears.
- 2 Enter 5 for Thickness.







4 Click .



Extruded Cut Feature

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

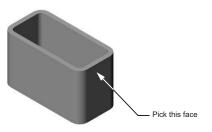
- □ Sketch plane In this exercise, the face on the right-hand side of the part.
- □ Sketch profile 2D circle

Open a Sketch

- 1 To select the sketch plane, click the right-hand face of the box.
- 2 Click View Orientation > Right 🗐 on the Heads-up View toolbar.

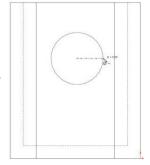
The view of the box turns. The selected model face is facing you.

3 Open a 2D sketch. Click **Sketch > Sketch**



Sketch the Circle

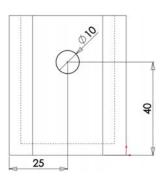
- 1 Click Sketch > Circle .
- 2 Position the pointer where you want the center of the circle. Click the left mouse button.
- 3 Drag the pointer to sketch a circle.
- 4 Click the left mouse button again to complete the circle.



Dimension the Circle

Dimension the circle to determine its size and location.

- 1 Click Sketch > Smart Dimension 2.
- 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 Features > Extruded Cut .
 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 **View Orientation > Isometric** on the Heads-up View toolbar.

Save the Part

- 1 Click **Save** on the Menu Bar.
- 2 Click File, Exit.

Be sure your part (.sldprt file) was saved in your ME9 folder!

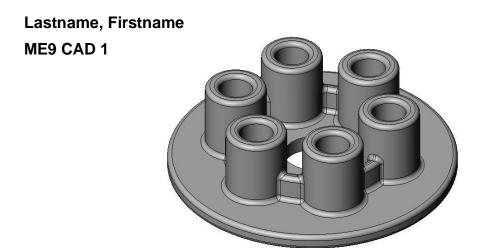
Save the view shown above in a PDF file (click Save As and select PDF as the file type).

Name the file **ME9 CAD 1a - Lastname**, **Firstname** and save it in your ME9 folder.

PART B

As a second introductory exercise, the online SolidWorks Tutorial "30 Minute Lesson" will guide you through the creation of a part and a drawing. Although this tutorial is labeled as a 30-minute lesson, the exercise typically takes a little longer. You should have plenty of time to complete this exercise, so relax and have fun. Remember, we will cover the details of model construction later. This is just a preview.

- 1. Start SolidWorks, click Help \rightarrow SolidWorks Tutorials \rightarrow Getting Started \rightarrow Introduction to SolidWorks.
- 2. Follow the steps in the tutorial to complete the part and a drawing of the object shown below. Save the files in your ME9 folder. Use the default SolidWorks drawing templates.
- 3. At the end of the tutorial, add the following note to the part (Insert \rightarrow Annotations \rightarrow Note...):



- 1. Save the part in your ME9 folder with a file name **ME9 CAD 1 Lastname**, **Firstname**.
- 2. Set the current view to Isometric. Save the view in a PDF file (click Save As and select PDF as the file type). Name the file **ME9 CAD 1 Lastname**, **Firstname** and save it in your ME9 folder.

Note: Do not select the box to save as a "3D PDF" file.

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