

Engineering Design and Technology Series

SAE[®] Design and Analysis Project with SolidWorks[®] Software



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Lesson 1 Introduction

When you complete this lesson, you will be able to:

- Describe the relationship between Parts, Assemblies and Drawings;
- Identify the principal components of the SolidWorks user interface;
- Download and extract the required companion files.

Introduction

SolidWorks Engineering Design and Technology Series

Using This Book

The SAE Design and Analysis Project with SolidWorks Software helps you learn the principles of assembly design and structural analysis using SolidWorks and SolidWorks Simulation as an integral part of a creative and iterative design process.

For this project, You will "learn by doing" as you complete modeling and structural analysis lessons.

What is SolidWorks Software?

SolidWorks is design automation software. In SolidWorks, you sketch ideas and experiment with different designs to create 3D models using the easy to learn Windows[®] graphical user interface.

SolidWorks is used by students, designers, engineers and other professionals to produce single and complex parts, assemblies and drawings.

Prerequisites

Before you begin the SAE Design and Analysis Project with SolidWorks Software you should complete the following online tutorials that are integrated in the SolidWorks software:

- Lesson 1 Parts
- Lesson 2 Assemblies
- Lesson 3 Drawings

You can access the online tutorials by clicking **Help**, **SolidWorks Tutorials**, **All SolidWorks Tutorials (Set 1)**. The online tutorial resizes the SolidWorks window and runs beside it.

As an alternative, you can complete the following lessons from *CAD Instructor Guide*:

- Lesson 1: Using the Interface
- Lesson 2: Basic Functionality
- Lesson 3: The 40-Minute Running Start
- Lesson 4: Assembly Basics
- Lesson 6: Drawing Basics

Conventions Used in This Book

This manual uses the following typographical conventions:

Convention	Meaning
Bold Sans Serif	SolidWorks commands and options appear in this style. For example, Insert, Boss means choose the Boss option from the Insert menu.
Typewriter	Feature names and file names appear in this style. For example, Sketch1.
17 Do this step.	The steps in the lessons are numbered in sans serif bold.

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 resources.

Click the SolidWorks Resources tab and click Student Curriculum.



Conventions Used in This Book

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3 SolidWorks Content.

Expand the SolidWorks Educator Curriculum folder.

Expand the proper Curriculum <year> folder.

Click **Design Library (a)** to open the design library task pane.

Click the SAE Car Design Project folder.

The lower pane will display an icon representing a Zip file that contains the companion files for this project.



4 Download the Zip file. Press Ctrl and click the icon.

You will be prompted for a folder in which to save the Zip file.

Ask your teacher where you should save the Zip file. Usually the C:\Temp folder is a good location. Click **OK**.

Tip: Remember where you saved it.



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SolidWorks

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5 Open the Zip file.

Browse to the folder where you saved the Zip file in step 4.

Double-click the SAE Project Files.zip file.

6 Click Extract.

Click **Extract** and Browse to the location where you want to save the files. The system will automatically create a folder named



_SAE_Project_ENG in the location you specify. For example, you might want to save it in My Documents. Check with your teacher about where to save the files.

You now have a folder named SAE Project Files on your disk. The data in this folder will be used in the exercises.

Tip: Remember where you saved it.

Analyzing a Structure Using SolidWorks and SolidWorks Simulation

During this session, you will learn how to analyze a structure using SolidWorks and SolidWorks Simulation.

Once you have had a chance to see how easy it is to use SolidWorks solid modeling software, you will use an assembly to check whether components fit properly.

You will then make a drawing of one of the components, complete with a cut list. If a printer is available, you can print out a hardcopy of your drawing.

Lesson 2 Using Assemblies

When you complete this lesson, you will be able to:

- Understand the difference between edit assembly and edit part modes;
- Create a virtual, in-context part;
- Open a part from the assembly;
- Create a new instance from an existing instance;
- Set the materials in a part;
- Use pack and go to manage the files.

SolidWorks Engineering Design and Technology Series

Creating Parts In-Context

A thorough knowledge of how to work with assemblies is critical to success with SolidWorks.

In this example, you will create a virtual part and use in-context techniques to mode a brake pad using geometry from both the Rotor - Cast Iron and Brake Caliper components.

The component will copied to create a second instance and it will be mated into the assembly.



Opening an Assembly with Quick View

Clicking the **Quick view / Selective open** when you open an assembly allows you to view only the components that you want to view.

1 Open Brake&Wheel.

Click File, Open and select the assembly Brake&Wheel from the Lesson 2\Brake&Wheel folder. Select Mode: Large Design Review and click Open. Click OK on the Large Design Review page.



Note: At the message: Graphics data might be out of date for the flagged components because these components reference inactive configurations, click **OK**.

Hiding and Showing Components

Components can be hidden or shown at any time to create an uncluttered display and make working on the assembly faster.

In addition, components that are hidden before opening the assembly are not loaded into memory, further decreasing the load on the machine.

Tip: The **Quick view / Selective open** settings are stored in a Display State.

There are many ways to hide and show components. Here are a few useful methods and where they are best used.

	Hiding Components	Showing Components
Many Components	Hide Others- Hides all visible components but the selected component(s).	Show Hidden Components- Shows all the hidden components for selection.
Single or Few Components	Hide/Show Components- Hides the selected visible components.	Hide/Show Components- Shows the selected hidden components.

2 Orientation.

The assembly opens in an Isometric orientation. Click in the graphics area and press **Shift+Arrow Up** to change the view orientation.

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Where to Find It

■ Shortcut Menu: Right-click a component and click Hide Others

3 Hide others.

Right-click the Rotor - Cast Iron and select Hide Others.



Where to Find It

- Shortcut Menu: Right-click and click Show Hidden Components
- 4 Show hidden.

Right-click in the graphics area and select **Show Hidden Components**. Click the Brake Caliper component as shown. Click **Exit Show-Hidden** to complete the command.





Using Assemblies

SolidWorks

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Where to Find It

- CommandManager: Large Design Review > Selective Open Selective Open
- 5 Selective open.

Click Selective Open **Mathematica**.

Click **All components displayed** and click **Open**. These are the only components that we need to view right now.

Contraction of the second	1,00001
Selected components	Cancel
All components displayed	

6 Message.

A message appears.

Because you used "Selective Open", hidden components are not loaded into memory. Therefore, when you first show a hidden component you might notice a delay while it is loaded. Also, a new Display State is created corresponding to the "Selective Open" state. Click **OK**.

7 Display state.

The changes are stored in the original display state Default_Display_State-1.



8 Save.

Click **File**, **Save I** to save the assembly and parts.

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9 Orientation changes.

Press the **Spacebar** and double-click the orientation *Isometric from the dialog. **Middle (wheel)+click** and drag the edge of the Rotor - Cast Iron as shown to rotate the geometry.



Using Assemblies



10 Zoom.

Click the Brake Caliper component in the FeatureManager design tree and click **Zoom to**

Selection 🔍.



The Assembly FeatureManager design tree

The assembly is comprised of components and mates. The components can be either part components or assembly components (sub-assemblies). The assembly FeatureManager design tree shows an accurate snapshot of the assembly using icons and text to describe the current settings.



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Where to Find It

- CommandManager: Evaluate > AssemblyXpert
- Menu: Tools, AssemblyXpert

11 AssemblyXpert.

Click **AssemblyXpert .** The dialog lists the number of parts, unique parts, sub-assemblies and unique sub-assemblies among other things.

Ð	Total number of components in Brake&Wheel:	33	
	Parts.	29	
	Unique Part Documents.	-	
	Subaccamblian:	4	
	Unique Subassambles	2	
	Unique Subassembly Documents	2	
	Maximum Denth	2	
	Number of too level components	7	
	Resolved components:	5	
	Resolved documents:	5	
	Lightweight components:	0	
	Suppressed components	28	
	Number of top level mates:	15	
	Number of bodies:	6	
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Click OK.

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Working In-Context

Working in-context means editing a component (part or assembly) while in the context of the assembly. The mode is toggled between **Edit Assembly** and **Edit Part**.

Edit Assembly Mode vs. Edit Part Mode

When an assembly is opened, it opens in the default **Edit Assembly** mode. To create or edit a component part in-context, **Edit Part** is used. You can toggle back and forth between the modes using **Edit Component [Solution**].



Tip: Colors are used to indicate which mode is currently active. See "Why do the Colors Change?" on page 17 for more information.

Below is a breakdown of some common actions that are typically performed in each mode:

Edit Assembly Mode	Edit Part Mode
Adding new components	Creating new sketches
Inserting mates	Creating sketch geometry
Moving components	Creating bosses or cut features

Using Assemblies

In-Context Parts and Virtual Parts

In-Context Parts are parts that are created or edited in the context of the assembly. In-context part names appear in the FeatureManager design tree appended with an arrow (In-Context->).

Virtual Parts are in-context parts that are saved within the assembly rather than as separate part files. Parts can be both in-context and virtual. Virtual part names appear in the FeatureManager design tree in square brackets [Virtual_Part^Test].



Why use In-Context and Virtual Parts?

In-Context parts reference other parts in the assembly and change automatically when the references change.

Virtual Parts are more flexible because they can be renamed, deleted or saved as external (part) files at any time.

Tip: If there are no references, do not create the part in-context.

Setup for Using Edit Part

There are settings under system options that can be used to determine how assemblies and virtual parts behave in Edit Part mode.

Where to Find It

- CommandManager: Assembly > Edit Component [5]
- Shortcut Menu: Right-click a component and click Edit Part

12 Virtual part setting.

Click Tools, Options, System Options, Assemblies and clear Save new components to external files.

Do not click OK yet.

13 In-context appearance setting.

Click **Display/Selection** and select **Opaque Assembly** from the pulldown menu under **Assembly transparency for in context edit**.

Do not click **OK** yet.

14 In-context part setting.

Click **Colors** and click **Use specified color when editing parts in assemblies**. That color is listed under the **Assembly, Edit Part** setting.

Click OK.

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Creating a New Part

Creating a in-context new part requires a few selections including a planar face or plane for use as a sketch plane.

The selected face or plane will establish the orientation and position of the Front plane of the new virtual part. This in turn defines the orientations of the Top and Right planes.



Using Assemblies

Where to Find It

- CommandManager: Assembly > Insert Components [], New Part []
- Menu: Insert, Component, New Part

15 New part.

Click **New Part** s and select the face of the Brake Caliper as shown.



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Edit Part Mode

Edit Part Mode is the opposite toggle of **Edit Assembly Mode** and allows the use of sketch and feature tools within the assembly. It is triggered by adding a new part component or editing an existing part in the assembly.

Editing Assembly

Why do the Colors Change?

Due to the settings made ("Setup for Using Edit Part" on page 15), the appearance of all the parts remains opaque. The part being edited will appear in the **Assembly, Edit Part** color and all others appear in the **Assembly, Non-Edit Parts** color.

Controlling the Display

The display includes the visibility and colors of the components in the assembly. Controlling the display is the first step in managing the assembly itself, and the Display Pane is one of the best tools.

Display Pane

The **Display Pane** is a portion of the FeatureManager design tree that has visual controls and is usually hidden. The columns show the current state of **Hide/Show**, **Display Mode**, **Appearance** and **Transparency** and allow you to change them. The options are described below.

- Hide/Show Toggles between Hide Component and Show Component.
- Display Mode Sets the display to Wireframe, Hidden Lines Visible, Hidden Lines Removed, Shaded With Edges, Shaded, or Default Display.
- **Appearance** Sets the appearance of the component. The lower triangle represents the part appearance while the upper triangle represents the component (assembly level) appearance.
- **Transparency** Toggles **Transparency** on and off.

Note: The display pane works independently of the mode.

Where to Find It

- Display Pane: Hide/Show in the component row
- **Note:** Click **Hide Display Pane w** to close the display pane.



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16 Display pane.

Click **Show Display Pane** to expand the display plane and change the appearance of components.

Expand the Brake Rotor Assembly folder. Click the Rotor - Cast Iron component in the **Hide/Show** column as shown to hide it.





Where to Find It

- CommandManager: Sketch > Convert Entities []]
- Menu: Tools, Sketch Tools, Convert Entities

17 Convert entities.

Select the face and click **Convert Entities** .



Using Assemblies

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Tip: If you see small green icons on the geometry of the sketch, you are seeing sketch relations. Click **View, Sketch Relations** to shut off their display.

18 Delete.

Delete three entities to open the end of the sketch.



Note: There is one large arc and two small ones connected to it. Only one small arc is shown here.

19 Convert edge.

Click **Show Display Pane** and show the Rotor - Cast Iron component. Select the edge of the Rotor - Cast Iron component and click **Convert Entities** .



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Where to Find It

- CommandManager: Sketch > Sketch Fillet
- Menu: Tools, Sketch Tools, Fillet

20 Sketch fillets.

Click **Fillet** , set the **Fillet Radius** to **2.5mm**, and select the first set of geometry by selecting the geometry inside of where they would intersect.





Repeat the selections for the similar geometry on the opposite side. Click \checkmark twice.

21 View normal to.

Click **View Normal To** 4 and zoom in as shown.



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22 Delete.

Box select from upper left to lower right as shown to select the three entities.

Delete the three selected entities.

Using Assemblies



23 Drag endpoints.

Drag the endpoint of the vertical line outside the geometry as shown.

Repeat for the other vertical line and stop where the endpoints are horizontal.



24 Tangent Arc. Click Tools, Sketch Entities,

Tangent Arc () and create the arc between the two endpoints as shown.



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25 Drag and drop.

Drag the centerpoint of the arc to the edge of the circular edge. Drop it on the centerpoint that appears.



26 Previous view.

Click **Previous View** store to go back to previous views and zoom states. Return to the zoomed isometric view.

27 Extrude.

Click **Extrude** $\boxed{\mathbf{a}}$ and set the **Depth** to **3mm**. Click \checkmark .



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From	Ŕ
Sketch Plane	~
Direction 1	\$
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3.00mm	1
TRAVE OURWOOT	

Using Assemblies

Understanding the Color Coding

The part turns blue when the extrusion is completed. The reason was explained earlier ("Why do the Colors Change?" on page 17) but until there is a solid body, it is difficult to notice. This is the **Assembly, Edit Part** color and it appears in both the graphics and in the FeatureManager design tree.



28 New sketch.

Select the face and click **Sketch [2]**.



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29 Convert entities.

Select the edge and click **Convert Entities C**.



Where to Find It

- CommandManager: Sketch > Offset Entities 🗊
- Menu: Tools, Sketch Tools, Offset Entities

30 Offset entities.

Hide the Rotor - Cast Iron component. Click **Offset Entities** $\boxed{\ensuremath{\mathbbmath\mathbbms}}$ and set the **Offset Distance** to **2mm**. Select the edge, click **Reverse** and click \checkmark .

Repeat the procedure for the opposite side.





Using Assemblies

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31 Convert.

Show the Rotor - Cast Iron component. Select the edge as shown and click **Convert Entities** . Click **<**.

Using Assemblies



32 Drag.

Drag the open endpoints out beyond the converted edge.



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33 Trim.

Click Trim Entities **[14]**,

Power trim E. **Click+Drag** across the sections of geometry using the paths shown to trim away excess geometry.

Using Assemblies



34 Sketch fillets.

Add sketch fillets **Radius 1mm** in four places as shown.

Tip: If corners are trimmed to a single endpoint, select the endpoint to add the fillet.

Extrusions In-Context

Extrusions can also be created in-context when external generator is referenced. In this example, the depth of an extrusion is measured as an offset from an existing face.



0

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35 Extrude.

Click Extrude and set the End Condition to Offset from Surface. Set the Offset Distance to 1.5mm. Click in Face/Plane field.





Using Assemblies

36 Select other.

Right-click the face as shown and select **Select Other**. Click the top selection, Face@[Brake Rotor Assembly<1>/Rotor - Cast Iron<1>].



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Tip: The top selection at the cursor is not listed. Why? It is assumed that if you wanted to select that face you would have selected it directly.

37 Offset distance.

Check the direction and click 🛩



Edit Assembly Mode

Edit Assembly Mode is the opposite toggle of **Edit Part Mode** and is the default state of the assembly where you can add components and mates. It is triggered by exiting the editing of a part in the assembly or opening of an assembly file.



