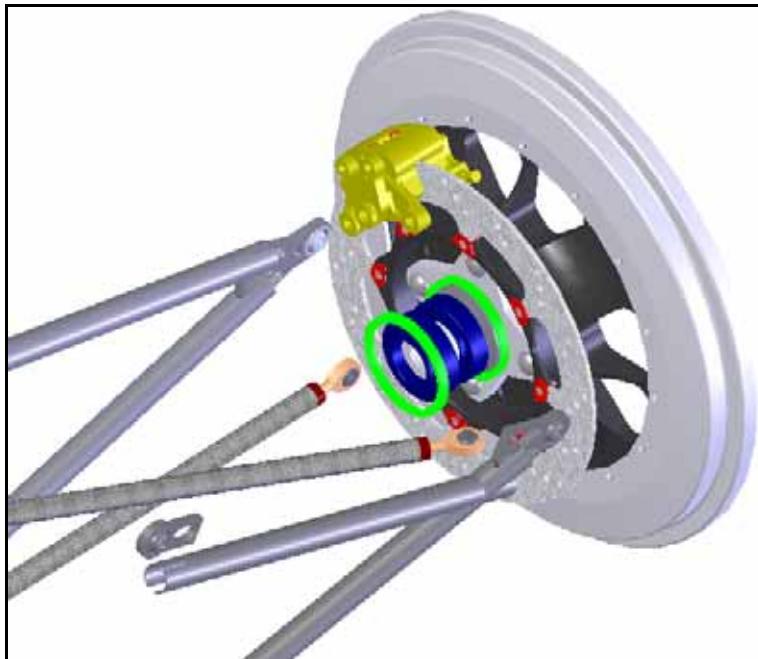


SAE® Design and Analysis Project with SolidWorks® Software



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SolidWorks**

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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.

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 *An Introduction to Engineering Design With SolidWorks*:

- 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 Content.

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

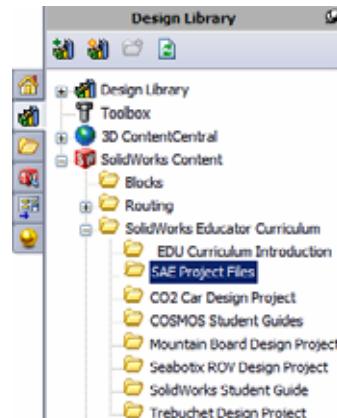
Click on SolidWorks Content to show the folders below it.

Click on SolidWorks Educator Curriculum.

Click SAE Project Files - English.

Note: There may be more curriculum folders listed in addition to the SAE Project Files.

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



3 Download the Zip file.

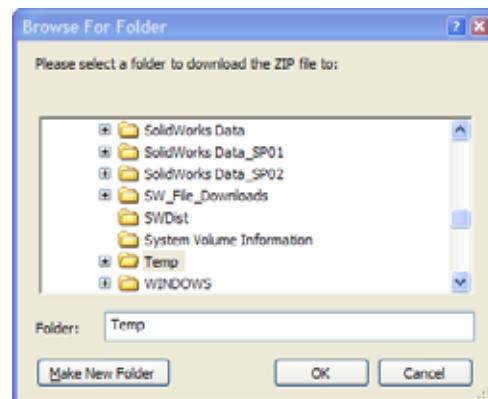
Press **Ctrl** and click the icon.

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.



4 Open the Zip file.

Browse to the folder where you saved the Zip file in step 3.

Double-click the SAE Project Files.zip file.

5 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:

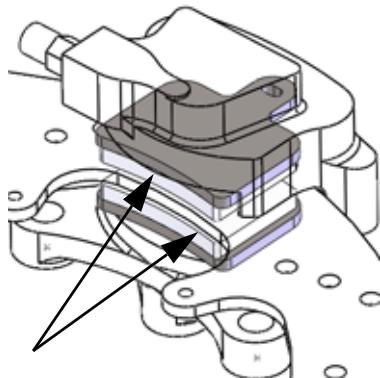
- Understand the difference between edit assembly and edit part modes;
- Create a virtual, in-context part;
- Open a part from the assembly;
- Create a new instance from an existing instance;
- Set the materials in a part;
- Use pack and go to manage the files.

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

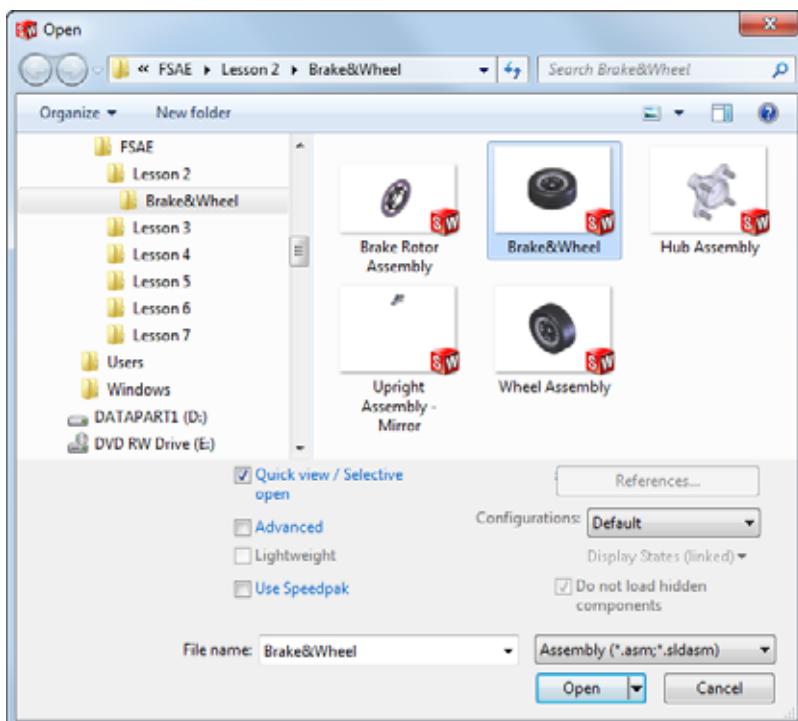


Opening an Assembly with Quick View

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. Click **Quick view / Selective open** and click **Open**.



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.

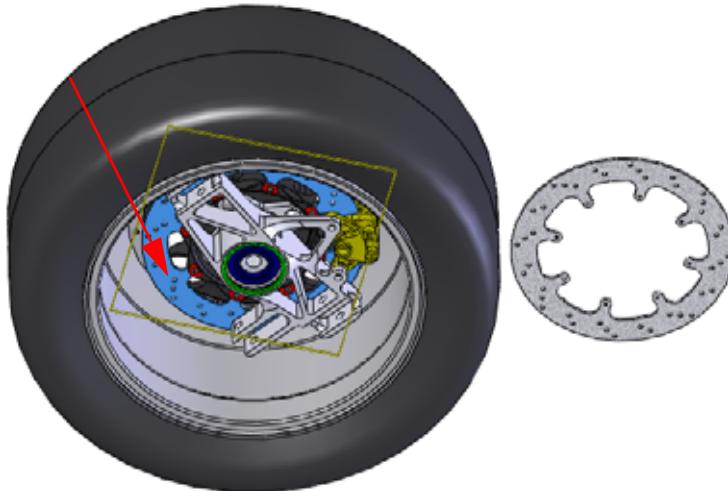
	Hiding Components	Showing Components
Many Components	Hide Others- Hides all visible components but the selected component(s).	Show Hidden Components- Shows all the hidden components for selection.
Single or Few Components	Hide/Show Components- Hides the selected visible components.	Hide/Show Components-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

2 Orientation.

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

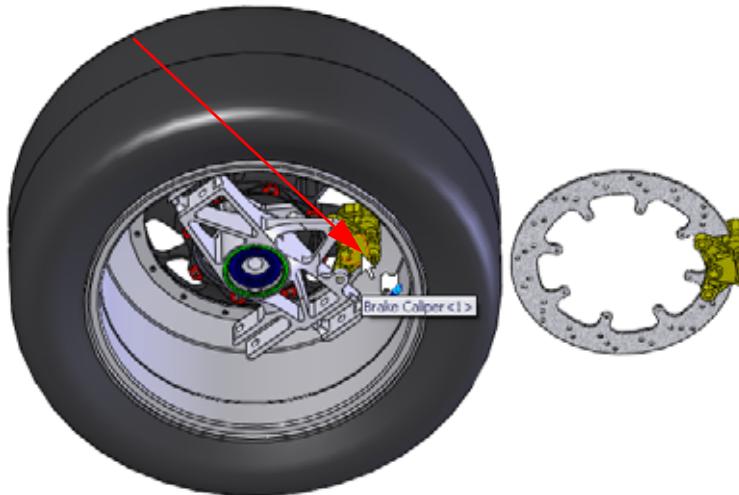
3 Hide others.

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



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.



5 Selective open.

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



6 Message.

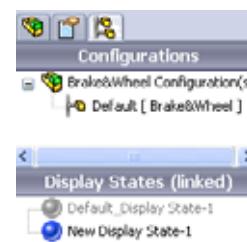
A message appears.

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 new Display State is stored under the current configuration Default and is named New Display State-1.

The original Display State Default_Display State-1 retains the settings of all components visible.

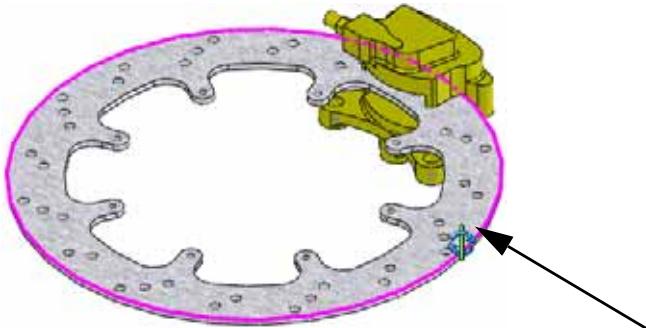


8 Save.

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

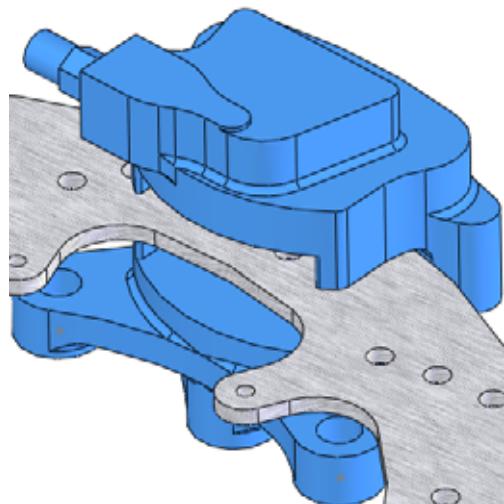
9 Orientation changes.

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.



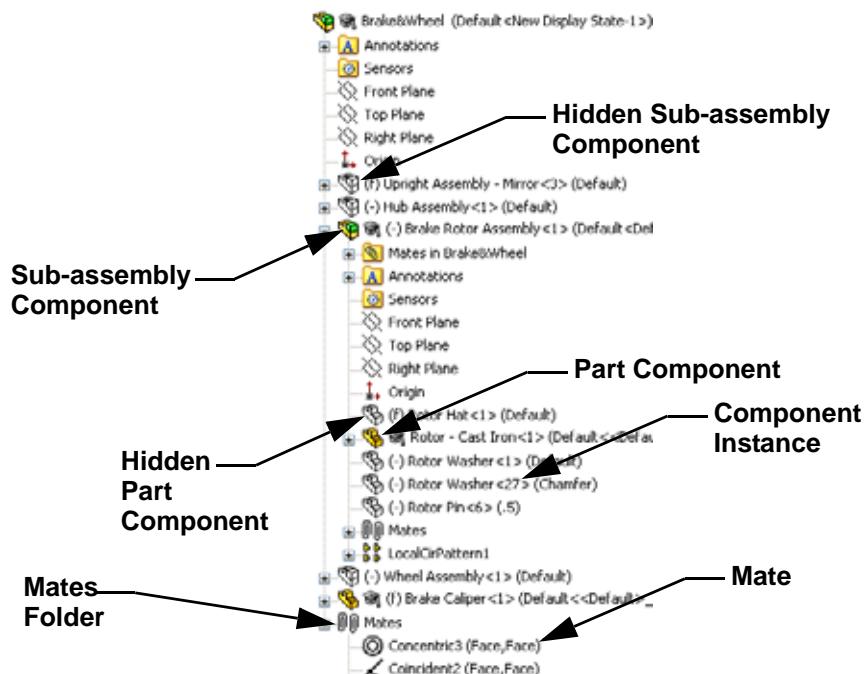
10 Zoom.

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



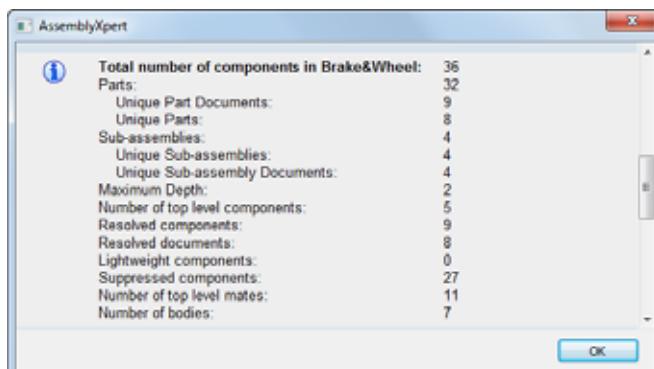
The Assembly FeatureManager design tree

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.



11 AssemblyXpert.

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



Click **OK**.

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** .



Edit Assembly

Edit Part

Tip: Colors are used to indicate which mode is currently active. See “Why do the Colors Change?” on page 14 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

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.

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.

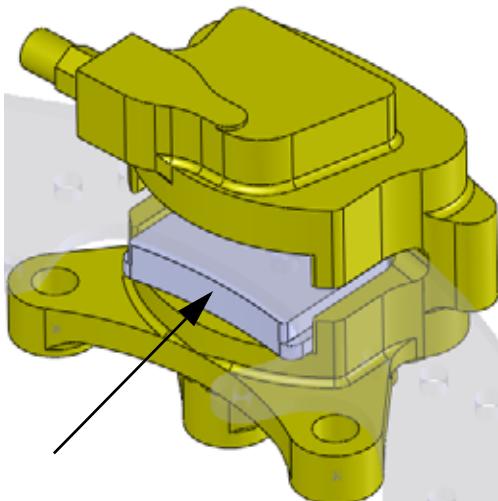
Click **Colors** and click **Use specified color when editing parts in assemblies**. That color is listed under the **Assembly, Edit Part** setting.

Click **OK**.

Creating a New Part

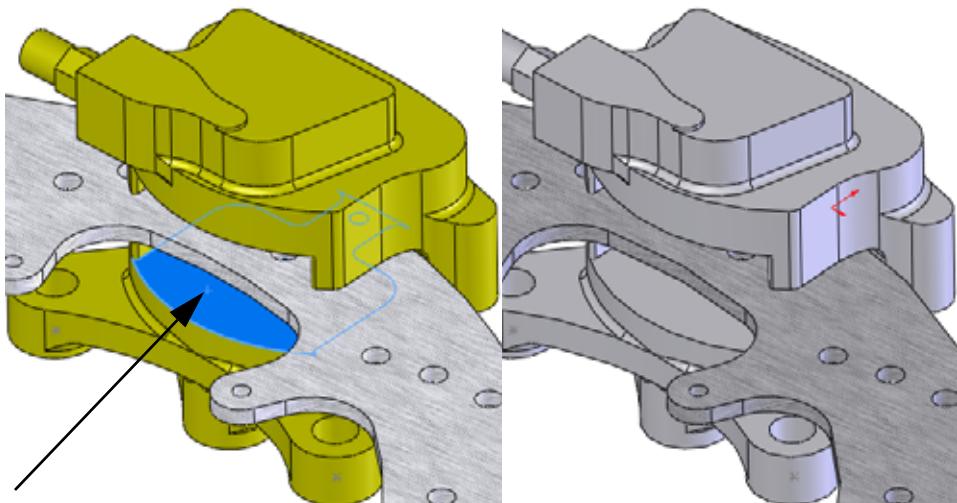
Creating a in-context new part requires a few selections including a planar face or plane for use as a sketch plane.

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.



15 New part.

Click **Insert, Component, New Part**  and select the face of the Brake Caliper as shown.



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.



Why do the Colors Change?

Due to the settings made (“Setup for Using Edit Part” on page 12), 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.

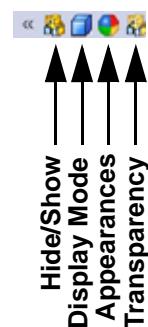
Controlling the Display

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.

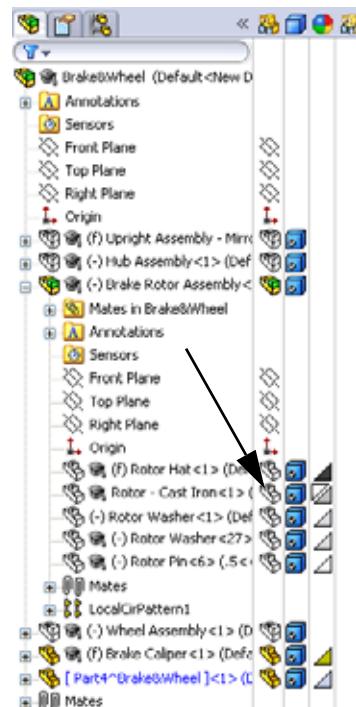
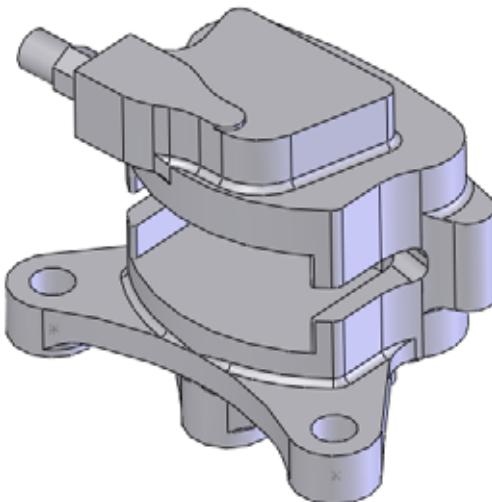


Note: The display pane works independently of the mode.

16 Display pane.

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

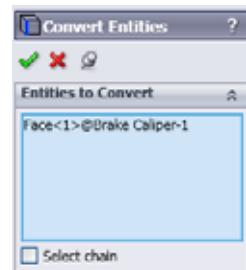
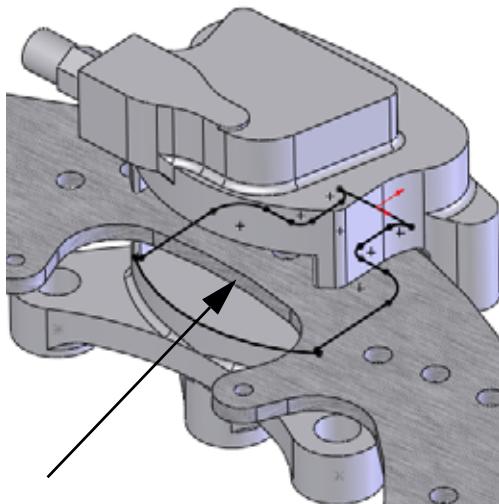
Expand the Brake Rotor Assembly folder.
Click the Rotor - Cast Iron component in the **Hide/Show** column as shown to hide it.



Click **Hide Display Pane**  to close the display pane.

17 Convert entities.

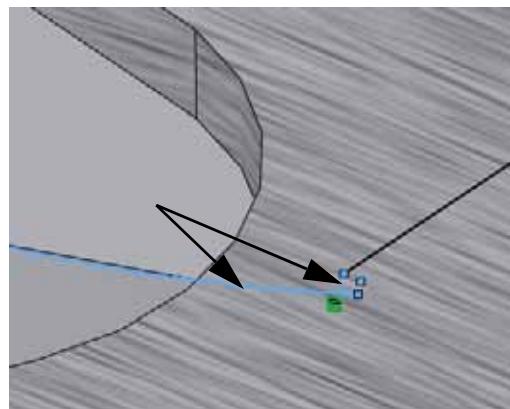
Click **Convert Entities**  and select the face and click  twice.



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.

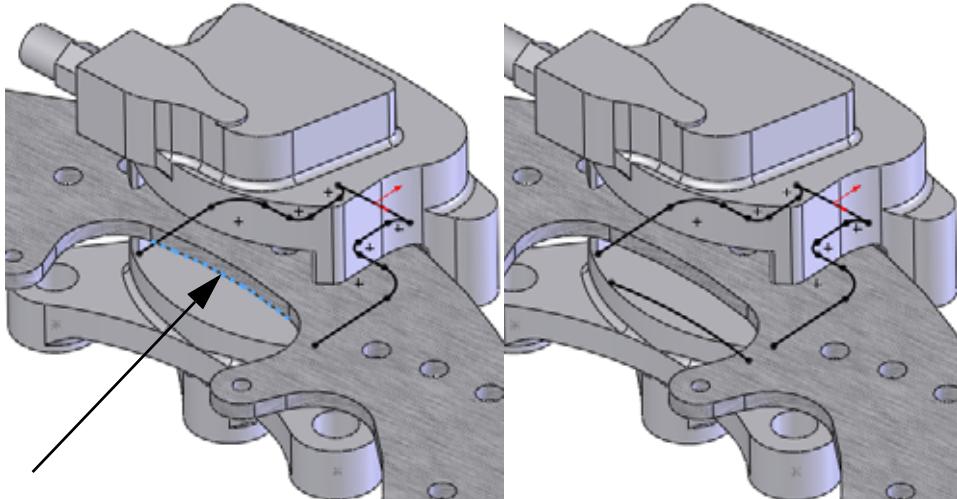


Note: There is one large arc and two small ones connected to it. Only one small arc is shown here.

19 Convert edge.

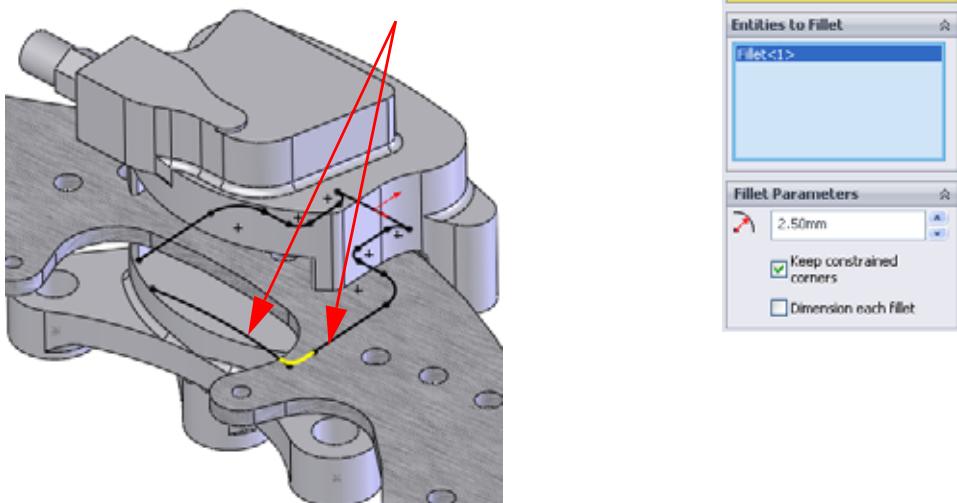
Click **Show Display Pane** » and show the Rotor - Cast Iron component.

Click **Convert Entities**  and select the edge of the Rotor - Cast Iron component. Click  twice.



20 Sketch fillets.

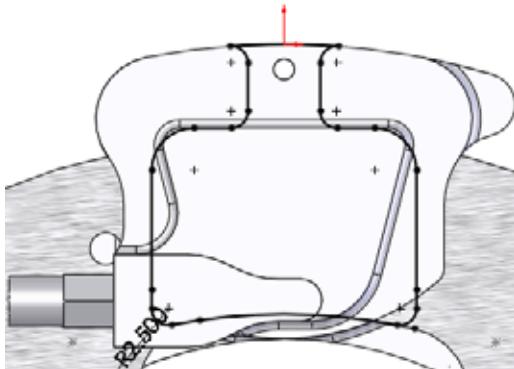
Click **Tools, Sketch Tools, Fillet** , set the **Fillet Radius** to **2.5mm**, and select the first set of geometry by selecting the geometry inside of where they would intersect.



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

21 View normal to.

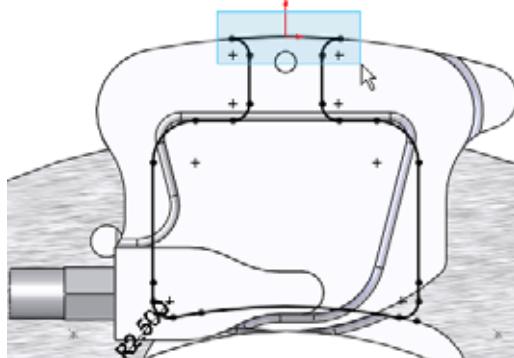
Click **View Normal To**  and zoom in as shown.



22 Delete.

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

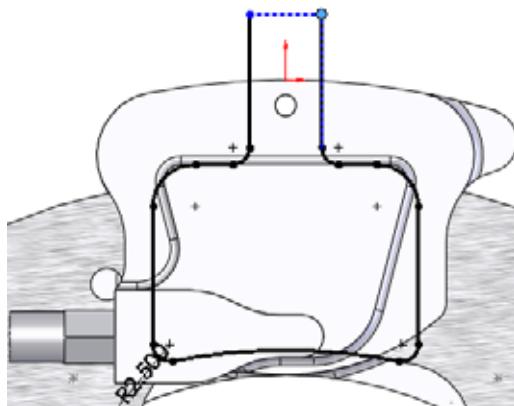
Delete the three selected entities.



23 Drag endpoints.

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

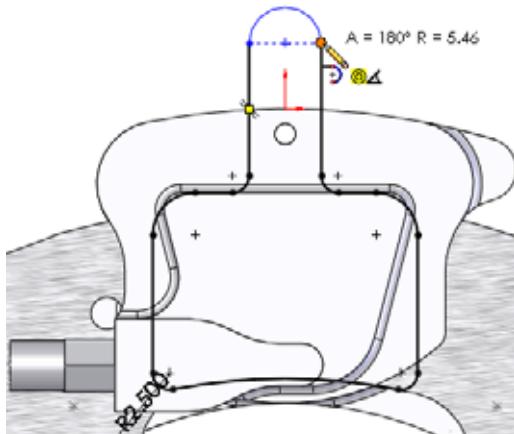
Repeat for the other vertical line and stop where the endpoints are horizontal.



24 Tangent Arc.

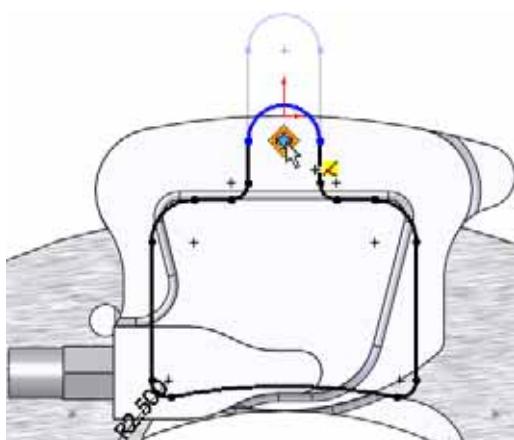
Click **Tools, Sketch Entities,**

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



25 Drag and drop.

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

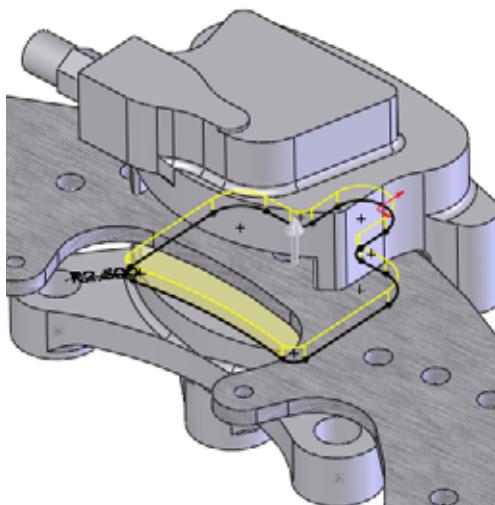


26 Previous view.

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

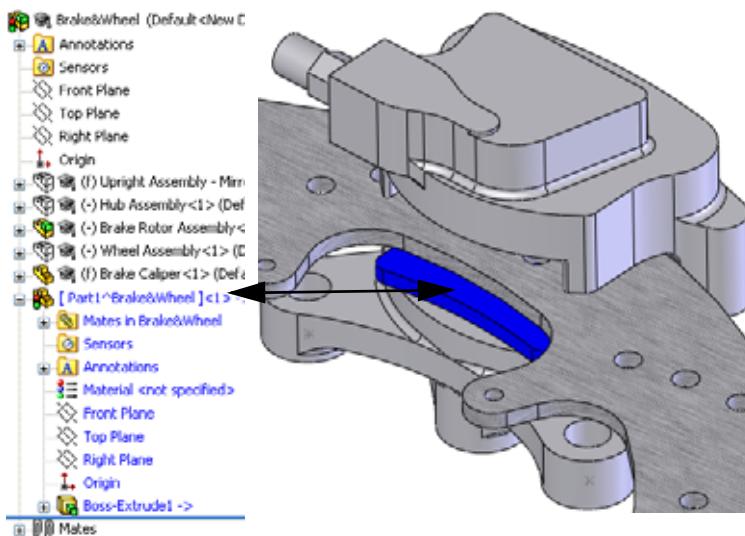
27 Extrude.

Click **Extrude**  and set the **Depth** to **3mm**. Click .



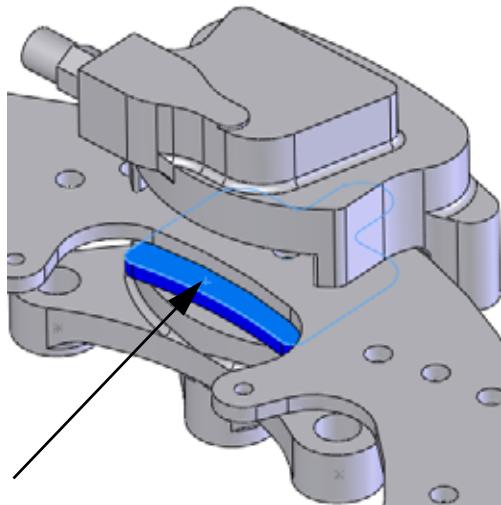
Understanding the Color Coding

The part turns blue when the extrusion is completed. The reason was explained earlier (“Why do the Colors Change?” on page 14) 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.



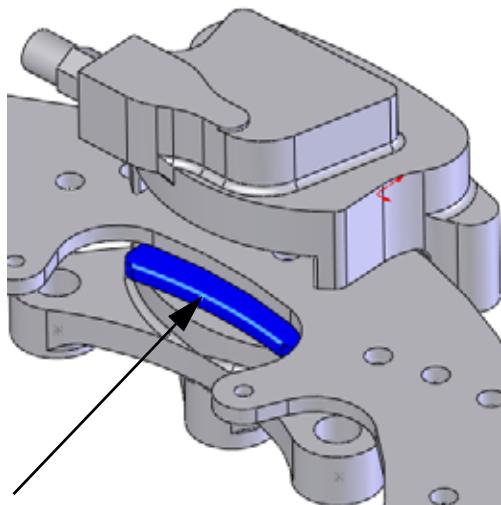
28 New sketch.

Select the face and click **Sketch** .



29 Convert entities.

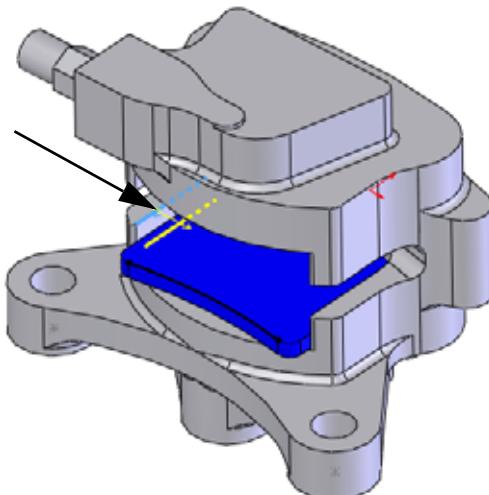
Select the edge and click **Convert Entities**  as shown. Click .



30 Offset entities.

Hide the Rotor - Cast Iron component. Click **Offset Entities**  and set the **Offset Distance** to **2mm**. Select the edge, click **Reverse** and click .

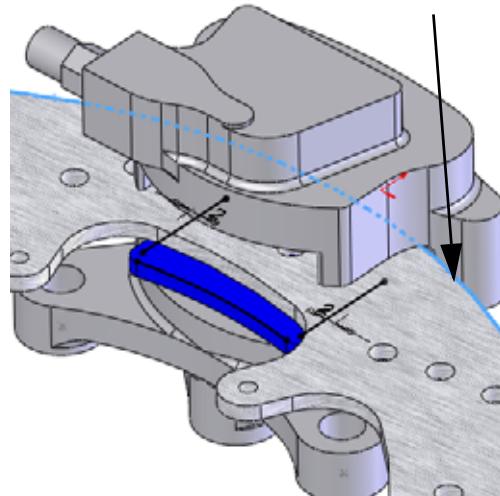
Repeat the procedure for the opposite side.



31 Convert.

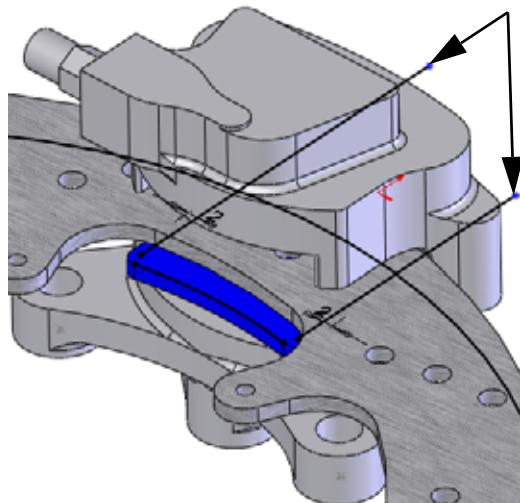
Show the Rotor - Cast Iron component. Select the edge as shown and click **Convert Entities** .

Click .



32 Drag.

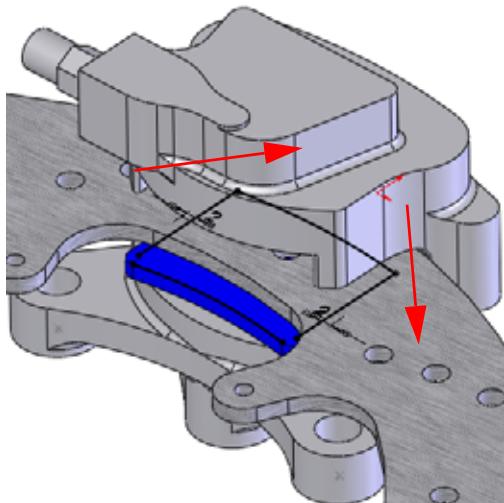
Drag the open endpoints out beyond the converted edge.



33 Trim.

Click **Trim Entities**  ,

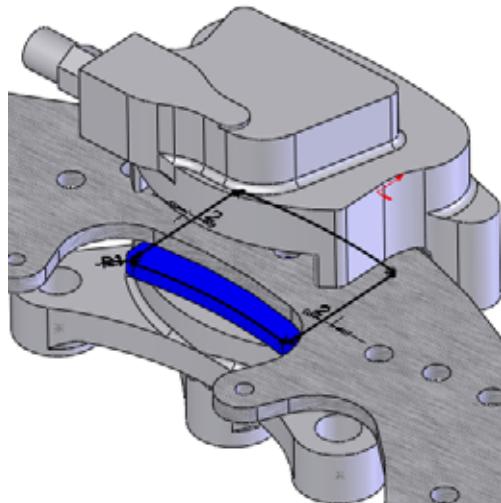
Power trim  . **Click+Drag** across the sections of geometry using the paths shown to trim away excess geometry.



34 Sketch fillets.

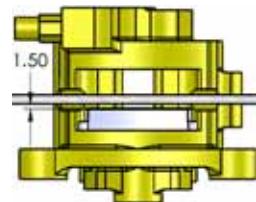
Add sketch fillets **Radius 1mm** in four places as shown.

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



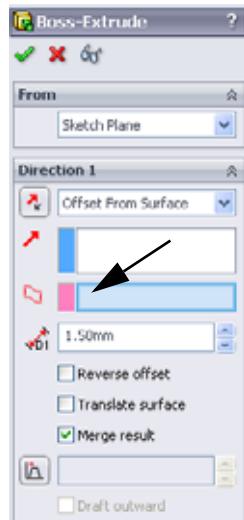
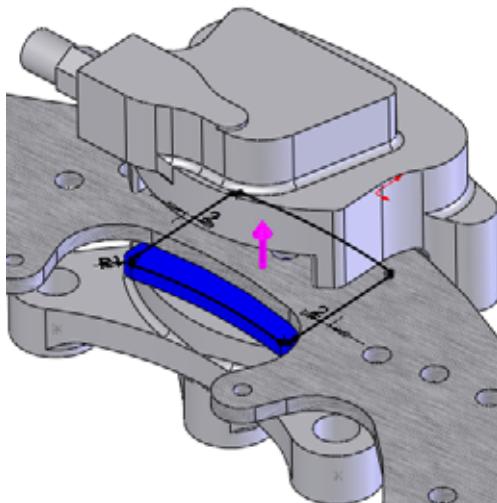
Extrusions In-Context

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.



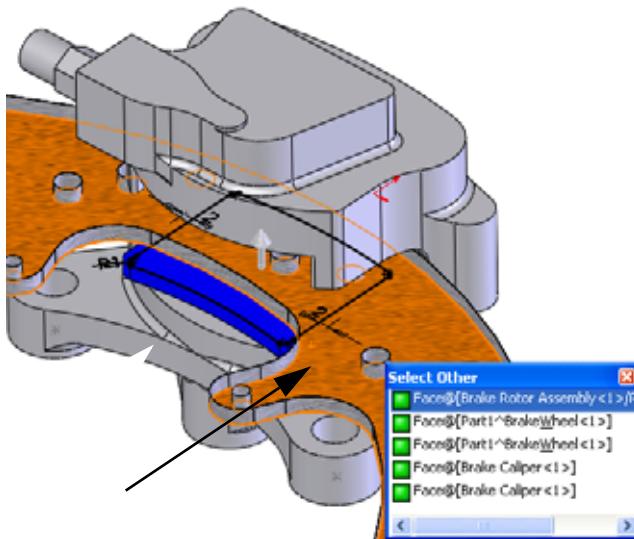
35 Extrude.

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



36 Select other.

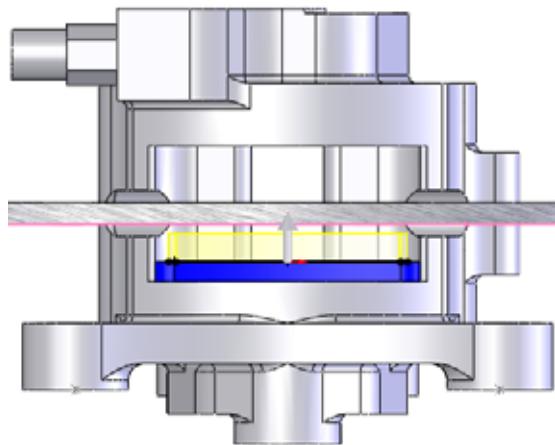
Right-click the face as shown and select **Select Other**. Click the top selection, Face@[Brake Rotor Assembly<1>/Rotor - Cast Iron<1>].



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

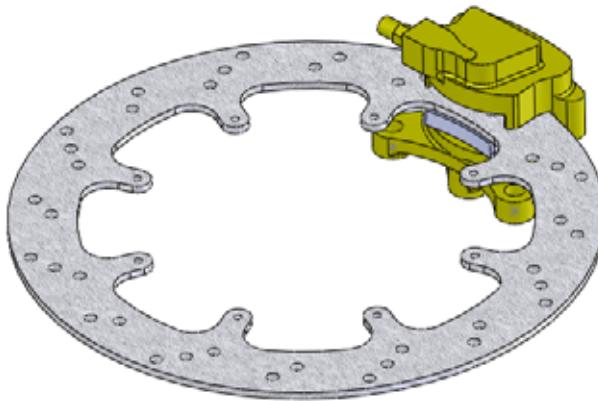
Edit Assembly Mode is the opposite toggle of **Edit Part Mode** and is the default state of the assembly where you can add components and mates. It is triggered by exiting the editing of a part in the assembly or opening of an assembly file.



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.



Working with Virtual Parts

The virtual part has been stored inside the assembly since it was created. Now that it is 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.

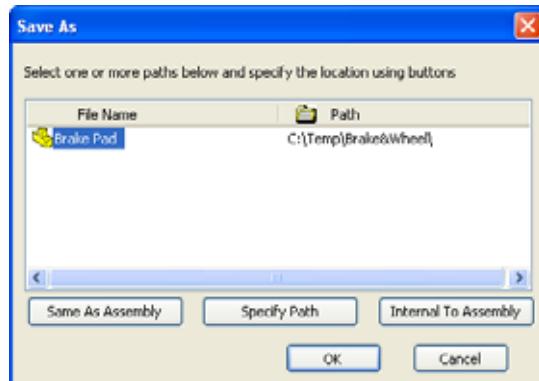
Saving a Virtual Part as an External Part

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**.



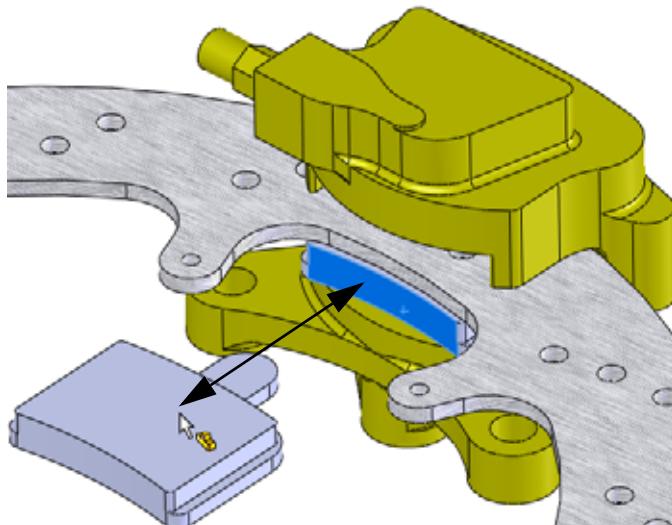
Tip: The part name appears without the square brackets.

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.

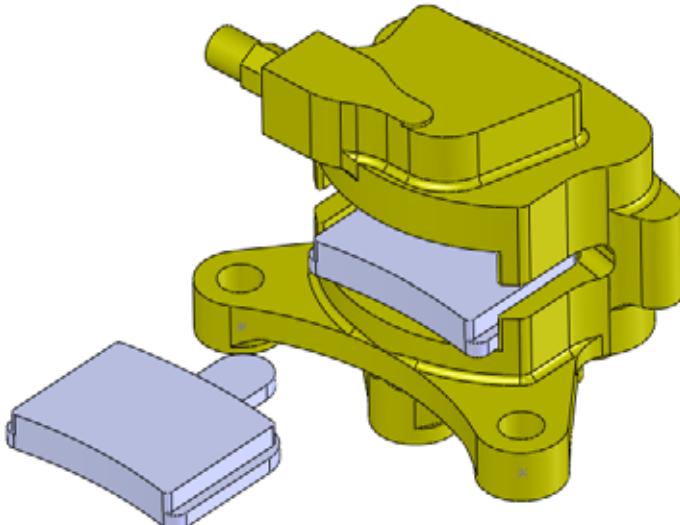
4 Copy an instance.

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



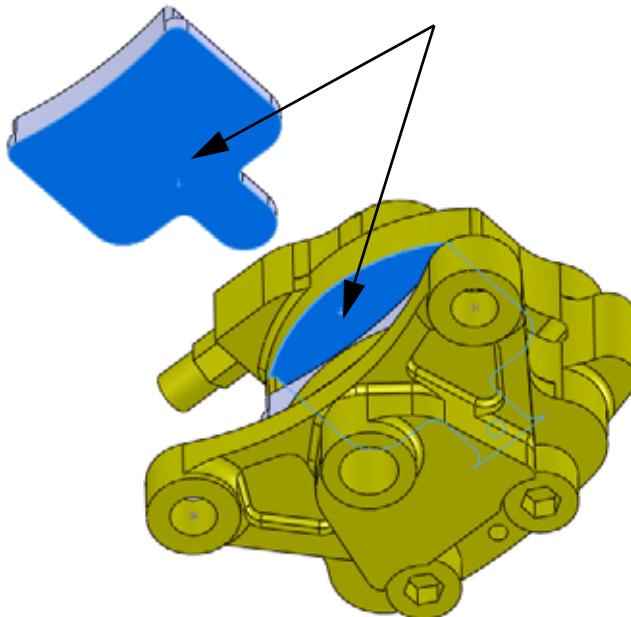
5 Hide component.

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



6 First mate.

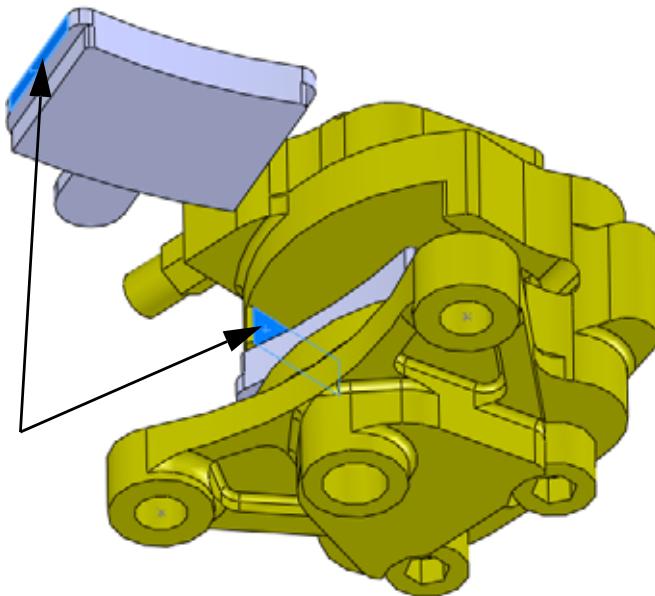
Press **Shift+Up Arrow** key. Click **Insert, Mate**  and select the faces as shown.



Click **Coincident**  and **Anti-Aligned** . Click .

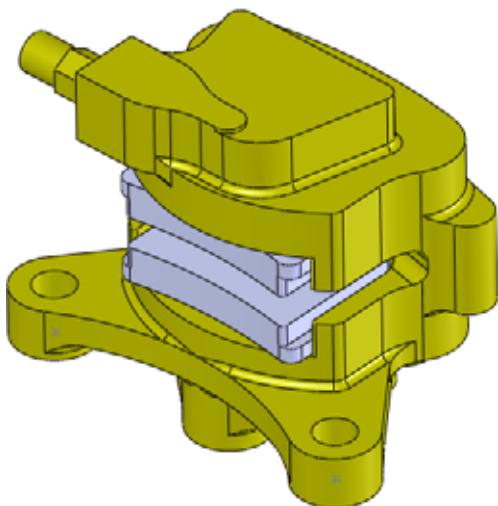
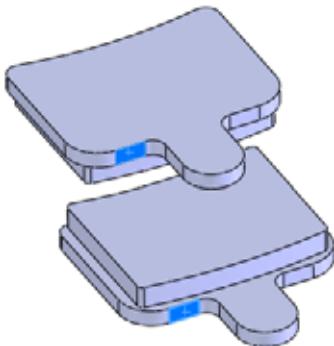
7 Second mate.

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



8 Third mate.

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



Show the Brake Caliper component.

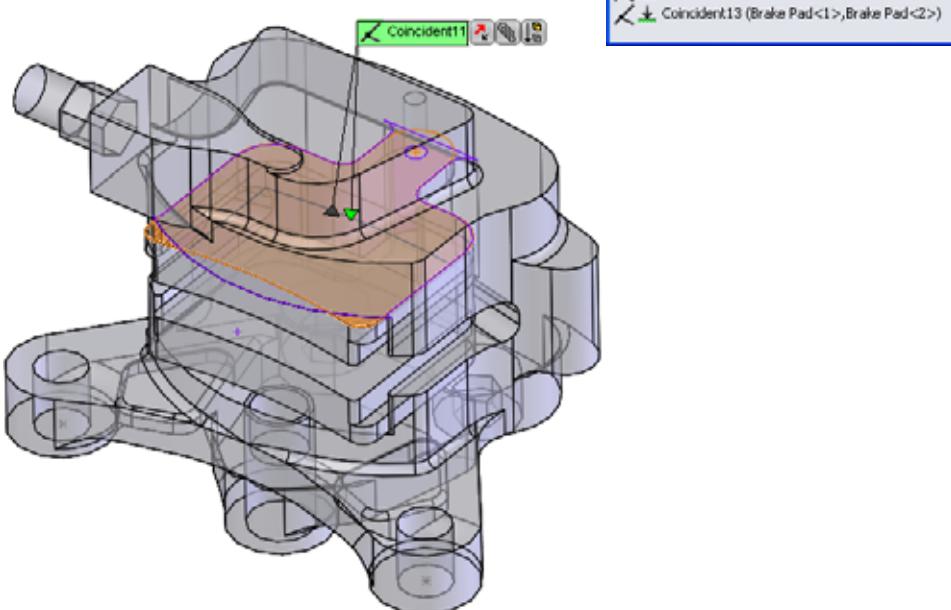
Viewing the Mates of a Component

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 indicates a path to ground. Mates marked like this are the ones that hold the component in place.

9 View mates.

Click the Brake Pad<2> and **View Mates** .



Click the “x” to close the dialog.

In-Context Part Editing

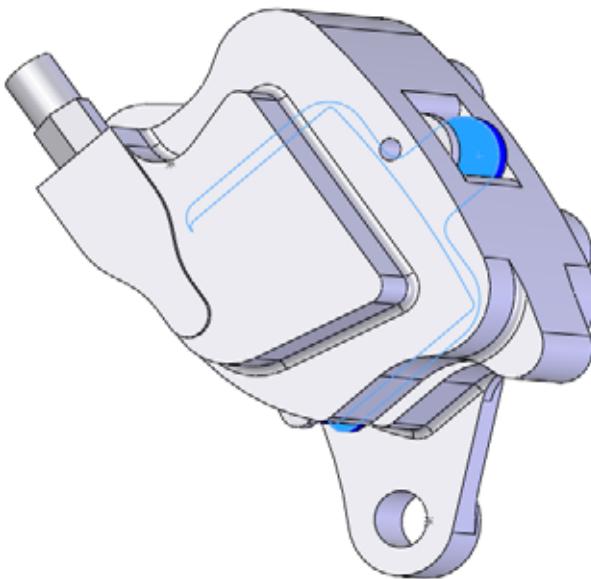
Any component part can be edited in the assembly whether it was created in the context of the assembly or not. Toggling back to edit part mode uses the same command; **Edit Part**.

10 Edit part.

Click Brake Pad<1> and **Edit Part** .

11 New sketch.

Click the face and click **Insert Sketch** . A new sketch has been created on the face.

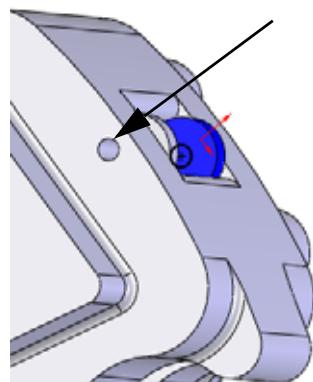


12 Convert edge.

Click the circular edge and click

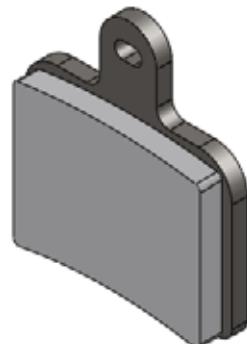
Convert Entities . Click .

Exit the sketch.



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.

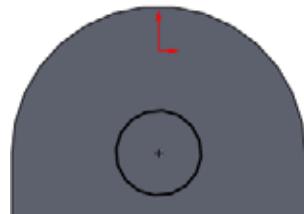


1 Open Brake Pad.

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

2 Edit a sketch.

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

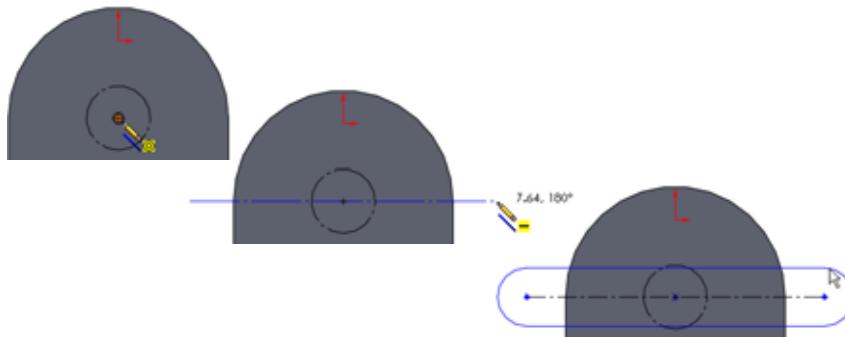


3 Construction.

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

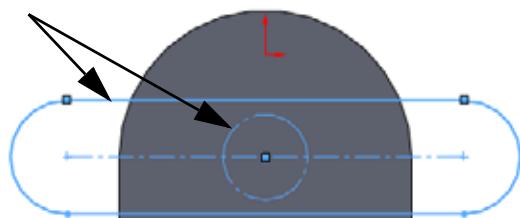
4 Slot.

Click **Tools, Sketch Entities, Centerpoint Straight Slot**  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 .



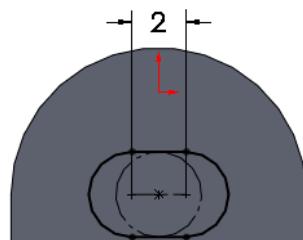
5 Relations.

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



6 Dimension and cut.

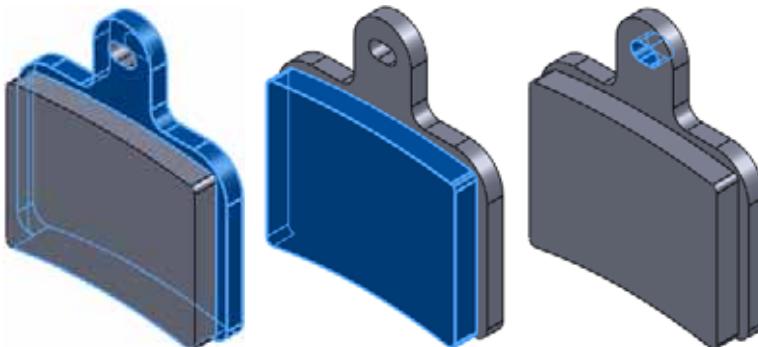
Add a **2mm** dimension as shown to fully define the sketch. Create a cut using a **Through All** end condition.



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.



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.

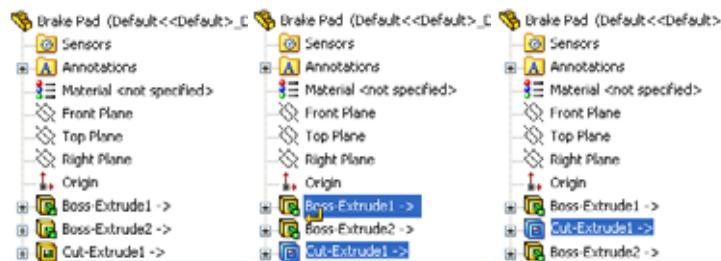
Right-click the Cut-Extrude1 feature and select **Parent/Child**. The dialog tells you that the Boss-Extrude1 and Sketch4 features are the parents of the selected feature. This also means that the Boss-Extrude2 feature is *not*. This means that the child can be moved to a position between the boss features.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



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

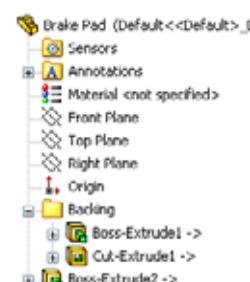
8 Reorder.

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



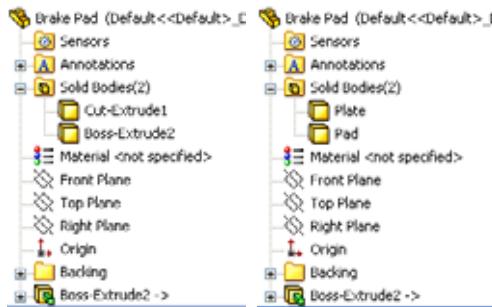
9 Folder.

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



10 Edit feature.

Click Boss-Extrude2 and **Edit Feature** . Clear **Merge result** and click . There are now two solid bodies named Cut-Extrude1 and Boss-Extrude2. Rename them to Plate and Pad as shown.



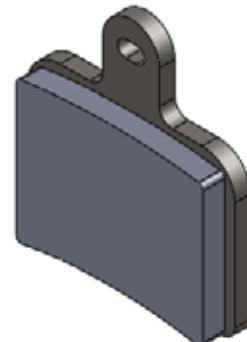
Tip: The default names were taken from the last feature that was applied to the body.

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.

11 Material for the Plate.

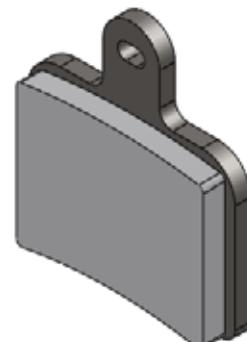
Right-click the body Plate and select **Material**, **Edit Material**. Under **Steel**, select **1023 Carbon Steel Sheet (SS)**. Click **Apply** and **Close**.



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.



13 Open assembly.

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

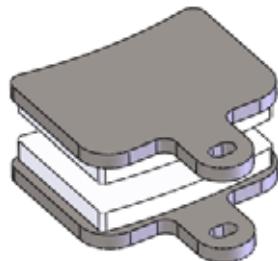


This message appears:

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.



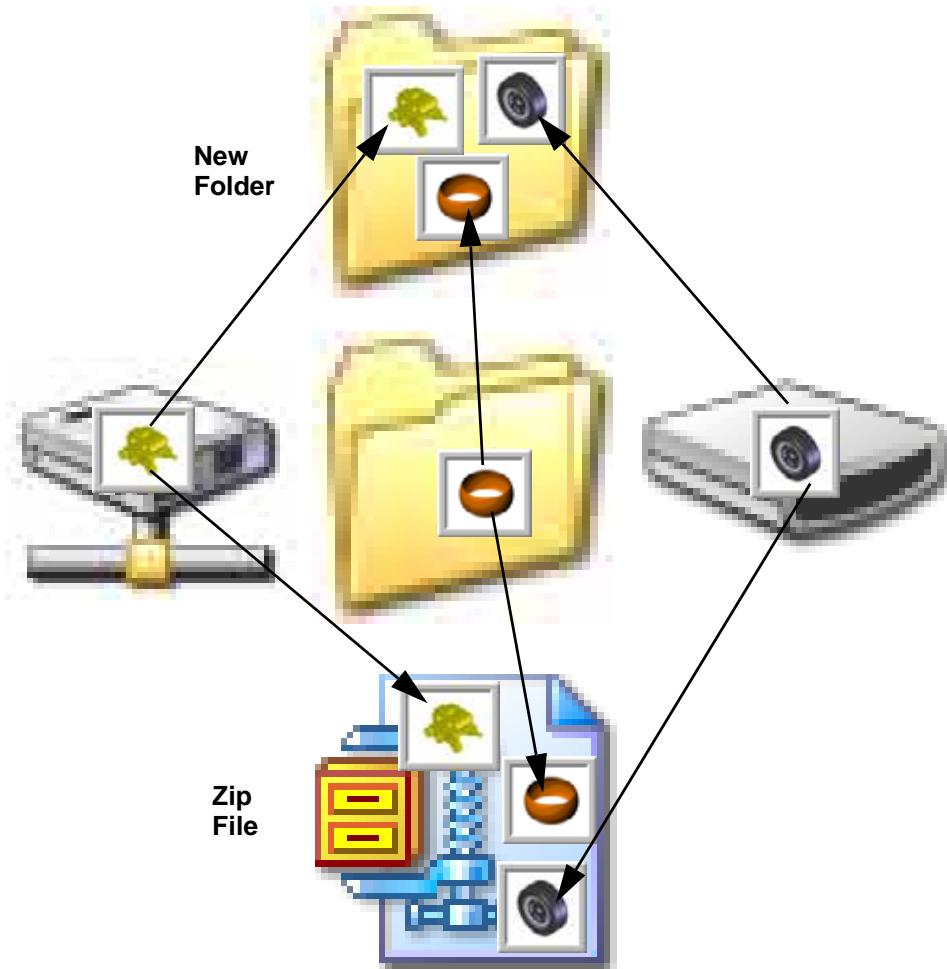
15 Display state.

Select the original Display State Default_Display State-1 to have all the components visible.



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

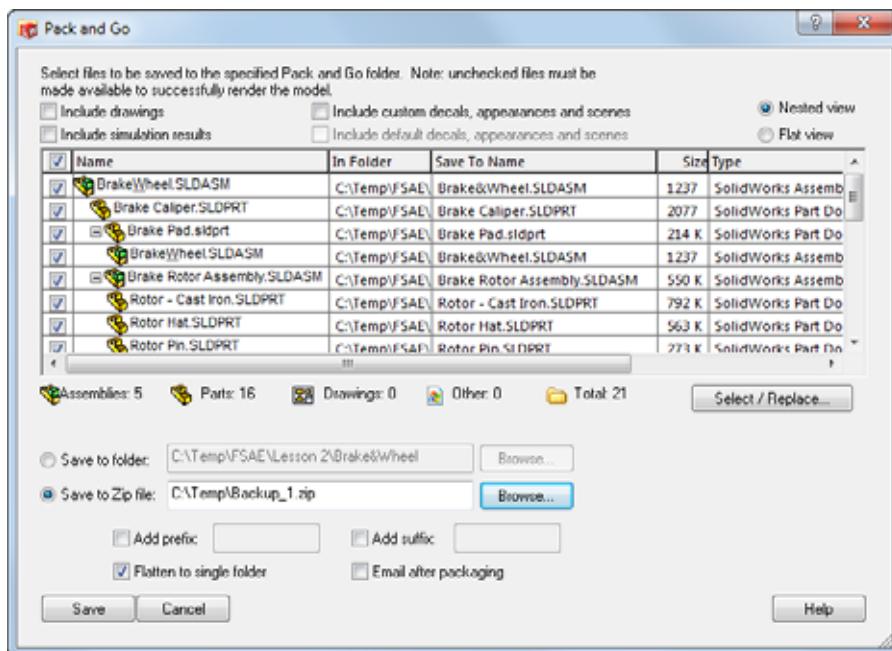
The work flow using Pack and Go creates multiple backups, using the last backup to start the next work session.

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.

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**.



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

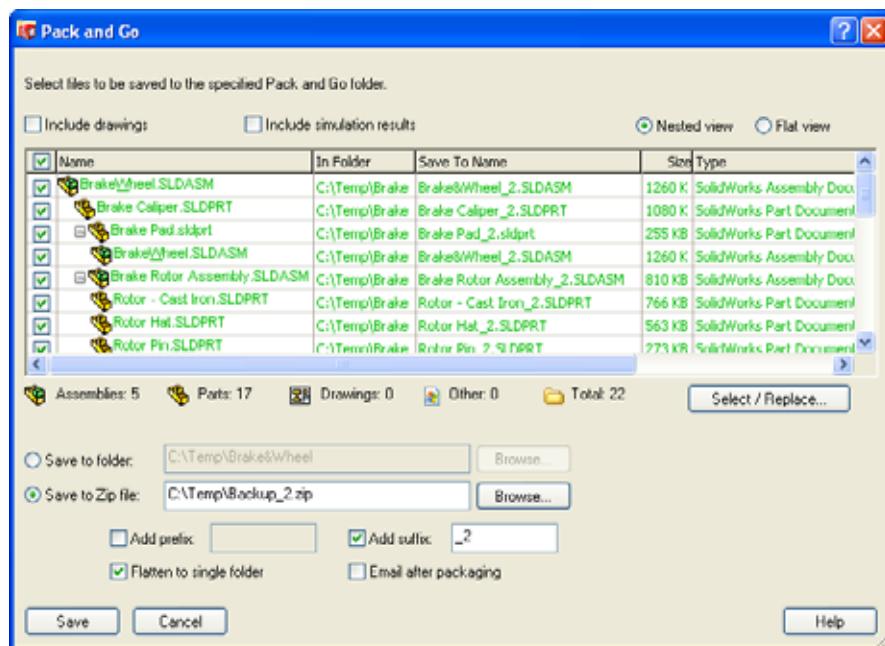
If you want to rename the files with each new backup, the **Add prefix** and **Add suffix** options can be used.

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.

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**.



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.

Lesson 3

Creating a Weldment

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

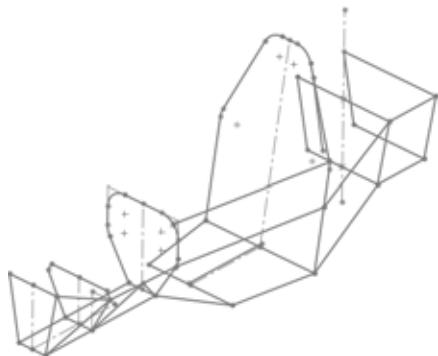
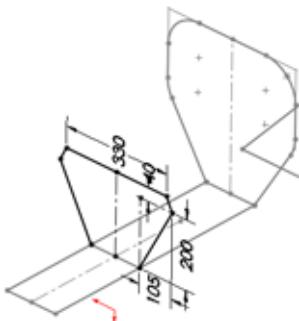
- Creating weldment parts;
- Using 3D sketches;
- Working with sub-assemblies;
- Creating custom profiles;
- Adding structural members;
- Trimming structural members;
- Editing structural members.

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.

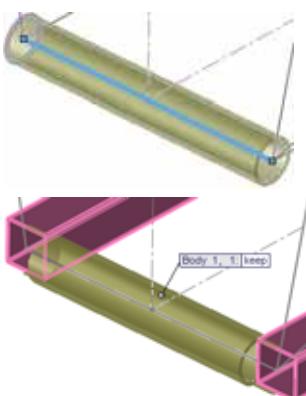
Planes and Sketches

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.



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.

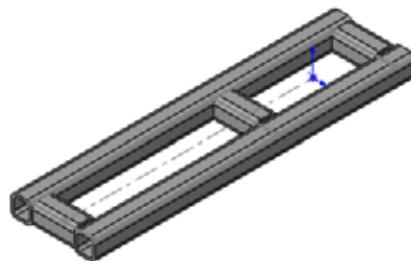
Tip: Most of the work to create a weldment is invested in the creation of the weldment sketches.

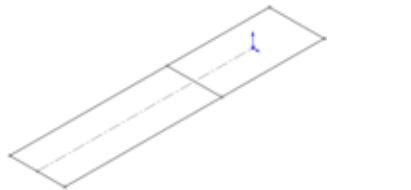
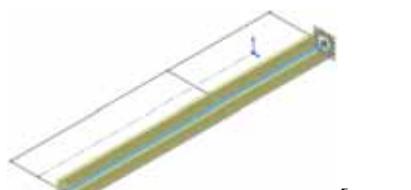
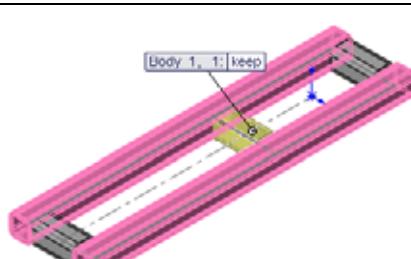
Creating a Weldment

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  and trim or extend the length of structural members to planes or other structural members.	