1 Edit assembly.

Click Edit Component 💿 or the confirmation corner 🥦 to edit the assembly.

This returns you to edit assembly mode and all the colors return to their original settings.



Edit Assembly Mode

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Using Assemblies

Working with Virtual Parts

The virtual part has been stored inside the assembly since it was created. Now that it nearly complete, we will save it externally and make it a real part.

Renaming a Virtual Part

2 Rename.

Right-click the [Part1^Brake&Wheel] component and select **Rename Part**. Type the name Brake Pad.

Tip: Although the part has been renamed, it remains a virtual part.

Saving a Virtual Part as an External Part

The virtual part requires a location on disk to store the new *.sldprt file.

3 Save as external file.

Right-click the part and select **Save Part (in External File).** The file Brake Pad.sldprt is added to the assembly folder.

Click Same As Assembly and OK.

File Marine	Path	
-		

Tip: The part name appears without the square brackets.

Adding Component Instances and Mating

Components can be added to the assembly in several ways. If there is already an instance of that component in the assembly, additional instances can be added using control+drag and drop.

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4 Copy an instance.

Click and **Control+drag** the Brake Pad<1> component. Drop the component outside the Brake Caliper as shown.



5 Hide component.

Click the Rotor - Cast Iron component and click Hide Components .



Adding Component Instances and Mating
SolidWorks

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6 First mate.

Press Shift+Up Arrow key. Click Insert, Mate $\$ and select the faces as shown. Click Coincident $\$ and Anti-Aligned $\$. Click $\$.



7 Second mate.

Press the **Down Arrow** key. Select the faces as shown, click **Coincident** \bowtie and click \checkmark .



Adding Component Instances and Mating

Engineering Design and Technology Series

8 Third mate.

Hide the Brake Caliper component. Select the faces as shown, click **Coincident** \swarrow and click \checkmark .



Using Assemblies

9 Show.

Show the Brake Caliper component.



10 Save.

Click **File**, **Save I** to save the assembly and parts.

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Viewing the Mates of a Component

The mates used to constrain a component can be listed and viewed using **View Mates**. It is a useful tool in understanding how components are used in the assembly.

Note: The arrow symbol \perp indicates a path to ground. Mates marked like this are the ones that hold the component in place.

Where to Find It

■ Shortcut Menu: Right-click a component and click View Mates

11 View mates.

Click the Brake Pad<2> and View Mates .



Note: Click the "**x**" to close the dialog

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In-Context Part Editing

Any component part can be edited in the assembly whether it was created in the context of the assembly or not. Toggling back to edit part mode uses the same command; **Edit Part**.

12 Edit part.

Click Brake Pad<1> and Edit Part [5].

13 New sketch.

Click the face and click **Insert Sketch** $\boxed{\mathbb{C}}$. A new sketch has been created on the face.



14 Convert edge.

Click the circular edge and click **Convert Entities □**. Click **✓**. Exit the sketch.

Adding Component Instances and Mating

SolidWorks Engineering Design and Technology Series

Using Assemblies

Opening a Part from an Assembly

In this example, we will create a slot is to match up with the hole in the Brake Pad component. The slot will be constructed from an existing hole and makes for easier fit up.



1 Open Brake Pad.

Click on the Brake Pad<2> in the FeatureManager design tree and select **Open** \ge .

2 Edit a sketch.

Right-click on Sketch3-> in the FeatureManager design tree and select **Edit Sketch 2**.



3 Construction.

Click the circle and click **For construction** to make it dashed.

Where to Find It

- CommandManager: Sketch > Straight Slot , Centerpoint Straight Slot
- Menu: Tools, Sketch Entities, Centerpoint Straight Slot
- 4 Slot.

Click **Tools, Sketch Entities, Centerpoint Straight Slot** \bigcirc and place the cursor at the center of the circle. Drag the cursor horizontally and click to create the centerline. Drag vertically and click to create the height. Click \checkmark .



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5 Relations.

Click the circle and a horizontal line of the slot. Add a **Tangent** S relation.



6 Dimension and cut.

Add a **2mm** dimension as shown to fully define the sketch. Create a cut using a **Through All** end condition.



Using Assemblies

Multi-body Materials

In order to have different materials in the same part there must be multiple solid bodies (multi-bodies) within that part.

This part is currently made up of three features; two bosses and one cut feature. They are listed in their order of creation. There is just one solid body because by default new boss features are merged to the current body. The part will be edited to create multi-bodies.



Boss-Extrude1->



Boss-Extrude2->

Cut-Extrude1->

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Reordering Features

Features can be reordered in the Feature Manager design tree using drag and drop. The rule to remember is that you cannot reorder a child feature before the parent feature. So how can you determine the parent/child relationships?

Parent/Child Relationships

The Parent/Child tool is used to determine the parents and children of any feature. In this case it will be used to determine the limits of where a feature can be reordered.

7 Parent/child.

Right-click the Cut-Extrude1 feature and select Parent/Child. The dialog tells you that the Boss-Extrude1 and Sketch4 features are the parents of the selected feature. This also means that the Boss-Extrude2 feature is *not*. This means that the child can be moved to a



position between the boss features. Click Close.

Note: The Sketch4 feature is embedded in the Cut-Extrude1 feature.

8 Reorder.

Drag the Cut-Extrude1 feature and drop it on the Boss-Extrude1 feature. This places it between the boss features.



9 Folder.

Right-click the first feature Boss-Extrude1, **Control+click** the second feature Cut-Extrude1 and select **Add to New Folder**. Name the folder Backing.



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10 Edit feature.

Click Boss-Extrude2 and Edit Feature $\textcircled{1}{8}$. Clear Merge result and click \checkmark . There are now two solid bodies named Cut-Extrude1 and Boss-Extrude2. Rename them to Plate and Pad as shown.

S Brake Pad (Default< <default>_C</default>	S Brake Pad (Default< <default> D</default>
Con Sensors	i Sensors
Annotations	Annotations
- (D) Sold Bodies(2)	= D Sold Bodies(2)
Cut-Extrude1	Plate-
Boss-Extrude2	🗖 Pað
3 ≦ Material <not specified=""></not>	3 ≝ Material <not specified=""></not>
S Front Plane	🛞 I ront Plane
🛇 Top Plane	🛞 Top Plane
🛇 Right Plane	🕄 Richt Plane
I, Origin	L. Origin
🗉 🔄 Backing	🗃 🛄 Backing
🕒 🔞 Boss-Extrude2 ->	🛞 🔞 Boss-Extrude2 ->

Tip: The default names were taken from the last feature that was applied to the body.

Materials

Materials can be added to the entire part or to selected solid bodies within the part. In this case we will take advantage of the multi-body format to assign different materials to each body.

Where to Find It

- Shortcut Menu: Right-click a component or body and click Edit Material
- 11 Material for the Plate.

Right-click the body Plate and select Material, Edit Material. Under Steel, select 1023 Carbon Steel Sheet (SS). Click Apply and Close.

12 Material for the pad body.

Right-click the body Boss-Extrude2 and select Material. Under Other Non-metals, select Ceramic Porelain. Click Apply and Close.

Note: Custom materials and custom material libraries can be created.



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13 Open assembly.

Press Control+Tab and move the cursor to the assembly.



This message appears:

Models contained within the assembly have changed. Would you like to rebuild the assembly now? Click **Yes**.

14 Edit assembly.

Click Edit Component 🛐. Hide the Brake Caliper and Rotor - Cast Iron components.



15 Display state.

Create a new display state and rename to ALL. Right-click in the graphics area and click **Show Hidden Components**. Window-select all the components and click **Exit Show Hidden**.



Materials

SolidWorks Engineering Design and Technology Series

Pack and Go

Pack and Go is utility that can be used to *copy* all files used by the assembly into a new folder or zip file, consolidating the file set into one location.



Work Flow

The work flow using Pack and Go creates multiple backups, using the last backup to start the next work session.

Creating a Zip File

Creating a zip file is a good way to consolidate the files and generate a backup in one step. The zip file can be used to start the next session and then be stored away.

Pack and Go

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1 Pack and go.

Click File, Pack and Go and click Save to Zip file. Using Browse, set the location to a temporary folder, name the file Backup_1.zip and click Save.

Include simulation results	Include custom Include detroit	decals, appearances and scenes decals, appearances and scenes		 Nested view Elst view
V Name	In Folder	Save To Name	Site	Туре
BrakeWheel SLDASM	C'Temp\/ SAE	Brake&Wheel.SLDASM	1237	SolidWorks Assemb
Brake Caliper:SLDPRT	C'(Temp\/FSAE)	Brake Caliper.SLDPRT	2077	SolidWorks Part Do
B Brake Pad.sklprt	C'\Temp\/FSAE\	Brake Pad sldprt	214 K	SolidWorks Part Do
BrakeWheetSLDASM	C'(Temp\/FSAE)	Brake&Wheel.SLDASM	1237	SolidWorks Assemb
Brake Rotor Assembly.SLDASM	C:\Temp\/FSAE\	Brake Rotor Assembly SLDASM	550 K	SolidWorks Assemb
Rotor - Cast Iron, SLDPRT	C:\Temp\/FSAE\	Rotor - Cast Iron.SLDPRT	792 K	SolidWorks Part Do
Rotor Hat SLDPRT	C:\Temp\/FSAE\	Rotor Hat.SLDPRT	963 K	SolidWorks Part Do
Rotor Pin.SLDPRT	Cotemn/FSAR	Rotor Pin SUDPRT	274 8	SolidWorks Part Do
Assemblies 5 🧐 Parts 16 🔯 I	Drawings 0	👷 Other 0 📴 Total 21	C	Select / Replace
Save to folder	(Brakes)wheel	hour.		
	-	Bioyve		
K Save to Zip file: C:\Temp\Backup_1 ap				
Norman Save to Zip file: C\Temp\Backup_1 zip	Add suffi	0		

2 Unzip.

At the start of the next session, unzip the file to a new folder and begin working. More files may be added from external drives or different folders.

3 Repeat the process.

At the start of the next session, unzip to a new folder and begin working. Repeat the process each time to keep all the files together.

Adding to File Names

If you want to rename the files with each new backup, the **Add prefix** and **Add suffix** options can be used.

For example, the file name Brake Caliper could become Brake Caliper_2 or 2-Hub Assembly with a suffix or prefix added.

Note: Using a prefix or suffix changes the name. It is not the same as using the SolidWorks Data Management product.

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4 Appending the names.

Use the same settings as in the previous step 1 but click Add suffix and type $_1$ in the box. Click Save.

	Include	simulation results	1	 Neste 	d view O Flat view
Name Name		In Folder	Save To Name	Size	Туре
C Stake Mireel S	ILD/ASM	C:\Temp\Brake	BrakesWheel_2.SLDASM	1260 K	SolidWorks Assembly Doc
🔽 🤏 Brake Calip	er SLOPRT	C:\Temp\Brake	Brake Calper_2.SLOPRT	1080 K	SolidWorks Part Document
🔽 🛛 👒 Brake Pa	al sidori	C:\Temp\Brake	Brake Pad_2.sklpit	255 kB	SolidWorks Part Document
🔽 🧐 Braket	ed SLDASM	C:\Temp\Brake	BrakesWheel_2.SLDASM	1260 K	SolidWorks Assembly Doc
Brake Ro	ker Assenbly SLDASM	C:\Temp\Brake	Brake Rotor Assembly_2.SLDASM	810 KB	SolidWorks Assembly Doc
Rolar C	Jack Iron SLDPRT	C:\Temp\Brake	Rotor - Cast Iron_2.SLDPRT	766 KB	SolidWorks Part Document
Rolor Ha	I SLOPRT	C:\Temp\Brake	Rotor Hat_2.SLDPRT	563 KB	SolidWorks Part Document
Rolor Pi	SLDPRT	CiTenniBrake	Robor Pin 7 S DPRT	273 KR	Selectionics, Parl Comment
Assemblies: 5	S Parts 17	Drawings 0	Binwen	1	Select / Replace
	CATampi Baskup 2 sin		Browse		
Save to Zip file:	erviceubapacenter st				

- **Tip:** Virtual parts that have not been saved as external files will appear greyed in the list with <internal to assembly> as the folder. They are stored within the assembly that was active when they were created.
 - 5 Save and close all files.

SolidWorks Engineering Design and Technology Series

Lesson 3 Creating a Weldment

When you complete this lesson, you will be able to:

- Creating weldment parts;
- Using 3D sketches;
- Working with sub-assemblies;
- Creating custom profiles;
- Adding structural members;
- Trimming structural members;
- Editing structural members.

Engineering Design and Technology Series

Creating a Weldment

Creating Weldment Parts

The Frame will be built as a in-context **Weldment** using references from existing assemblies. The weldment process includes creating sketches, adding structural members and trimming the structural members.

Planes and Sketches

Planes and sketches, both 2D and 3D, are used to define the locations of the structural members in the weldment.





Structural Members

The structural members, square and round tubes in this example, are added to the lines and arcs of the sketches. After they are added they are trimmed to fit.





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Weldments

A **Weldment** is a multi-body part composed of structural members. The centerlines of the structural members are sketched and profiles are selected from a library and are applied to the sketches.

Tip: Most of the work to create a weldment is invested in the creation of the weldment sketches.

Creating a Weldment

Creating a typical weldment involves several steps to create and trim the weldment members.

For example, this weldment is made up of five structural members; two of the longer length and three of the shorter length.

The following chart defines the basic steps in creating a weldment part.



Sketch	Create a new sketch that defines the centerlines of the weldment profiles. The sketch can be a 2D or 3D sketch. Close the sketch.	
Create Structural Members	Click Structural Member and select geometry. Select a profile from the library for use with the geometry.	
Trim and Extend Structural Members	Click Trim/Extend P and trim or extend the length of structural members to planes or other structural members.	Booy 1, 1; Ivep

Some Important Points about Weldments

Here are some important points to remember about weldments:

- Weldment parts are multi-body parts. Each structural member is an individual solid body.
- The profiles used are selected from a library. You must create your own profiles and library folders if the standard profiles are not what you need.
- Weldments can make use of symmetry. Structural members can be patterned using Mirror with the Bodies to Pattern option.
- A Cut-List folder is created to store information about the structural members and their lengths of the structural members. They can also be sorted by equal lengths.
- A Weldment Cut List table can be generated in a drawing of the weldment.
- The drawing can also have balloons like a BOM.

Some Details About Weldment Sketches

Weldment sketches are used to define the centerlines of beams used in the weldment. Continuous beams should be created using single pieces of geometry. If they are not, smaller separate pieces will be created.

Sketches used for weldments may be different than sketches used for other features. For example, the sketch shown here would not be useful for a boss extrude or revolve.



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1 Open Frame&Suspension.

Click **File**, **Open** and select the assembly Frame&Suspension from the Lesson 3\Frame&Suspension folder.



2 Show sketches.

Open any instance of the A-Arm Spherical Ball component.

Right-click Sketch1 and select **Show**.

Return to the assembly.



Note: The A-Arm Spherical Ball components can be rotated for a better view of the sketch.

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Using Different Planes and Sketches

In the process of adding a new part in-context, planar face or plane must be selected. This plane is used as a sketch plane and a new sketch is opened. In fact, several things happen at once when the plane is selected:

- A new in-context part is created.
- The new part is being edited.
- A new sketch is created on the selected plane.
- The new sketch is open.
- The selected plane becomes the Front plane of the new part.

The new sketch may not be sketch you want. Although it is a requirement to select something, you do not have to use the sketch. You can just exit the sketch and create the plane or sketch that you need.

3 New part.

Click **Insert, Component, New Part** and select the plane Ground. A new sketch, in the new part, is automatically started on the selected plane. See "Creating a New Part" on page 16 for more information.



Note: The Ground plane is selected, but in theory any plane or planar face could have been selected because none of the existing planes or planar faces is sufficient.

4 Exit sketch.

Right-click in the graphics area and select **Exit Sketch [2]** to leave the current sketch. You are still editing the new part.

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Creating a Weldment

5 Reference plane.

Click Insert, Reference Geometry, Plane 🔯.

Select the plane Front plane from the active part [Part1^Frame&Suspension]<1> and Parallel.

Select the centerpoint of an arc from A-Arm Spherical Ball (connected to the A-arm - Upper assembly) as shown. Click and rename the plane Upper_Arm.





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6 Lower plane.