Some Important Points about Weldments

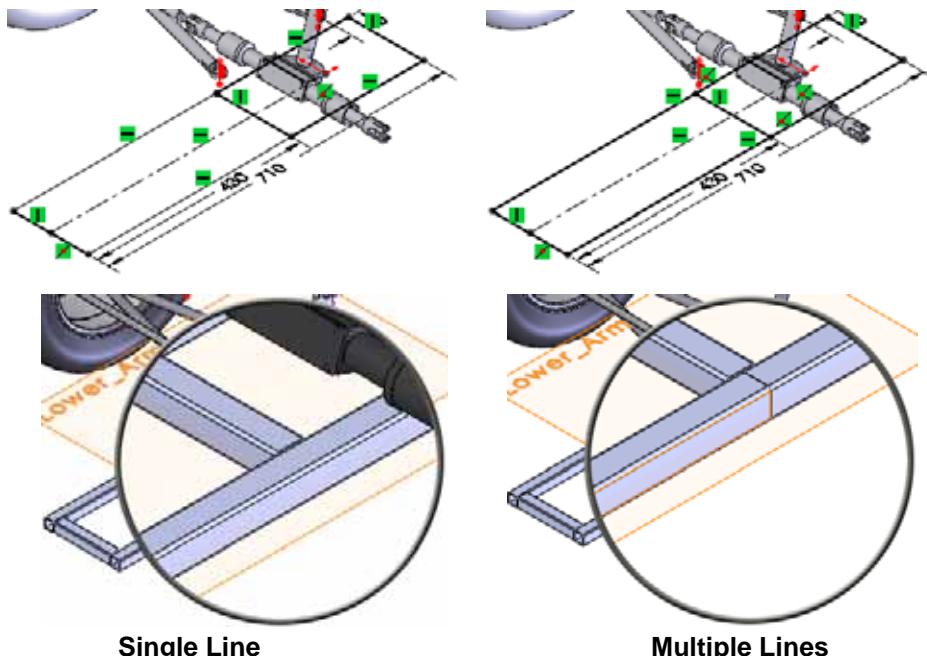
Here are some important points to remember about weldments:

- 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

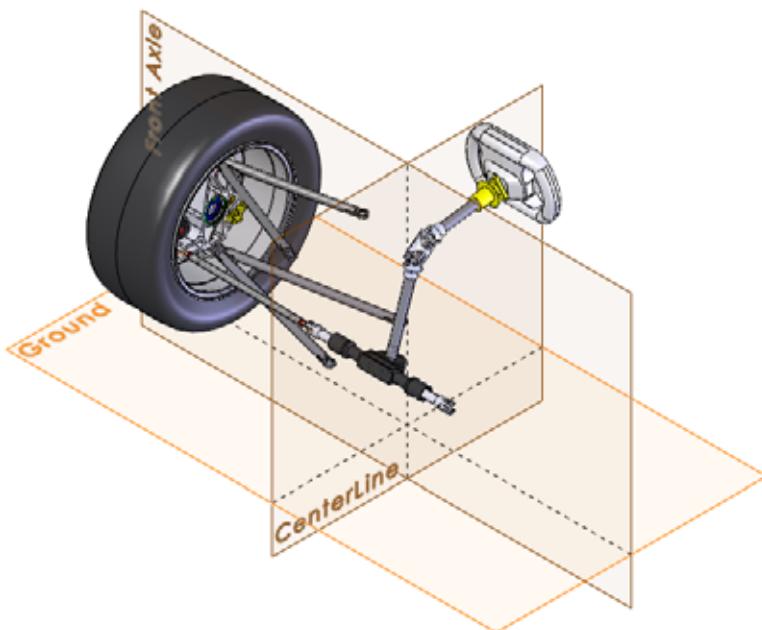
Weldment sketches are used to define the centerlines of beams used in the weldment. Continuous beams should be created using single pieces of geometry. If they are not, smaller separate pieces will be created.

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.



1 Open Frame&Suspension.

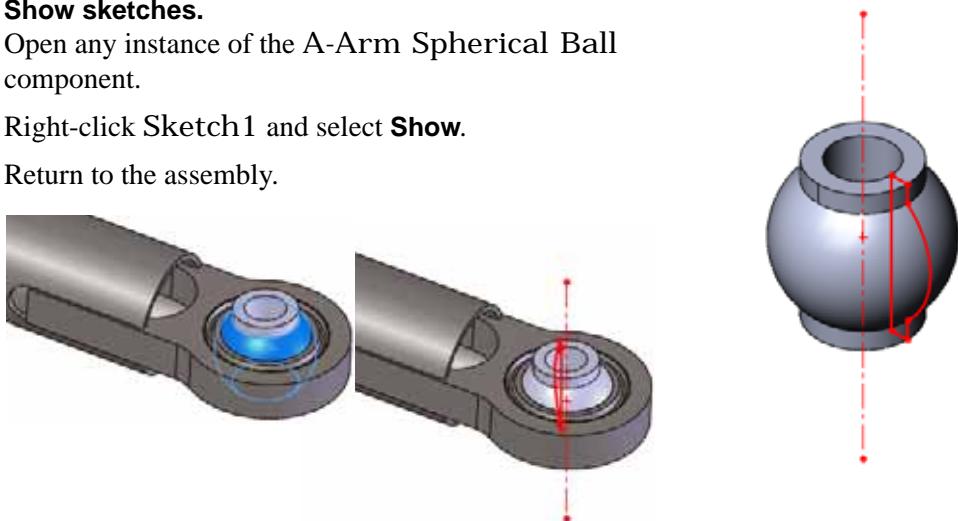
Click **File, Open** and select the assembly Frame&Suspension from the Lesson 3\Frame&Suspension folder.

**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.

Using Different Planes and Sketches

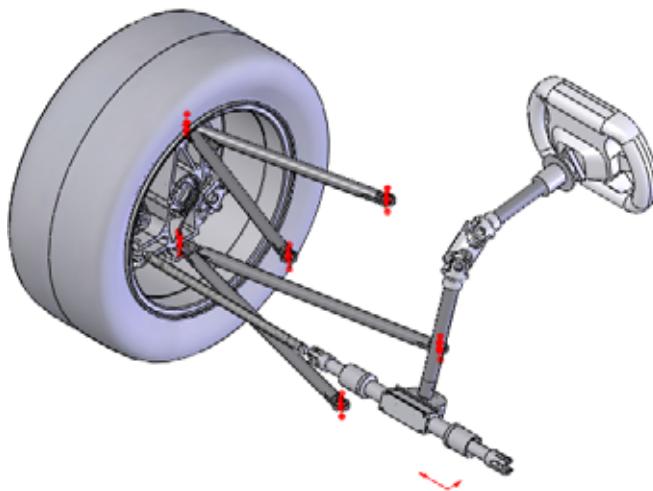
In the process of adding a new part in-context, planar face or plane must be selected. This plane is used as a sketch plane and a new sketch is opened. In fact, several things happen at once when the plane is selected:

- 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 13 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.

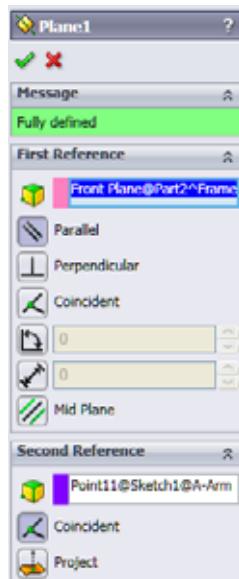
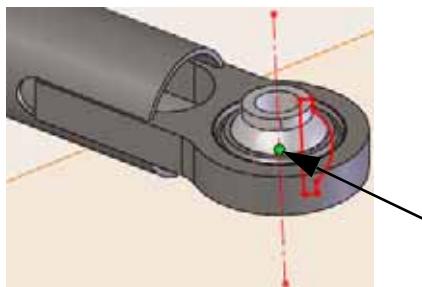
Right-click in the graphics area and select **Exit Sketch** to leave the current sketch. You are still editing the new part.

5 Reference plane.

Click **Insert, Reference Geometry, Plane** .

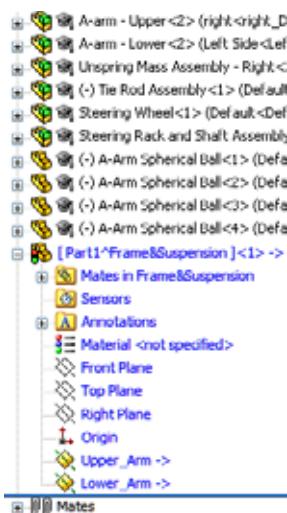
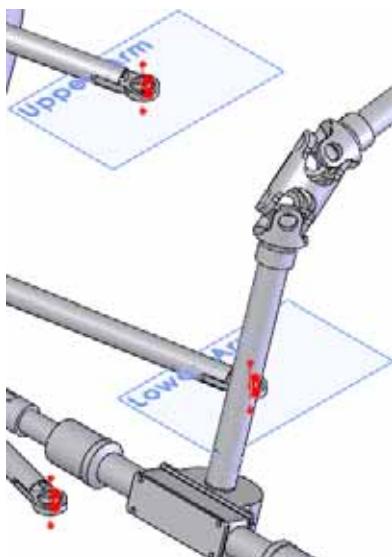
Select the plane Front plane from the active part [Part1^Frame&Suspension]<1> and **Parallel**.

Select the centerpoint of an arc from a A-Arm Spherical Ball component of the A-arm - Upper assembly as shown. Click  and rename the plane Upper_Arm.



6 Lower plane.

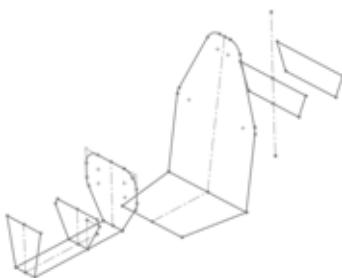
Using the same procedure as the previous step, create a new plane using the plane Front and a centerpoint of an arc from a A-Arm Spherical Ball component of the A-arm - Lower assembly. Click  and rename the plane Lower_Arm.



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.



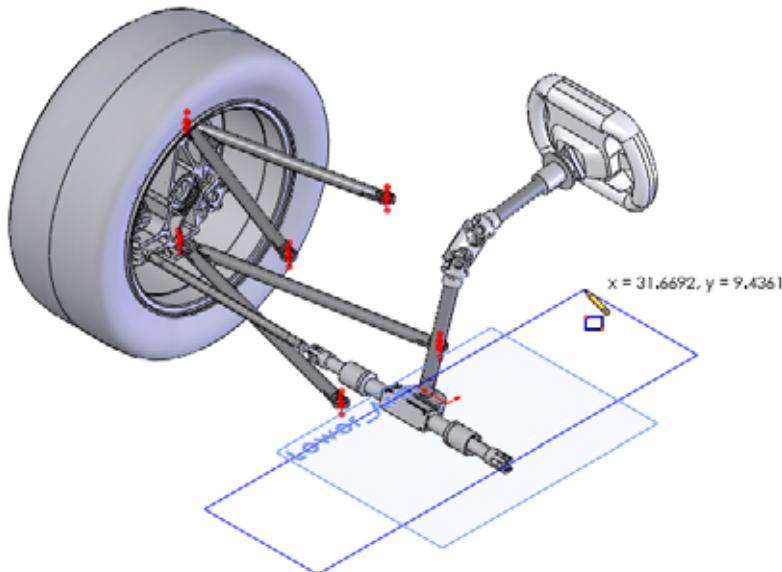
Note: Many of the 2D sketches, like this one, will require the creation of new planes.

7 New sketch.

Right-click the plane Lower_Arm and **Sketch** .

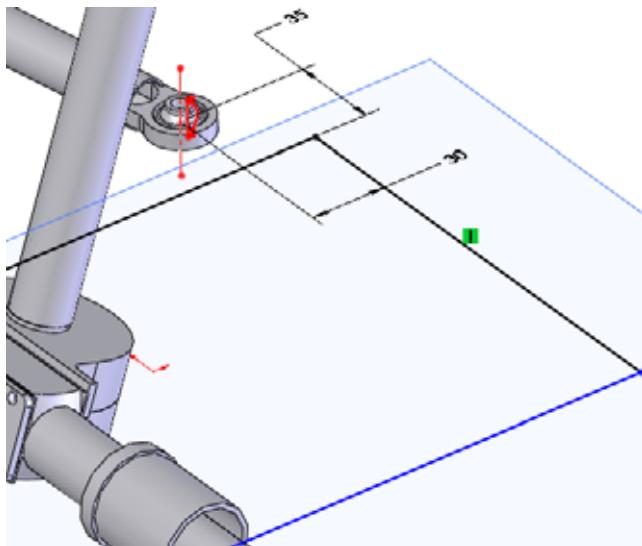
8 Rectangle.

Click **Rectangle** and sketch as shown.



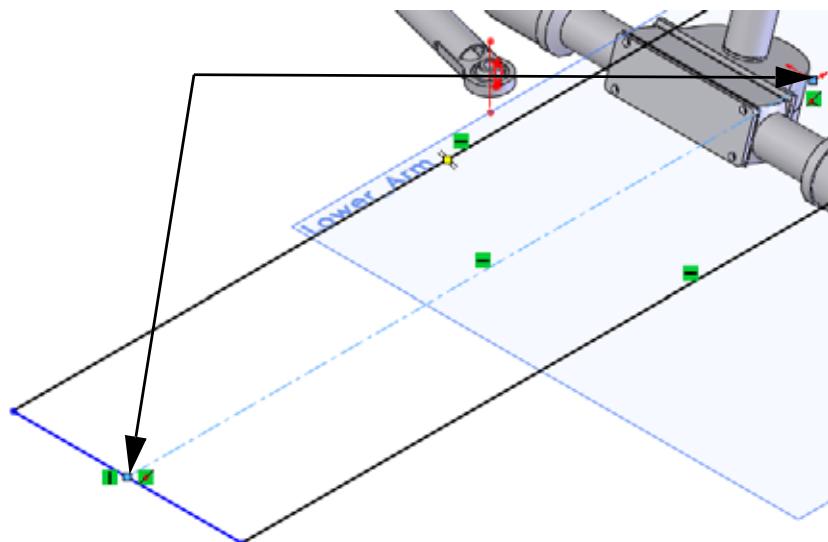
9 Reference to existing geometry.

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.

**10 Centering geometry.**

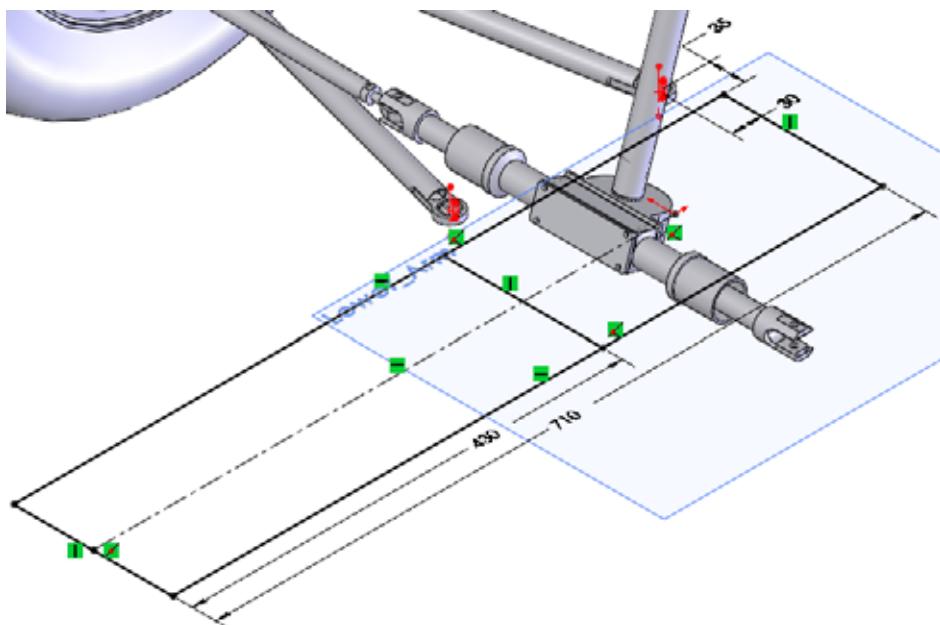
Sketch a centerline connecting the midpoint of the line and the origin as shown.

Add a **Horizontal** relation.



11 Added line and dimension.

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

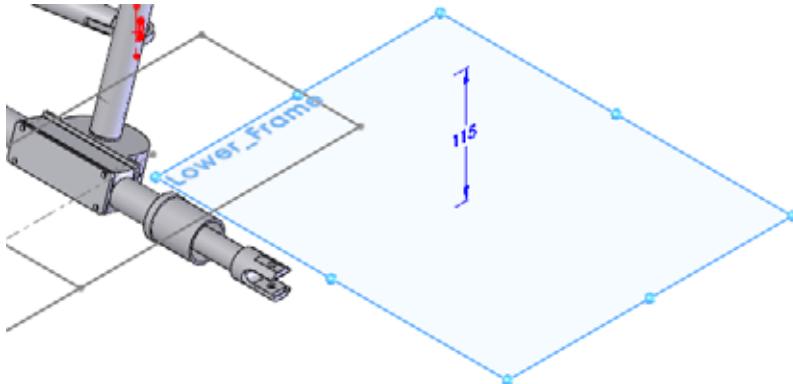


12 Exit sketch.

Right-click in the graphics area and select **Exit Sketch**  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.

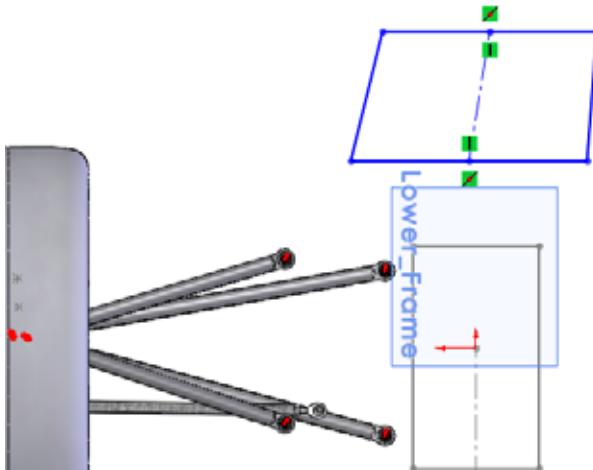


14 Hide components.

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

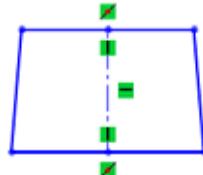
15 New sketch.

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



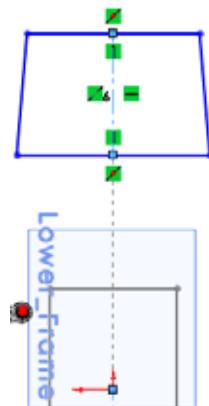
16 Symmetry.

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



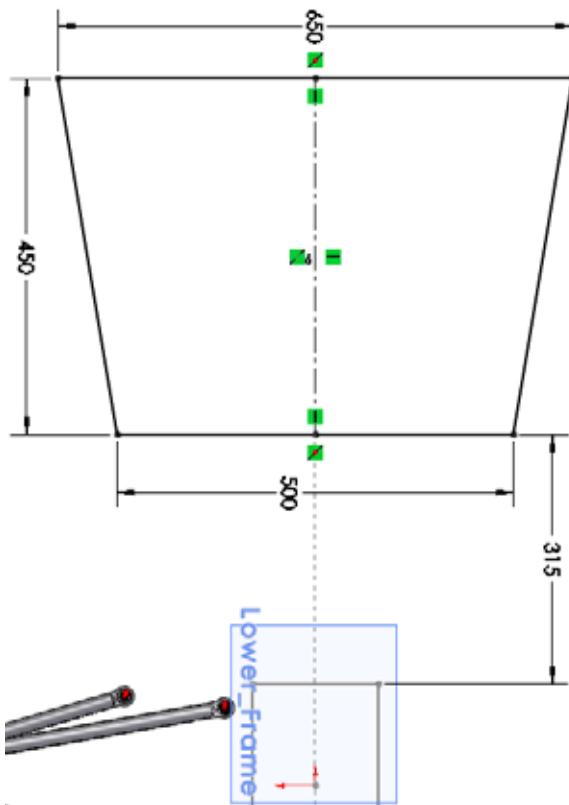
17 Relation to inactive sketch.

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



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.

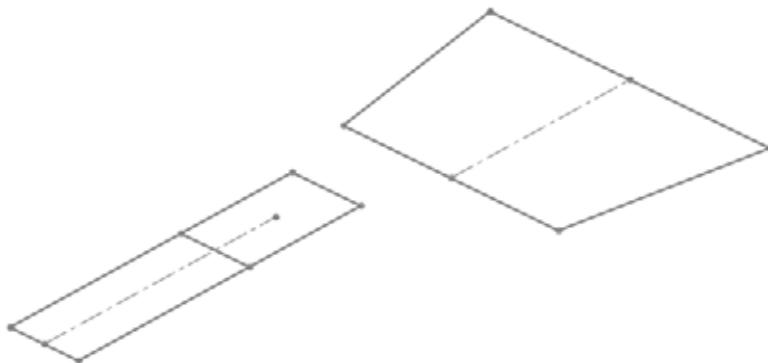
Right-click the virtual part Frame^Frame&Suspension and select **Open Part**.

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.

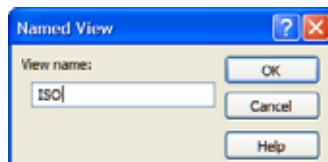
Click **View, Modify, Rotate**  and rotate the view orientation so that it looks similar to that of the assembly.



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**  icon on the **Orientation** dialog.

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



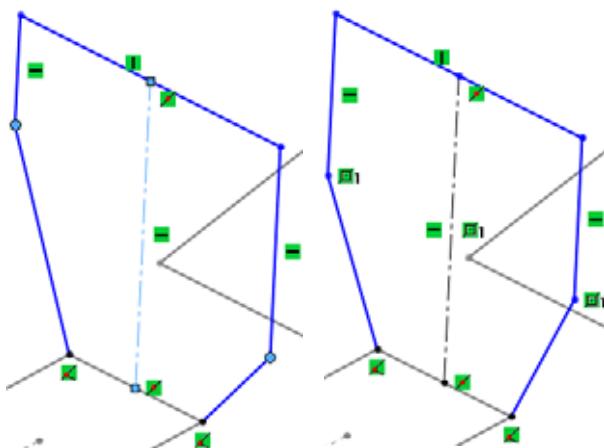
The view name ISO will be added to the list and can be recalled by double-clicking.

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**  relation.



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

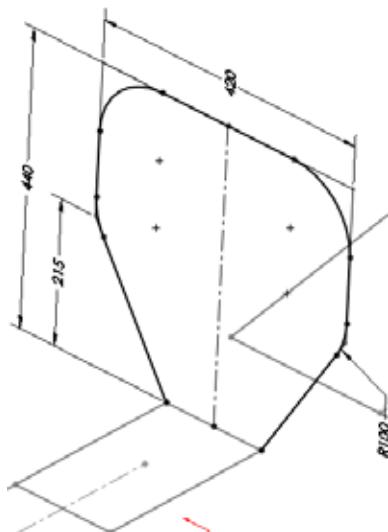
7 Dimensions and fillets.

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

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

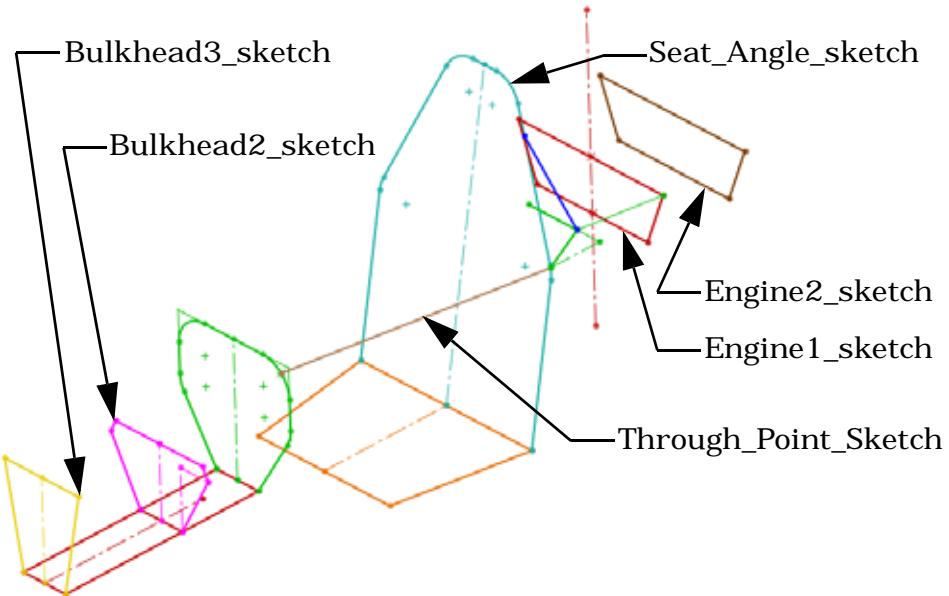
Tip: If this message appears: At least one segment being filleted has a midpoint or equal length relation. Geometry may have to move to satisfy this relation when the fillet is created. Do you want to continue? Click **Yes**.

Close the sketch and rename it to Bulkhead1_Sketch.



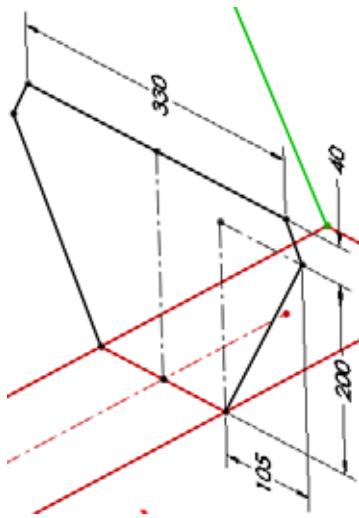
Additional Planes and 2D Sketches

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

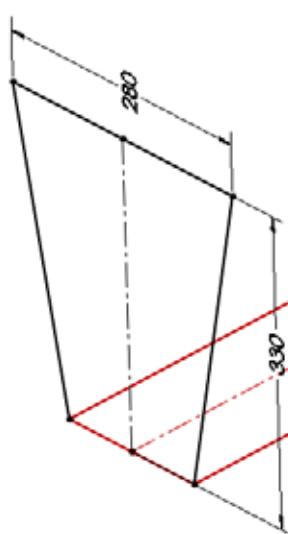


8 Bulkhead2 and Bulkhead3 planes and sketches.

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



Bulkhead2_sketch



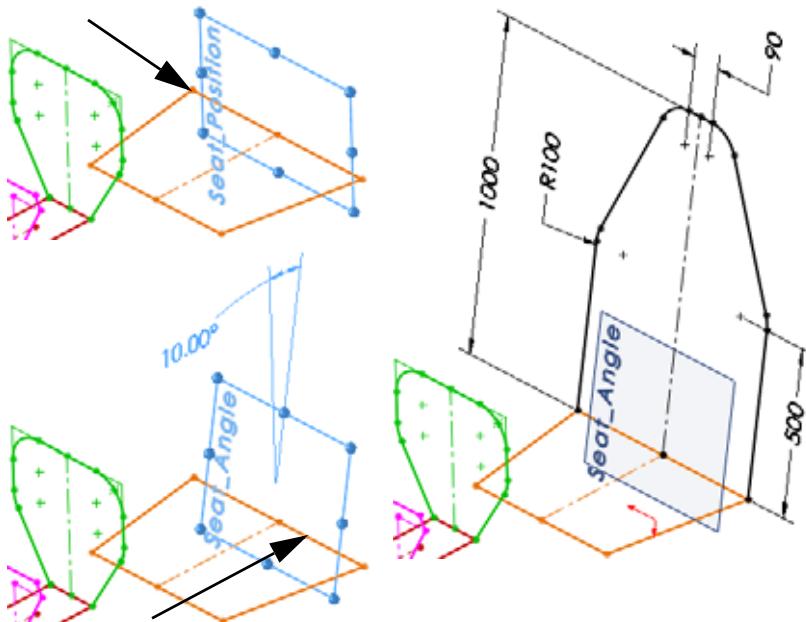
Bulkhead3_sketch

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.

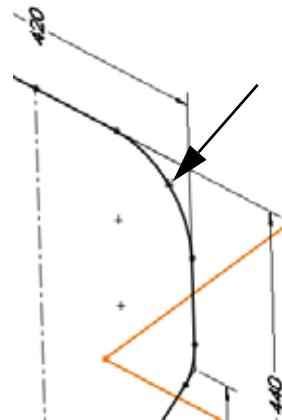
Sketching with Pierce Relations

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.

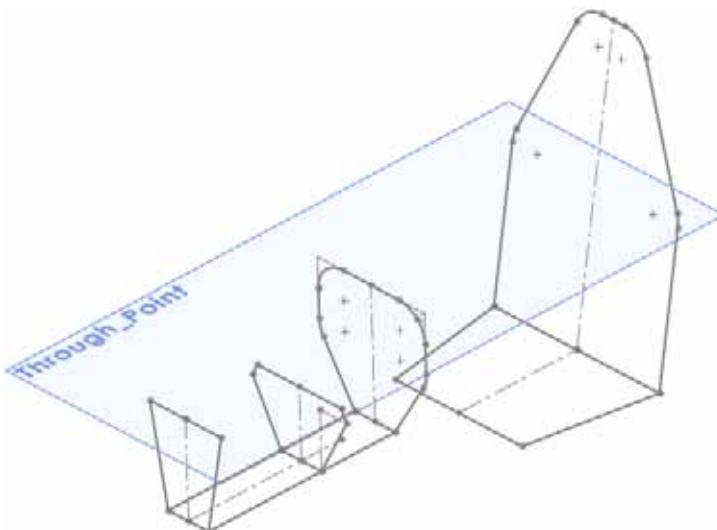
10 Edit sketch.

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.



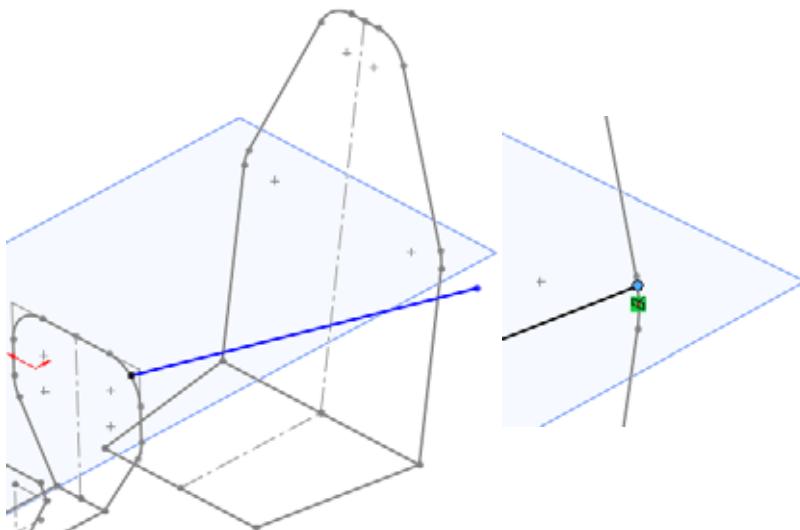
11 Plane through point.

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

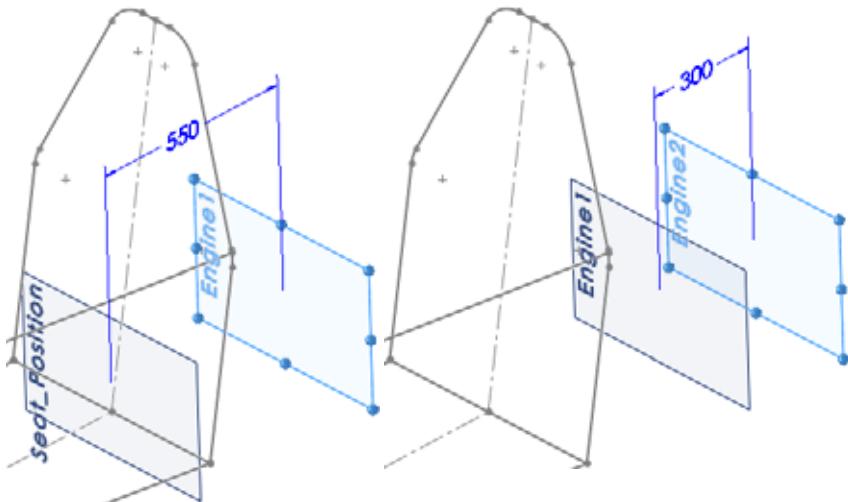


12 Pierce.

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.

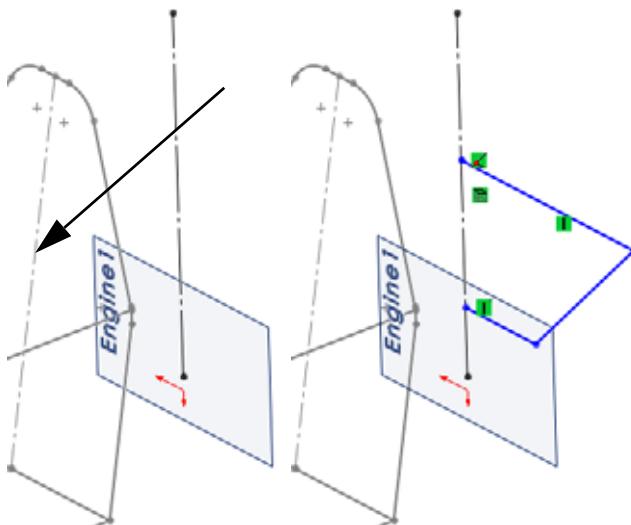
**13 Planes.**

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



14 Converting a single sketch entity.

Create a new sketch on the Engine1 plane. Click **Convert Entities**  and select the centerline in the Seat_Angle_sketch as shown. Click  and . Click the line and select **Construction Geometry** . Add the lines to the centerline as shown.



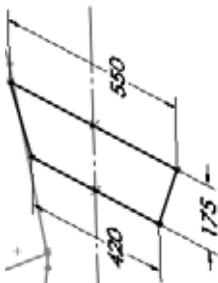
15 Mirror in the sketch.

Click **Tools, Sketch Tools, Mirror**  and select the three sketched lines. Right-click and select the centerline and click .



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.

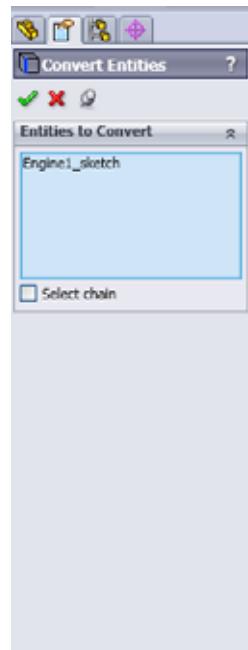
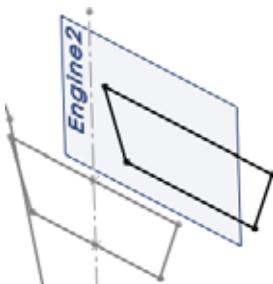


17 Converting an entire sketch.

Create a new sketch on the Engine2 plane.

Click **Convert Entities**  and select the sketch Engine1_sketch using the flyout FeatureManager design tree as shown. Click .

Close the sketch and rename it Engine2_sketch.



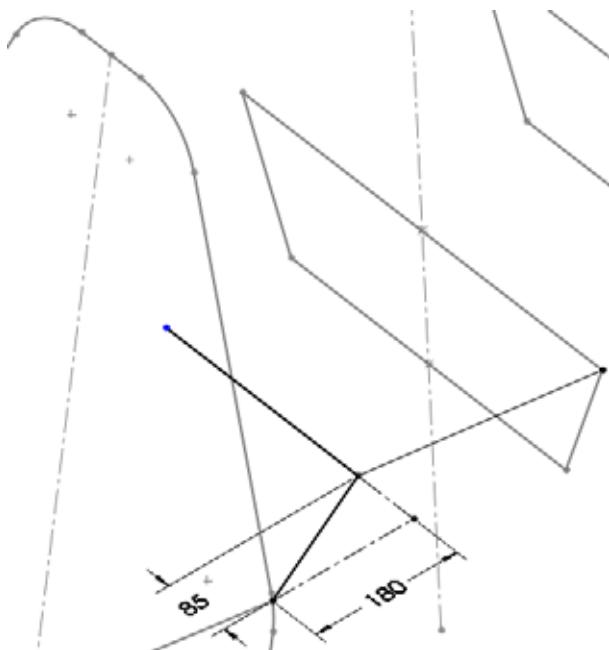
Adding Bracing to the Frame

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 61.

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.



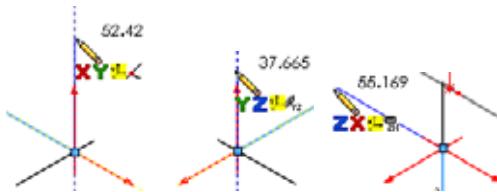
Note: This sketch is another example of a valid weldment sketch that would be unsuitable for a standard extrusion. For more information, see “Some Details About Weldment Sketches” on page 43.

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.

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.



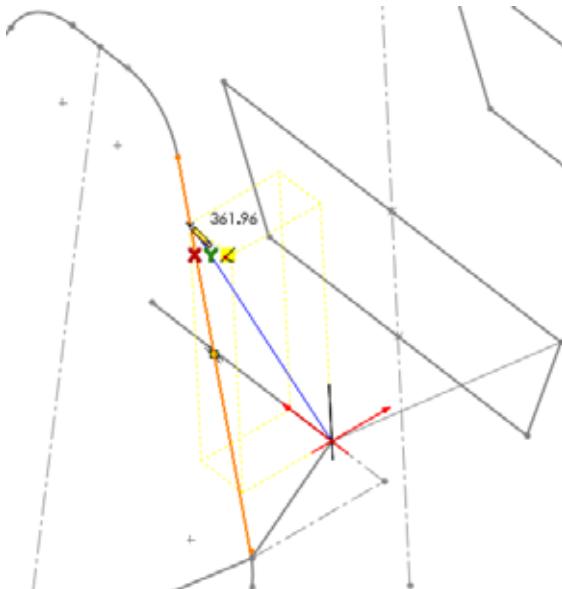
Note: The space handles can be ignored to sketch between endpoints in 3D space.

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.

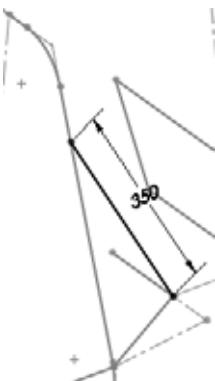
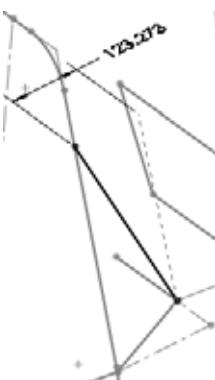
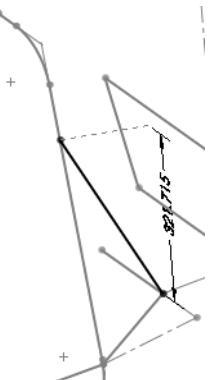
19 3D sketch.

Click **Insert, 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.



3D Sketch Dimensions

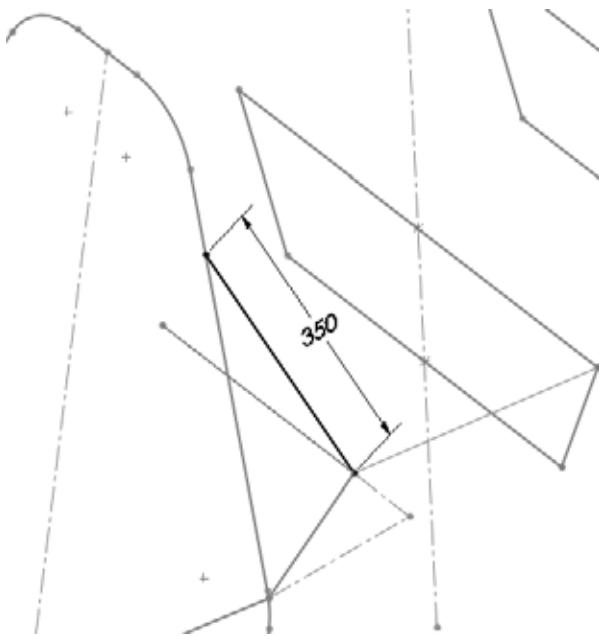
The **Tab** key can be pressed while creating a dimension to switch between the true distance (absolute) and distances along the X, Y and Z axes.

Select the line and place the dimension... (Absolute)		
press Tab once (Along X)	press Tab twice (Along Y)	press Tab three times (Along Z)
		

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

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.

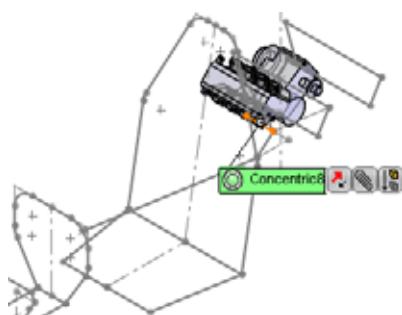
Click the ConfigurationManager tab in the FeatureManager design tree and double-click the display state Full_Display State-1. Click the FeatureManager design tree tab .

25 Hide planes.

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

Placing a Component Using Sketch Geometry

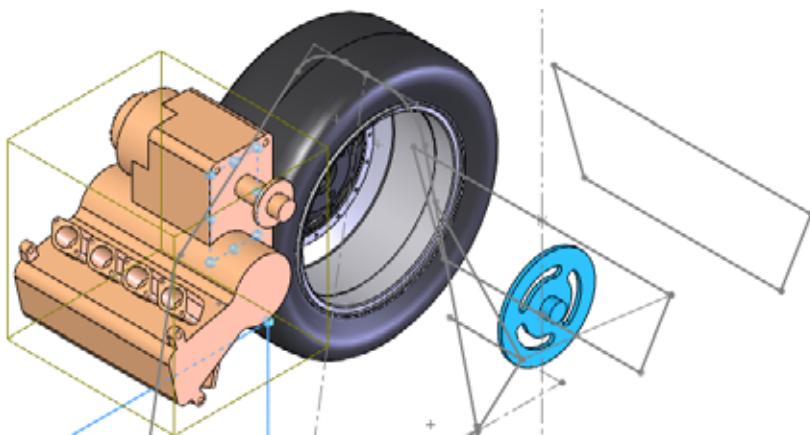
The geometry of the frame, an in-context virtual part, can be used to create mates as well as place and orient components.



26 Insert component and mate.

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

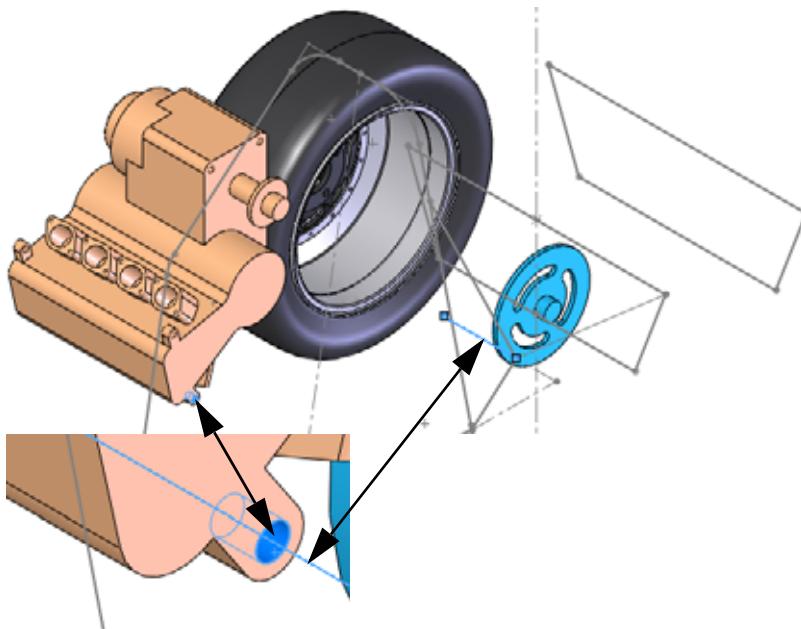
Mate the Right plane of the ENGINE and the Centerline plane of the assembly using a **Coincident** mate.



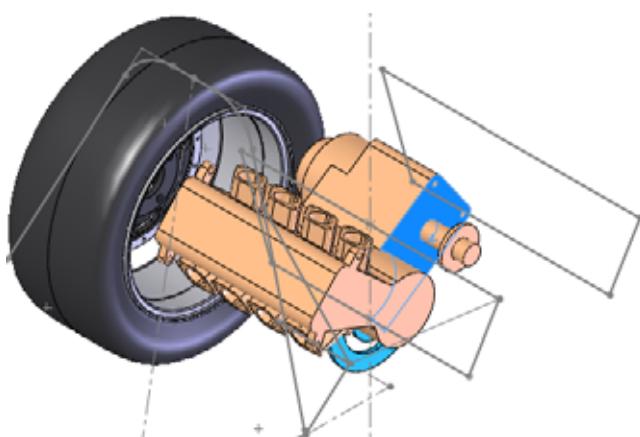
Note: Use the **Aligned** and **Anti-Aligned** options to get the mate alignment as shown.

27 Concentric mate.

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

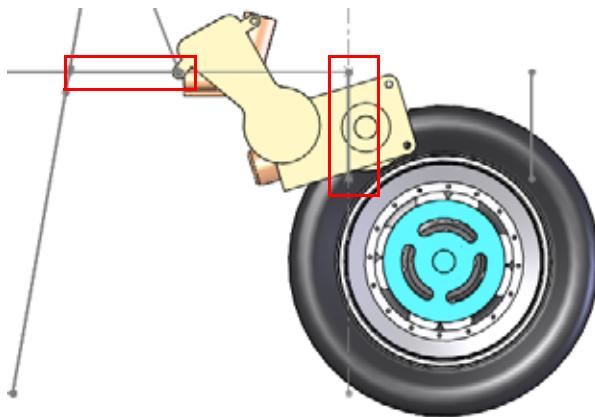
**28 Rotate.**

Rotate the under defined component as shown.



Making Changes

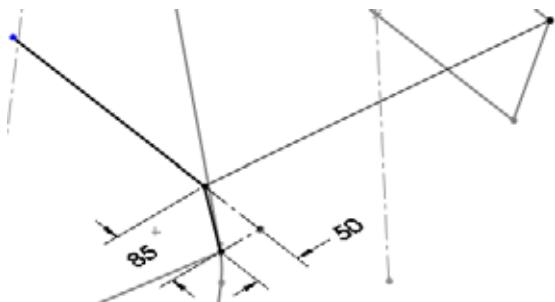
With the addition of the ENGINE, we can see that it appears to be too close to the Engine1_sketch (inside right box). One solution may be to pull the attachment point back by shortening the length of a line in the Engine_Mount sketch (inside left box).



29 Change dimension value.

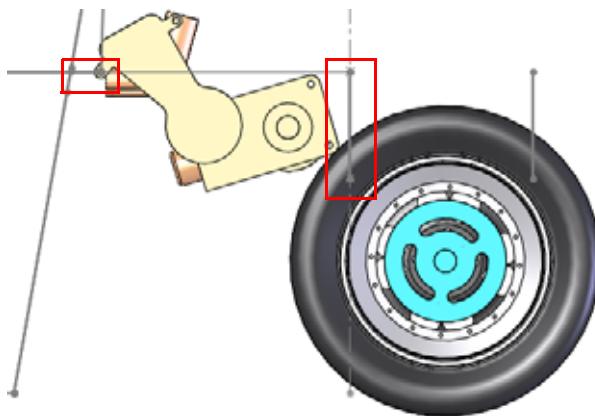
Return to the part by pressing **Ctrl+Tab** and clicking the part.

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



30 Return to the assembly.

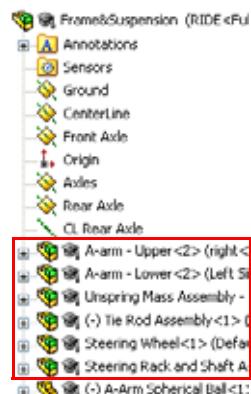
Press **Ctrl+Tab** and click the assembly. With the change in the sketch geometry, the engine has been pulled back to a better position within the frame as shown.



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 FeatureManager 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**

Rigid vs. Flexible Sub-Assemblies

All sub-assemblies have the option of being solved as **Rigid** or **Flexible**. The **Component Properties** dialog is used to toggle between them.

Rigid

Rigid  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.

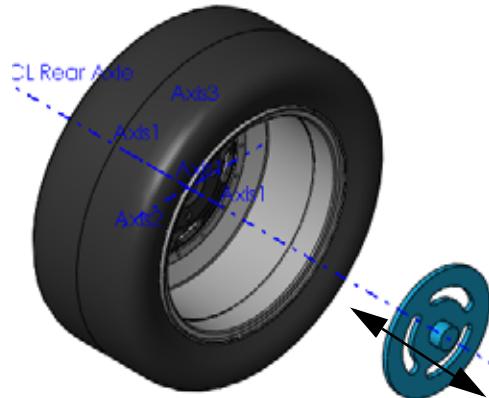
Flexible

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

1 Test component.

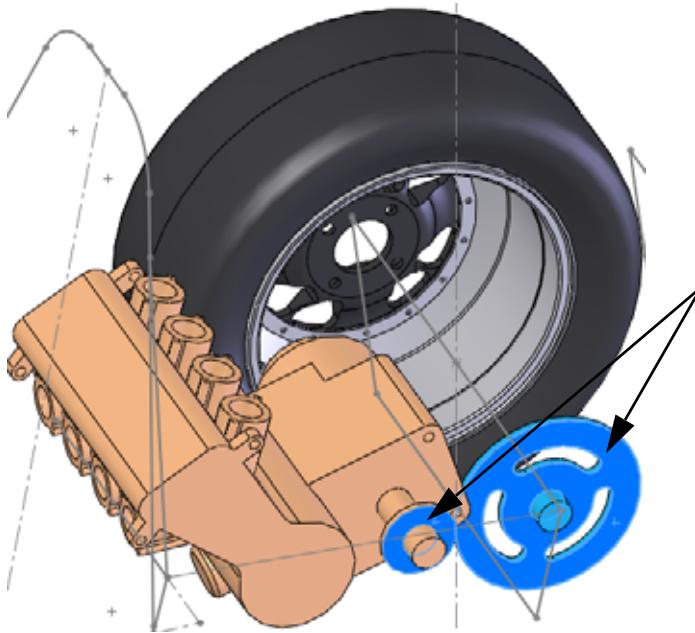
The assembly is designed so that the Sprocket is free to move along the axis CL Rear Axle. Move it a small amount to test it.

Close the sub-assembly without and return to the main assembly.



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 .



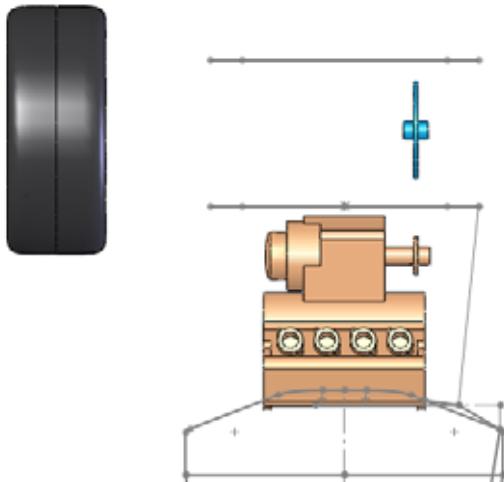
Why Does the Warning Appear?

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.



Bracing Sketches

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

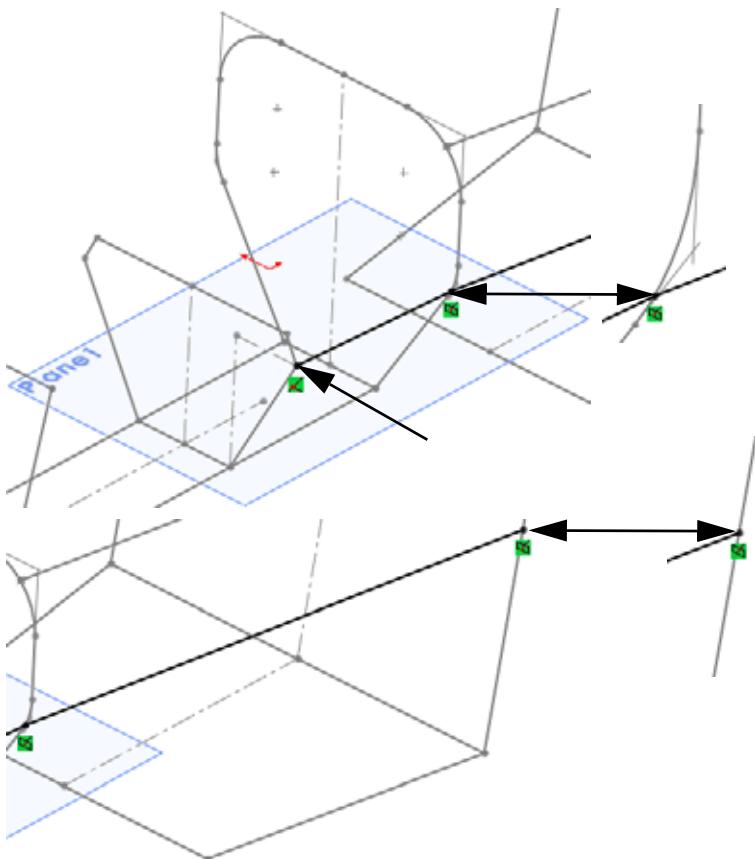
4 Open part.

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.

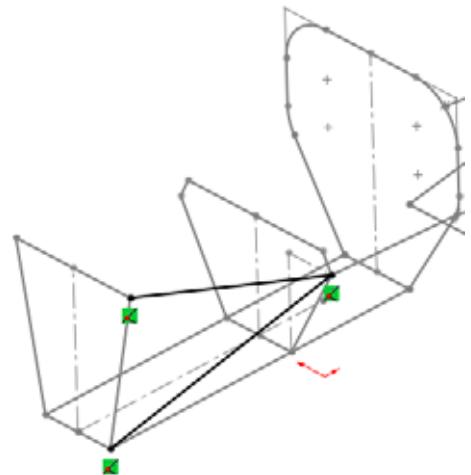
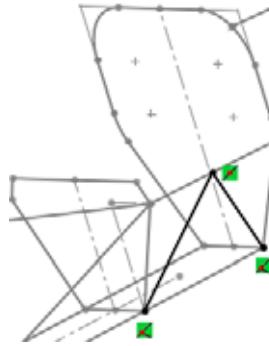
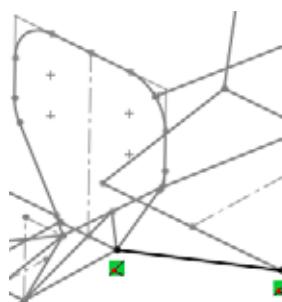
In the each sketch, use one **Coincident** and one **Pierce** relation.

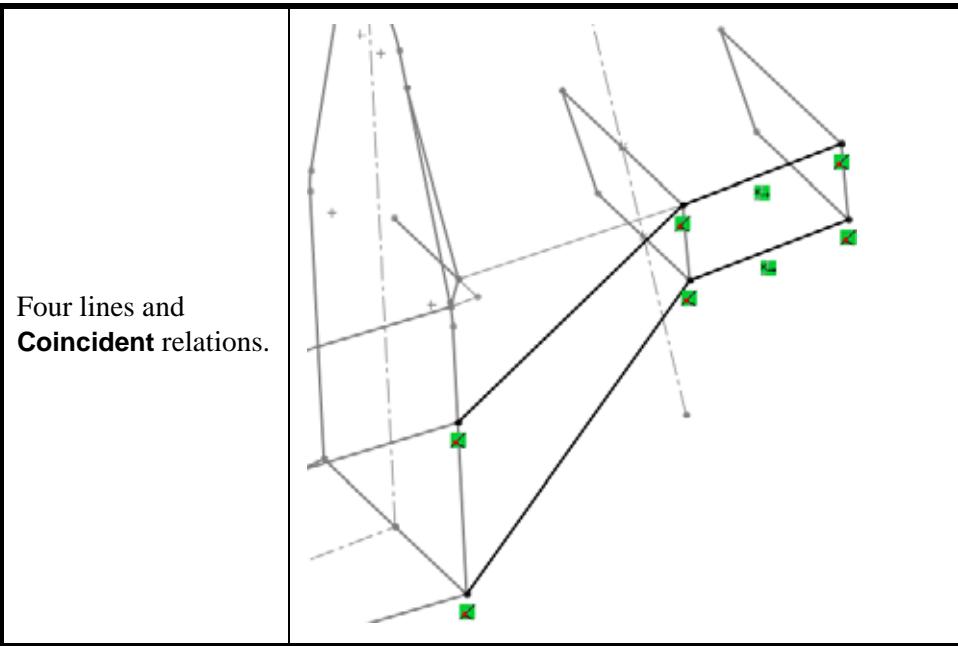


Note: The pierce relation in the first sketch uses an arc, the second uses a line.

Remaining Sketches

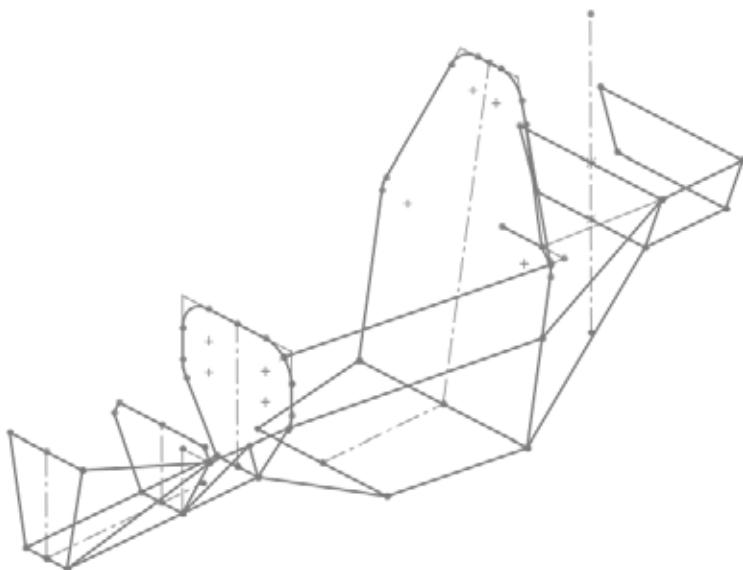
The remaining sketches are all 3D sketches and will be presented with a brief description and an image as shown below.

Two lines and Coincident relations.	 A 3D wireframe model of a structure. Two specific lines are highlighted in black. Green square markers with a cross indicate coincident relations between these lines and other surfaces or features in the model. A red curved arrow points from one of the markers to a specific point on one of the black-lined lines.
Two lines, Coincident and Midpoint relations.	 A 3D wireframe model of a structure. Two specific lines are highlighted in black. Green square markers with a cross indicate coincident relations, and a green circle with a cross indicates a midpoint relation between these lines and other features.
One line and Coincident relations.	 A 3D wireframe model of a structure. One specific line is highlighted in black. A green square marker with a cross indicates a coincident relation between this line and another feature in the model.



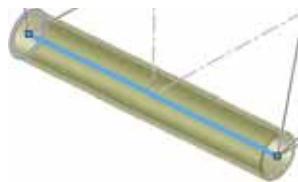
Why is the Bracing One Sided?

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 86.



Weldment Structural Members

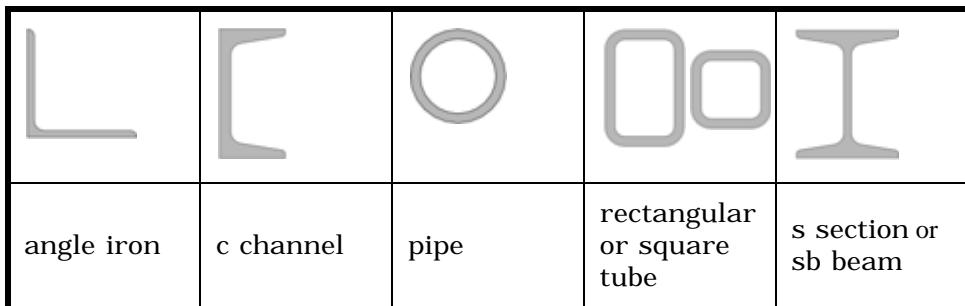
Weldments use standard profile shapes that are applied to sketch geometry to create structural members. Each structural member is a separate sold body in the multi-body part.



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

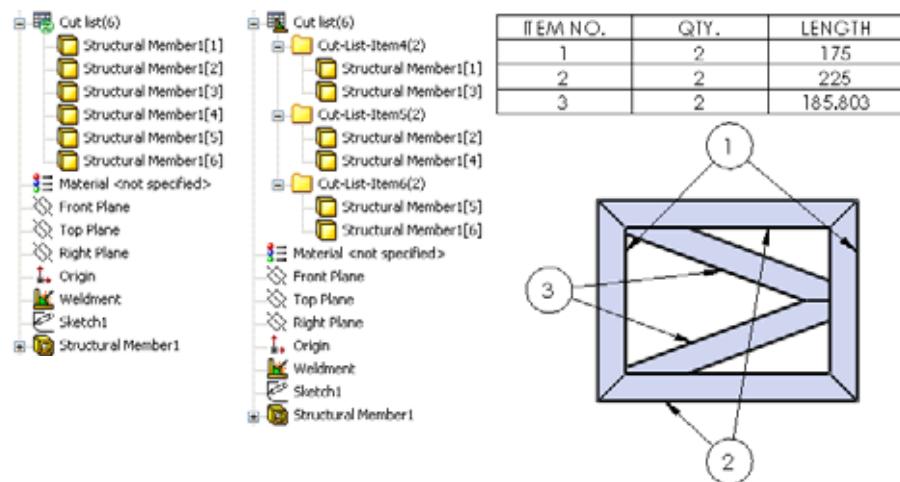
A Quick Look at the Standard Profiles

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:



Weldment Cut List

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



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.



Note: The weldment profiles are set to specific folders by the option **Tools, Options, File Locations** and **Weldment Profiles**.

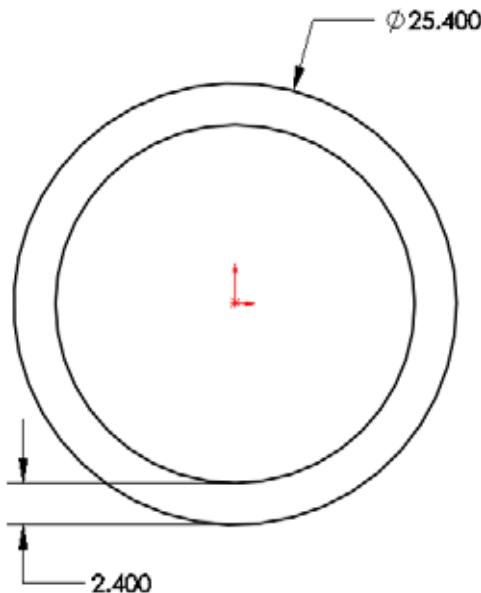
1 Open a new part.

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.



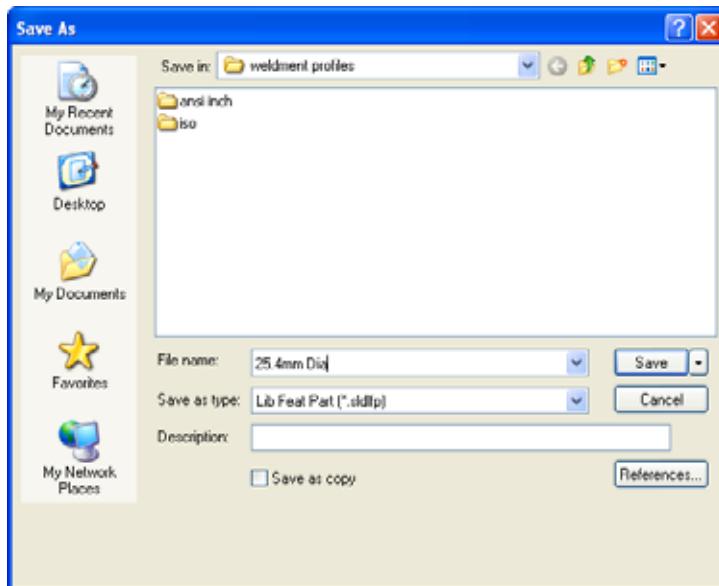
3 Save as library feature.

Select the sketch Sketch1. Click **File, Save As**.

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

For **Save in**, navigate to the default weldment library folder located in the <Install Directory>\Program Files\SolidWorks Corp\SolidWorks\data\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.

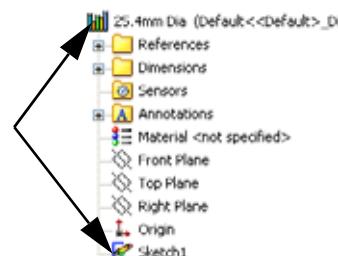


Library Feature Folders and Icons

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.

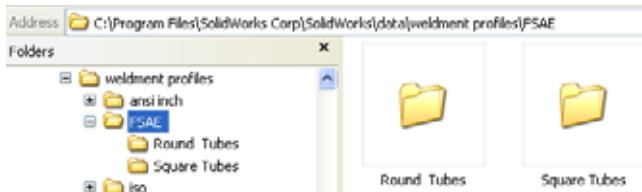


5 Create new Folders.

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
\SolidWorks\data\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.

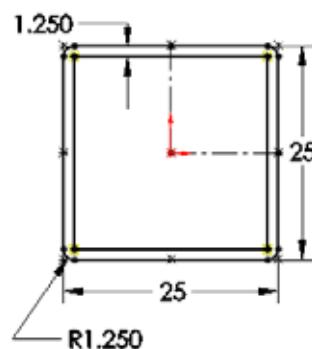
**6 Move existing file.**

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.



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 84 for more information.

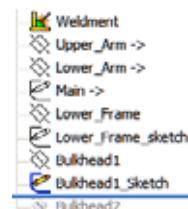
1. Move the rollback bar to a position after the sketch feature that you want to use for creating structural members.
2. Select the geometry of inactive sketches to define the centerlines of the structural members.
3. Add only structural members of the same type in the same feature.

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.

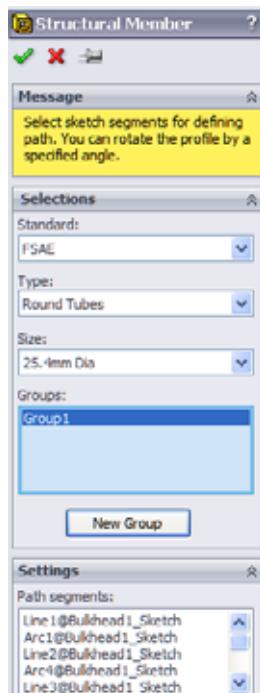
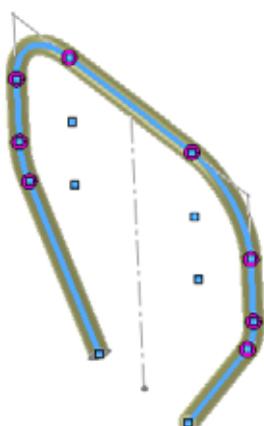


2 Structural member.

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

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

Select all the lines and arcs as shown and click .



Rename the structural member feature to Structural Member_Bulkhead1.