Using the same procedure as the previous step, create a new plane using the plane Front and a centerpoint of an arc from A-Arm Spherical Ball (connected to

the A-arm - Upper assembly) as shown. Click *and rename the plane Lower_Arm.*



Creating a Weldment



Using 2D Sketches

A series of 2D sketches will be created to form the basic shape of the frame.

3D sketches will be used to fill in much of the bracing and support structure after the basic shapes are complete.



Note: Many of the 2D sketches, like this one, will require the creation of new planes.

7 New sketch.

Right-click the plane Lower_Arm and Sketch **[2]**.

Weldments

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Engineering Design and Technology Series

Creating a Weldment

8 Rectangle.

Click **Rectangle** and sketch as shown.



9 Reference to existing geometry.

Add dimensions between the centerpoint of the arc and the edge of the rectangle as shown. Set the values to **35mm** and **30mm**. Selection of A-Arm Spherical Ball geometry for the dimensions creates external references to that component.



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10 Centering geometry.

Sketch a centerline connecting the midpoint of the line and the origin as shown. Add a **Horizontal** — relation.



11 Added line and dimension.

Add the line and dimensions **710mm** and **430mm** as shown.



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12 Exit sketch.

Right-click in the graphics area and select **Exit Sketch [2]** to leave the current sketch. Rename the sketch Main.

13 New plane.

Control+drag the plane Lower_Arm downwards. In the **Plane** dialog set the offset to **115mm**. Rename the plane Lower_Frame.



14 Hide components.

Hide the Steering Wheel and Steering Rack and Shaft Assembly assemblies.

15 New sketch.

Create a new sketch on the plane Lower_Frame. Sketch the lines and a centerline connecting the midpoints of both lines as shown.



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16 Symmetry.

Add a **Horizontal** — relation to the centerline. The two angled lines are now symmetric to each other.

1 1

Creating a Weldment



17 Relation to inactive sketch.

Select the centerline of the current sketch, **control+select** the centerline of the inactive sketch and add a **Collinear** \swarrow relation.



Creating a Weldment

Engineering Design and Technology Series

18 Complete sketch.

Add the dimensions as shown. Exit the sketch. Rename the sketch Lower_Frame _sketch.



Renaming and Opening a Virtual Part

It is easier to work on the part outside the assembly provided that the assembly geometry is not being referenced. The part can be opened even though it is a virtual part.

1 Rename the virtual part.

Right-click the virtual part Part1^Frame&Suspension in the FeatureManager design tree and select **Rename Part**. Type Frame.

2 Open the virtual part.

Right-click the virtual part Frame[^]Frame&Suspension and select **Open Part** *▶*.

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Why is the View Orientation Different?

When a new part is created in-context, the initial plane or planar face selected defines the position and orientation of the Front plane of the new part. This may cause the new part to an unexpected orientation, different than that of the assembly. Saving a view state is a great solution.

3 Orientation.

Click **View**, **Modify**, **Rotate** and rotate the view orientation so that it looks similar to that of the assembly.



4 Save a view state.

In order to save the view orientation and zoom, save a view state. Press the **Spacebar** and click the **New View S** icon on the **Orientation** dialog.

anned Anna	12
ew name:	OK
150	Cancel
	Help

Type the name ISO and click **OK**.

The view name ISO will be added to the list and can recalled by double-clicking.

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5 Plane.

Create a new plane using Right and **Parallel** for the **First Reference** and the endpoint of the Main sketch as the **Second Reference**. Rename the plane Bulkhead1.

6 Sketch geometry.

Create a new sketch on the plane Bulkhead1. Add lines to the existing endpoints of the Main sketch. Select the centerline, control+select the endpoints and add a **Symmetric** *s* relation.



Note: Endpoints or geometry can be selected for the addition of a symmetry relation.

7 Dimensions and fillets.

Add the three linear dimensions (**440mm**, **215mm** and **420mm**) as shown.

Click Sketch Fillet \square and set the Fillet Radius to 100mm. Select four corners as shown. Click \checkmark .

Tip: If this message appears: At least one segment being filleted has a midpoint or equal length relation. Geometry may have to move to satisfy this relation when the fillet is created. Do you want to continue? Click **Yes**.

Close the sketch and rename it to Bulkhead1_Sketch.



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Additional Planes and 2D Sketches

A series of small 2D planes and sketches are needed to define the shape of the frame.



8 Bulkhead2 and Bulkhead3 planes and sketches.

Create planes parallel to Right through the endpoints in the Main sketch and create the following sketches.



Bulkhead2_sketch

Bulkhead3_sketch

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9 Seat position and angle.

Create a new plane using Right and **Parallel** for the **First Reference** and the endpoint of the Main sketch as the **Second Reference**. Rename to Seat_Position.

Using Seat_Position and the sketch line as shown, create a new plane. Click **Angle** and use **10** degrees. Rename it to Seat_Angle.

Create the sketch and rename it to Seat_Angle_sketch.



Tip: Add the 100mm sketch fillets before the 90mm and 500mm dimensions.

Sketching with Pierce Relations

So far, we have connected geometry to endpoints and midpoints. But what if the structural members are connected to an arc or elbow shape? This method utilizes a lesser known relation called **Pierce**. The pierce relation positions an endpoint where the sketch plane intersects (pieces) the selected geometry.

- **Note:** Sometimes it is necessary to create a bracing member in order to create bulkheads and other sketches. In this case the position of the bracing member is required to sketch the position of a bulkhead.
 - 10 Edit sketch.

Edit the sketch Bulkhead1_Sketch. Click **Point** and add a point near the arc. Add a **Midpoint** relation between the point and the arc as shown. Exit the sketch.



11 Plane through point.

Create a new plane using Front and **Parallel** for the **First Reference** and the point as the **Second Reference**. Rename to Through_Point.



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12 Pierce.

Create a new sketch on the Through_Point plane. Sketch a line from the point towards the arc as shown. Select the endpoint and the arc and add a **Pierce** relation. Rename to Through_Point_Sketch.



13 Planes.

Create a new plane offset **550mm** from the Seat_Position plane. Rename to Engine1. Offset **300mm** from the Engine1 to create Engine2.



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14 Converting a single sketch entity.

Create a new sketch on the Engine 1 plane. Click **Convert Entities** $\boxed{1}$ and select the centerline in the Seat_Angle_sketch as shown. Click \checkmark . Click the line and select **Construction Geometry** $\boxed{1}$. Add the lines to the centerline as shown.



15 Mirror in the sketch.

Click **Tools, Sketch Tools, Mirror** $\boxed{\square}$ and select the three sketched lines. Right-click and select the centerline and click \checkmark .





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16 Collinear with a plane.

Add a **Collinear** relation between the Through_Point plane and the top line and add the dimensions as shown. Exit the sketch and rename it Engine1_sketch.



17 Converting an entire sketch.

Create a new sketch on the Engine2 plane.

Click **Convert Entities** and select the sketch Engine 1_sketch using the flyout FeatureManager design tree

as shown. Click 🗹.

Close the sketch and rename it Engine2_sketch.







Weldments

Creating a Weldment

SolidWorks Engineering Design and Technology Series

Adding Bracing to the Frame

Bracing is added between the existing sketches using more sketches. The best practice is to use the type that is most suitable in the situation, 2D or 3D. For more information regarding 3D sketches, see "Using 3D Sketches" on page 65.

Tip: All of the previous sketches could have been created as 3D sketches.

18 Sketch.

Create a new sketch on the Through_Point plane. Add the two centerlines, three lines and two dimensions as shown. Rename to Engine_Mount. Exit the sketch in an under defined state.



Note: This sketch is another example of a valid weldment sketch that would be unsuitable for a standard extrusion. For more information, see "Some Details About Weldment Sketches" on page 46.

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Using 3D Sketches

3D sketches are very useful in creating sketch geometry that does not lie on an existing or easily definable plane. Bracing lines that cross through 3D space between existing endpoints are best created using 3D sketches.

Where to Find It

- CommandManager: Sketch > Sketch 🛃 > 3D Sketch 🐉
- Menu: Insert, 3D Sketch

Sketching in the X, Y and Z Directions

The **Space Handles** that appear in a 3D sketch when creating lines can be used to sketch along the **X**, **Y** and **Z** directions. The direction is shown at the cursor.



Note: The space handles can be ignored to sketch between endpoints in 3D space.

How do you know if you are in a 3D sketch?

Look at the **Status Bar** near the bottom right corner of the window. It will say Editing 3DSketch1 for a 3D sketch or Editing Sketch6 for a 2D sketch.

19 3D sketch.

Click **3D Sketch** and add a line. Start the line at the lower endpoint and end it on the angled line as shown. A **Coincident** relation is added in the process.



Using 3D Sketches

SolidWorks Engineering Design and Technology Series

3D Sketch Dimensions

The **Tab** key can be pressed while creating a dimension to switch between the true distance (absolute) and distances along the X, Y and Z axes.



Note: Once created, you can use **Edit Sketch [2]** to edit either a 2D or 3D sketch.
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20 Dimension.

Add an absolute dimension of **350mm** that defines the true length of the line. Exit the sketch.



21 Assembly.

Press **Ctrl+Tab** and click the assembly. At the message: Models contained within the assembly have changed. Would you like to rebuild the assembly now? Click **Yes**.

22 Edit assembly.

Edit the assembly by right-clicking in the graphics area and selecting **Edit Assembly: FrameSuspension**.

23 Save.

Save the assembly and click **Save All**. At the message: This assembly contains unsaved virtual components which must be saved click **Save internally (inside the assembly)** click **OK**.

24 Show display state.

There are many components that have been hidden. They will be shown now.

Click the ConfigurationManager tab R in the FeatureManager design tree and double-click the display state Full_Display State-1. Click the FeatureManager design tree tab 8.

25 Hide planes.

If there are planes visible, click View, Planes to turn of the display.

Using 3D Sketches

SolidWorks Engineering Design and Technology Series

Placing a Component Using Sketch Geometry

The geometry of the frame, an in-context virtual part, can be used to create mates as well as place and orient components.



26 Insert component and mate.

Click Insert, Component, Existing Part/Assembly 2 and select the ENGINE part from the Lesson 3 folder.

Mate the Right plane of the ENGINE and the Centerline plane of the assembly using a **Coincident** mate.



Note: Use the Aligned and Anti-Aligned options to get the mate alignment as shown.

SolidWorks

Engineering Design and Technology Series

27 Concentric mate.

Select the cylindrical face of the engine and the sketch line as shown. Add a **Concentric** mate between them.



28 Rotate.

Rotate the under defined component as shown.



SolidWorks Engineering Design and Technology Series

Making Changes

With the addition of the ENGINE, we can see that it appears to be too close to the Engine1_sketch (inside right box). One solution may be to pull the attachment point back by shortening the length of a line in the Engine_Mount sketch (inside left box).



29 Change dimension value.

Return to the part by pressing **Ctrl+Tab** and clicking the part.

Double-click the Engine_Mount sketch in the FeatureManager design tree and change the dimension to **50mm** as shown. Click **Rebuild 1**.



SolidWorks

Engineering Design and Technology Series

30 Return to the assembly.

Press **Ctrl+Tab** and click the assembly. With the change in the sketch geometry, the engine has been pulled back to a better position within the frame as shown.



Working With Sub-Assemblies

Large assemblies often have multiple sub-assemblies within them. This is good practice and helps make the assembly easier to work with, shortening the FeatureManager design tree and breaking the components into logical groups.

This section will show how to open a sub-assembly from the main assembly and set Component Properties.



Opening a Sub-assembly from the Assembly

Sub-assembly components can be opened directly from the assembly FeaureManager design tree like part components.

Tip: Right-clicking a component part in the sub-assembly and trying to open it will open the part only.

31 Open sub-assembly.

Right-click the sub-assembly Rear Axle Assembly from the FeatureManager design tree and select **Open assembly 2**.

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Rigid vs. Flexible Sub-Assemblies

All sub-assemblies have the option of being solved as **Rigid** or **Flexible**.

Where to Find It

Shortcut Menu: Right-click a sub-assembly and click Component
Properties 2. Change the Solve as option to Rigid or Flexible.

Rigid

Rigid **(s)** treats the entire sub-assembly as a single rigid component when used as a sub-assembly. Rigid is the default condition for all sub-assemblies.

Flexible

Flexible <u>allows</u> the components that can move or rotate in the assembly to move or rotate when used as a sub-assembly.

1 Test component.

The assembly is designed so that the Sprocket is free to move along the axis CL Rear Axle. Move it a small amount to test it.

Close the sub-assembly without changes and return to the main assembly. Click **Discard**.



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2 Warning.

To line up the Sprocket with the engine output, select the faces indicated and try to add a **Coincident** mate. A warning saying that the coincident mate cannot be added will appear. Click \ge .



Why Does the Warning Appear?

The sub-assembly is solved as the default rigid condition which does not allow components in the sub-assembly to move independently.

3 Component properties. Right-click the Rear Axle Assembly and select Component

Properties 😭. Under Solve as, click Flexible and OK. Using the same procedure as in step 2, add the Coincident mate.



Working With Sub-Assemblies

SolidWorks

Engineering Design and Technology Series

Bracing Sketches

There are several more planes and sketches required to complete the frame geometry.

- **4 Open part.** Open the virtual part frame.
- 5 Pierce relation sketches.

Create a new plane through endpoint indicated and **Parallel** to Front plane. Use the plane to create two new sketches.

In the each sketch, use one **Coincident** and one **Pierce** relation.



Note: The pierce relation in the first sketch uses an arc, the second uses a line.

Bracing Sketches

SolidWorks Engineering Design and Technology Series

Remaining Sketches

The remaining sketches are all 3D sketches and will be presented with a brief description and an image as shown below.



SolidWorks Engineering Design and Technology Series

Creating a Weldment



Why is the Bracing One Sided?

The resulting structural members will be mirrored across a plane to create the bracing on the opposite side. For more information, see "Mirroring Structural Members" on page 90.



Bracing Sketches

SolidWorks Engineering Design and Technology Series

Weldment Structural Members

Weldments use standard profile shapes that are applied to sketch geometry to create structural members. Each structural member is a separate sold body in the multibody part.

Weldments also track the quantities and lengths of the structural members using a weldment cut-list.



A Quick Look at the Standard Profiles

The weldment profiles are divided into two **Standard** folders: ansi inch and iso. Profiles must be library features and be present in the folders to be used. The standard profiles use these shapes:

		0		I
angle iron	c channel	pipe	rectangular or square tube	s section or sb beam

Weldment Cut List

The Weldment Cut List tracks the structural member types and lengths.



SolidWorks Engineering Design and Technology Series

Creating Custom Profiles

We need two profiles: a round (pipe) tube and a square tube each with an outside measurement of **25.4mm**. This size is different than the standard sizes, so they will need to be created and placed in the proper folders.



Creating a Weldment

- **Note:** The weldment profiles are set to specific folders by the option **Tools**, **Options**, **File Locations** and **Weldment Profiles**.
 - 1 Open a new part.

Open a new part with mm units.

2 Sketch.

Create a new sketch on the Front plane and sketch a **25.4mm** circle and dimension as shown.

Add an offset of **2.4mm** as shown and exit the sketch.



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3 Save as library feature. Select the sketch Sketch 1. Click File, Save As.

Select the **Save as type Lib Feat Part (*.sldlfp)**, type the file name 25.4mm Dia, and click **Save**.

For **Save in**, navigate to the default weldment library folder located in the C:\Program Files\SolidWorks Corp\lang\english\ weldment profiles folder.

Note: If you receive a message that you are not allowed to save in that folder, try saving the file in a local folder and moving the file to that folder. Also, the full path used in this lesson is based upon the default installation on SolidWorks onto the C drive. If SolidWorks has not been installed using the default settings, then the path will require some adjustment.

Save As			2 🔀
- A	Save in: 🤤	weldsent profiles	1 G # P II-
My Recent Documents Desktop	ansinch iso		
My Documents			
*	File name	25 4mm Dia	Save -
Foylanca	Save as type:	Lib Feat Part (".sldtp)	Cancel
	Description		
My Network. Places		Save as copy	References
1			

Library Feature Folders and Icons

The library feature is a different file type and uses different icons for the top level component and the sketch.