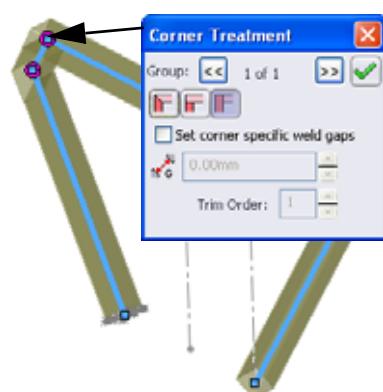
Corner Treatments

The **Corner Treatments** **End Miter**  **End Butt1**  or **End Butt2**  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

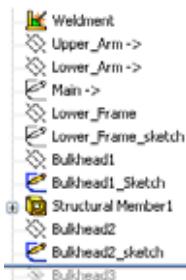
Individual corner treatments can be set by clicking the circular corner markers and setting the dialog that appears.



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.



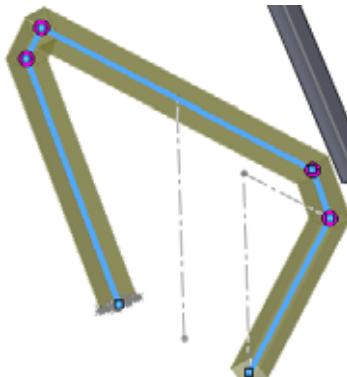
4 Structural member.

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

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

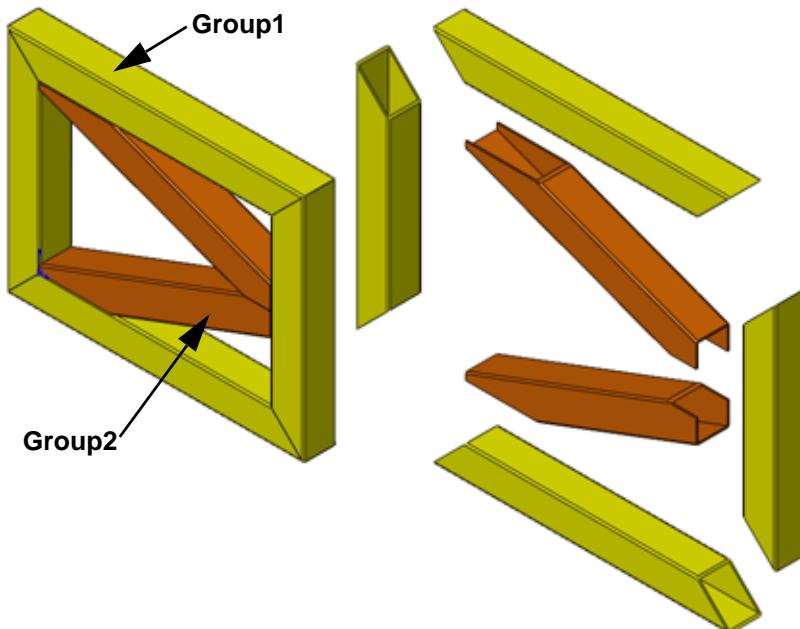
Make sure **Apply corner treatment** and **End Miter**  are clicked. In this example, all corner treatments will remain as end miters.

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



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.

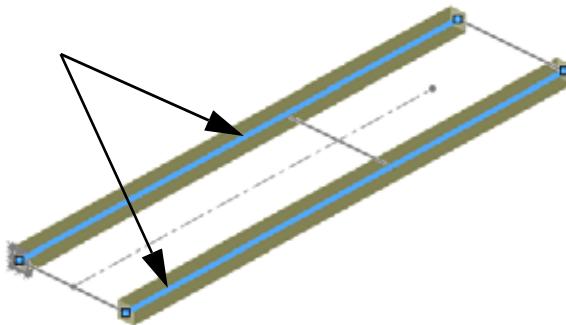
Drag the rollback bar to a position after the Main-> sketch.



6 Structural member.

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

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



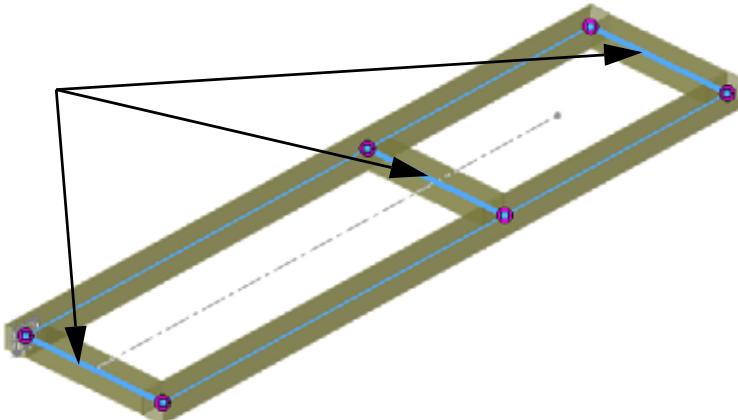
These selections are added to for Group 1.



7 Group.

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

Click .



Using Trim/Extend

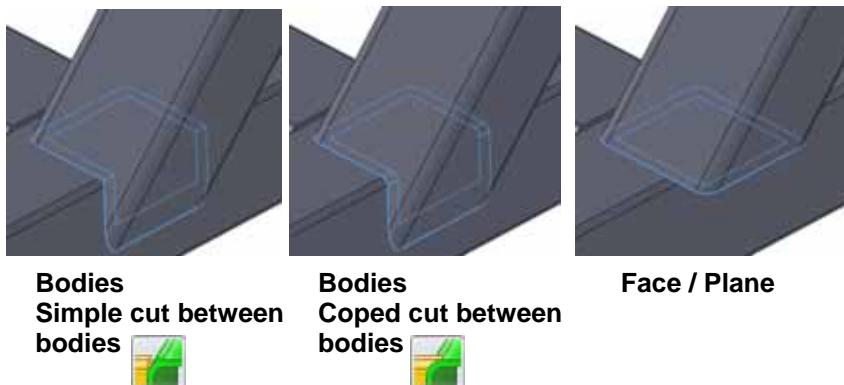
The **Trim/Extend**  tool is used to trim the end of a structural member against a face/plane or one or more other structural members.

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 84 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 94.

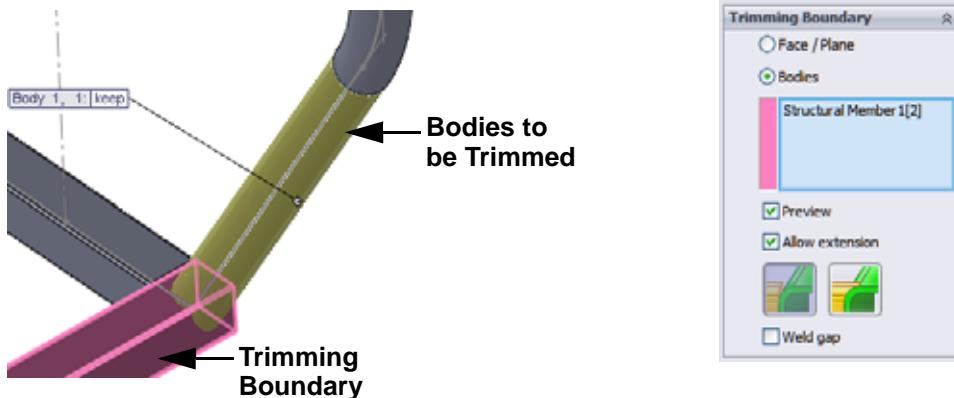
8 Trim.

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

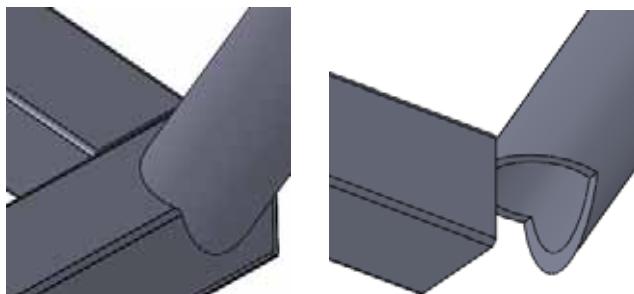
Click **Trim/Extend**  and click **End Trim** .

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

Click **Simple cut between bodies**  and .



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



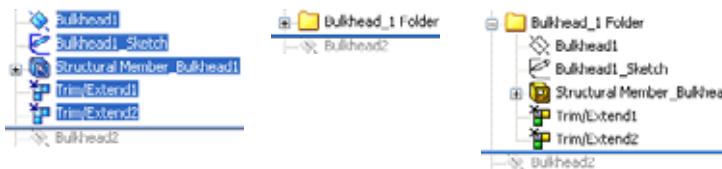
Using Folders

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.

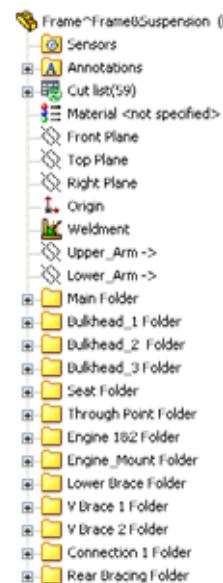
Note: Features can also be dragged and dropped into the folder but they must remain in the same sequential order.

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.

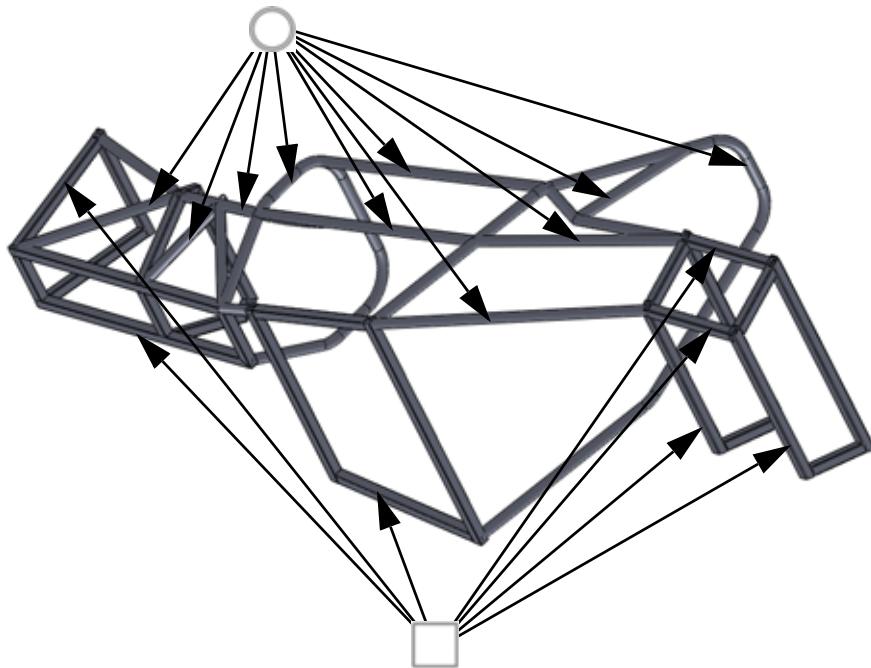


If this procedure is used throughout the FeatureManager design tree it will significantly shorten the length.



Map for Structural Member Types and Trim/Extend

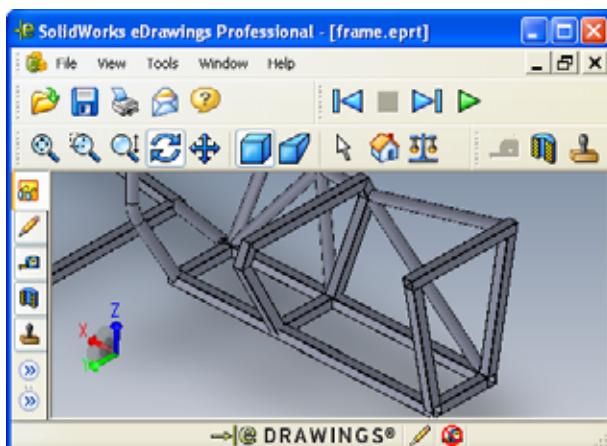
The structural member types are either **FSAE, Round Tubes, 25.4mm Dia**  or **Square Tubes, 25mm Sides** . Create them and add the trimming and folders using the map below or the included eDrawings files (below). Use **End Trim**  and **Trimming Boundary Bodies** for all features.



eDrawings File for Member Types and Trim/Extend

Double-click the eDrawings file frame.eprt  or the html file frame.html . 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.

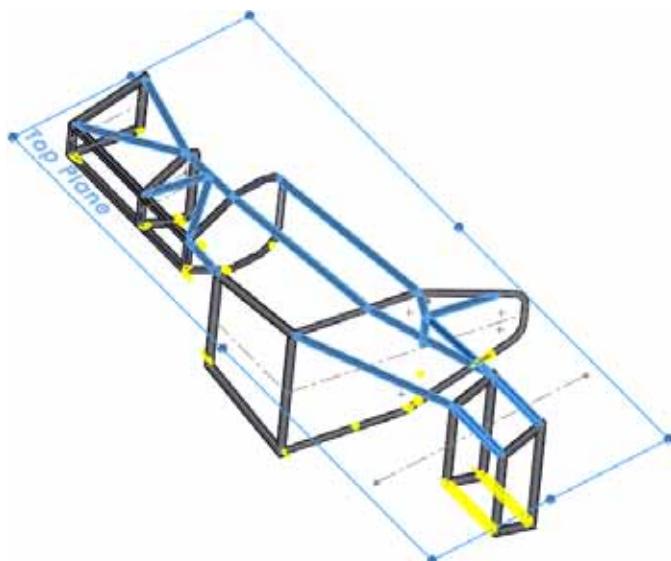


Mirroring Structural Members

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.

Click **Insert, Pattern/mirror, Mirror**  and click the Top plane as the **Mirror Face/Plane**. Click **Bodies to Mirror** and select the bodies shown in blue. Click .



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

Editing

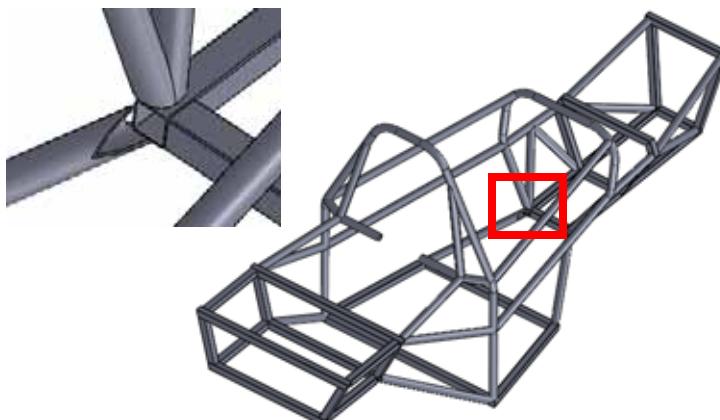
A big advantage of creating geometry in SolidWorks is the editing capability. You can edit sketches and features to change the design at any time. In this section, several types of changes will be introduced.

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.

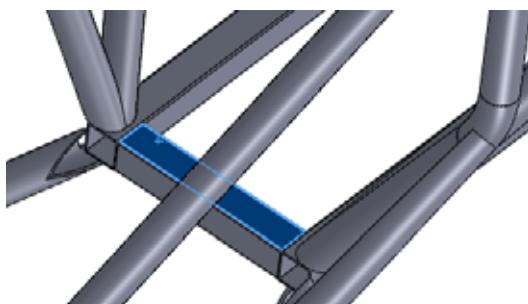
11 Zoom in.

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



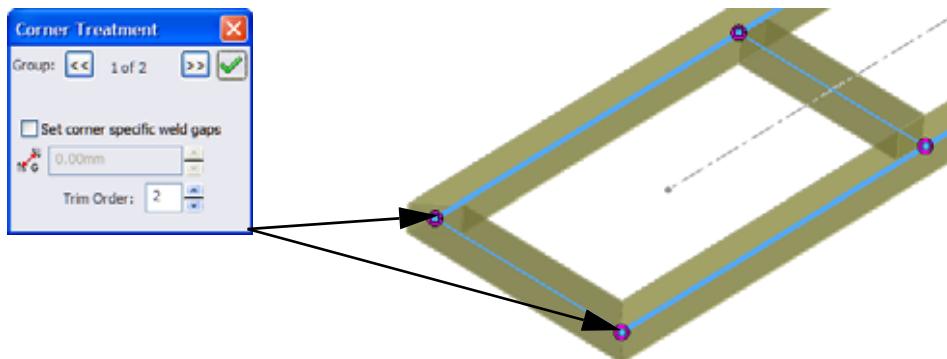
12 Edit structural member feature.

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** .



13 Corner treatment.

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



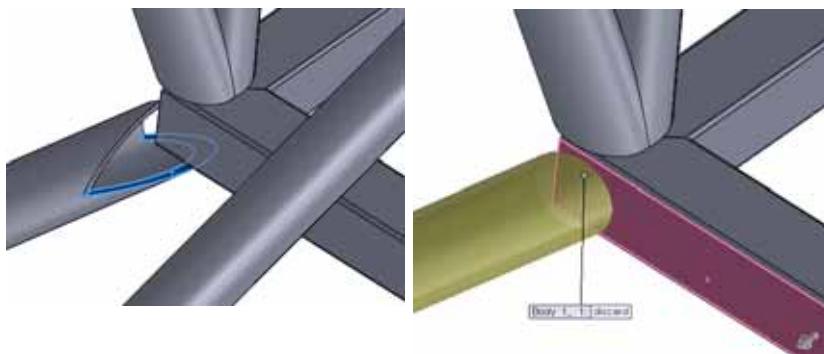
Note: The potential changes that can be made are limited by how the feature was created. This feature was created using groups so miters are the only other choice.

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** . Under **Trimming Boundary**, click **Face / Plane** and select the face as shown. Click .



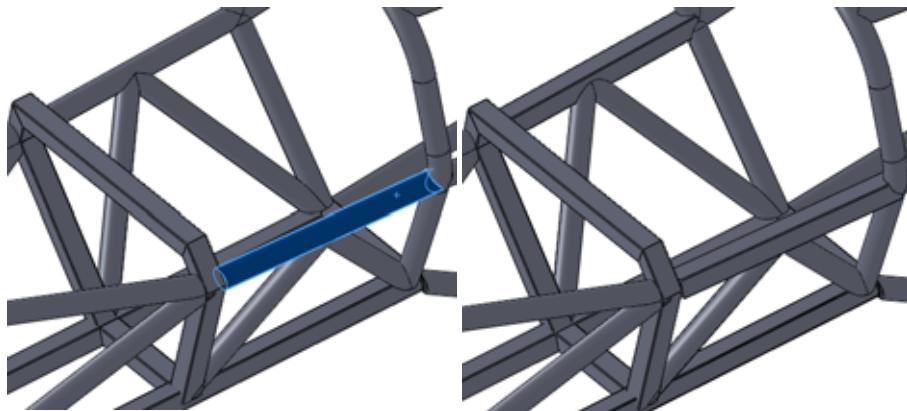
Editing the Type of Structural Member

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 **Standard 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.

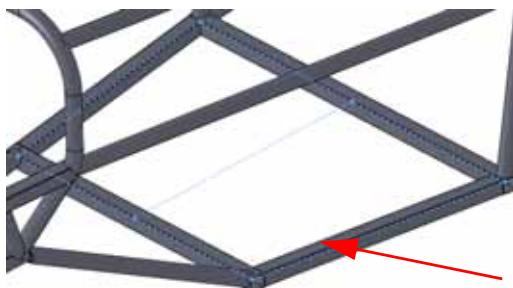
Consider a case where structural members of the same type and size are stacked on top of one another. Is it necessary to sketch a centerline offset by the exact amount?



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 76.

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**.



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** .



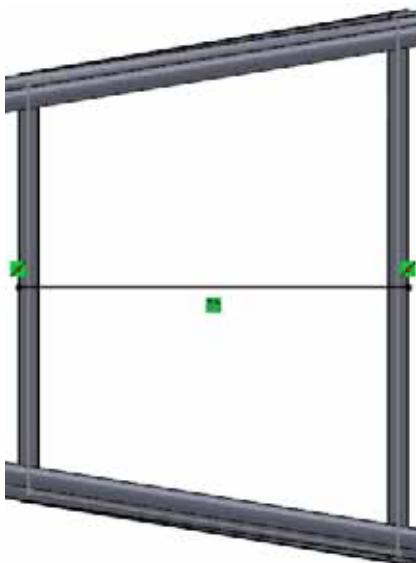
19 Convert.

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.

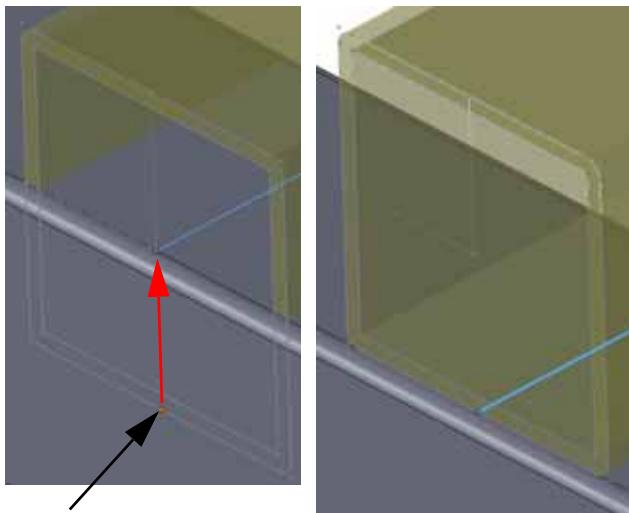
Close the sketch.



20 Locate profile.

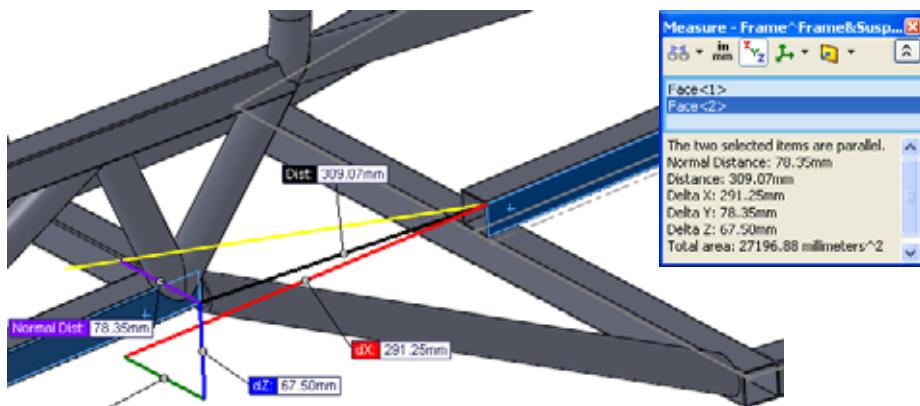
Click **Insert, Weldments, Structural Member** and the following options:

Standard 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 .



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.

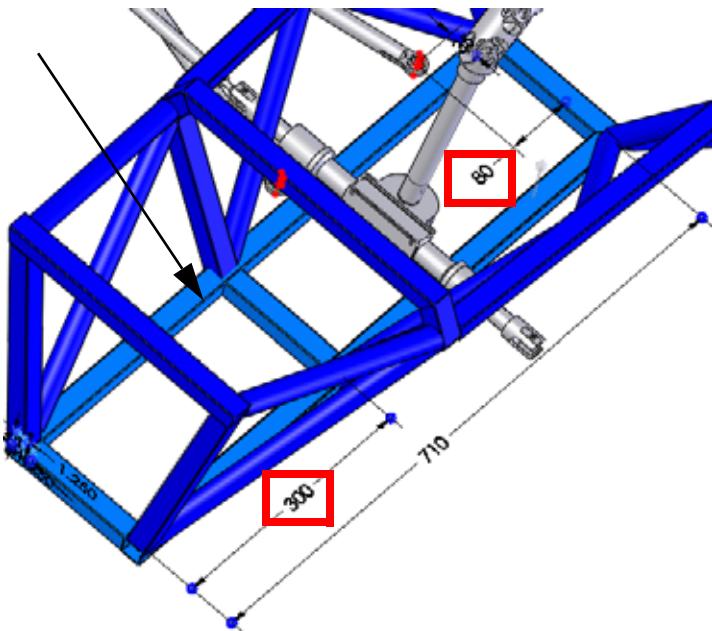
Using Instant 3D

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

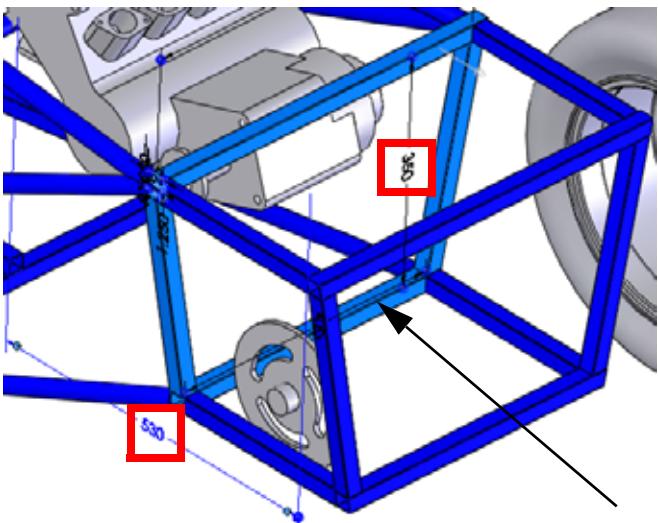
Single-click a dimension to change the value.	
Drag the dimension handle near the ruler to lock onto whole number ruler values.	

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.



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**  tool.

24 Edit the Assembly.

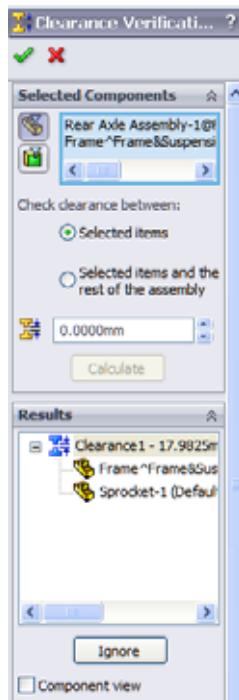
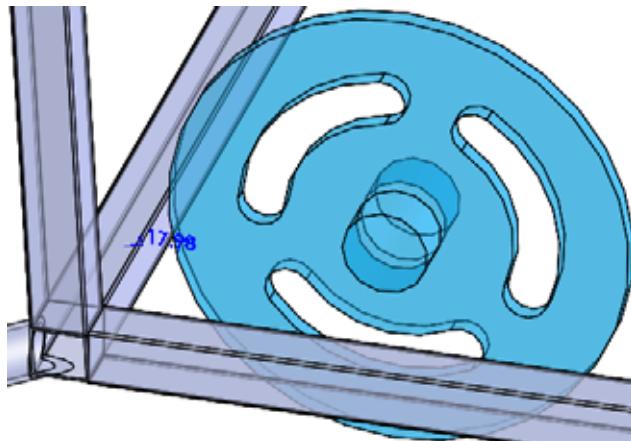
Click confirmation corner  to edit the assembly.

25 Clearance.

Click **Tools, Clearance Verification** .

Click **Selected Items** and select the Sprocket and frame.

Type **0mm** in **Minimum Acceptable Clearance** and click **Calculate**. The clearance is close to **18mm**. Click .



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 97.

Fully Defining a Component

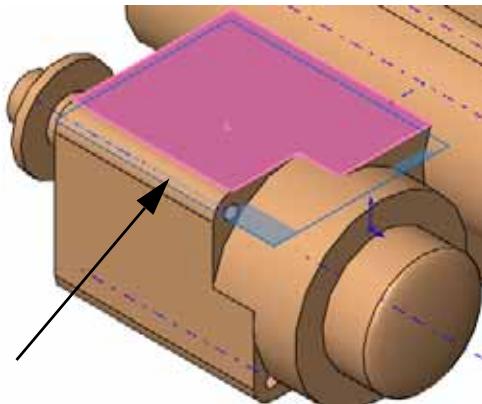
The ENGINE component remains under defined, still able to rotate freely. It should be fully defined to prevent inadvertent movement.

26 Open the ENGINE.

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

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

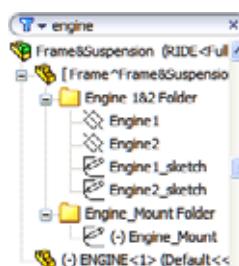
The FeatureManager design tree filter can be used to filter by name. Typing a name into the filter will result in a list of only the features, sketches, components or mates that use those characters.

28 Filter.

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

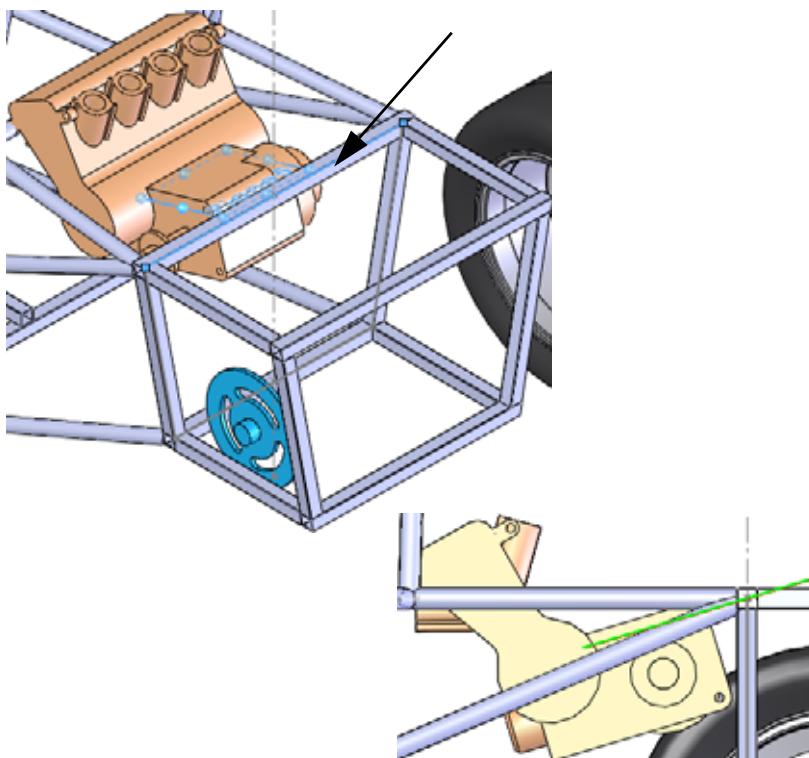
Right-click Engine1_sketch and select **Show**.

Click “x” to stop the filter.



29 Mate.

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



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**.

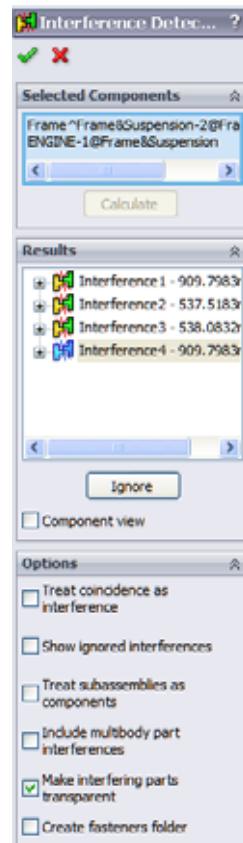
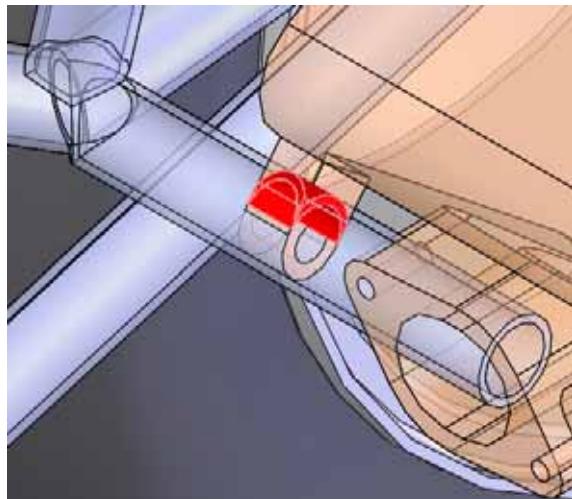
30 Interferences.

Click **Tools, Interference Detection** .

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 .



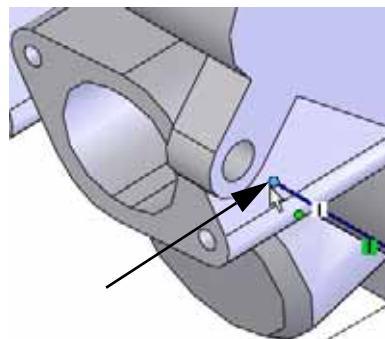
31 Edit sketch.

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** . Click “x” to stop the filter.

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

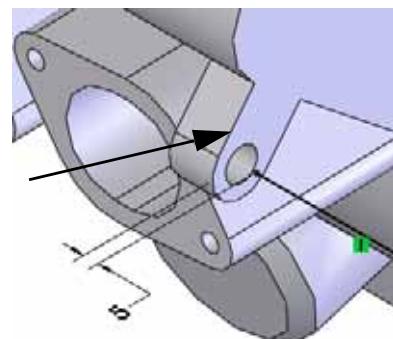
32 Drag.

Drag the open endpoint short of the ENGINE component.



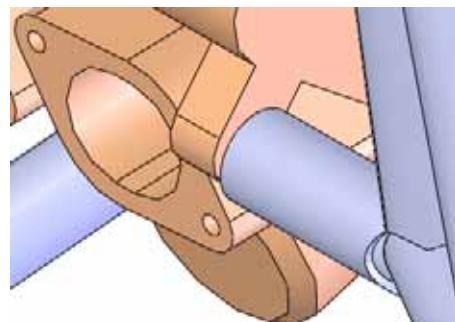
33 Dimension.

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



34 Edit assembly.

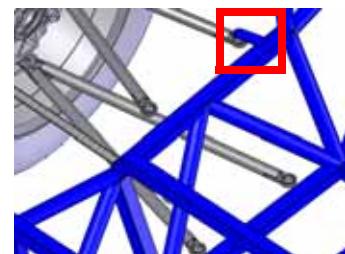
Click **Edit Component**  to exit the sketch and return to edit assembly mode.



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

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.

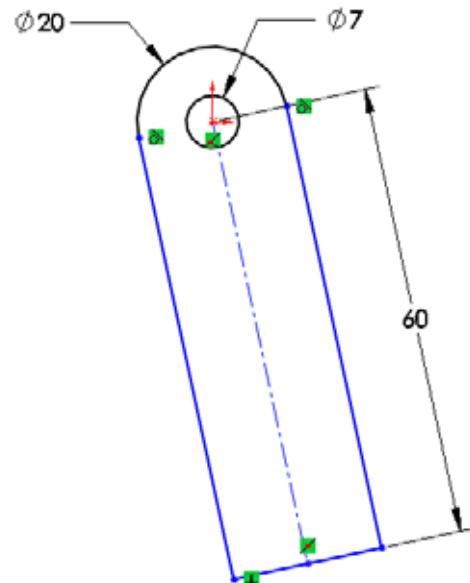
1 New part.

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:

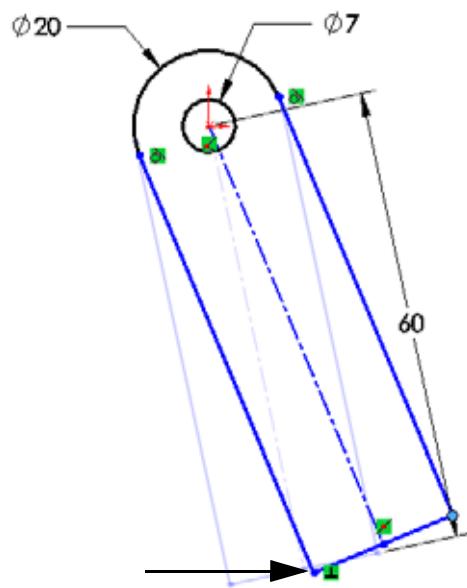
1. Create a centerline from the origin on a slight angle.
2. Offset the centerline with the bi-directional option (add dimensions off) to create the sides.
3. Add a tangent arc to close the upper end.
4. Add a line between endpoints to close the lower end.
5. Add a circle.
6. Add dimensions.



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

3 Drag.

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



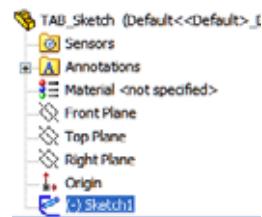
4 Save.

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.

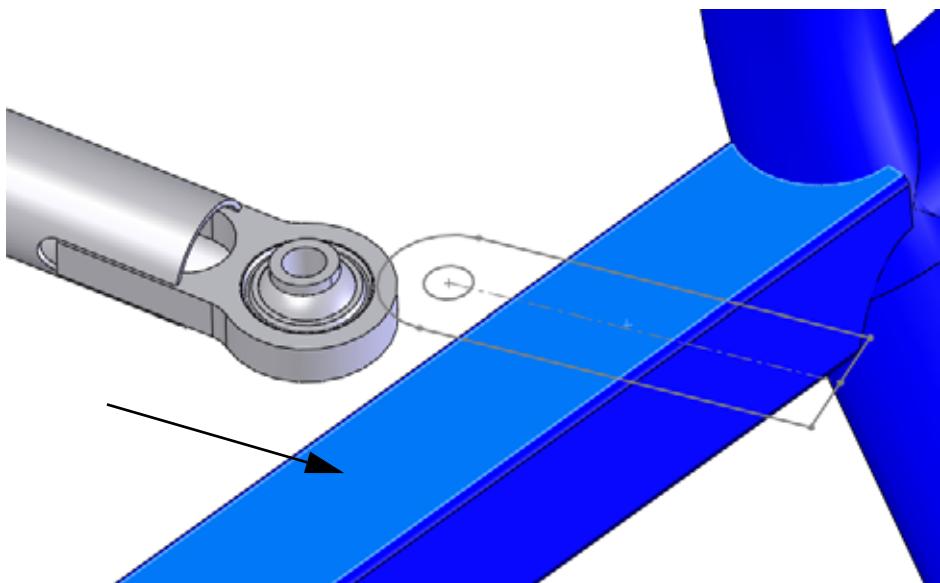
Tip: A common mistake in this situation is to paste the sketch before editing the part. The sketch can be pasted, but is only useful as an assembly feature. An assembly feature is a feature that exists only at the assembly level and can only be used to create a cut.

6 Edit part.

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

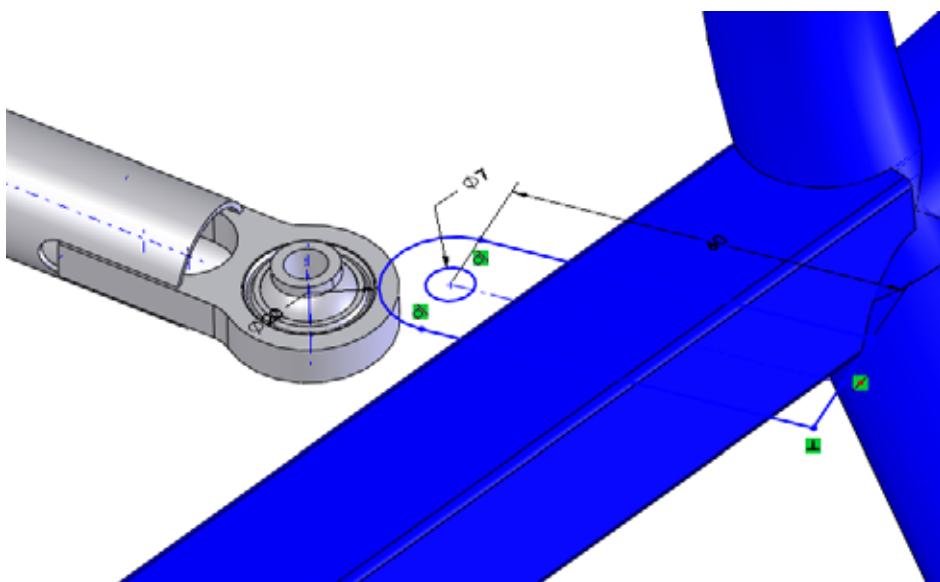
7 Paste.

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



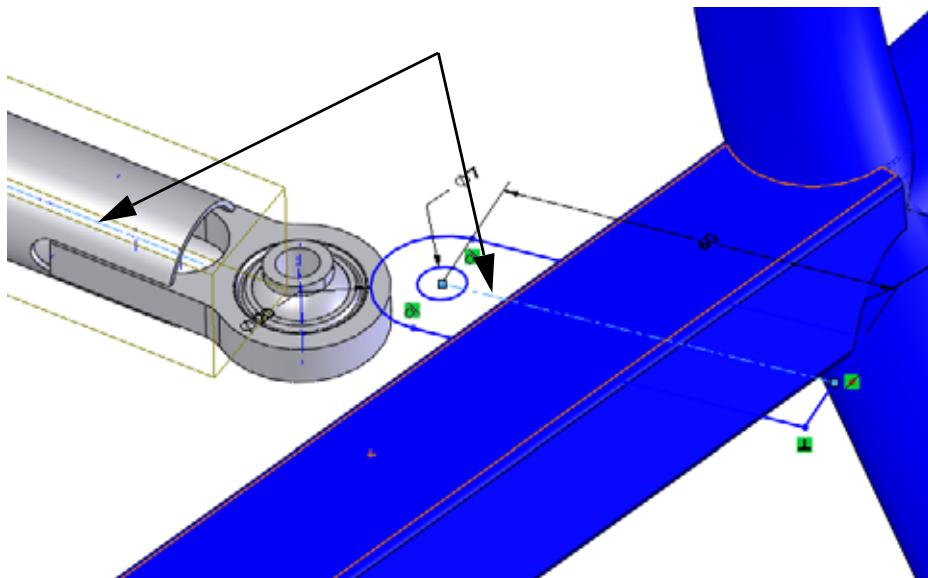
8 Edit sketch.

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



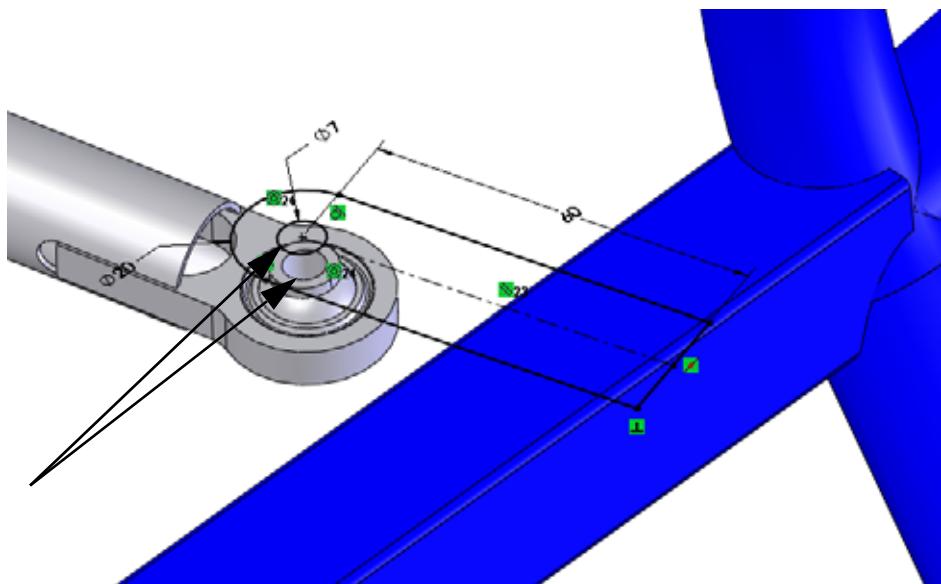
9 Temporary axes.

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



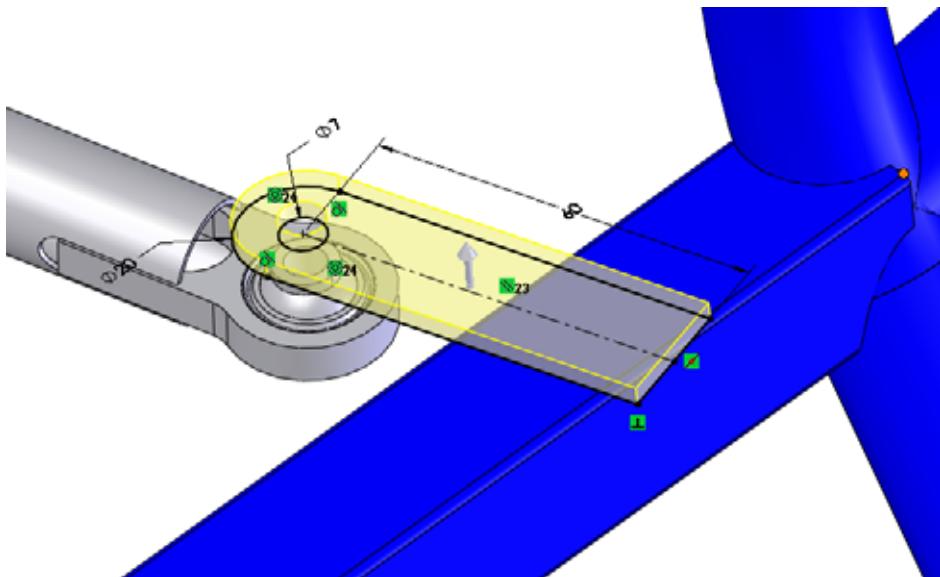
10 Concentric.

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



11 Extrude.

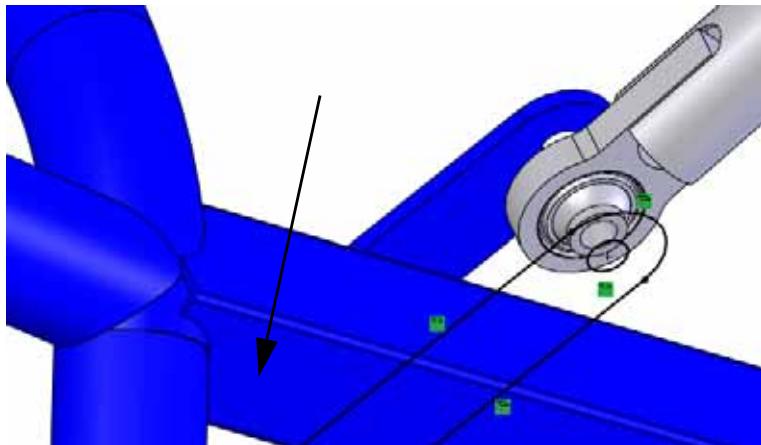
Extrude the sketch **3mm** as shown.



Note: This part has been marked as a weldment **Weldment**, so the **Merge Result** option in extrude remains cleared.

12

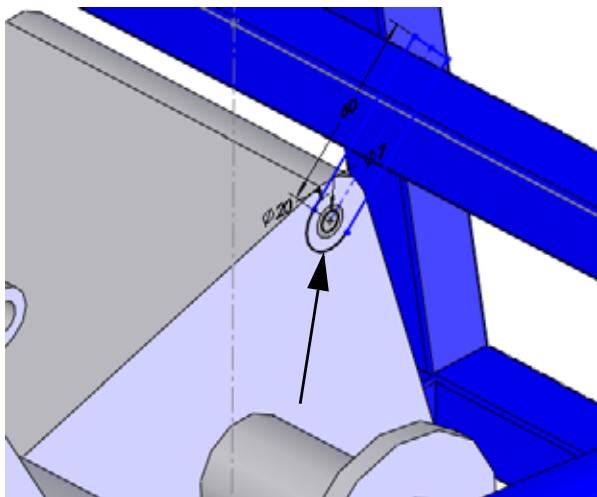
Select the bottom face and create a new sketch. Convert edges from the previous body to create the sketch and extrude the sketch **3mm** as in the previous step.



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

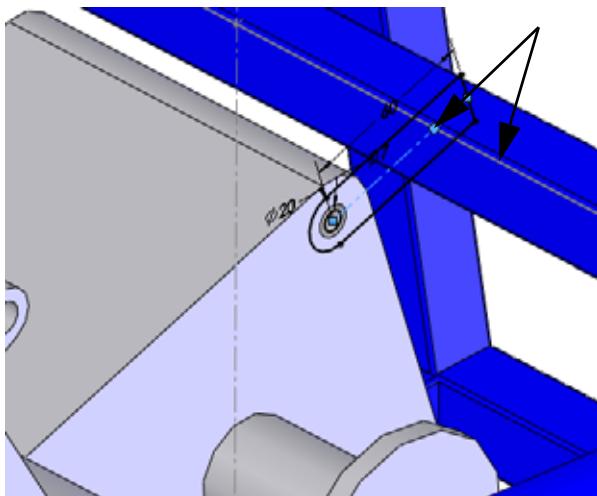
13 Paste sketch.

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.

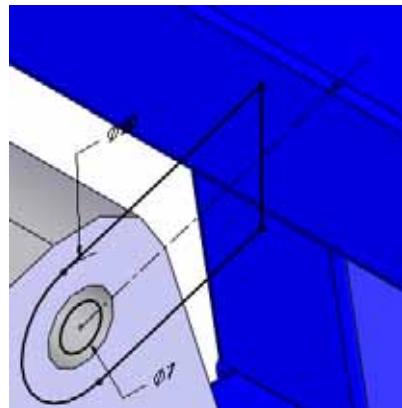
Add a point on the centerline. Add a **Coincident** relation between the centerline and the point. Use the point to add a **Pierce** relation between the point and the inactive sketch line.



15 Trim and extrude.

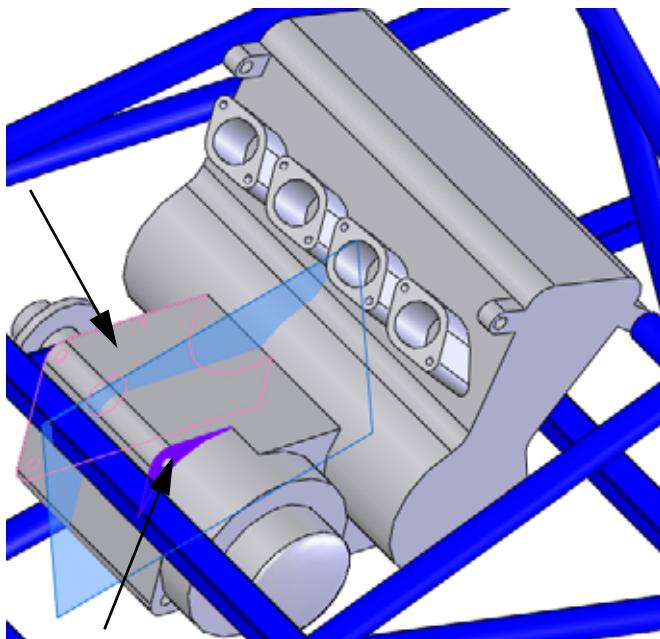
Create the front edge of the structural member using **Convert Entity**. Trim the geometry leaving the centerline and point intact as shown.

Extrude the sketch **3mm**.



16 Plane.

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

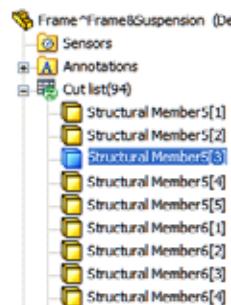


17 Mirror.

Mirror the body using the in-context plane.

Weldment Cut Lists

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

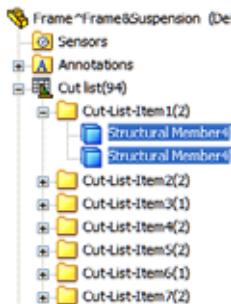
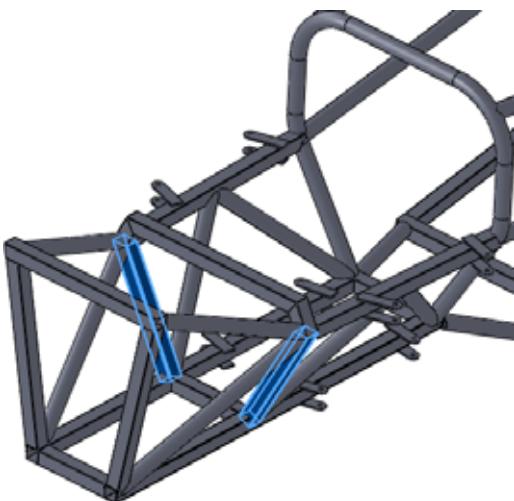


Note: In a non-weldment multi-body part the folder is named Solid Bodies.

18 Updating the folder.

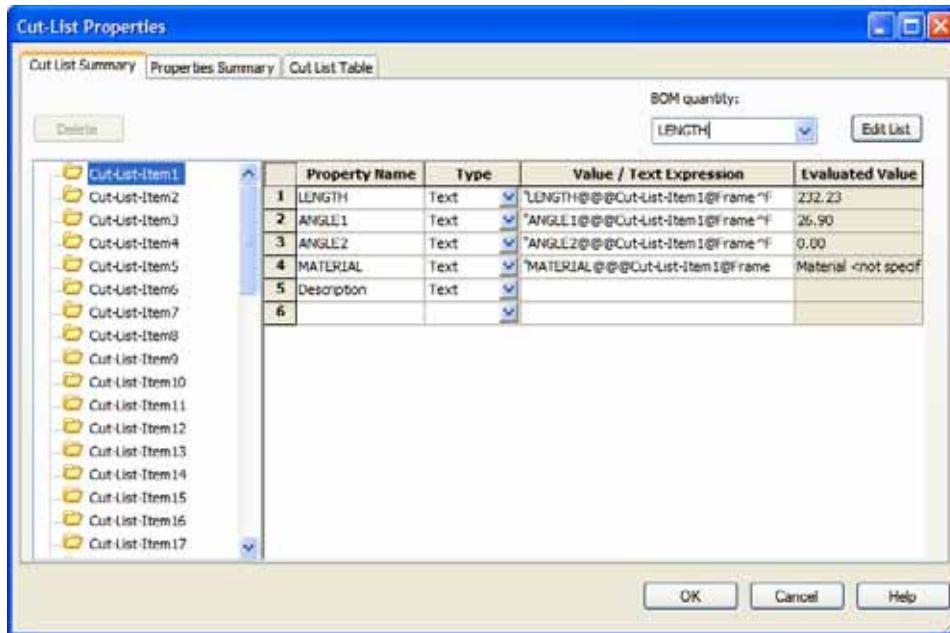
Open the part Frame.

Right-click the folder and select **Update** to group similar structural members in folders.



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**.

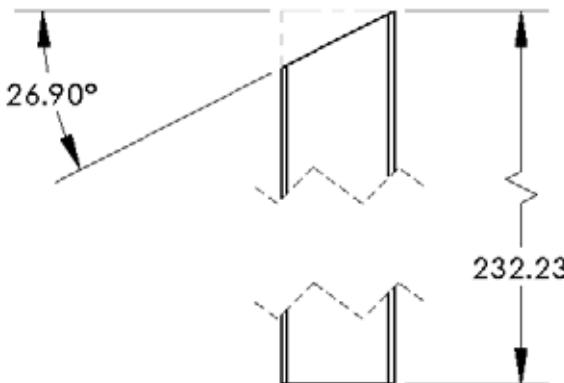


For example, Cut-List-Item1 the following property values and are listed:

LENGTH = 232.23

ANGLE1 = 26.90

A drawing of the structural member would look like this:



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.

Lesson 4

Using Molds and Surfaces

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

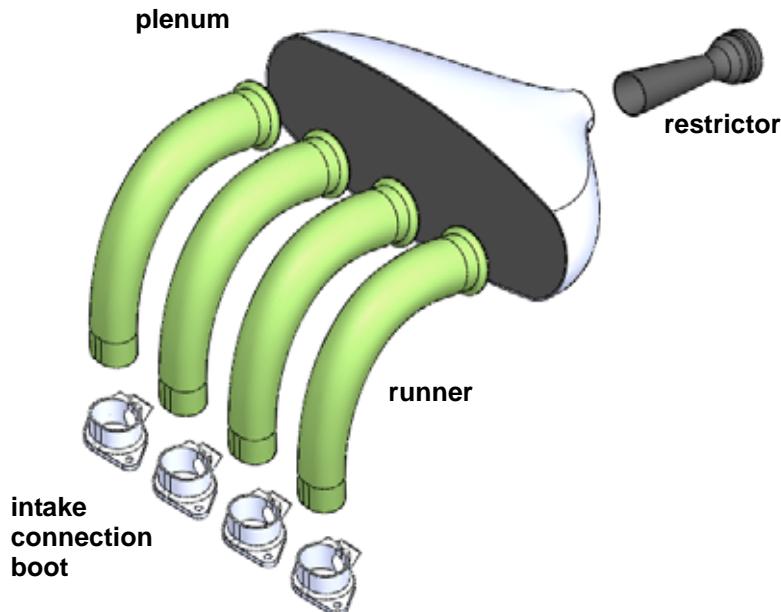
- Understand the loft feature;
- Describe the bodies created using the mold tools;
- Use surfaces to add to the mold tools;
- Use symmetry to create additional molds.

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.

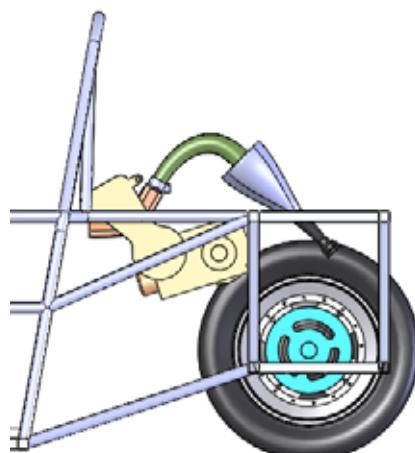


2 Open assembly.

Open the assembly Frame&Suspension from the
Lesson 3\Frame&Suspension folder.

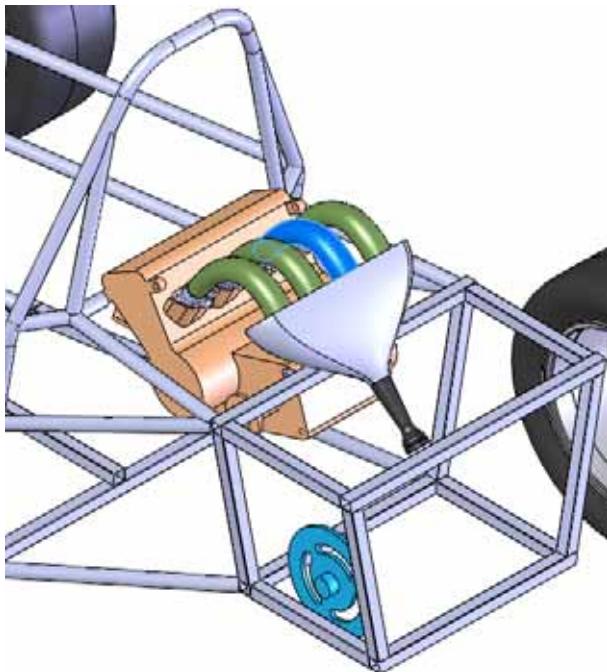
This is the assembly used in the previous
lesson.

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



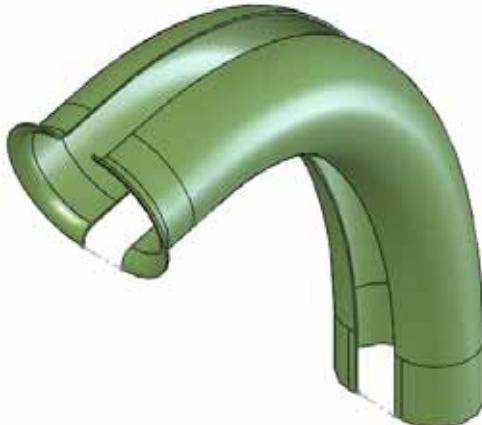
3 Open runner.

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



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.



Features in the Part

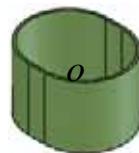
The model of the runner part is created using three main boss features: the Bell, the Straight Boot Section and the Center Guide Curve.

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.



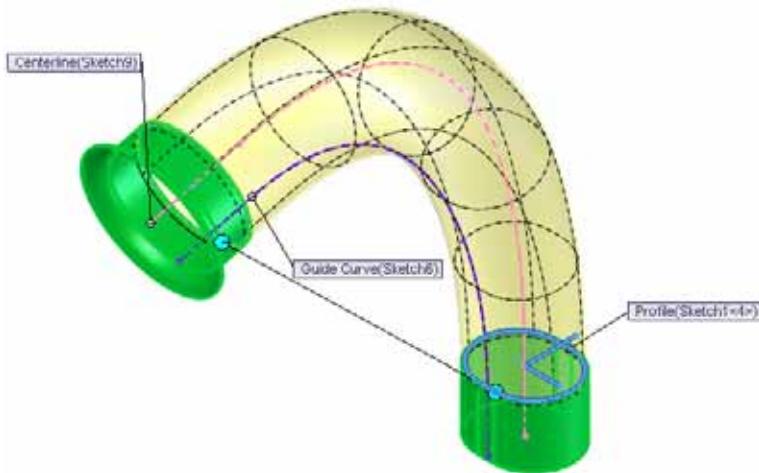
For more information about loft features, see “Anatomy of a Loft” on page 113.

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.

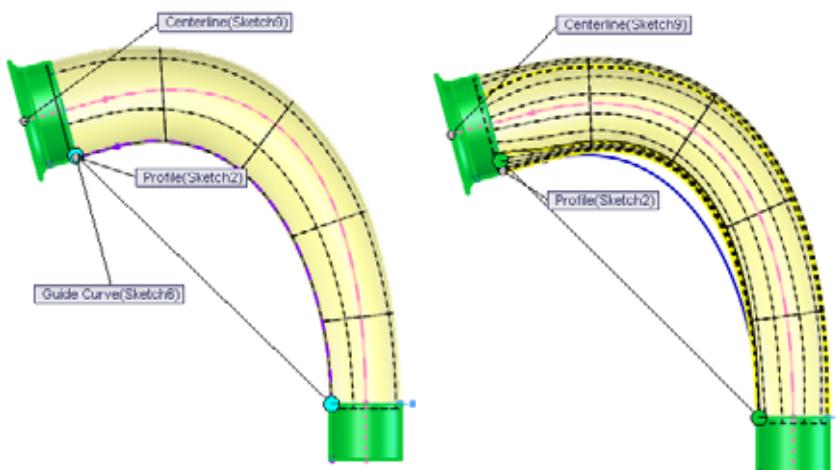
Profiles

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.



Loft with Guide

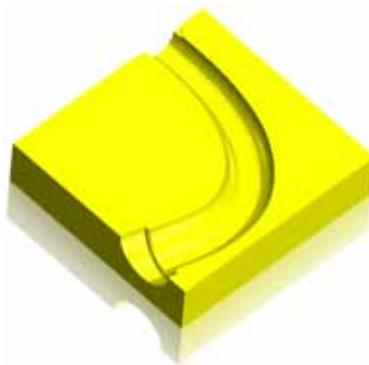
Loft without Guide

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.

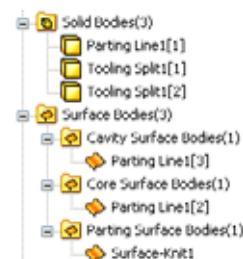
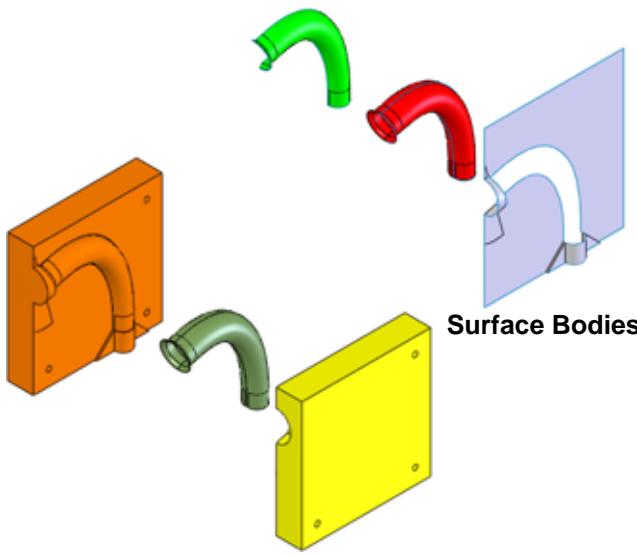
If carbon fiber is used, the cavity solid body is the desired mold. When the carbon fiber is created, the thickness will be applied to the inside maintaining the outside dimensions that fit into the intake connection boot components.

This example assumes carbon fiber is used.



Surface and Solid Bodies

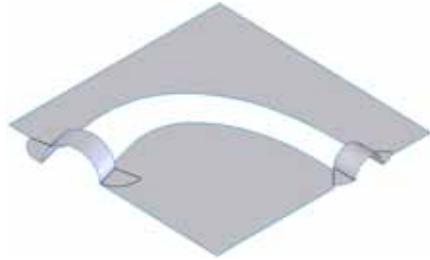
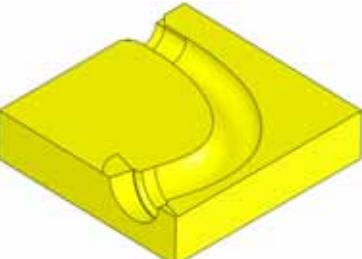
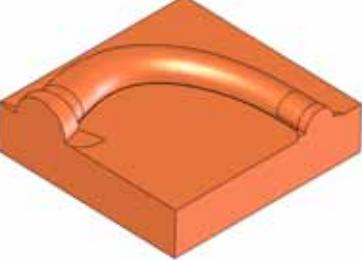
The mold tool procedure creates multiple bodies, including sets of both surfaces and solids, in a single part file. For more information see “Surface and Solid Bodies” on page 114.



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

Description of the Bodies

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

Molded Part 	Parting Surface 
Cavity Solid 	Cavity Surface 
Core Solid 	Core Surface 

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  and Undercut Analysis  are used to see if the part can be removed from the mold. Parting Line Analysis  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 	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.

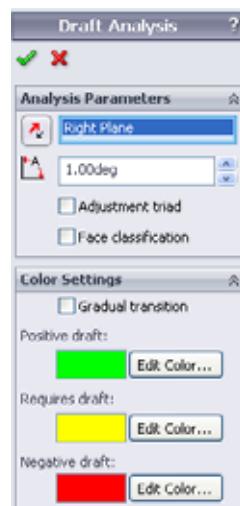
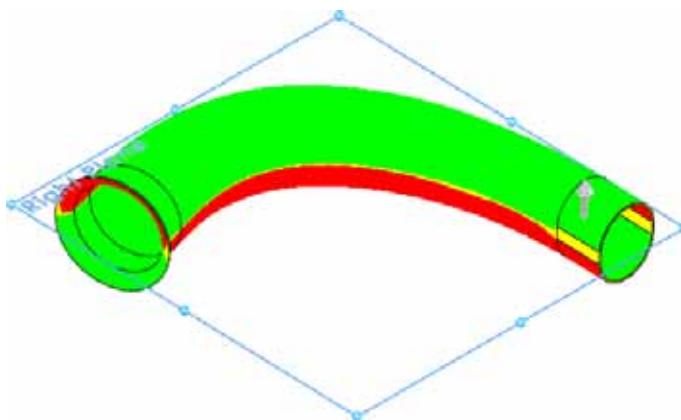
4 Configuration.

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 **View, Display, Draft Analysis** and select the Right plane. Set the **Draft Angle** to **1deg** and click .



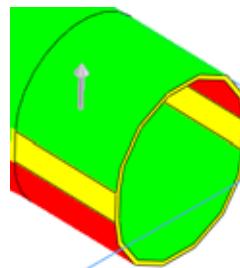
What Does This Mean?

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.