4 Close library feature. Close the library feature part.



Weldment Structural Members

Engineering Design and Technology Series

5 Create new Folders.

Adding the weldment profile to the weldment profiles folder is not enough to make use of it.

Create a new folder FSAE in the

<Install Directory>\Program Files\SolidWorks Corp \lang\english\weldment profiles folder. Open that folder and create two folders named Square Tubes and Round Tubes.

Move the 25.4mm Dia.SLDLFP file into the Round Tubes folder.



6 Move existing file.

Move the existing 25mm Sides.SLDLFP file into the Square Tubes folder.

Note: The file can be found in the Lesson 3 folder.

Why Do Some Profiles Contain Points?

Profiles formed using lines typically contain points at the corners like this tube. They are useful in moving the profile to a position other than the centerline.



Weidment

Main ->

♦ Upper_Arm → ♦ Lower_Arm →

SolidWorks Engineering Design and Technology Series

Adding Structural Members

Adding the structural members is the simplest part of the process. Selecting the existing sketch geometry gives the structural members position and length.

It is helpful to keep all the related features together in the FeatureManager design tree for future editing. In order to do this, it is best to rollback. See "Using Folders" on page 88 for more information.

- 1. Move the rollback bar to a position after the sketch feature that you want to use for creating structural members.
- 2. Select the geometry of inactive sketches to define the centerlines of the structural members.
- 3. Add only structural members of the same type in the same feature.
- 1 Rollback.

In the Frame part, right-click the Bulkhead2 plane and select **Rollback**

This will place the Structural Member feature just after the Bulkhead1_Sketch sketch, near the related plane.

2 Structural member.

Click Insert, Weldments, Structural Member i and the following options:

Standard FSAE, Type Round Tubes and Size 25.4mm Dia.

Select all the lines and arcs as shown and click \checkmark .





Rename the structural member feature to Structural Member_Bulkhead1.

Adding Structural Members

SolidWorks Engineering Design and Technology Series

Corner Treatments

The **Corner Treatments End Miter**, **End Butt1 c** or **End Butt2 r** are used to determine the trimming of the members meeting at a corner within the same feature.



Corner treatments are applied to all corners in the Structural Member dialog and individually

Individual corner treatments can be set by clicking the circular corner markers and setting the dialog that appears.



Adding Structural Members

Engineering Design and Technology Series

- **3 Rollback.** Drag the rollback bar to a position after the Bulkhead2 sketch.
- **Note:** The **Weldment** feature is added to the part automatically. This defines the part as a weldment part.

4 Structural member.

Click **Insert, Weldments, Structural Member** and the following options:

Standard FSAE, Type Square Tubes and Size 25mm Sides.

Make sure **Apply corner treatment** and **End Miter F** are clicked. In this example, all corner treatments will remain as end miters.

Select all the lines of the sketch as shown and click \checkmark .





Creating a Weldment

SolidWorks Engineering Design and Technology Series

Using Groups

The **Group** option allows you to select multiple "groups" of edges in the same dialog. The structural members of each group are automatically trimmed against each other; for example the members of Group1 are trimmed by the members of Group2.



Note: The use of groups may limit the selection of corner treatments.

5 Rollback.

Drag the rollback bar to a position after the Main-> sketch.



Engineering Design and Technology Series

6 Structural member.

Click **Insert, Weldments, Structural Member** and the following options:

Standard FSAE, Type Square Tubes and Size 25mm Sides.



These selections are added to for Group1.

😥 Structural Member X fessage Select sketch segments for defining path. You can rotate the profile by a specified angle. Selections 2 Standard: ESAE v Type: Square Tubes Sizer 25mm Sides Groups Group New Group Settings Puth segments: Line1@Main Line3@Main

7 Group.

Click **New Group** and select the three remaining edges for Group2. Click



Creating a Weldment

SolidWorks Engineering Design and Technology Series

Using Trim/Extend

The **Trim/Extend Trim/Extend** tool is used to a trim the end of a structural member against a face/plane or one or more other structural members.

It is helpful to keep the related trim/extend features together with the in the FeatureManager design tree. In order to do this, it is best to rollback first similar to adding structural members. See "Using Folders" on page 88 for more information.

- 1. Move the rollback bar to a position after the Structural Member feature that you want to trim.
- 2. Add one trim/extend feature for each end of the structural member. In the simplest cases, the trim/extend feature may include both ends of the structural member.

Trimming Boundary

The trimming boundary can be either a Face/Plane or other Bodies. Options with other bodies include Simple cut between bodies and Coped cut between bodies.



Tip: The Trim/Extend tool eliminates interferences between solid bodies by shaping the ends and creating the correct fit. Interferences between solid bodies in the multibody part can be checked at the assembly level. For more information, see "Checking Clearances" on page 98.

Engineering Design and Technology Series

8 Trim.

Drag the rollback bar to a position after the Structural Member_Bulkhead1 feature.

Click Trim/Extend **P** and click End Trim **F**.

Select the **Bodies to be Trimmed** and **Trimming Boundary Bodies** as shown.







Creating a Weldment

The Bodies to be Trimmed is shortened using the Trimming Boundary body.



Using Trim/Extend

Engineering Design and Technology Series

Creating a Weldment

Using Folders

Rollback was used in the previous steps to place the trim/extend feature just after the structural member being trimmed. This allows us to group a series of related features into a **Folder**, effectively reducing the length of the FeatureManager design tree.

Note: Features can also be dragged and dropped into the folder but they must remain in the same sequential order.

9 Add folder.

Shift+select the sequence of features as shown. Right-click and select Add to New Folder, naming the folder Bulkhead_1 Folder.



If this procedure is used throughout the FeatureManager design tree it will significantly shorten the length of the tree.



Using Trim/Extend

SolidWorks Engineering Design and Technology Series

Map for Structural Member Types and Trim/Extend

The structural member types are either FSAE, Round Tubes, 25.4mm Dia \bigcirc or Square Tubes, 25mm Sides \square . Create them and add the trimming and folders using the map below or the included eDrawings files (below). Use End Trim is and Trimming Boundary Bodies for all features.



eDrawings File for Member Types and Trim/Extend

Double-click the eDrawings file frame.eprt a or the html file frame.html a. This will open an eDrawing or browser that can be zoomed, scrolled and rotated using the same tools as in SolidWorks.

Use the eDrawing to visually determine the type of structural member used for each sketch and how it is to be trimmed.



Using Trim/Extend

SolidWorks

Engineering Design and Technology Series

Mirroring Structural Members

Mirror can be used to pattern features, faces or bodies across a plane. In this example solid bodies will be mirrored to complete the frame.

10 Mirror bodies.



Note: Mirroring the bodies *after* trimming eliminates the need to trim the mirrored bodies.

SolidWorks Engineering Design and Technology Series

Editing

A big advantage of creating geometry in SolidWorks is the editing capability. You can edit sketches and features to change the design at any time. In this section, several types of changes will be introduced.

Editing the Corner Treatment

The corner treatments used on some of the earlier structural members may seem inadequate after all the bracing is added. In this case, an open end makes the connection with the brace difficult.

11 Zoom in.

Zoom in on the are as shown. This area shows a potential problem where the brace member connects to an open face of a square tube.



12 Edit structural member feature.

It is not necessary to know the name of the feature. Right-click on a face of any structural member of the feature and select **Edit Feature [6]**.



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13 Corner treatment.

Click the corner marker as shown. In the **Corner Treatment** dialog, set the **Trim Order** to **2** and click \checkmark . Repeat the process for the other corner and click \checkmark in the main dialog. This changes the selections to mitered corners.



Note: The potential changes that can be made are limited by how the feature was created. This feature was created using groups so miters are the only other choice.

Editing the Trimming

With the changes to mitered corners, the trimming can now be changed. In this case a planar face will be used instead of a body.

14 Edit trim feature.

Right-click the face of the structural member shown and select **Edit Feature (16)**. Under **Trimming Boundary**, click **Face / Plane** and select the face as shown. Click **(**.



SolidWorks Engineering Design and Technology Series

Editing the Type of Structural Member

The standard, type or size of the structural member can be changed. In this example, structural members that will have plates attached will be changed to square profiles.

15 Edit feature.

Right-click the face of the structural member shown and select Edit Feature 🔞.

Click FSAE, Type Square Tubes and Size 25mm Sides. Click



Using Locate Profile

In creating all of the structural members so far, the profiles have been placed directly at the default center position.

Consider a case where structural members of the same type and size are stacked on top of one another. Is it necessary to sketch a centerline offset by the exact amount?



Locate Profile allows you to place the profile within the sketch using the points built into the profile. For more information see "Why Do Some Profiles Contain Points?" on page 80.

Engineering Design and Technology Series

16 Find the parents.

Right-click a face of the structural member as shown and select **Parent/Child**. The **Parents** and **Children** of the feature are listed. Click **Close**.





Creating a Weldment

17 Show sketch.

Right-click the Lower_Frame_sketch in the dialog and select **Show**.

18 New sketch.

Right-click the top face of the structural member and select **Sketch [**.



Engineering Design and Technology Series

19 Convert.

Click **Convert Entities ()** and select the centerline of the visible sketch.

Drag the endpoints outside the structural members and exit the sketch.

Add relations to span the full width as shown.

Close the sketch.



Creating a Weldment

20 Locate profile.

Click Insert, Weldments, Structural Member and the following options: FSAE, Type Square Tubes and Size 25mm Sides. Select the line and click Locate Profile. Click the point at the lower center position on the profile and click \checkmark .



Creating a Weldment

Engineering Design and Technology Series

21 Measure.

Click Tools, Measure and select the two faces shown.



Repeat the measurement on the opposite side to ensure that the structural member is centered.

22 Save.

Save the part and close it. Return to the assembly and click **Yes** to rebuild the assembly.

SolidWorks Engineering Design and Technology Series

Using Instant 3D

Instant 3D can be used to make dynamic changes to a model by dragging or directly changing dimensions.



Where to Find It

CommandManager: Assembly > Instant 3D S

23 Change dimensions.

Click **Instant 3D** and double-click the feature shown. Right-click the Frame part and select **Edit Part** . Double-click the feature and drag the handles on the dimensions to resize them to the values **80mm** and **300mm** as shown.



Editing

Engineering Design and Technology Series

Double-click the feature and change the dimensions to resize them to **350mm** and **530mm** as shown.



Checking Clearances

What is the minimum clearance between the Sprocket and frame? This question can be using **Clearance Verification S** tool.

24 Edit the Assembly.

Click confirmation corner $\overset{\textcircled{}}{\overset{\textcircled{}}{\overset{}}}$ to edit the assembly.

Engineering Design and Technology Series

Click Tools, Clearance Verification 🚟.

Click Selected Items and select the Sprocket and

25 Clearance.

frame.

Creating a Weldment

Rear Axle Assembly-1@

Frame*Frame&Sus

众

3

~

🚼 Elearance Verificati...

Selected Components

×

RS.



Note: A visual check shows that there is no interference between the Sprocket and frame components, but they can be checked. For more information, see "Interference Detection" on page 102.

SolidWorks

Engineering Design and Technology Series

Fully Defining a Component

The ENGINE component remains under defined, still able to rotate freely. It should be fully defined to prevent inadvertent movement.

26 Open the ENGINE.

Right-click the ENGINE component and select **Open Part 2**.

27 Add new plane.

Click **View, Temporary Axes**. Add a **Parallel** plane between the selected face and the temporary axis through the hole as shown.





Filtering the FeatureManager Design Tree

The FeatureManager design tree filter can be used to filter by name. Typing a name into the filter will result in a list of only the features, sketches, components or mates that use those characters.

28 Filter.

Return to the assembly. Click in the filter at the top of the FeatureManager design tree and type engine.

Right-click Engine1_sketch and select Show.

Click "**x**" to stop the filter.



Engineering Design and Technology Series

29 Mate.

Add a **Coincident** mate between the plane and the line in the sketch.



Editing

Creating a Weldment

SolidWorks

Engineering Design and Technology Series

Interference Detection

Interference Detection is useful in finding interferences or clashes, between static component parts in the assembly. The results show the interfering volumes in red.

Tip: To check interferences between solid bodies in the same multibody part, select the multibody part only and click **Include multibody part interferences**.

30 Interferences.

Click Tools, Interference Detection [3].

By default, the entire assembly is selected for interference detection, so right-click in the **Selected Components** field and select **Clear Selections**.

Select the ENGINE and frame components and click **Calculate**.

A total of four interferences are found combined into two symmetrical sets. Click \checkmark .



✓ X
Selected Components
Frame "Frame&Suspension-2@Fra ENGINE-1@Frame&Suspension
¢ 2
Colculate
Results 🔅
🗉 🙀 Interference 1 - 909.7983
🖹 🙀 Interference2 - 537.5183r
Interference4 - 909.7983r
Component view
Options A
Treat coincidence as interference
Show ignored interferences
Components
Include multibody part interferences
Make interfering parts transparent

31 Edit sketch.

Click in the filter at the top of the FeatureManager design tree and as before type engine. Right-click the under defined (-) Engine_Mount and select **Edit Sketch 2**. Click "**x**" to stop the filter.

Tip: This is a shortcut that puts you in Edit Part and Edit Sketch mode directly.
Engineering Design and Technology Series

32 Drag.

Drag the open endpoint short of the ENGINE component.

Creating a Weldment





33 Dimension.

Add a dimension between the edge of the ENGINE and the endpoint as shown.

34 Edit assembly.

Click **Edit Component** st the sketch and return to edit assembly mode.



Note: The sketch name has changed to Engine_Mount-> denoting that it is fully defined and contains external references.

Creating a Weldment

SolidWorks Engineering Design and Technology Series

Mounting Plates

The frame is currently placed with reference to the suspension points, but it is not attached. To create an actual attachment, mounting tab plates are required.



Creating a Reusable Sketch

The mounting tabs used to attach the suspension to the frame are plates that are rounded on one end. Although all the tabs have a similar shape, the length and ends may vary. For this reason, creating a sketch that can be easily copied and modified is a good solution.

1 New part.

Open a new part with units mm. Create a new sketch on the Front plane.

2 Geometry and dimensions.

Create the geometry and dimensions using these guidelines:

- 1. Create a centerline from the origin on a slight angle.
- 2. Offset the centerline with the bidirectional option (add dimensions off) to create the sides.
- 3. Add a tangent arc to close the upper end.
- 4. Add a line between endpoints to close the lower end.
- 5. Add a circle.
- 6. Add dimensions.



Note: The centerline is on a slight angle (not horizontal or vertical) and under defined to make the placement flexible.

Engineering Design and Technology Series

3 Drag.

Drag the lower corner endpoint. The under defined sketch should rotate but maintain the shape.



Creating a Weldment

4 Save.

Save the part as TAB_Sketch.

Exit the sketch but *do not* close the part.

5 Copy sketch.

Select the Sketch1 feature in the FeatureManager design tree and **Edit**, **Copy**.



Pasting the Sketch

After editing the frame part, the sketch will be pasted onto the face of a structural member.

Tip: A common mistake in this situation is to paste the sketch before editing the part. The sketch can be pasted, but is only useful as an assembly feature. An assembly feature is a feature that exists only at the assembly level and can only be used to create a cut.

6 Edit part.

Return to the assembly, right-click the Frame part and select Edit Part 🛐.

Engineering Design and Technology Series

7 Paste.

Select the face of the structural member as indicated. Click **Edit**, **Paste**.



8 Edit sketch.

Right-click an edge of the sketch and select **Edit Sketch [2]**.



Mounting Plates

Creating a Weldment

Creating a Weldment

SolidWorks

Engineering Design and Technology Series

9 Temporary axes.

Show the temporary axes by clicking **View**, **Temporary Axes**. Select the centerline and the temporary axis and add a **Parallel** relation.



10 Concentric.

Select the circular edge and the circle and add a **Concentric** relation.



Mounting Plates

Engineering Design and Technology Series

11 Extrude.

Extrude the sketch **3mm** as shown.



Note: This part has been marked as a weldment <u>wedden</u>, so the **Merge Result** option in extrude remains cleared.

12 Repeat procedure.

Select the bottom face and create a new sketch. Convert edges from the previous body to create the sketch and extrude the sketch **3mm** as in the previous step.



Repeat for the remaining tabs on the same side and mirror.