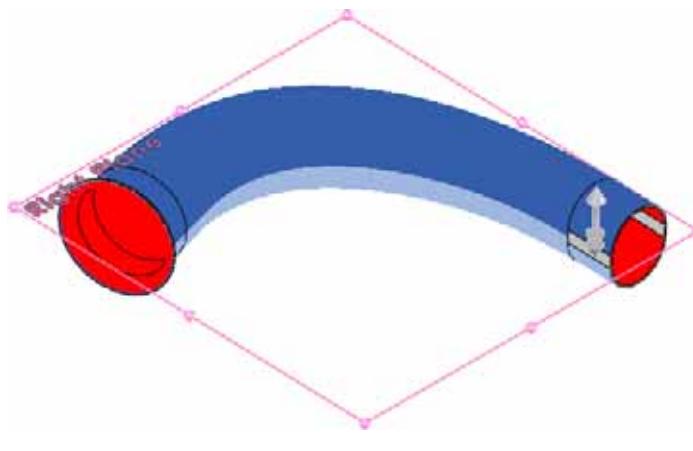
6 Display off.

Click **View, Display, Draft Analysis** .



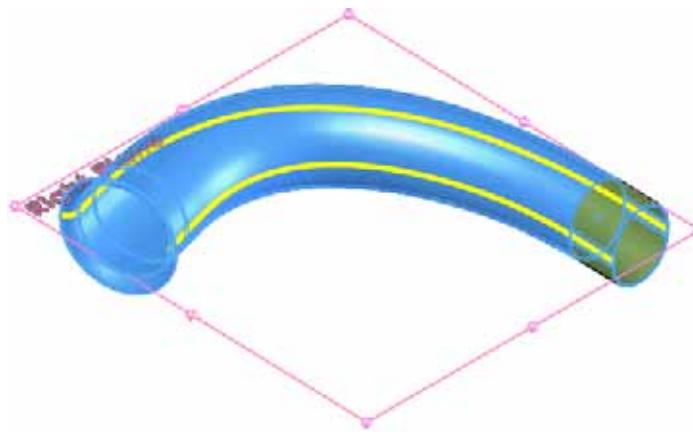
7 Undercut Analysis.

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



8 Split line.

Click **Insert, Curve, Split Line**  and click **Intersection**. Select Right plane and the faces that intersect that plane. Click .



9 Parting line.

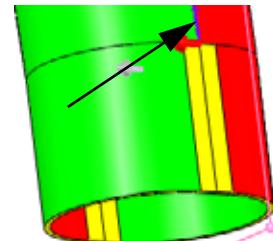
Click **Insert, Molds, Parting Line** , Select Right plane and set the **Draft Angle** to **1deg**.

Click **Draft Analysis**.

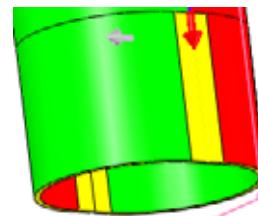


10 Edge selections.

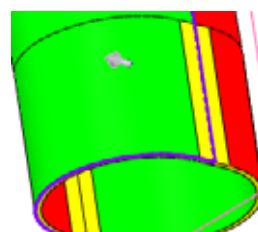
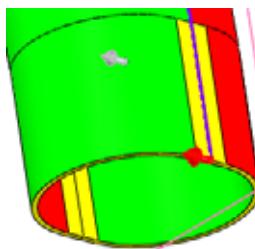
Select the starting edge as shown.



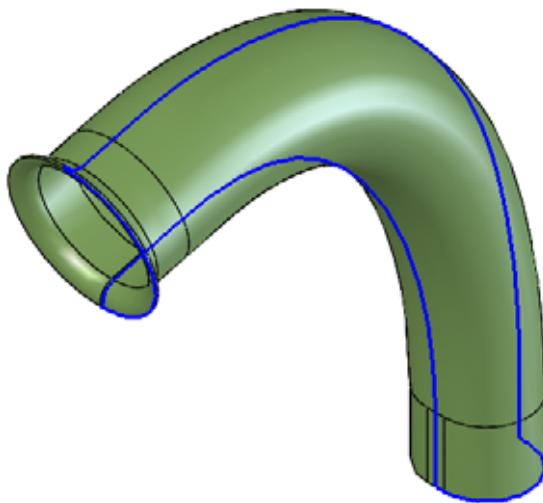
Click **Select Next Edge**  (or type "n") and then **Add Selected Edge**  (or type "y").



Continue the selections using the selection tools.

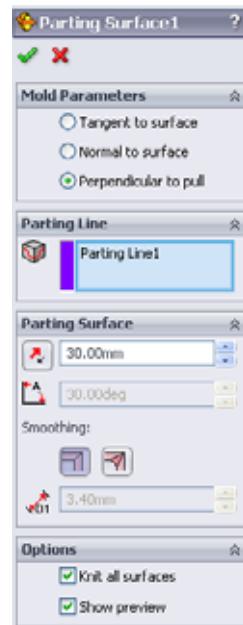
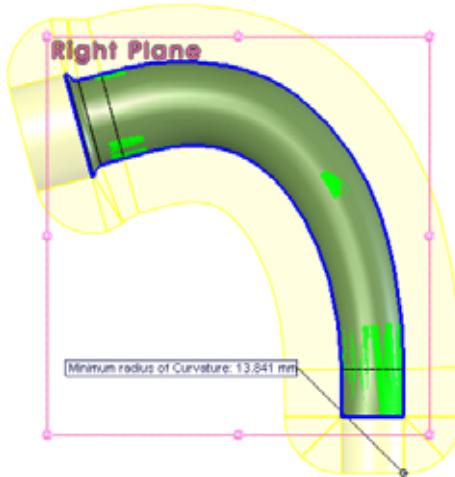


Continue selecting edges to complete the loop of selections as shown. Click .



11 Parting surface.

Click **Insert, Molds, Parting Surface**  and select **Perpendicular to pull**. Set the **Distance** to **30mm** and click **Knit all surfaces**. Click **Sharp**  and click .



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

Using Surfaces

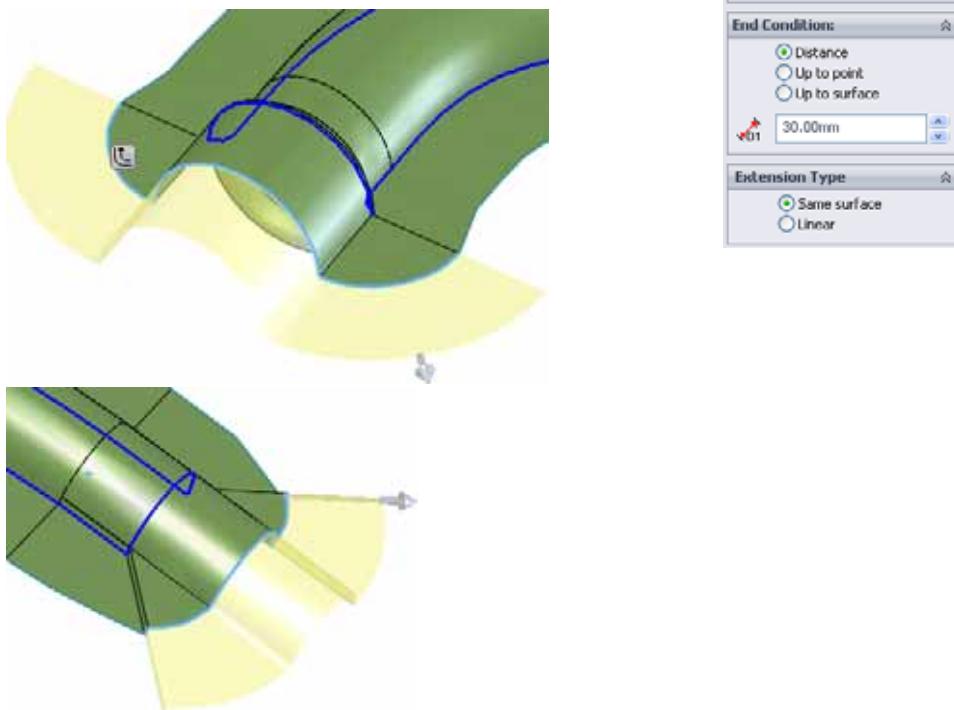
Surfaces often must be used to fill in or complete the surface geometry started by parting and shut-off surfaces. Here is a partial list of surface tools found in the **Insert, Surface** menu.

Extruded	Planar	Ruled
Revolved	Knit	Filled
Swept	Extend	Offset
Lofted	Trim	

12 Extend edges.

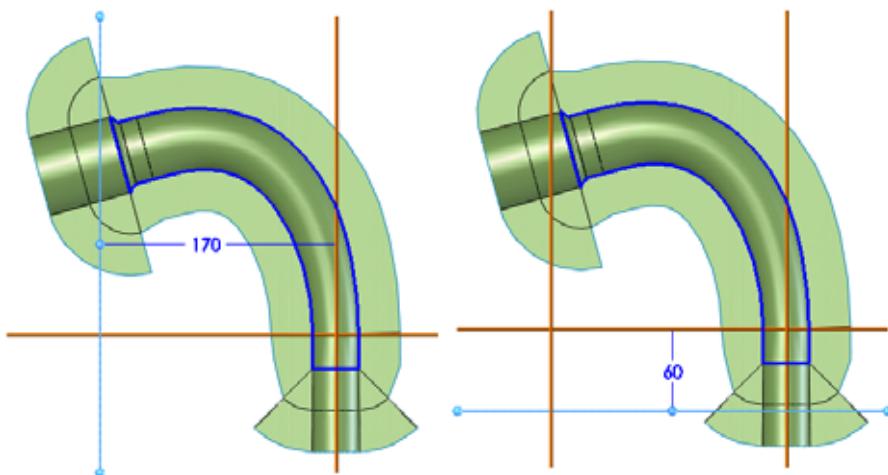
Click **Insert, Surface, Extend** , select three edges as shown. Click **Distance**, set the value to **30mm** and click .

Repeat the process for five edges on the opposite end.



13 Planes.

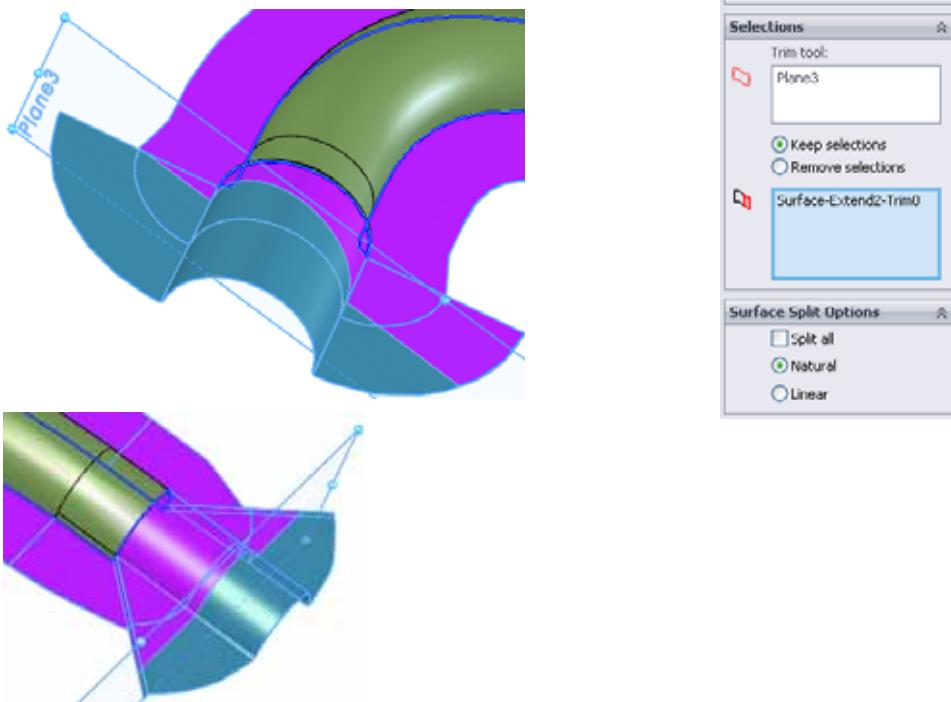
Create planes offset **170mm** from the Front and **60mm** from the Top as shown.



14 Trim.

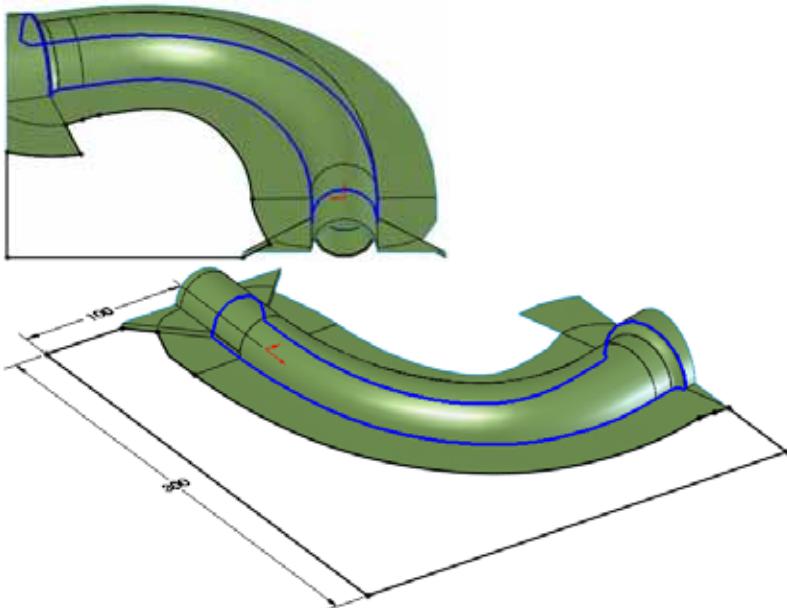
Click **Insert, Surface, Trim** and select **Standard**. Select the Plane3, **Keep selections** and select inside the plane.

Click . Repeat the process for the opposite end.



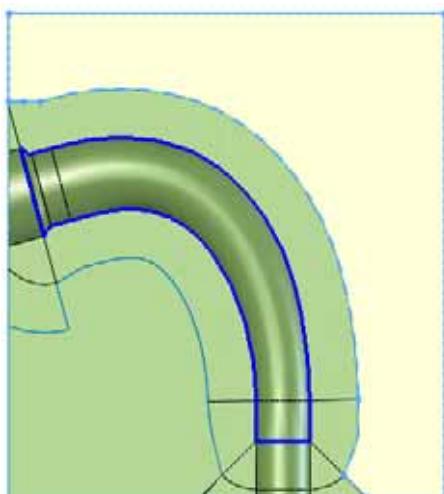
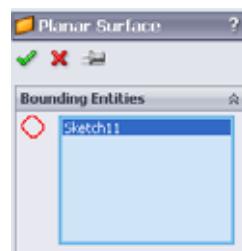
15 Sketches.

Right-click the parting surface and select **Sketch** . Using converted entities and lines, create the two sketches as shown. Exit both sketches.



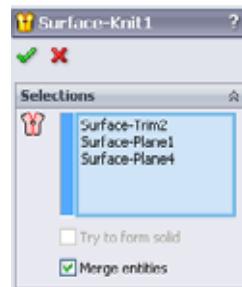
16 Planar surfaces.

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



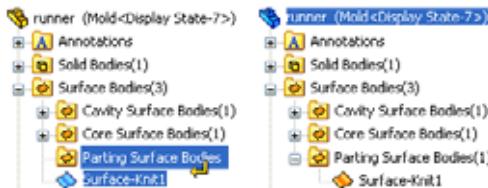
17 Knit surface.

Click **Insert, Surface, Knit Surface**  and select the three surfaces. Click **Merge entities** and click .



18 Drag and drop.

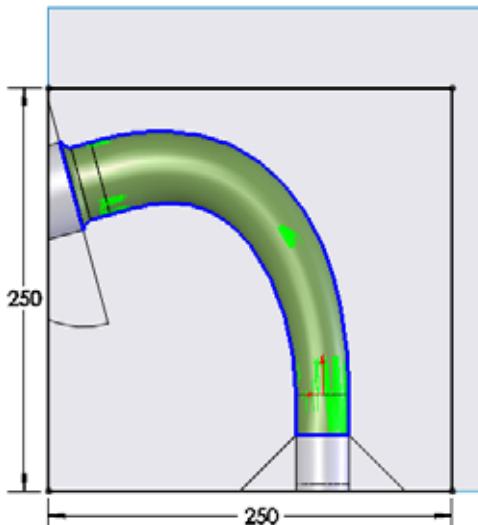
Drag and drop the Surface-Knit1 body into the Parting Surface Bodies folder as shown.



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

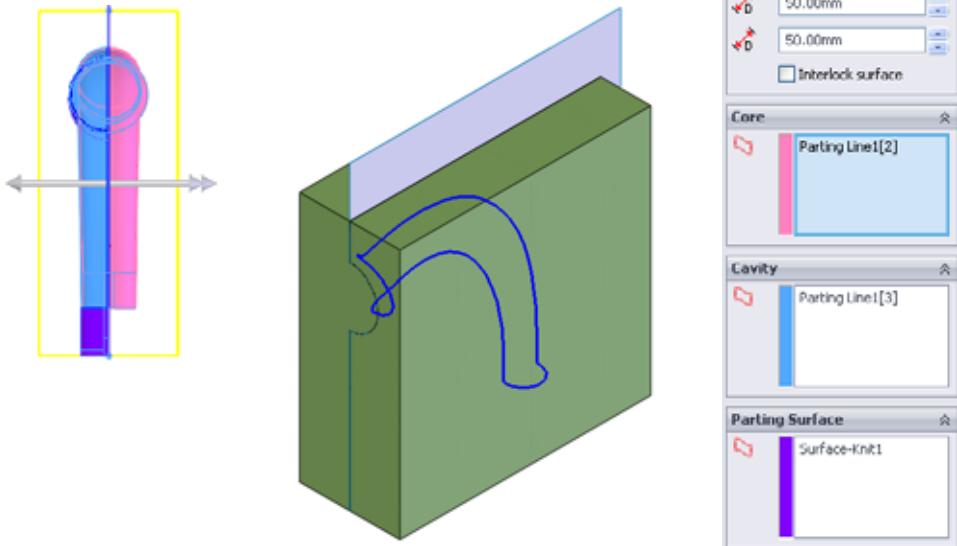
19 Tooling split sketch.

Click **Insert, Molds, Tooling Split**  and select the knitted surface as the sketch plane. Create the sketch and dimensions as shown.



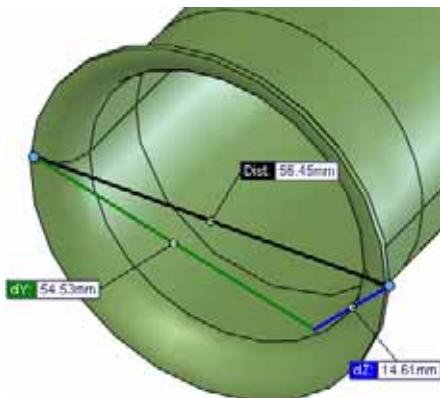
20 Tooling split sizing.

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



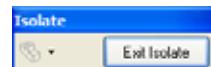
21 Test measurements.

Right-click the first (Parting Line 1) 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.



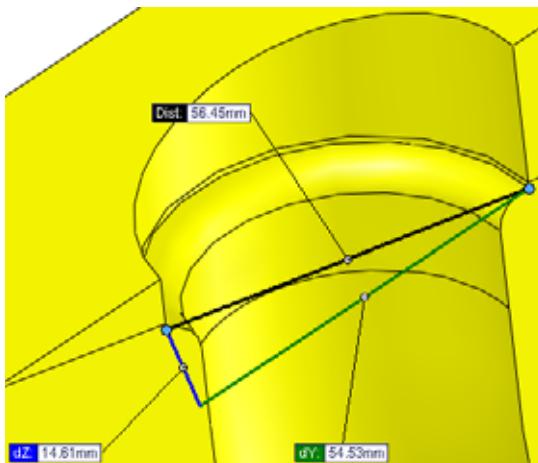
22 Exit isolate.

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



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.



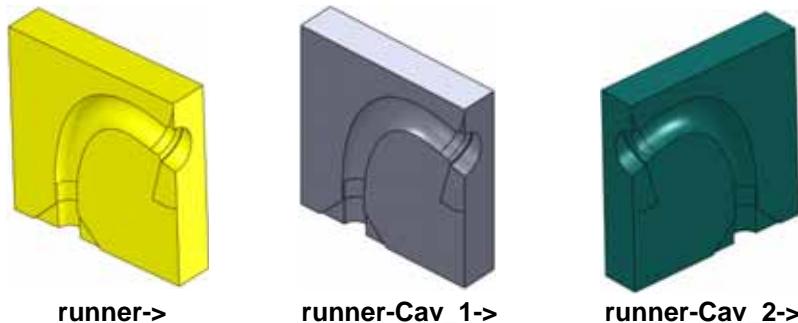
This shows that the outside faces are used to create the mold. Click **Exit Isolate**.

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.

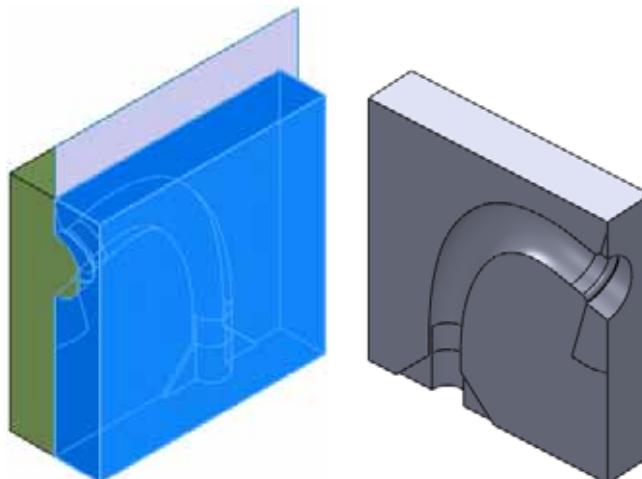
runner -> mirrored part



Note: 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.

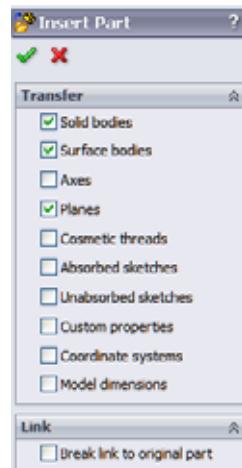
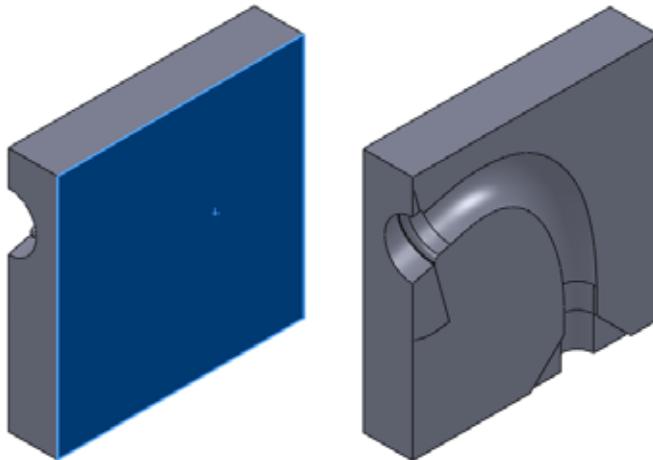
Right-click the cavity solid body and select **Insert into New Part**. Use the name runner-Cav_1. A stock feature Stock-runner-1 is created.



25 Mirror part.

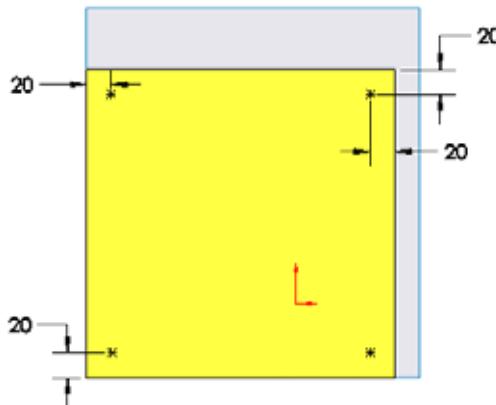
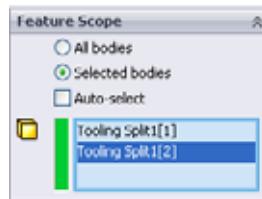
Select the Right plane and click **Insert, Mirror Part**. Click **Solid bodies, Surface bodies** and **Planes**. Click .

Use the name runner-Cav_2.



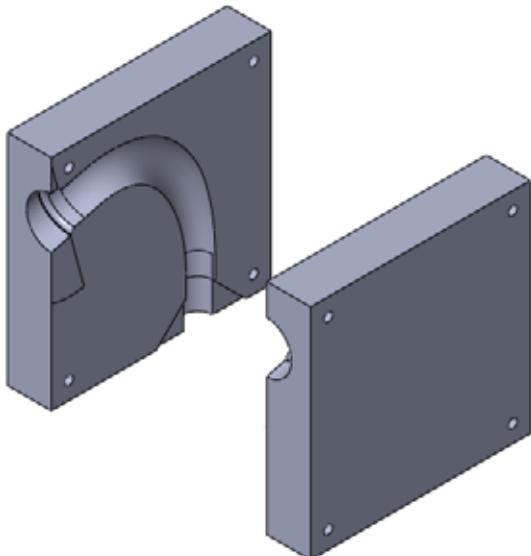
26 Holes.

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



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.**

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 Design

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.

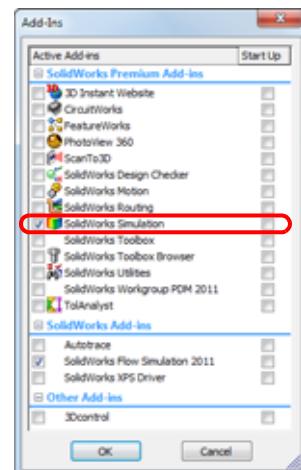
Click **File**, **Open** and select the assembly Brake Rotor Assembly. Click **Open** to open the assembly.

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.

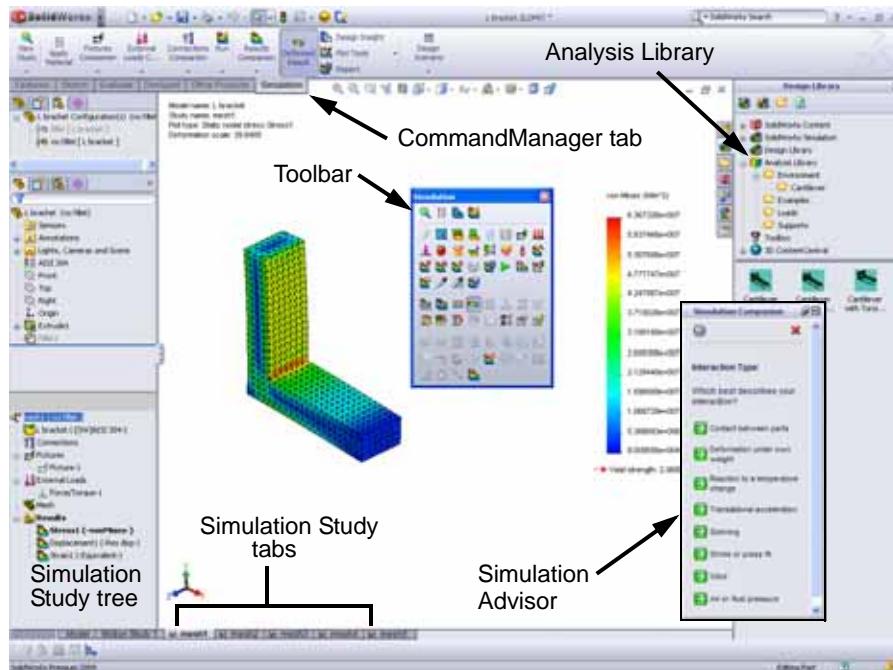
Click **Tools**, **Add-Ins**. Select **SolidWorks Simulation**.

Click **OK**.



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.



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.

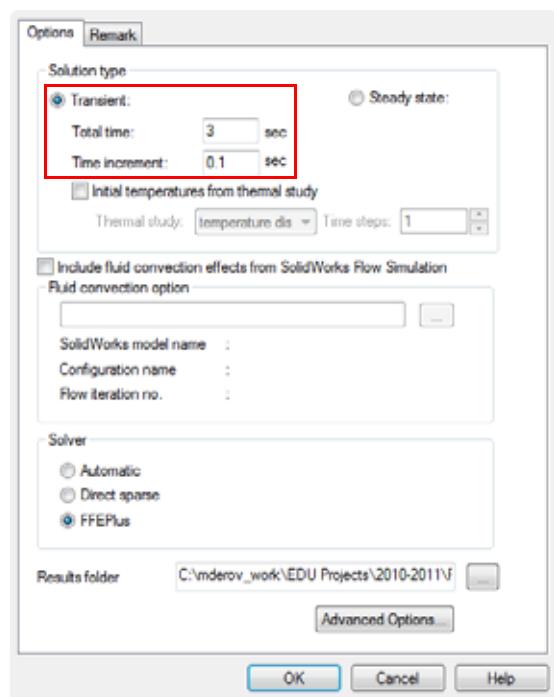
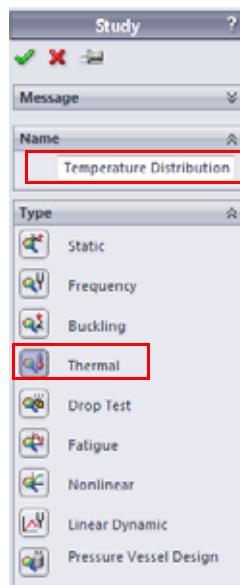
The setup of the study is done using the Simulation study tree.

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 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 breaking 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.

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

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

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

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

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

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 convection 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.

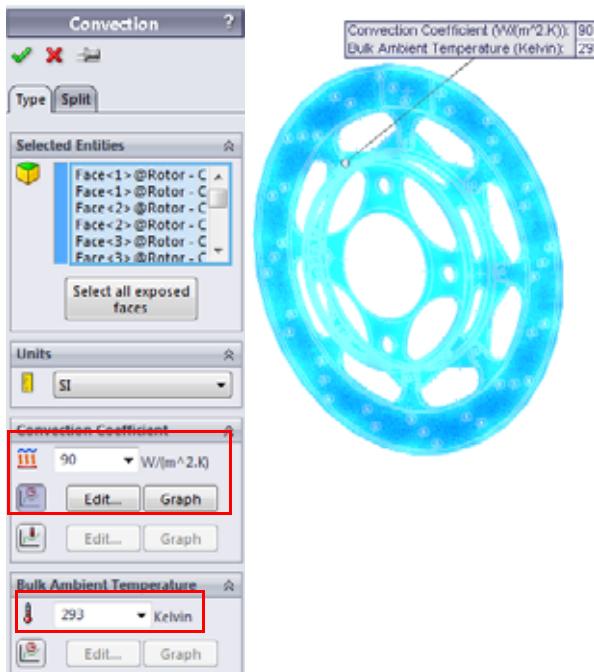
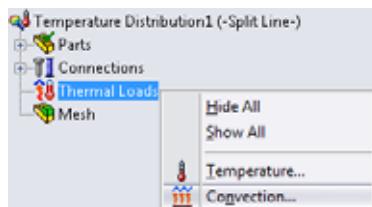
6 Apply thermal load.

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.

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

Enter **293 Kelvin** as the **Bulk Ambient Temperature**.



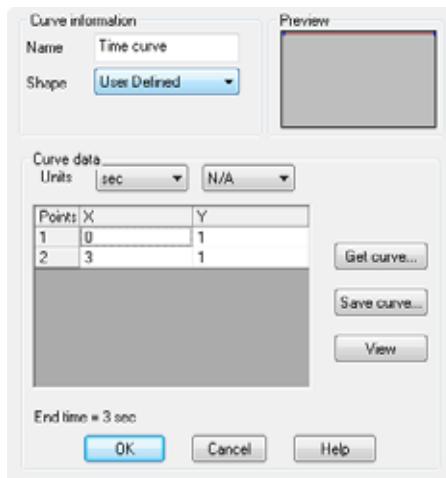
7 Edit Time Curve.

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**.



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:

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

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

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

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.

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

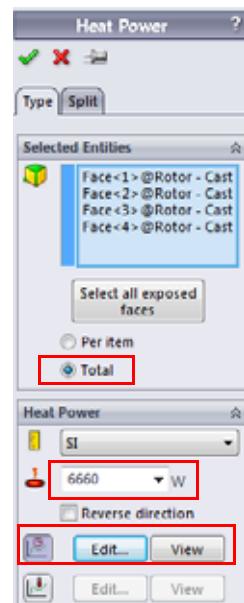
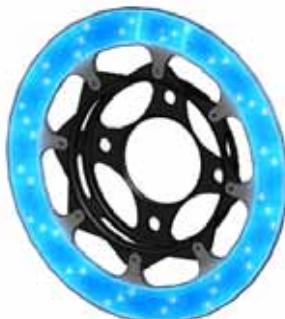
8 Apply thermal load.

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**.



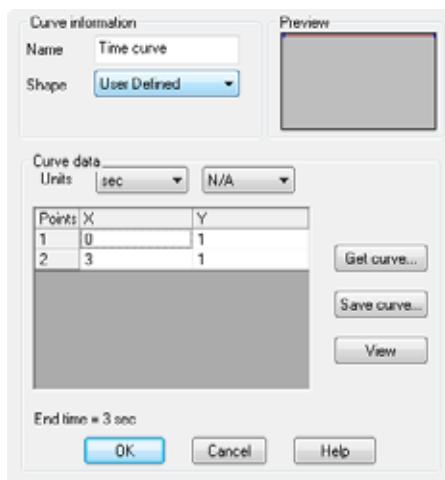
9 Edit Time Curve.

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**.



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.

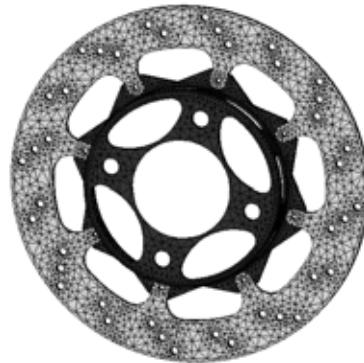
Enter **25 C** as the **Temperature**.

Click .

11 Mesh the model.

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

Click .



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.