Creating a Weldment

Creating a Weldment

SolidWorks

Engineering Design and Technology Series

13 Paste sketch.

Show Engine1_sketch. Select the face, click **Edit**, **Paste** and edit the sketch. Add a **Concentric** relation between the arc and circular edge as shown.



14 Point.

Add a point on the centerline. Add a **Coincident** relation between the centerline and the point. Use the point to add a **Pierce** relation between the point and the inactive sketch line.



Engineering Design and Technology Series

15 Trim and extrude.

Create the front edge of the structural member using **Convert Entity**. Trim the geometry leaving the centerline and point intact as shown.

Extrude the sketch **3mm**.



Creating a Weldment

16 Plane.

Click **Insert, Reference Geometry, Plane**, select the faces and click **Mid Plane**. Rename the plane Centered.



17 Mirror. Mirror the body using the in-context plane.

Creating a Weldment

SolidWorks Engineering Design and Technology Series

Weldment Cut Lists

The Cut list folder list of all structural members in the weldment.





18 Updating the folder.

Open the part Frame. Right-click the Cut list folder and select **Update** to group similar structural members in folders.





SolidWorks Engineering Design and Technology Series

Cut List Properties

The **Cut-List Properties** can be used to view detailed information about the individual structural members including lengths and angles. Right-click a Cut-List-Item folder and select **Properties**.

	Summary	Cut	List Table				
						BOM quantity	
Delete						LENGTH . Ede L	at.
Cut-List-Item1			Property Name	Type		Value / Text Expression	Evaluated Value
Cid-List-Bern2	- 19	1	LENGTH	Text		1.ENGTH@@@Cut-List-Item7@Frame^Frame&S	184.21
Cut-List-tern3		2	ANGLET	Text	-	'ANGLE1@@@Cut-List-Item7@Frame *Frame3S	45.00
Cut-List-Item4	-	3	ANGLE2	Text		*ANGLE2@@@Cut-List-Item7@Frame *Frame&S	45.00
Cut-List-Rem5		4	Description	Text	+		
Cut-List-ben6	1.00	5	MATERIAL	Text		'MATERIAL@@@Cut4.st-ttem7@Frame frame	Material criot specifier
- Cut-List-tern7		6	<type a="" new="" propert<="" td=""><td>1</td><td></td><td></td><td>1</td></type>	1			1
Cut-List-Rem8							
Cus-List-Bern9							
10 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1							
Cut-Let-tern 10							
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Cut-List item 10 Cut-List item 11 Cut-List item 12							
Cut-List-tern10 Cut-List-tern12 Cut-List-tern12 Cut-List-tern13							
Cut-Lut-tem10 Cut-Lut-tem11 Cut-Lut-tem12 Cut-Lut-tem13 Cut-Lut-tem14							
Curtue ten10 Curtue ten11 Curtue ten12 Curtue ten12 Curtue ten13 Curtue ten14 Curtue ten15							
Cur-Lus tem 10 Cur-Lus tem 11 Cur-Lus tem 12 Cur-Lus tem 12 Cur-Lus tem 13 Cur-Lus tem 15 Cur-Lus tem 15							

For example, Cut-List-Item7 the following property values and are listed:

LENGTH = 184.21

ANGLE1 = 45.00

A drawing of the structural member would look like this:



Engineering Design and Technology Series

Save as External File

The virtual part can be saved externally, creating a part file outside of the assembly file.

Note: You cannot create a drawing of the virtual part until it is saved to an external file.

19 Edit the Assembly.

Return to the assembly.

Click confirmation corner 🧠 to edit the assembly.

20 Save external.

Right-click the virtual part and select **Save Part (in External File)**. Click **Same As Assembly** and **OK**.

Note: The name changes slightly. The braces surrounding the name ([,]) are removed.

21 Save and close all files.

SolidWorks Engineering Design and Technology Series

Lesson 4 Using Molds and Surfaces

When you complete this lesson, you will be able to:

- Understand the loft feature;
- Describe the bodies created using the mold tools;
- Use surfaces to add to the mold tools;
- Use symmetry to create additional molds.

Engineering Design and Technology Series

Molds and Surfaces

The Mold Tools commands can be used to create the mold tooling for a part. The mold tooling is created using a set of surface and solid bodies in the same part. Like the weldment, it is a multibody part.

1 Open assembly.

Open the assembly Intake Assembly from the Mold folder.



2 Open assembly.

Open the assembly Frame&Suspension from the

Lesson 3\Frame&Suspension folder. This is the assembly used in the previous lesson.

Add the Intake Assembly assembly to the main assembly. Mate the sub-assembly using one concentric and two coincident mates.



Engineering Design and Technology Series

3 Open runner.

Right-click one of the instances of the part runner and select **Open Part [2]**.

Using Molds and Surfaces



The runner part

The actual runner part is created in two pieces, a right and a left, that are later assembled into one. Each piece requires a mold.



Using Molds and Surfaces

SolidWorks Engineering Design and Technology Series

Features in the Part

The model of the runner part is created using three main boss features: the Bell, the Straight Boot Section and the Center Guide Curve.

The revolve feature named Bell is used to connect the runner components to the plenum.

The extrude feature named Straight Boot Section is used to connect the runner components to the intake connection boot components.

This feature must fit inside the intake connection boot.

The loft feature named Center Guide Curve smoothly connects the revolve and extrude features.



For more information about loft features, see "Anatomy of a Loft" on page 118.

SolidWorks Engineering Design and Technology Series

Using Molds and Surfaces

Anatomy of a Loft

The loft feature is used to transition between different profile shapes. In this example, the starting profile is a slot shape that transitions to a circular shape.

Profiles

Profiles represent the starting, ending and optionally intermediate shapes along the loft. They are closed contour sketches or curves.



Guides

A centerline is a guide used to orient the profiles in transition. A guide curve is used to shape the loft. Guide curves are open contour sketches or curves.



Using Molds and Surfaces

SolidWorks Engineering Design and Technology Series

Creating Mold Tooling

SolidWorks contains a series of mold tools that can be used to analyze and create mold tooling starting with the molded part.

If carbon fiber is used, the cavity solid body is the desired mold. When the carbon fiber is created, the thickness will be applied to the inside maintaining the outside dimensions that fit into the intake connection boot components.

This example assumes carbon fiber is used.

Surface and Solid Bodies

The mold tool procedure creates multiple bodies, including sets of both surfaces and solids, in a single part file. For more information see "Surface and Solid Bodies" on page 119.



Note: A multibody part, like a weldment, is created.





Engineering Design and Technology Series

Description of the Bodies

Each of the resulting solid and surface bodies are created for a specific purpose. They are described below.



Using Molds and Surfaces

SolidWorks Engineering Design and Technology Series

Mold Tools

The mold tools are designed for use with plastic injection molding, but they can be adapted for use with other manufacturing methods. The sequence shown here is typically used. The result is a multibody part with bodies representing the molded part, the core, and the cavity.

Scale 🍓	Scales the molded part to account for shrinkage in some materials. <i>Not used in this example.</i>
Analysis Tools	 Draft Analysis s and Undercut Analysis s are used to see if the part can be removed from the mold. Parting Line Analysis s is used to visualize potential parting lines.
Split Line 📴	Breaks up model faces by adding edges.
Parting Lines 😔	Uses model geometry to define the parting line edges that define the parting surface.
Shut-off Surfaces ৰ	Adds surfaces to close holes in some molded parts. <i>Not used in this example.</i>
Parting Surfaces 💊	Surfaces created from the parting lines to separate the mold cavity from the core.
Surfaces	Many surfaces types can be used to supplement or replace the parting surface depending on the complexity of the model.
Tooling Split 🜌	Creates both the core and cavity solid bodies by splitting a solid.

Note: The options scale and shut-off surfaces are not used in this example.

Tip: Most of these tools are on the **Mold Tools** tab of the CommandManager. Rightclick an existing tab and select **Mold Tools** to display the tab.

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4 Configuration.

Create a new configuration, Mold, to hold all the mold features. Make sure that **Suppress new features and mates** under **Advanced Options** is checked. The new features will be suppressed in the Default configuration.

5 Draft analysis.

Click **Draft Analysis** \aleph and select the Right plane. Set the **Draft Angle** to **1deg** and click \checkmark .







Using Molds and Surfaces

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What Does This Mean?

This means that the angle between the Right plane and the yellow faces is **1deg** or less. It is in fact **0deg** because they are perpendicular.

- **Note:** This small undrafted area is not a problem using carbon fiber, but it would cause problems for other materials.
 - 6 Display off. Click Draft Analysis 🔊.
 - 7 Undercut Analysis.

Click **Undercut Analysis** and select the Right plane. Click **X**. This confirms that the mold should be created in two pieces.







8

Using Molds and Surfaces

Engineering Design and Technology Series

Split line. Click Split Line ⓐ and click Intersection. Select Right plane and the faces that intersect that plane. Click ✓.



9 Parting line.

Click **Parting Lines** Select Right plane and set the **Draft Angle** to **1deg**.

Click Draft Analysis.



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10 Edge selections. Select the starting edge as shown.



Using Molds and Surfaces

Click Select Next Edge (or type "n") and then Add Selected Edge (or type "y").

Continue the selections using the selection tools.



Continue selecting edges to complete the loop of selections as shown. Click



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11 Parting surface.

Click Parting Surfaces → and select Perpendicular to pull. Set the Distance to 30mm and click Knit all surfaces. Click Sharp → and click ✓.



Parting Surface	-
V X	
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Select a parting line feat model edges to create a surface	ture or parting
Mold Parameters	1
C Tangent to surf	ace
🗇 Normal to surfa	ce
Perpendicular to	pull
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Parting Line1	
Parting Surface	\$
30.00mm	\$
1 30.00deg	1
Smoothing:	
Kpr 3.40mm	0
Options	1
V Knit all surfaces	
Show preview	
Manual mode	

Using Molds and Surfaces

Note: The parting surface is not always large enough to be able to split the mold.

Using Surfaces

Surfaces often must be used to fill in or complete the surface geometry started by parting and shut-off surfaces. Here is a partial list of surface tools found on the **Surfaces** tab of the CommandManager.

- Extruded 🐼, Revolved 📠
- Swept **G**, Lofted
- Planar
- Ruled *§*
- Knit 10
- Filled
- Extend 🧟, Trim
- Offset <a>S

Engineering Design and Technology Series

12 Extend edges.

Click **Extend Surface** and select three edges as shown. Click **Distance**, set the value to **30mm** and click \checkmark .

Repeat the process for five edges on the opposite end.



🗩 Su	rlace-Extend1	2
1	¢	
Edge	s/Faces to Extend	á
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End C	ondition:	À
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-	30.00mm	<u>x</u> 9
Exte	nsion Type	-
	⊙Same surface OLinear	

Using Molds and Surfaces

Engineering Design and Technology Series

13 Planes.

Create planes offset **170mm** from the Front and **60mm** from the Top as shown.



14 Trim.

Click **Trim Surface** and select **Standard**. Select the Plane3, **Keep selections** and select inside the plane. Click . Repeat the process for the opposite end.





Using Molds and Surfaces

Engineering Design and Technology Series

15 Sketches.

Click the parting surface and select **Sketch [**]. Using converted entities and lines, create the two sketches as shown. Exit both sketches.



16 Planar surfaces.

Click **Planar Surface** \bowtie and select a sketch. Press **Return** to repeat the command and repeat the procedure for the other planar surface.





Using Molds and Surfaces

Using Molds and Surfaces

Engineering Design and Technology Series

17 Knit surface.

Click Knit Surface $\boxed{100}$ and select the three surfaces. Click Merge entities and click \checkmark .





18 Drag and drop.

Drag and drop the Surface-Knit1 body into the Parting Surface Bodies folder as shown.

😘 runner (Mold <display state-75)<="" th=""><th>👒 rimmi (Mald et splay State 7.5)</th></display>	👒 rimmi (Mald et splay State 7.5)
Annotations	Annotations
(a) Sold Bodies(1)	(a) Sold Bodies(1)
- 😥 Surface Badies(3)	- 😥 Surface Bodies(3)
+ 💓 Cavity Surface Bodies(1)	+ 🥑 Covity Surface Bodies(1)
+ el Core Surface Bodies(1)	+ el Core Staface Bodies(1)
Parking Surface Bridges	😑 🛃 Parting Surface Bodies(1)
Contractions 💝	🚸 Surface-Knit1

Note: The surface must be moved because it was created manually using extend, planar and knit surfaces.

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Using Molds and Surfaces

19 Tooling split sketch.

Click **Mold Tools > Tooling Split** and select the knitted surface as the sketch plane. Create the sketch and dimensions as shown.



20 Tooling split sizing.

Exit the sketch. Set the **Depth In Direction1** and **Depth In Direction2** to **50mm** as shown. Click





Engineering Design and Technology Series

21 Test measurements.

Right-click the first (Parting Line1) solid body in the Solid Bodies folder and select **Isolate**. Click **Tools**, **Measure** and measure the distance between endpoints as shown. The key dimension is **Dist 56.45mm** as shown.



22 Exit isolate.

Click **Exit Isolate** on the **Isolate** dialog.



Using Molds and Surfaces

23 Isolate cavity solid.

Right-click the third solid body in the Solid Bodies folder and select **Isolate**. The key dimension is the again **Dist 56.45mm** as shown.



This shows that the outside faces are used to create the mold. Click **Exit Isolate**.

SolidWorks Engineering Design and Technology Series

Making Use of Symmetry

Individual bodies can be saved into new part files. This creates an external reference from that single body to the part file.

The other half of the mold is symmetric to the current one. It can be created using a mirrored part. The mirrored part is derived from the original part but mirrored using a plane or planar face.

runner -> mirrored part



Note: The if the other mold is *not* symmetrical to the first, a similar procedure can be followed using slightly different selections in the Parting Line feature.

24 Stock feature.

Right-click the cavity solid body in the Solid Bodies folder of the FeatureManager design tree and click **Insert into New Part**. Use the name runner-Cav_1. A stock feature Stock-runner-1 is created.



Making Use of Symmetry

Engineering Design and Technology Series

25 Mirror part.

Select the Right plane and click Insert, Mirror Part. Click Solid bodies, Surface bodies and Planes. Click \checkmark .

Use the name runner-Cav_2.





Using Molds and Surfaces

26 Holes.

Return to the runner part. Select the outer face and click **Insert, Features, Hole, Wizard (a)**. Add **10mm** through all holes placed as shown and select the two mold bodies with the **Feature Scope**. Click **(a)**.





Engineering Design and Technology Series

27 New assembly.

Create a new assembly and add the runner-Cav_1 and runner-Cav_2 parts. The holes have been transferred to these parts using the external references.



28 Save and close all files.

SolidWorks Engineering Design and Technology Series

Lesson 5 Brake Rotor Analysis

When you complete this lesson, you will be able to:

- Set up and run a thermal analysis;
- Post process thermal results;
- Set up and run a static analysis;
- Apply thermal loading to structural analysis;
- Post process static analysis.

Brake Rotor Analysis

Brake Rotor Design

Engineering Design and Technology Series

SolidWorks

When a vehicle is in motion, it is said to have kinetic energy. Brakes are designed to stop the vehicle by absorbing the kinetic energy of the vehicle and dissipating it in the form of heat.

The kinetic energy of a vehicle is given by the following equation:

$$KE = \frac{1}{2}mv^2$$



where m is the mass of the vehicle and v is the velocity of the vehicle. If the mass of the vehicle is large, or the velocity is large, this can amount to a large amount of energy dissipated in the form of heat. The braking mechanism must be able to withstand both the temperatures generated by this heating as well as the forces incurred during breaking.

In this lesson, we will investigate the design of the braking rotor. The vehicle will be moving at full speed and will be braked to a complete stop. First, a thermal analysis will be run to calculate the temperature distribution in the brake rotor during braking. Next, a static analysis will be run to see the affect of the thermal load as well as the forces that come with the brake pads.

1 Open Brake Rotor Assembly.

Click **File**, **Open** and select the assembly Brake Rotor Assembly. Click **Open** to open the assembly.

Activate the configuration called Split Line. It contains the model with necessary changes so that we can run the analysis properly.

2 Start SolidWorks Simulation.

Click Tools, Add-Ins. Select SolidWorks Simulation. Click OK.



Brake Rotor Design

SolidWorks Engineering Design and Technology Series

SolidWorks Simulation Interface

SolidWorks Simulation functions are accessed in the same way as SolidWorks. When a simulation study is created, a Simulation Study tree appears beneath the FeatureManager design tree. Each new study that gets created is represented by a tab at the bottom of the screen. Like SolidWorks functions, Simulation functions can be accessed from the Simulation toolbar, CommandManager, or **Simulation** drop-down menu. Additionally, functions can be selected by right-clicking geometry or items in the Simulation Study tree.


Engineering Design and Technology Series

3 Create a study.

From the Simulation drop-down menu, select Study.

Select Thermal as the Type.

Enter Temperature Distribution as the Name.

Click 🖌.

The FeatureManager design tree will split and a Simulation study tree will be created below the FeatureManager design tree.

The setup of the study is done using the Simulation study tree.

4 Adjust study properties.

Right-click the study name at the top of the Simulation study tree and select **Properties**.

Under Options, select Transient and enter **3 sec** as the Total time.

Click **OK**.

Transient:			② Steady state:
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Time increment:	0.1	sec	
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Brake Rotor Analysis



Brake Rotor Design

Transient Thermal Analysis

This model is run as a transient analysis because we want to study the heat distribution during braking from a speed of 22 m/s to a complete stop. We can easily calculate the time it takes to stop by first calculating the force it takes to stop the vehicle. We know that the vehicle is 275 kg and we assume the coefficient of friction between asphalt and rubber is 0.72. To stop the vehicle in the shortest time, the maximum braking force can not be larger than the maximum friction force between the tires and the ground, which can be transmitted to the ground. This maximum friction force is calculated as follows.

$$F_f = \mu \cdot m \cdot g = (0.72)(275kg) \left(9.81 \frac{m}{s^2}\right) = 1942.4N$$

Now that we know the friction force, we can calculate the average acceleration of the car during braking.

$$a_x = \frac{F_f}{m} = \frac{1942.4N}{275kg} = 7.06\frac{m}{s^2}$$

Finally, we can calculate the time it takes to stop.

$$t = \frac{v_x}{a_x} = \frac{22\frac{m}{s}}{7.06\frac{m}{s^2}} = 3s$$

5 Apply material.

_

Right-click the Parts folder and select Apply Material to All.

Select Gray Cast Iron under Iron as the material. Click Apply, then Close.

Thermal Boundary Conditions

SolidWorks Simulation solves for the temperature distribution in the solid using the conduction equations and boundary conditions applied to the boundaries of the model. SolidWorks Simulation has several thermal boundary conditions that can be applied to thermal studies.

Temperature

Allows for the definition of a temperature on a certain entity or body.

Convection

Applies a convection boundary condition to the selected faces. The convection coefficient and ambient temperature are specified and the heat lost due to convection is calculated automatically.

Heat Flux

Applies some amount of heat into a face per unit area.

Heat Power

Applies some amount of heat to a vertex, edge, face or component.

Radiation

Allows surface-to-surface or surface-to-ambient radiation.

In our model, we will apply convection to all faces because all of the faces will be exposed to the air. In addition, we will apply a heat power to the faces that the brake pads touch.

Convection

Convection is the transfer of thermal energy between a surface and a fluid. The amount of heat transferred through conduction is proportional to the convection coefficient, h, the surface area, A, and the temperature difference between the surface and the surrounding fluid.

$$Q_{convection} = hA(T_s - T_f)$$

In this lesson, we will assume a convection coefficient of 90 W/m².K and an ambient temperature of 20°C, which are approximations. Actual convection coefficients and ambient temperature could be computed by running a CFD analysis in SolidWorks Flow Simulation or from experiments.

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6 Apply thermal load.

Right-click Thermal Loads in the Simulation Study tree and select **Convection**.

Select Select all exposed faces in the

Convection PropertyManager. This will choose all of the exposed faces for the convection boundary condition.

Temperature Distribution1 (-Split Line-) Parts Different Londs Hide All Show All Temperature... Cogyvection...

Enter 90 W/m².K as the Convection Coefficient.

Enter 293 Kelvin as the Bulk Ambient Temperature.



Brake Rotor Analysis

Brake Rotor Analysis

SolidWorks

Engineering Design and Technology Series

7 Edit Time Curve. Click Use Time Curve and select Edit.

Column X denotes the time and column Y denotes the multiplication factor that will be applied to the convection coefficient that was entered.

Enter (0, 1) and (3, 1) in the table. This will represent the convection always being turned ON.

Click OK.

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Click 🖌.

Note: Similar to the time dependence of the loading using time curves, any thermal load can be made temperature dependent using temperature curves. Solutions with temperature curves are significantly more time consuming as convergence iterations are required because each finite element can generate a different level of heat power based on its average temperature.

Heat Power

As the vehicle is braked, the rotor is spinning and the brake pads are rubbing against the surface of the rotor, creating friction and heat energy. Much of the kinetic energy of the car is being transferred to thermal energy through the brake pads. The heat power will be applied to the brake rotors in the area that the pads touch.

The amount of heat power can be calculated from the amount of kinetic energy carried by the car. If we assume the mass of the car is 275 kg and the car is travelling at 25 m/s, the kinetic energy of the car is as follows:

$$KE = \frac{1}{2}mv^2 = \frac{1}{2}(275kg)\left(22\frac{m}{s}\right)^2 = 66.55kJ$$

If we assume all of that kinetic energy is transferred to thermal energy during braking that lasts 3seconds, we can calculate the heat power.

$$HeatPower = \frac{KE}{\Delta t} = \frac{66.55kJ}{3s} = 22.18kW$$

Since we will analyze only one pad, and about 60% of the mass of the vehicle will be on the front, the heat power is reduced.

HeatPower= $\frac{42.95kW(0.60)}{2} = 6.66kW$

8 Apply thermal load.

Right-click Thermal Loads in the Simulation Study tree and select **Heat Power**.

Select the four faces of the rotor that the brake pads touch.

Enter 6660 W as the Heat Power.

Select Total.





Transient Thermal Analysis

Brake Rotor Analysis

SolidWorks

Engineering Design and Technology Series

9 Edit Time Curve. Click Use Time Curve and select Edit.

Column **X** denotes the time and column **Y** denotes the multiplication factor that will be applied to the heat power that was entered.

Enter (0, 1) and (3, 1) in the table. This will represent the heat power always being turned ON.

Click OK.

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ind tit	NO # 3 SOC		-	_

Click 🖌.

10 Initial temperature.

Right-click Thermal Loads in the Simulation Study tree and select **Temperature**.

Select Initial temperature.

Select the assembly from the fly-out FeatureManager design tree.

Enter **25 C** as the **Temperature**.

Click 🖌.

Engineering Design and Technology Series

11 Mesh the model.

Right-click Mesh in the Simulation Study tree and select **Create Mesh**. Use **Curvature based mesh** and default element size.

Click 🖌.

Brake Rotor Analysis



12 Run the study.

Click **Run** from the Simulation drop-down menu.

- **Note:** The study should take several minutes to run. A calculation is performed at each time step as specified in the study properties. Results will be available for each time step performed.
- **Tip:** Be careful when specifying the time step magnitude so that accurate resolution of your loading curve is obtained.

Post Processing

We will now look at the various post processing options associated with transient thermal analysis.

13 Plot Temperature distribution.

Right-click the Thermal1 plot of the temperature distribution and select **Edit Definition**.

Change the Units to Celsius.

Make sure the **Plot Step** is set as **30**.

Click 🖌.



Post Processing

Engineering Design and Technology Series

Brake Rotor Analysis



14 Probe.

Right-click on the temperature plot in the Result folder and select **Probe**.

Select any location on the brake rotor.

Click the **Response** button *L* under **Report Options**.

A plot of temperature vs. time will appear.





Click 🖌.



Post Processing

Engineering Design and Technology Series

15 Plot Resultant Heat Flux.

Right-click Results in the Simulation Study tree and select **Define Thermal Plot**.

Select HFLUXN: Resultant Heat Flux as the Component.

Make sure the Plot Step is set as 30.

Click 🖌.



- **Tip:** You can also select Plot Bounds across All Steps from the Plot Step menu. This will look through all solution time steps and plot the maximum or minimum values.
- **Note:** You can also plot the temperature gradient. In addition, you can plot the directional components of each thermal result quantity. You are encouraged to try these plots.



Static Study

We now know the temperature distribution after the vehicle brakes from 22 m/s to a complete stop. This temperature distribution will be transferred into our static study as a thermal loading condition and the material will be allowed to expand or contract as a result of the temperature distribution. In addition, we will apply a loading condition to simulate the load that the brake pad would apply to the rotor. We want to insure that the brake pad will not warp significantly during this extreme braking situation.

Brake Rotor Analysis

SolidWorks

Engineering Design and Technology Series

16 Create a study.

From the Simulation drop-down menu, select Study.

Select Static as the Type.

Enter Thermal Stress as the Name.

Click 🖌.

17 Apply material.

Right-click the Parts folder and select Apply Material to All.

Select Gray Cast Iron under Iron as the material. Click Apply, then Close.

Tip: You can also copy the materials from the thermal study by selecting the Parts folder in the thermal study, then drag and drop it onto the static study. Other study parameters can be copied similarly.

Temperature Dependent Material Properties

Material properties are often temperature dependent. You can make your material properties temperature dependent in SolidWorks Simulation by creating a custom defined material and selecting **Temperature Dependent** under where you enter the parameter value.

Fixtures

The fixtures in your study will represent how the structure is attached to the real world. It is always best to pick the type of fixture that best represents the attachment to the real world. The following type of fixtures are available in SolidWorks Simulation:

Fixed Geometry

Fixes all degrees of freedom. Also known as a rigid support.

Roller/Slider

Specifies that a planar face can move freely in the plane, but is restrained normal to the plane.

Fixed hinge

Can be applied to a cylindrical face and only allows movement about the axis of the cylindrical face.

Advanced Fixtures

Allows for different types of restraints in various directions. Feel free to review the help menu to investigate these types.

Engineering Design and Technology Series

18 Apply fixture.

Right-click Fixtures in the Simulation Study tree and select **Fixed Geometry**.

Select the face on the inner diameter of the rotor hub where it would be mounted to a shaft.

Click 🖌.

Brake Rotor Analysis



Note: We are applying a rigid restraint to the shaft mounting location. With this, comes the assumption that the shaft will not deform under any applied loading transferred through this mounting location. For this example, we are assuming the shaft is much more rigid than the rotor hub. If we expect the shaft to deform, we would need to include it in the analysis.

19 Apply fixture.

Right-click Fixtures in the Simulation Study tree and select **Roller/Slider**.

Select the split face on one side of the rotor where the fixed brake pad will rub the rotor.

Click 🖌.



Discussion

Typically, the brake calipers are designed such that one pad applies a load to the rotor to push it into the other pad. With this restraint, we are simulating the stationary pad and assuming that the pad will not deform under the loading. We will now apply the loading.

Fixtures

Brake Rotor Analysis

SolidWorks Engineering Design and Technology Series

Loading

Like the fixtures, the loading in your structure should best represent the in-service loading conditions. The following types of structural loading is available in SolidWorks Simulation.

Force

Applies a force to an edge, face, or vertex in the direction defined by reference geometry.

■ Torque

Applies a torque about a reference axis.

- Pressure Applies a pressure to a face.
- Gravity

Applies linear acceleration to parts or assemblies.

Centrifugal Force

Applies an angular velocity and acceleration.

Bearing Load
 Defined between contacting cylindrical faces.

20 Apply load.

Right-click External Loads in the Simulation Study tree and select **Pressure**.

Select the split face where the brake pad will push against the rotor.

Enter **1e6 N/m^2** as the **Pressure Value**. This value can be known from experiments.





Click 🖌.

Loading

Braking Force

In addition to the load applied to the rotor through the brake pad, there is also a frictional component of the braking force in the circumferential direction. If we know the normal load applied to the bracket (1 MPa) and the friction coefficient between the rotor and pads (0.6), the frictional force can be calculated using the area of the pad application.

$$F_f = \mu \cdot F_N = (0.6) \cdot \left(1 \times 10^6 \frac{N}{m^2} \cdot 8.2781 \times 10^{-4} m^2\right) = 497N$$

Note: The frictional coefficient between the pad and rotor can often vary depending on the temperature. Using 0.6 as the friction coefficient is a simplification made in this example.

21 Apply load.

Right-click External Loads in the Simulation Study tree and select **Force**.

Select the two split faces on the rotor where the brake pads make contact.

Select Selected Direction.

Select Axis1 of the Rotor Hat part as the reference.

Select the **Circumferential** direction and enter **497 N**.





Click 🖌.

Note: When an axis is selected as a reference, the coordinate system changes to cylindrical coordinates and our load can be applied circumferentially.

Braking Force

Engineering Design and Technology Series

Thermal Loading

Now that the structural loadings have been applied, we need to apply the thermal loading. We want to apply the maximum amount of thermal loading seen during the braking. This is obviously the temperature distribution at the end of the braking. The temperatures solved for in the thermal study will then be transferred to the static study and the material will be able to respond to the change in temperature. This will cause additional displacement and thermal stresses to develop in the structure.

22 Adjust study properties.

Right-click the study name at the top of the Simulation study tree and select **Properties.**

Select the Flow/Thermal Effects tab.

Select **Temperatures from thermal study** and select **Temperature Distribution**, **Time step 30** as the Thermal study.

Enter 298 Kelvin as the Reference temperature at zero strain.

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-					
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Click OK.

The Thermal load will appear in the External Loads folder.

Engineering Design and Technology Series

23 Mesh the model.

Right-click Mesh in the Simulation Study tree and select **Create Mesh**. Use **Curvature based mesh** and default element size.

Click 🛹 .

Brake Rotor Analysis



24 Run the study.

Click **Run** from the Simulation drop-down menu.

Post Processing

We will now learn about the various post processing options available for static studies.

25 Plot stress.

Activate the Stress1 plot by double-clicking it from the Results folder. It is a plot of the von Mises stress in the model.



Engineering Design and Technology Series

Editing Plots

To edit a plot, right click on the plot and select Edit Definition.

The **Display** dialog lets you specify a stress component and units.

The Advanced Options lets you choose to plot either a Node or Element value. For the nodal values, the stresses are averaged and displayed at the nodes. For the elemental values, the stresses in a given element are averaged and the element is given this averaged stress and it is displayed.

The Show as tensor plot option allows you to plot the orientation as well as the magnitude of the stresses.

The **Deformed Shape** dialog allows you to show the deformed shape and to choose the scaling of the graphics window.

Chart Options

The chart options can be accessed by right-clicking on the plot and selecting **Chart Options** or by double-clicking the legend. The chart options control the annotations as well as other options such as the color, type of units (scientific, floating, etc), number of decimals shown in the legend.

Settings

The plot settings can be accessed by right-clicking on the plot and selecting Settings. They are used to control various display options.



Brake Rotor Analysis





Engineering Design and Technology Series

26 Display model maximum.

Right-click on the plot and select Chart Options.

Click Show max annotation.

Click 🖌.



Discussion

Notice that the maximum occurs in a sharp corner where the mesh is quite coarse. If stresses are of interest in this location, significant mesh refinement would be required. In addition, this may be an area of singular stress due to the boundary conditions and the sharp corner. If this was the case, this stress value could be ignored.

Post Processing

Brake Rotor Analysis

SolidWorks

Engineering Design and Technology Series

27 Plot displacement.

Double-click the Displacement 1 plot to show the displacement plot.



Conclusions

In this lesson, we did an analysis of a brake rotor. We learned how to set up and run both a thermal and static study. We also learned about some of the post processing options available in SolidWorks Simulation.

In order to perform this analysis, some significant assumptions were made. First of all, the convection was assumed a constant 90 W/m^2.K throughout the analysis. This is a bit of an aggressive assumption due to the fact that as the car slows down, there will be less air flow over the rotor, therefore less heat lost due to convection. As mentioned before, SolidWorks Flow Simulation could be used to more accurately calculate the convection around the rotor.

Another assumption made in this modeling is that the heat power is applied to the entire rotor surface rather than simply the location the pad touches. In actuality, the heat power is only generated where the pad touches, and as the car moves, this location rotates around the entire surface of the rotor. By applying it to the entire rotor surface, we "smear" the heat power evenly along the surface in what could be considered a conservative assumption. Can you think of a way to apply the heat power to better match the real model?

Additional assumptions were made in this model such as the friction coefficient, material properties, and structural boundary conditions. Therefore, this analysis should serve as a first-pass type of analysis where further testing or investigation could be done for more conclusive results.

Conclusions

Lesson 6 Frame Analysis

When you complete this lesson, you will be able to:

- Set up an analysis using beam elements;
- Create truss elements;
- Calculate the torsional rigidity of your frame;
- Post process results in cylindrical coordinates;
- Define beam shear and moment diagrams;
- Evaluate your frame design.

Frame Analysis

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Torsional Rigidity

The torsional rigidity is defined as the torsional response (defined as some deflection angle) of a structure to an applied torque loading. The image to the right shows the deformation of a structure due to some applied torque loading. The torsional rigidity would be mathematically formulated as follows:

 $TorsionalRigidity = \frac{TorqueLoad}{AngularDeflection}$



With respect to frame design, the torsional rigidity is an important characteristic of the vehicle for several reasons. As we will see, the general rule of thumb for torsional rigidity is "the stiffer, the better."

Imagine strapping wheels to a mattress and trying to drive the mattress through twists and turns. A mattress car, as you can imagine, would not handle the lateral loading well and it would be very difficult to keep the wheels of the car on the ground. Torsional stiffness drastically affects the car's handling.

The lateral loading on a vehicle is taken up in two places; the frame and the suspension. Now, consider tuning the torsional stiffness of your car to drive on different road types. The suspension can be adjusted, the frame cannot. Ideally then, your suspension will take up much of the lateral loading. Adjusting the stiffness of your suspension will determine how the car handles different types of loading. If the loading is taken by the frame, it will be very difficult to tune the torsional stiffness of your car.

In this lesson, we will use SolidWorks Simulation to evaluate the torsional rigidity of our frame design. Perhaps most importantly, we will also be able to evaluate how design changes affect the torsional rigidity (i.e. is this design stiffer or more flexible than the previous?).

Element Types

To this point, we have used SolidWorks Simulation to analyze the thermal and structural characteristics of our brake rotor using solid elements. Solid elements worked well in the analysis of the rotor because the structure was relatively thick and our computer had the necessary resources to mesh and run the analysis.

In this lesson, we will be analyzing the frame structure that we modeled in a previous lesson. We could use solid elements in this analysis, however we will see another element type that will allow us to greatly simplify our calculations.

Shell Elements

When the structure becomes thin in one direction, such as a sheet metal part, SolidWorks Simulation can use shell elements to significantly simplify the calculations. The shell element in SolidWorks Simulation is a two dimensional triangular element. Each node of the shell element features six degrees of freedom (3 translational, 3 rotational), giving the nodes the ability to transfer moments. The thickness of the geometry is taken into account automatically in the formulation of the element.

Shell elements can be created in SolidWorks Simulation the following ways:



Frame Analysis

- Shell elements are automatically created in
 SolidWorks Simulation when a sheet metal part is used.
- If any surface geometry exists in your part, SolidWorks Simulation will automatically recognize them as shell elements as well.
- To manually define shell elements, you can right-click the solid body from the Parts folder of the Simulation Study tree and select **Define Shell By** Selected Faces.

Beam Elements

The beam element is another type of structural element available in SolidWorks Simulation. It is a one dimensional element with 2 nodes. Like shells, each node of a beam element has six degrees of freedom. The cross-sectional characteristics of the beam are taken into account in the formulation of the element. These characteristics are automatically computed in the software, therefore greatly simplifying the model setup.

Beam elements can be created in SolidWorks Simulation in the following ways:

- Beam elements are automatically created in SolidWorks Simulation when a weldment part is used.
- Right-click the solid body in the Parts folder of the Simulation Study tree and select Treat as Beam.



SolidBody 1(1	nm/Extend21
SolidBody	Apply/Edit Material
SolidBody	Treat as Beam
SolidBody	Treat as Remote Mas

Preparing for Analysis

As previously discussed, the first step in the analysis process is to simplify the model. Care must be taken when simplifying the model such that we do not remove something that could have a significant effect on our results.

1 Open Frame.

Click File, Open and select the part Frame. Click Open to open the part.

This is the Frame part that was created in the previous lesson. There are a couple of modifications to the model in order to make it a bit more convenient for meshing.

Tip: The model used for finite element analysis is often simplified from its final production state. Therefore, it is often beneficial to have multiple configurations of your models, one for analysis and one for production.

2 Add a configuration.

From the ConfigurationManager, right-click the part and select **Add Configuration**.

Type FEA as the **Configuration name**.

Click 🖌.



The new configuration will be created and activated in the ConfigurationManager. We

can now suppress unwanted features in our part that are unnecessary for our analysis.

In addition, there may be features to unsuppress that are not part of the final model, but are necessary to the analysis.

Note: In this model, we are doing an analysis on a part. If we were analyzing an assembly, it would be necessary to create the alternate configurations at the part level, then have these configurations active or inactive in the separate configuration at the assembly level.

3 Suppress features.

In the previous lesson, mounting locations were created for the suspension and the engine. In the analysis, these will not have any affect on the torsional stiffness of the vehicle.

Right-click on the Tabs folder and select **Suppress**.

Right-click on the EngineMount feature and select Suppress.

Experimental Design

It is important when preparing a finite element model to consider the physical representation of what you are trying to analyze. Loading and fixtures are going to be applied to the model based on the most accurate representation of the physical model. These boundary conditions will introduce assumptions into the model and it is critical that the assumptions are understood and reasonable with respect to what the model is trying to accomplish.

There are a number of techniques for experimentally measuring torsional rigidity. In the simulation, we will try to most accurately represent the experiment in the analysis setup. In the experiment, the front and rear wheels are mounted on beams and the suspension components are assumed to be fixed so that all of the load applied is transferred to the frame itself. The rear of the vehicle is held stationary (fixed) while a load is applied to the beam with the front wheels to simulate the torque as shown in the figure below.



Note: Our analysis will be setup with respect to this method for measuring torsional rigidity. Other methods exist and you would want to design your analysis according to the physical setup of the experiment in order to compare the results properly.

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4 Unsuppress features.

In this model, the suspension has not been included in the analysis. We need a location to apply our torque loading as well as a place to measure the angular displacement for calculating the torsional rigidity. For this, beam members have been created to connect the frame to the location where the wheel is mounted.

Right-click on the Rigid_Supports folder and select **Unsuppress**.



Frame Analysis

We are now ready to begin the analysis. Make sure that the SolidWorks Simulation Add-In is active from the **Tools, Add-Ins** menu.

5 Create a study.

From the Simulation drop-down menu, select Study.

Select Static as the Type.

Enter Torsional Rigidity as the Name.

Click 🖌.

Beam Meshing

As mentioned before, weldment features are automatically treated as beam elements in SolidWorks Simulation. If solid elements are desired for the analysis, right-click the beam feature in the Simulation Study tree and select, **Treat as Solid**.

6 Examine the **Frame** folder.

The Frame folder should contain sub-folders that list all the solid bodies that will be meshed as beams. You can tell that bodies the will be beams because of the beam icon **a** next to the body name.





Any Solid Bodies that will be meshed as solids should be excluded from the analysis. To do this, right-click the solid body and select **Exclude from analysis**.

Section Properties

We mentioned that all of the cross-sectional characteristics of the beam are taken into account automatically by SolidWorks Simulation. The section properties that are not computed automatically are constants for the torsional shear and shear due to bending calculation. These constants must be entered manually if this information is required from the analysis. To enter the constants, right-click the beam and select **Edit Definition**. The required contacts for torsional shear calculations are as follows:

Torsional Constant, K

The value for the torsional constant can be calculated or obtained from literature.

Distance for Maximum Shear

Distance from the center of the section to the point of maximum torsional shear.

Shear Factor in direction 1

Shear factor to account for non-uniform shear stress in direction 1 of the beam coordinate system.

Shear Factor in direction 2

Shear factor to account for non-uniform shear stress in direction 2 of the beam coordinate system.

End Conditions

At the end of each beam falls a node that will connect to either another beam joint or a boundary condition. As previously mentioned, the nodes of beams have six degrees of freedom (three translational and three rotational). These degrees of freedom may be restrained or released to reflect various structural connection configurations. To manually configure these degrees of freedom, right-click the beam and select **Edit Definition**. The graphics window will show the beam with two ends as shown in the figure.





Section Properties A For Forsional Shear Stress Calculation: Units: mm T Torsional Constant: 0 mm^4 Distance for Max Strear Stress: 0 mm

For Shear Stress Calculation:

Shear Factor:

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Reset

Beam Meshing

Frame Analysis

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Frame Analysis

The following options are allowed for beam end conditions:

Rigid

All six degrees of freedom are tied to the joint. All translational forces and rotational moments will be transferred from the beam element to the joint and vice versa.

Hinge

Three translational degrees of freedom are tied to the joint. All translational forces will be transferred from the beam element to the joint and vice versa. Rotational moments will not be transferred.

Slide

Three rotational degrees of freedom are tied to the joint. All rotational moments will be transferred from the beam element to the joint and vice versa. Translational forces will not be transferred.

Manual

A custom defined connection can be defined.

Trusses

In the same location, the beam can be defined as a **Truss** $\overline{\mathbf{a}}$ which can only resist axial loading.

We want to apply the torsional loading to the location where the wheels would mount so that we can best represent the experiment. Weldment features were created that defined this location. We want the loading to be transferred directly to the joints of the frame where the suspension mounts, however, we do not want any moments to transfer, only forces. To only allow the transfer of forces, we will make these members trusses.

7 Define trusses.

Select one of the truss members in the graphics window. It should highlight the member in the Frame folder of the Simulation Study tree.

Right-click the member in the Simulation Study tree and select **Edit Definition**.

Select **Truss** as the **Type**.

Repeat this procedure for the seven remaining truss members that mount to the suspension mounting points.



8 Apply material.

Right-click the Frame folder and select Apply Material to All Bodies.

Select Alloy Steel as the material. Click **Apply**, then **Close**.

Joint Group

The beam mesh is made out of a series of one dimensional lines that may be connected. The end points of these lines are called joints. SolidWorks Simulation automatically detects the joint positions, Torsional Rigidity (-FEA <As Machined>-) S Spint group Connections Fictures External Londs Mesh

Frame Analysis

however some joints can be very close and it could be desirable to merge them (or un-merge) them. In this case, the joint locations can be modified manually using the Joint Group. We will practice this in this lesson.

The beam joints are shown as yellow or magenta spheres in the graphics window.

- joints are connected to two or more beam members.
- joints are connected to a single member.

9 Edit Joint Group.

When a beam is defined in SolidWorks Simulation a folder called Joint Group is created in the Simulation Study tree.

Right-click the Joint Group folder and select Edit.





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10 Examine joints.

Right-click a joint icon to examine the beams that are connected by the joint.

To add or remove a beam member from the list, click the beam in the graphics window.

To save the new joint, simply close the **Select Joint Members** window.

11 Edit joint.

Examine the joints in the rear of the vehicle frame.

Notice that at one of the rear suspension mounting points, there may be two joints. This location should only have one joint that connects all of the beams.

Right-click one of the joints and add the omitted beams.

Right-click the other joint and remove all of the beams. This will delete that joint since it is redundant.

Click **Calculate** to recalculate the joints.



- **Note:** This procedure may be repeated as necessary until the joints are calculated properly. Additionally, this procedure may not be necessary if the joints are initially calculated correctly.
- **Tip:** Be sure to check your joints to insure that they have been computed accurately by the software. It can be helpful to thoroughly examine the mesh after meshing.

Beam Meshing



Frame Analysis

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Merging Joints Automatically

The software automatically calculates joint locations based on the ends of beams. At the joint end, a hypothetical sphere is drawn with a diameter chosen automatically with respect to the model geometry. If two joints fall within one sphere, the beam ends will be merged forming one joint. It is possible to modify the diameter of this hypothetical sphere using **Treat as joint for clearance less than**. Re-calculating the joints is necessary to get them to merge.

Fixtures

SolidWorks Simulation has a number of different fixtures that can be applied to beam joints:

- **Fixed Geometry** Fixes all six degrees of freedom (translational and rotational).
- Immovable (No translation)
 Fixes translational degrees of freedom only. Rotational degrees of freedom are left unconstrained.
- Use Reference Geometry

The user can specify a reference and choose which degrees of freedom are fixed (translational or rotational) with respect to that reference.

It was mentioned that in the experiment, the rear wheels are held stationary. To measure the torsional rigidity, the entirety of the load must be transferred directly to the frame. Therefore any components attached to the frame that transfer the load must be assumed to be rigid. If this is the case, it would make sense to fix the rear joints where the rear suspension would attach.

Frame Analysis





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12 Apply fixtures.

Right-click Fixtures in the Simulation Study tree and select **Fixed Geometry**.

Select the eight joints at the rear of the frame where the suspension would mount.

Click 🖌.





Frame Analysis

Note: You can tell which degrees of freedom are being fixed by viewing the arrows. An arrow in a given direction means that the translation in that direction is restrained. If the arrow has a tail, the rotation about that direction is restrained as well.





Immovable

Frame Analysis

Force/Torque

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Loading

SolidWorks Simulation allows for the application of forces and torques to beams or beam joints. The force is applied to the beam or joint and the direction is defined by choosing a reference.

As with our fixtures, we will assume that the loading from the experiment is transferred directly to the frame beam joints at the suspension mounting points.

With the wheel structure assumed rigid, the load would be applied to the front axle axis and the frame would rotate about the center of the frame on the front axle axis. This is the location where the angular displacement must be measured about.



To simplify the analysis, we have not included our suspension components, therefore we must define representative loading condition that will simulate our experiment. The application of our loading to our truss members will transfer the equivalent torque to the frame through the suspension mounting points.

Note: The truss members are used only in the finite element model to directly transfer the loading. They are not part of the actual frame design.

Loading

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13 Apply loading.

Right-click External Loads in the Simulation Study tree, and select **Force**.

Select Joints under Selection.

Select the beam joint at the termination of the truss members.

Select the Front plane as the reference.

Select Normal to Plane and enter 169.7 N.

Click 🖌.



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Frame Analysis

Repeat this procedure for the opposite side.

Make sure that the direction is reversed on the opposite side.

Note: The loading magnitude was calculated such that a 100 N-m torque is applied to the frame.

$$Force = \frac{100Nm}{0.5892m}$$

Loading

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14 Mesh the model.

Right-click Mesh in the Simulation Study tree and select Create Mesh.

The beam mesh will automatically be created.



Note: Truss members are meshed as a single element because they will not have bending deformation since their nodes only transfer axial forces.

15 Run the study.

Click **Run** from the Simulation drop-down menu.

Post Processing

After the study is run, the Results folder in the Simulation Study tree will become active and our results are accessible. In this lesson, we will investigate the different post processing options available for beams.

The first result that we would like to compute is the torsional rigidity. To do this, we must know the angular deformation of the frame as a result of the torsional loading.



Cylindrical Coordinates

By default, SolidWorks Simulation plots the results using a cartesian coordinate system. Knowing this, one could convert the results to another, more desirable coordinate system.

SolidWorks Simulation also allows for the plotting of results in cylindrical coordinate. We will use this method to measure the angular deformation of the frame required to calculate torsional rigidity.

To change to cylindrical coordinates, a reference axis must be selected in the **Advanced Options**. When in cylindrical coordinates, the following changes are made to the coordinates with respect to the selected reference axis:

- X direction = Radial direction
- Y direction = Circumferential direction
- $\blacksquare \quad Z \text{ direction} = Axial \text{ direction}$

In addition, the cylindrical coordinate system appears at the bottom right of the graphics window to inform you of your coordinate selection.

Frame Analysis





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16 Plot angular displacement.

Right-click the Results folder and select **Define Displacement Plot**.

Select UY: Y Displacement as the Component.

Expand the **Advanced Options** and select Axis2 as the reference.

Select the Automatic scaling.



Note: The unit system of this plot is still in mm where we might expect it to have an angular unit since it is a circumferential displacement. In actuality, the software is taking the angular deformation about the axis and multiplying it by the distance away from the axis. To compute the actual angular deformation, we need to divide by its distance away from the axis.

Frame Analysis

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Frame Analysis

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Results

SolidWorks



17 Probe.

Right-click the Displacement plot in the Results folder and select Probe.

Select one of the ends of the trusses where the load is applied.



We now know the angular deformation of the end of the frame.

We must divide this by the distance away from the axis to measure the angle.

$$\psi = \frac{3.306mm}{589.2mm} = 0.00561rad$$

Click 🖌.

Note: The angular deformation will always be calculated in radians, not degrees.

Torsional Rigidity

We can now calculate the torsional rigidity of the frame.

$$\tau = \frac{100Nm}{0.00561rad} = 17825\frac{Nm}{rad}$$

Post Processing

SolidWorks Engineering Design and Technology Series

Beam Stresses

There are a number of different components of stress that develop in beams. Axial, torsional, shear, and bending stresses are all components of stress seen by beams. In SolidWorks Simulation, you can plot any of these.

By default, SolidWorks Simulation creates a stress plot called **Axial and bending** which looks at each beam element and plots whichever component of stress is the highest in the plot. This type of plot is useful in evaluating the maximum stress seen in your beams.

18 Plot stress.

Activate the Stress1 plot that was created by default by SolidWorks Simulation.



This is a plot of the highest stress (axial or bending) in each beam element. We can view these stress components separately.

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19 Plot axial stress.

Right-click the Results folder and select **Define Stress** Plot.

Select Axial and Automatic.

Click 🖌.



Avial P/A (N/m*2)

633,099.5 39,977.3 -553,145.0 1,146,267.3 -1,739,389 5 2,332,511.0 2,925,834.0 -3,518,756.3 **Frame Analysis**

Notice that some members are in tension and some members are in compression. You can use this information in design change decisions.

Cross Section Directions 1 and 2

To post process bending and shear stress, the beam directions 1 and 2 must be defined. SolidWorks Simulation defines direction 1 to be along the longest side of the cross section and direction 2 is it's perpendicular.

Frame Analysis

SolidWorks

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20 Plot bending stress.

Right-click the Results folder and select **Define Stress Plot**.

Select Bending in local direction 1 and Automatic.



Note: If you probe on one of the truss members, you will see that it has zero bending stress. This makes sense because the trusses do not transfer moments.

Similarly, the user can plot **Torsional**, **Shear in DIR 1** and **Shear in DIR 2** stresses.

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Shear and Bending Diagrams

SolidWorks Simulation also has the ability to plot shear and bending diagrams in the different beam directions. These can be used to study how the internal bending moments and shear forces vary along the length of the beam.

21 Plot bending moment.

Right-click the Results folder and select **Define Beam Diagrams**.

Select Moment in Dir1.

Under **Selected Beams**, choose **Select** and select the beam member shown at the front of the frame.

Click 🖌.





22 List beam forces.

Right-click on the Results folder and select List Beam Forces.

Click 🖌.

A list of the forces in each beam element are listed. You can save this list to be used later.

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Conclusion

In this lesson, we evaluated the torsional rigidity of our frame design. We learned how to set up an analysis using beam element. We also learned about the various post processing options available when beams are used.

At this point, it could be advantageous to change the frame design and re-evaluate the torsional rigidity. The most common goal for frame design is a torsionally stiff and light frame. Design changes could include different beam lengths, cross sections, or material properties.

It is important to note that the torsional rigidity was evaluated and set up with respect to the experiment that would be performed once the frame is made. We can only compare experiment to analysis if the boundary conditions are set up in the same manner. Additionally, assumptions are made in the finite element model that might not reflect the real world situation perfectly. Therefore, it is most advantageous to evaluate different frame designs using the software to decide which is the stiffest, not one that matches the experiment perfectly.

SolidWorks Engineering Design and Technology Series

Lesson 7 Intake Analysis

When you complete this lesson, you will be able to:

- Set up and run a Flow Simulation;
- Post-process a flow simulation.

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Intake Manifold Design

The common goal of intake manifold design is even distribution of flow to the piston heads. This is to insure optimum engine efficiency. In this lesson, SolidWorks Flow Simulation will be used to investigate the design of our intake manifold when the car is moving at 22 m/s (50 mph).



This lesson will introduce the complete set up of a SolidWorks Flow Simulation project. We will prepare our model for analysis, set up boundary conditions and engineering goals, run our project, and learn to post process our results.

Model Preparations

In structural analysis, it is typically necessary to simplify the SolidWorks geometry to allow the simulation to run. The same is true for flow analysis. A simplified model will result in faster mesh and run times providing you with a quicker result. Part of model preparation is deciding what type of model you will be running. SolidWorks Flow Simulation categorizes flow analysis in terms of the flow being either internal or external.

External Flow Analysis

This type of analysis involves the study of flow around a region not necessarily bounded by solid geometry. Typically, this is used for the study of flow over aircrafts, automobiles, buildings, etc.

Internal Flow Analysis

Internal flow analysis deals with the study of flow inside a region bounded by solid geometry. A typical example of an internal flow analysis would be the flow through and HVAC system. In this type of analysis, flow enters the model through some inlet and leaves the model through the outlet. Flow Simulation requires the model to be fully closed to perform an internal analysis, therefore some geometry modifications may be necessary prior to setting up the model. This will be the case in our analysis of the intake manifold.

1 Open Intake Assembly.

Click **File**, **Open** and select the part Intake Assembly. Click **Open** to open the part.

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2 Start SolidWorks Flow Simulation. Click Tools, Add-Ins. Select SolidWorks Flow Simulation.

Click OK.



Model Preparations

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SolidWorks Flow Simulation Interface

SolidWorks Flow Simulation functions are accessed in the same way as SolidWorks. When a simulation study is created, a Flow Simulation analysis tree appears as a tab next to the FeatureManager design tree tab. Each new study that gets created is linked to a specific configuration created in the ConfigurationManager. Like SolidWorks functions, Flow Simulation functions can be accessed from the Flow Simulation toolbars, CommandManager, or **Flow Simulation** drop-down menu. Additionally, functions can be selected by rightclicking geometry or items in the Flow Simulation analysis tree.



Lids

As mentioned previously, internal analysis requires the geometry of the model to be fully closed. In the intake manifold, there is an opening for the inlet flow and four ports for the flow to the cylinders. We must close these openings with lids. We will then apply the proper boundary conditions to the surface of these lids to tell Flow Simulation how the fluid will be entering or leaving through the lid surface.

Face +1 > @runner-10 Face +1 > @runner-11 Face +1 > @runner-14 Face +1 > @runner-15

1.00mm

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3 Create lids.

From the Flow Simulation drop-down menu, select Tools, Create Lids.

Select the planar faces on the inlet and the four outlet ports that the lid will cover.

Select Adjust Thickness and enter 1mm as the Thickness.

Click 🖌.



You'll notice nine new parts in the FeatureManager design tree. The lids are blind extrusions from the selected planar faces into the openings with a distance that was specified as the **Thickness**. The annular lids can be neglected as they would not come in contact with the internal fluid.

- **Note:** The thickness of the lid for an internal analysis is usually not important however, the lid should not be so thick that the flow pattern is affected down stream. For most cases, the lid thickness could be the same thickness used to create the neighboring walls.
- **Tip:** If the lid face is not planar, the lid tool cannot be used. In this case, simply create the lid manually using a mid plane extrusion.



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Check Geometry

SolidWorks Flow Simulation has a tool called **Check Geometry** that allows users to check the solid geometry to make sure that it is ready for analysis. We know that the geometry must be fully closed for an internal analysis.

Additionally, we need to make sure that no invalid contacts exist in our model. An invalid contact will cause the **Check Geometry** tool to report zero internal volume and the Flow Simulation will fail to solve. Some examples of invalid contact are shown in the figure.



4 Check geometry.

From the Flow Simulation menu choose: Tools, Check Geometry.

Make sure the **Exclude Cavities without** flow conditions check box is cleared.

Click Check.

The tool should calculate a fluid volume correctly and there should be no invalid contacts.

If invalid contacts exist, you must correct them before beginning the flow simulation project.

Click **Close** when you are satisfied.



Creating the Project

Now that our geometry has been properly modified, we can proceed with the Flow Simulation project.

5 Create a project.

From the Flow Simulation menu, choose: Project, Wizard.

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6 Create new project.

Under Configuration, click Create new to create a new configuration.

In the Configuration name box, enter Project 1.

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	+ Dask	Next 2 Cancel Help	

Click Next.

Note: SolidWorks Flow Simulation will create and activate a new configuration called Project 1 when the wizard is finished. All data associated with running the analysis will be stored in a separate folder in the model directory numbered sequentially, i.e. "1", "2", "3"... etc. based on how many projects are defined on this model.

SolidWorks

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7 Select units.

Select SI (m-kg-s) as the Unit System for this project.

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8 Select analysis type.

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Select Internal under Analysis Type. Click Next.

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9 Select fluid.

Expand the Gases tree and double-click Air to add it to the Project Fluids list.

Click Next.



10 Select wall conditions.

Accept the default conditions and click Next.



SolidWorks

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11 Initial conditions.

Accept the default conditions and click **Next**.

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70 - 20	Parameter	Value	
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12 Results and Geometry Resolution. Set the **Result Resolution** to **5**.

Click Finish.



Engineering Design and Technology Series

13 Review Flow Simulation analysis tree. SolidWorks Flow Simulation has created and activated an additional configuration called Project 1. In addition, the Flow Simulation analysis tree tab has been created in the FeatureManager.



Select the Flow Simulation analysis tree tab **a**.

If changes need to be made to the settings of the project, you can right click the Input Data folder and select the appropriate option.

In the graphics window, the Computational Domain is shown as a wireframe box enveloping the model. This is the area where the model will solve the flow simulation. Because this is an internal analysis, the model will be solved inside the solid geometry inside the box.

14 Insert boundary condition.

In the Flow Simulation analysis tree, under Input Data, right-click Boundary Conditions and select **Insert Boundary Condition**.

Select the inside surface of the lid covering the inlet as shown in the figure.

To access the inner face, right click the outer face on the lid and click **Select Other**. In the **Select Other** window, cycle through the faces by moving the pointer to highlight each face dynamically in the graphics window.



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15 Set up boundary condition. Under Type select **Flow Openings**.

Select Inlet Velocity.

Under Flow Parameters, click Normal to Surface and enter 22 m/s.

Click 🖌.



16 Insert boundary condition.

In the Flow Simulation analysis tree, under Input Data, right-click Boundary Conditions and select **Insert Boundary Condition**.

Select the inside face of one of the lids covering the outlet ports as shown in the figure.



Intake Analysis

SolidWorks Engineering Design and Technology Series

17 Set up boundary condition. Under Type select Pressure Openings.

Select Environment Pressure.

Click 🖌.



18 Set up boundary condition.

Repeat this procedure to create Environment Pressure boundary conditions for the remaining three outlet ports.

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Engineering Goals

SolidWorks Flow Simulation contains built in criteria to stop the solution process. In addition, it is best to use your own criteria by using engineering goals. Engineering goals are user specified parameters of interest which users can display while the solver is running and obtain information about after convergence is reached. Goals can be set throughout an entire domain (Global), in a selected area (Surface, Point), or within a selected volume (Volume). Finally, mathematical expressions using other goals can be written in an equation goal.

19 Surface goal.

In the SolidWorks Flow Simulation analysis tree, right-click Goals, and select **Insert Surface Goals**.

To select the inlet surface for the Surface Goals, split the feature pane and in the upper portion click the boundary condition Inlet Velocity 1 item in the Flow Simulation analysis tree to input the face where the surface goal is to be applied.

In the **Parameter** list, locate **Volume Flow Rate** and check the box.

Click 🖌.



20 Rename goal.

Rename the goal so that it appears as Inlet Volume Flow Rate.

Engineering Design and Technology Series

21 Surface goal.

Repeat the previous steps to apply a surface goal for the volume flow rate at the outlets.

When selecting the Environment Pressure boundary conditions, hold the control key and select all of the outlet conditions.

Click Create a separate goal for each

surface. This will create a separate goal for each outlet.

Click 🖌.

Rename each goal to reflect the outlets.



22 Equation goal.

Right-click Goals and select Insert Equation Goal.

Select the Outlet Volume Flow Rate 1 surface goal that was defined in the previous step to add it to the **Expression** box.

Click + in the Equation Goal window.

Repeat this procedure to sum all of the outlet goals.

Click OK.

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Creating the Project

Intake Analysis

Engineering Design and Technology Series

23 Rename goal.

Rename the goal so that it appears as Sum of Outlet Volume Flow Rates.

24 Solve.

In the Flow Simulation analysis tree, right-click the project and select **Run**.

Make sure that **Load Results** is selected.

Click **Run** with the default settings.

It could take up to an hour to run this study.

Post-Processing

We can begin to post-process the results once the solver is completed. In this part of the lesson, we will learn about various post processing options available in SolidWorks Flow Simulation. We first must change the transparency of the model so that we can view the results.

25 Change transparency.

In the Flow Simulation menu, select Results, Display, Transparency.

Move the slider to 0.75.

Note: The transparency can also be changed by right-click the parts in the FeatureManager design tree.

26 Cut plot.

In the Flow Simulation analysis tree, right-click Cut Plots and select Insert.

Select the Top plane of the plenum assembly.



Click 🖌.

Post-Processing

SolidWorks Engineering Design and Technology Series

Color Bar

In the **Color Bar** dialog box, you can specify a physical parameter for the display. You also have control over various settings of the display. You can access the **Color Bar** double-clicking on the legend in the plot.

27 Adjust view settings.

Double-click the legend to bring up the **Color Bar** dialog.

Change the **Parameter** to **Velocity**.

Change the **Number of colors** to **110**.

Click **OK**.





Hide the Cut Plot when you are through.

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28 Flow trajectory.

In the Flow Simulation analysis tree, right-click Flow Trajectories and select **Insert**.

Select the Inlet Velocity 1 boundary condition.

Enter **50** as the Number of Points. Click



Hide the Flow Trajectories when you are through.

29 Surface plot.

In the Flow Simulation analysis tree, right-click Surface Plots and select **Insert**.

Select Use all faces.

Make sure **Contours** is selected and the **Parameter** is **Pressure**. Click



Post-Processing

Engineering Design and Technology Series

30 Goals plot.

In the Flow Simulation analysis tree, right click Goal Plots and select Insert.

Click All in the Goal Plots window and click OK.

An Excel spreadsheet will be automatically created containing information about the goals.

Goal Name	Unit	Value	Averaged Value	Minimum 1
Inlet Volume Flow Rate	[m*3/s]	0.028454182	0.026444043	0.0284
Outlet Volume Flow Rate 3	[m+3/s]	-0.005352675	-0.005734393	-0.0062
Outlet Volume Flow Rate 1	[m*3/s]	-0.004508713	-0.004661057	0.0049
Outlet Volume Flow Rate 4	[m*3/s]	-0.008217182	-0.008125199	-0.0083
Outlet Volume Flow Rate 2	[m+3/s]	-0.010793548	-0.010353091	-0.0110
Sum of Outlet Volume Flow	[m/3/s]	0.028872117	0 028873741	0.0288

Discussion

As we can see from the goals plot, the inlet volume flow rate matches the outlet volume flow rate. This is a good sanity check in that all the flow entering the model is leaving the model. In addition, we notice the different volume flow rates exiting each outlet condition. Much of the flow is leaving through one of the outlets in the center. This was also seen with the flow trajectory plot. This could be because the outlet port is slightly off-set from the center of the plenum, causing more flow to leave through one side. Re-design of the plenum could better distribute the flow amongst the outlet ports.

In the real world, each piston would be firing at a different time as the engine is running. To more accurately represent this situation, a transient study could be set up with the outlet ports turning on and off with respect to when they would fire using time curves. To set this up in SolidWorks Flow Simulation, **Time Dependent** would need to be selected in the **Analysis Type** window of the **Wizard**. In addition, modifications to the outlet boundary conditions would be needed to specify when they turn on (flow is allowed) and off (flow is restricted).

Conclusions

In this lesson, we investigate the design of the intake manifold using SolidWorks Flow Simulation. We learned how to setup and run a flow analysis. We also learned good post-processing techniques for evaluating our designs. You are encouraged to continue you investigations of the post-processing options available on your own. In addition, more information in the form of tutorials and technical writings are available through the help menus.

Conclusions