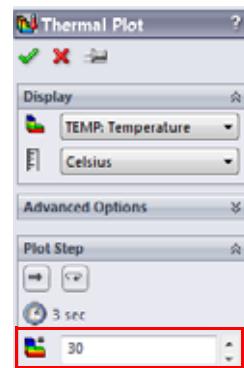
13 Plot Temperature distribution.

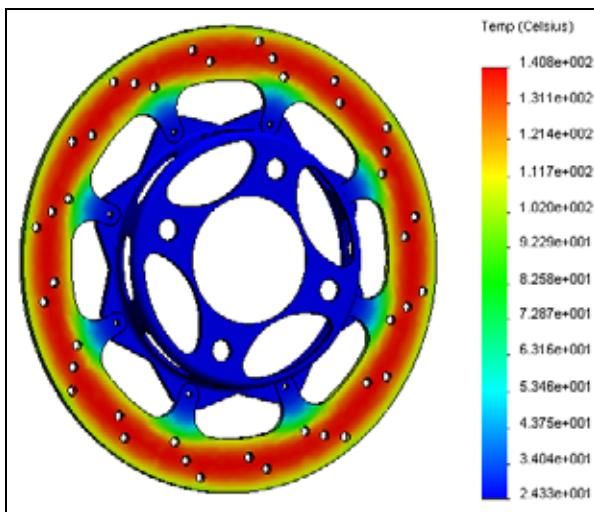
Right-click the Thermal1 plot of the temperature distribution and select **Edit Definition**.

Change the **Units** to **Celsius**.

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

Click .





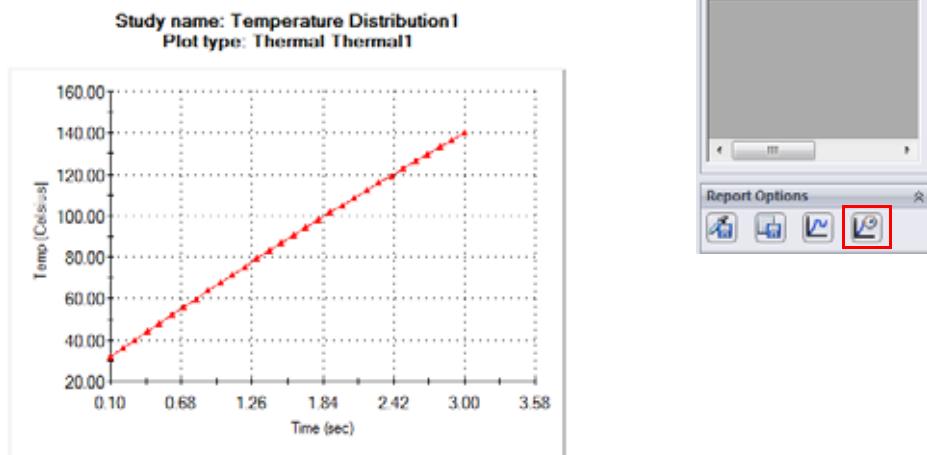
14 Probe.

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

Select any location on the brake rotor.

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

A plot of temperature vs. time will appear.



Click .

Note: You can do this with any of your created plots in a transient thermal analysis.

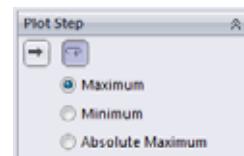
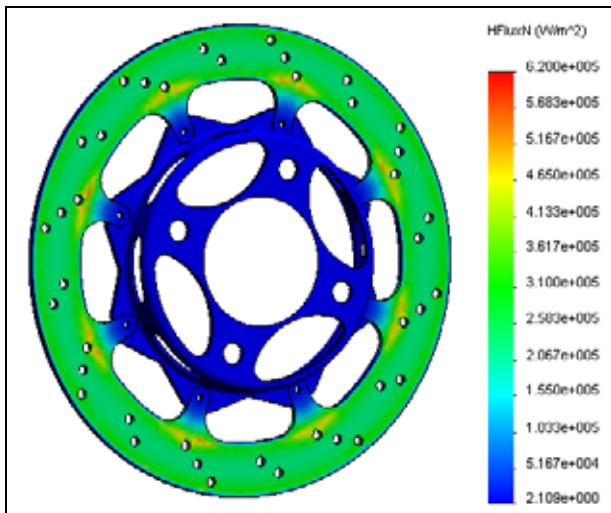
15 Plot Resultant Heat Flux.

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.

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**

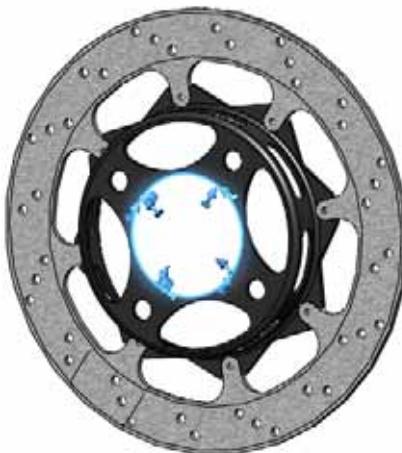
Allows for different types of restraints in various directions. Feel free to review the help menu to investigate these types.

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 .



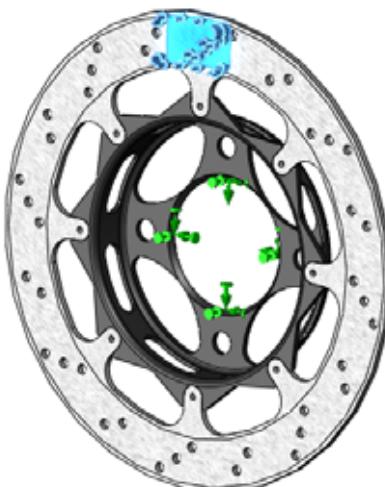
Note: We are applying a rigid restraint to the shaft mounting location. With this, comes the assumption that the shaft will not deform under any applied loading transferred through this mounting location. For this example, we are assuming the shaft is much more rigid than the rotor hub. If we expect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

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**

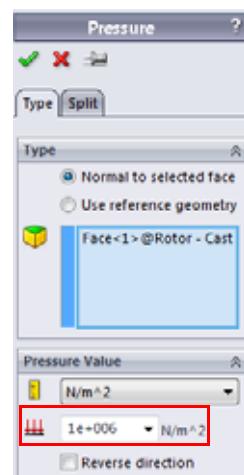
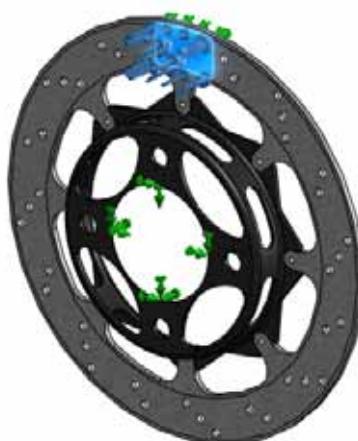
Defined between contacting cylindrical faces.

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/m²** as the **Pressure Value**. This value can be known from experiments.



Click .

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 that 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.

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

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.

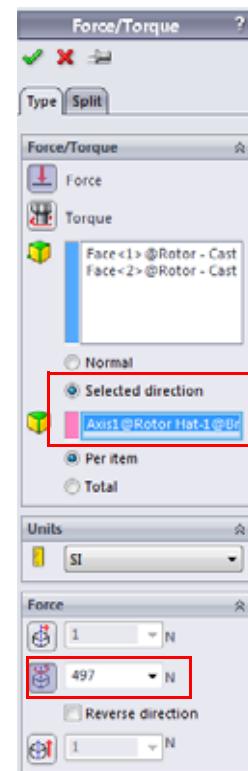
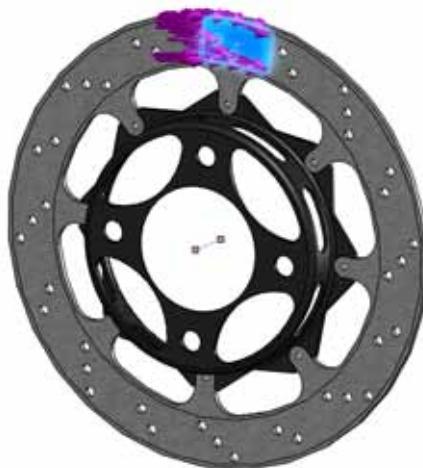
Right-click External Loads in the Simulation Study tree and select **Force**.

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.

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.

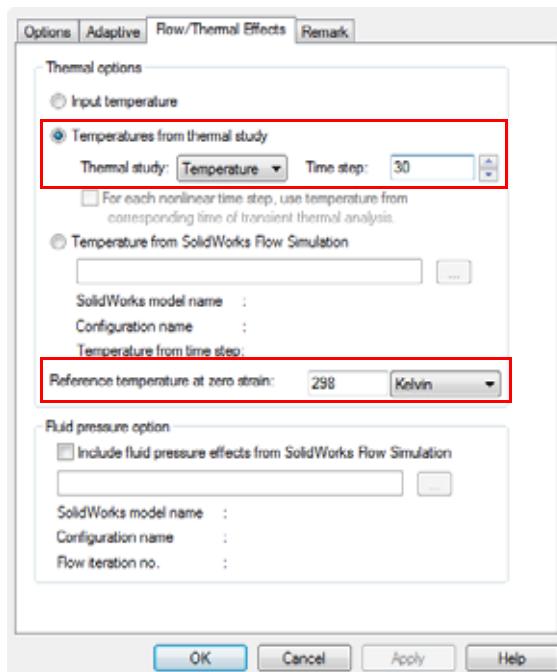
22 Adjust study properties.

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**.



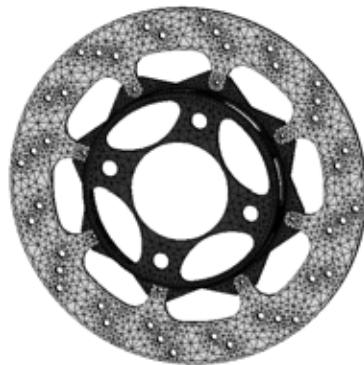
Click **OK**.

The Thermal load will appear in the External Loads folder.

23 Mesh the model.

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

Click .



24 Run the study.

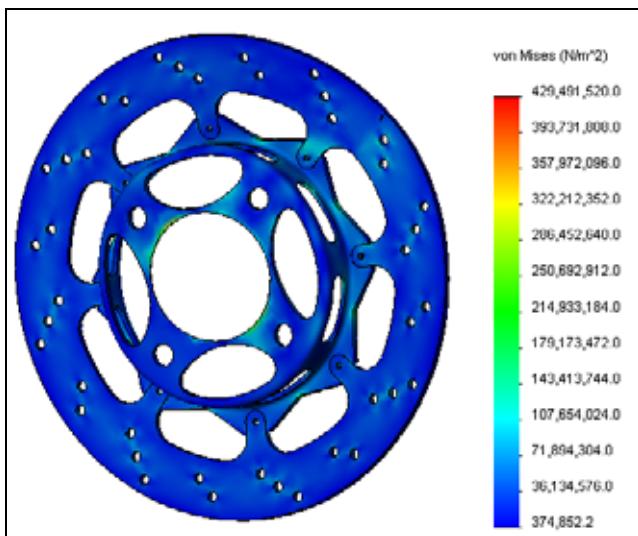
Click **Run** from the Simulation drop-down menu.

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.



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.

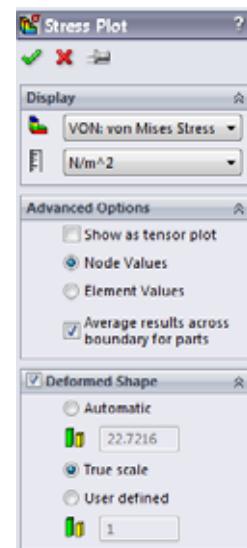
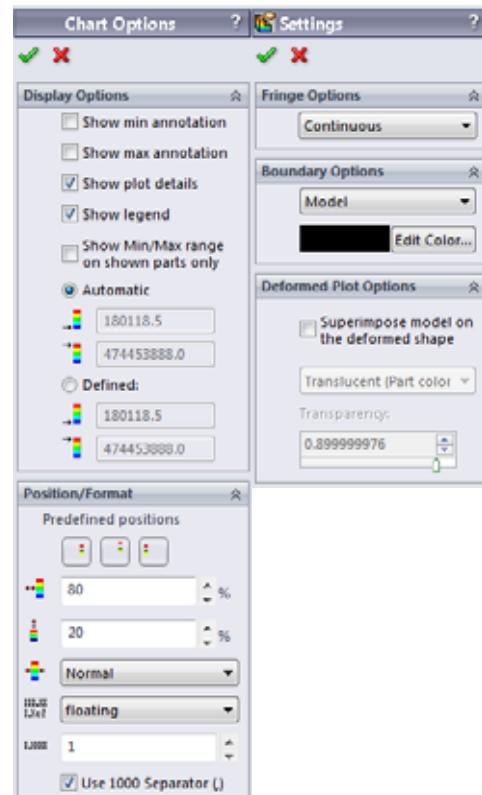


Chart Options

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.

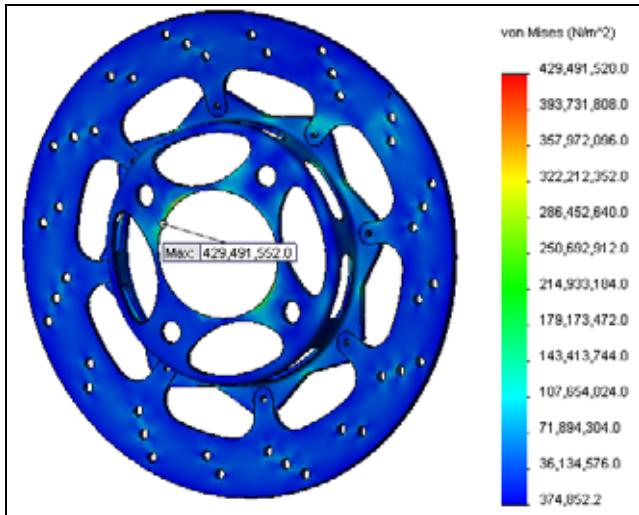


26 Display model maximum.

Right-click on the plot and select **Chart Options**.

Click **Show max annotation**.

Click .

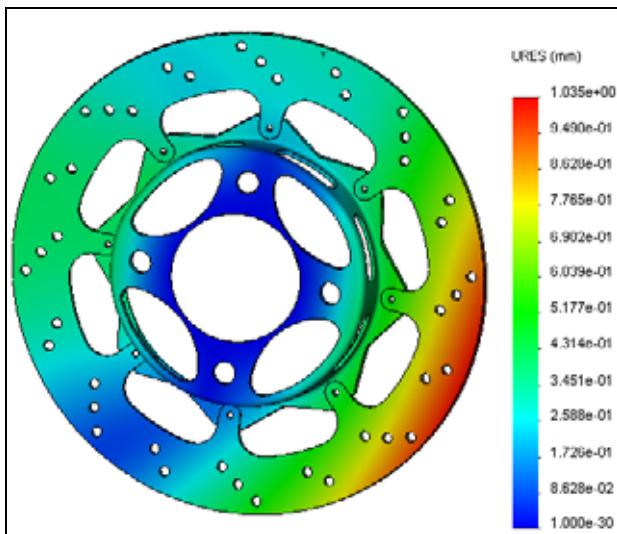


Discussion

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.

27 Plot displacement.

Double-click the Displacement1 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 $90 \text{ W/m}^2\text{K}$ 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.

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.

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:

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



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.

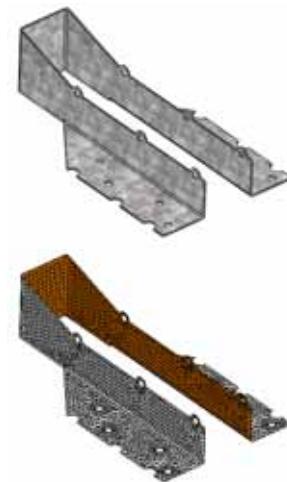
In this lesson, we will be analyzing the frame structure that we modeled in a previous lesson. We could use solid elements in this analysis, however we will see another element type that will allow us to greatly simplify our calculations.

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:

- 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**.

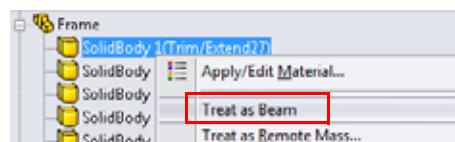
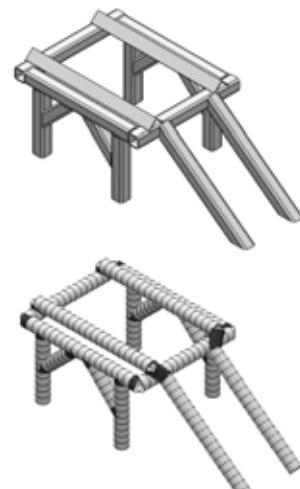


Beam Elements

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**.



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.

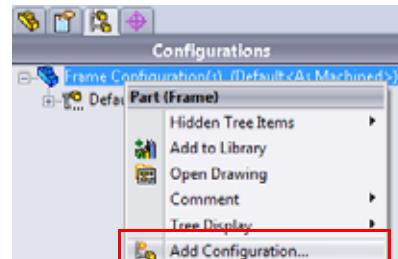
2 Add a configuration.

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.

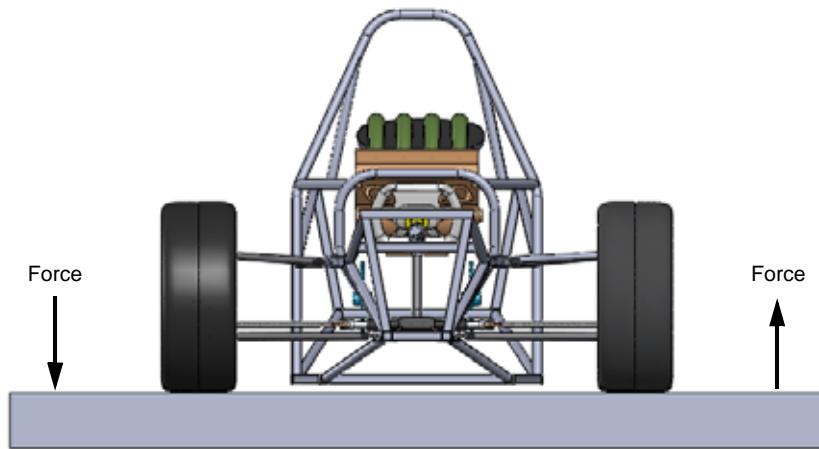
Right-click on the Tabs folder and select **Suppress**.

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.

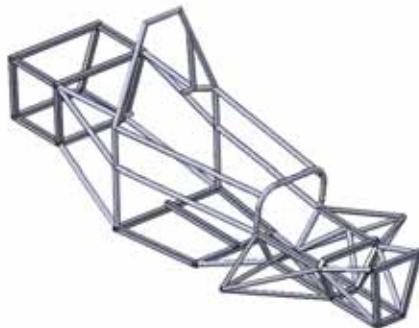


Note: Our analysis will be setup with respect to this method for measuring torsional rigidity. Other methods exist and you would want to design your analysis according to the physical setup of the experiment in order to compare the results properly.

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.

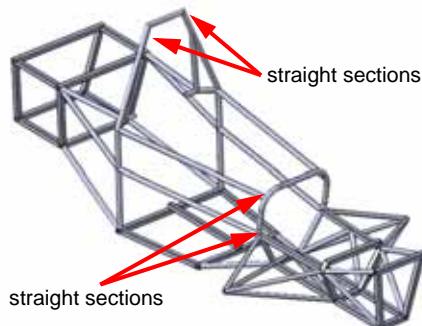
Right-click on the Rigid_Supports folder and select **Unsuppress**.



Additional Model Modifications

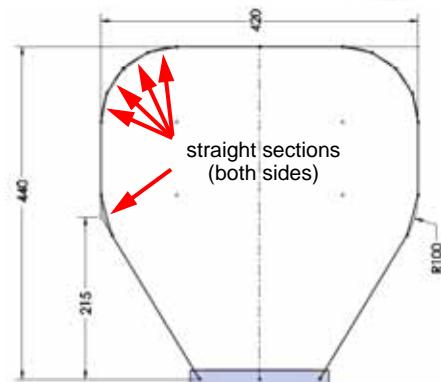
A few additional modifications to the original frame also have been made in preparation for analysis. When the beam is curved, it is approximated by short linear sections. When these linear sections become so short relative to the cross section of the beam, the assumptions on the stiffness of the beam element tend to become less accurate. Additionally, SolidWorks Simulation can have difficulties in meshing this type of curved geometry.

In our model, we have tried to avoid curved geometry altogether. Two sections of the frame have been modified so that they can be meshed appropriately. First, at the top of the Seat section, we have removed the curves. These are not going to significantly affect the stiffness of the vehicle frame, so this approximation is valid.



Second, we have modified the Bulkhead_1 section again to avoid any curves. Instead of the curves, we are using short straight line segments in our sketch as seen in the image. This is valid as well, because as we already know, the beam will be meshed as a short, straight section.

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**.

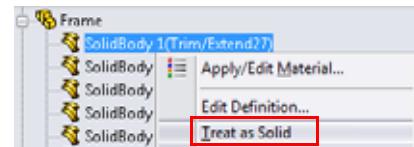
Select **Static** as the **Type**.

Enter Torsional Rigidity as the **Name**.

Click .

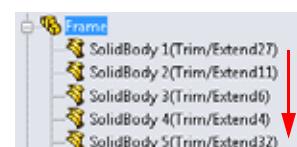
Beam Meshing

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 solid bodies that will be meshed as beams. You can tell that the will be beams because of the beam icon  next to the body name.

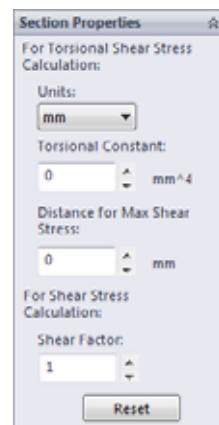


Any Solid Bodies that will be meshed as solids should be excluded from the analysis. To do this, right-click the solid body and select **Exclude from analysis**.

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**

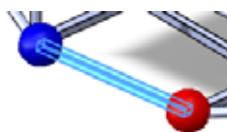
Distance from the center of the section to the point of maximum torsional shear.

■ **Shear Factor**

Ratio of effective area under shear and the cross section area of the beam. The quantity is computed in both local cross sectional directions.

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.



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**

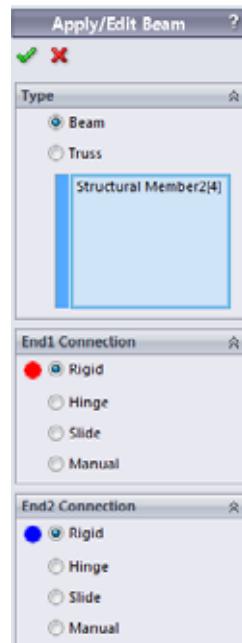
Three translational degrees of freedom are tied to the joint. All translational forces will be transferred from the beam element to the joint and vice versa. Rotational moments will not be transferred.

- **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**  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.

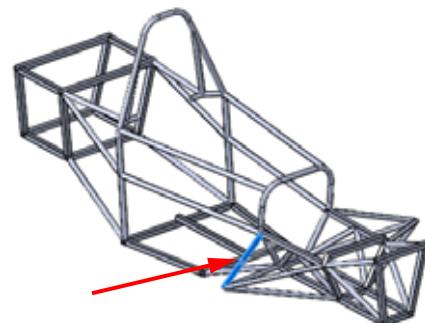
7 Define trusses.

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, 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.



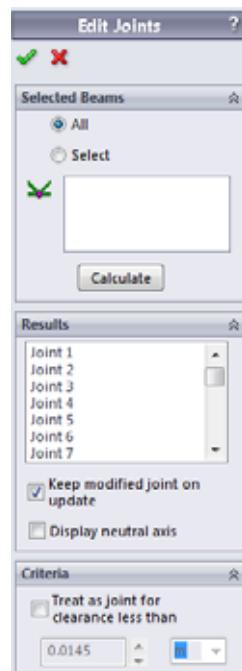
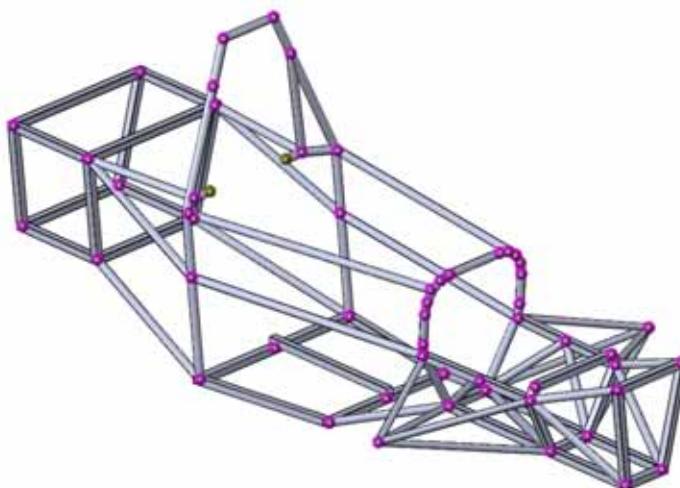
The beam joints are shown as yellow or magenta spheres in the graphics window.

-  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**.

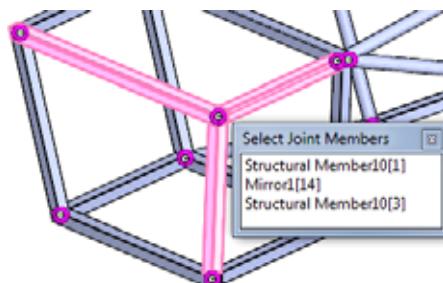


10 Examine joints.

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.

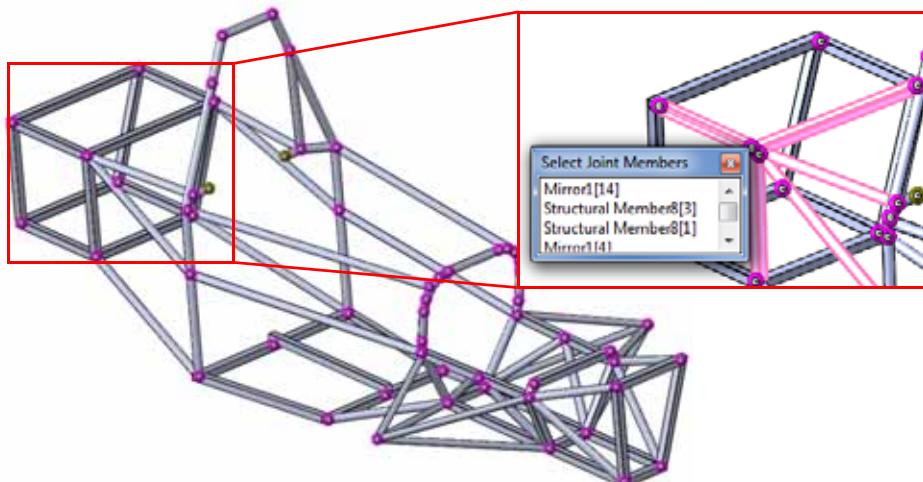
Examine the joints in the rear of the vehicle frame.

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.

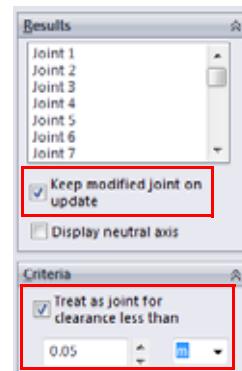


Note: This procedure may be repeated as necessary until the joints are calculated properly. Additionally, this procedure may not be necessary if the joints are initially calculated correctly.

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.

Merging Joints Automatically

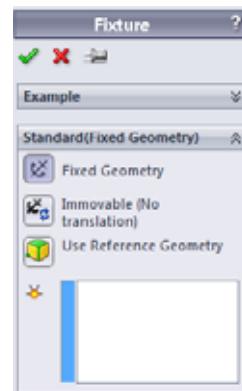
The software automatically calculates joint locations based on the ends of beams. At the joint end, a hypothetical sphere is drawn with a diameter chosen automatically with respect to the model geometry. If two joints fall within one sphere, the beam ends will be merged forming one joint. It is possible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



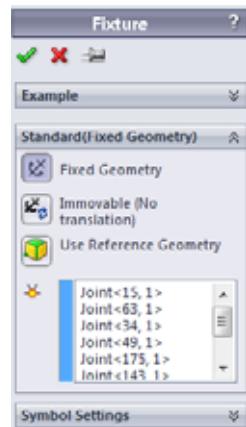
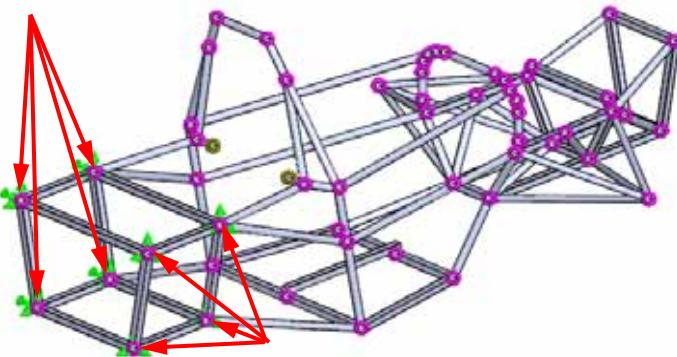
It was mentioned that in the experiment, the rear wheels are held stationary. To measure the torsional rigidity, the entirety of the load must be transferred directly to the frame. Therefore any components attached to the frame that transfer the load must be assumed to be rigid. If this is the case,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

12 Apply fixtures.

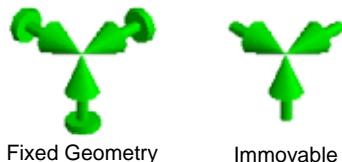
Right-click Fixtures in the Simulation Study tree and select **Fixed Geometry**.

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

Click .



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.

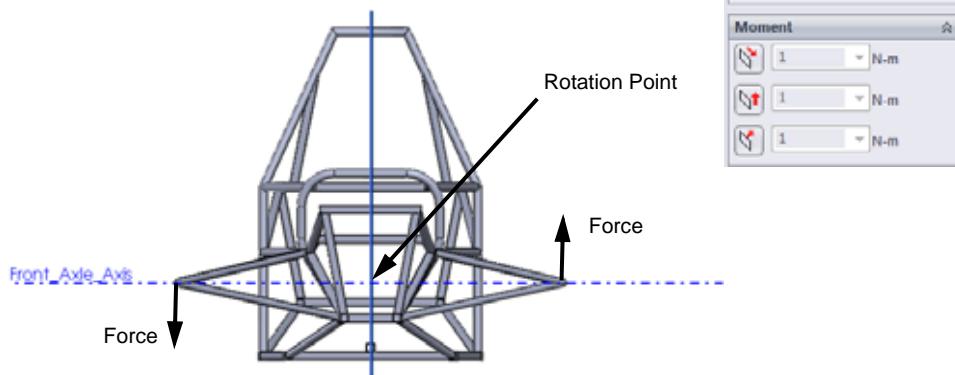


Loading

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.

13 Apply loading.

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

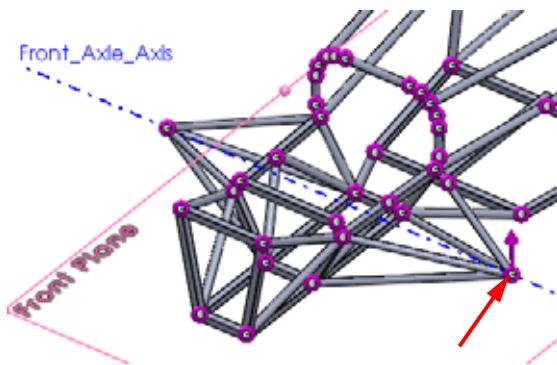
Select **Joints** under **Selection**.

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 .



Repeat this procedure for the opposite side.

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

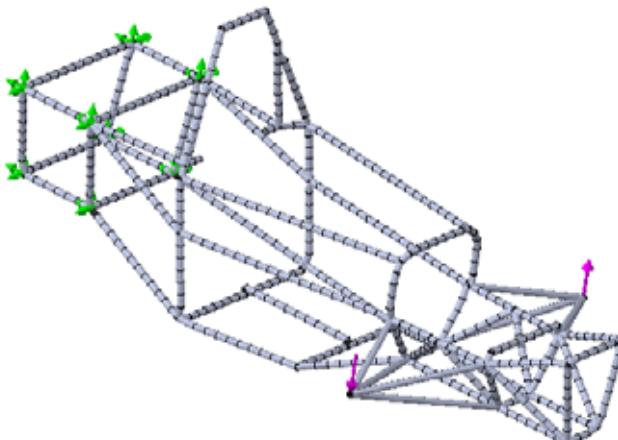
Note: The loading magnitude was calculated such that a 100 N-m torque is applied to the frame.

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

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.

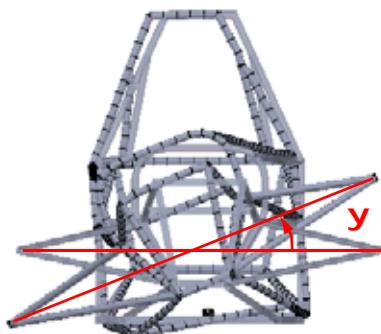
15 Run the study.

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

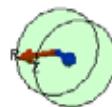
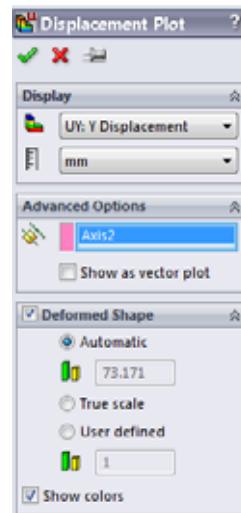
By default, SolidWorks Simulation plots the results using a cartesian coordinate system. Knowing this, one could convert the results to another, more desirable coordinate system.

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
- Z direction = Axial direction

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



16 Plot angular displacement.

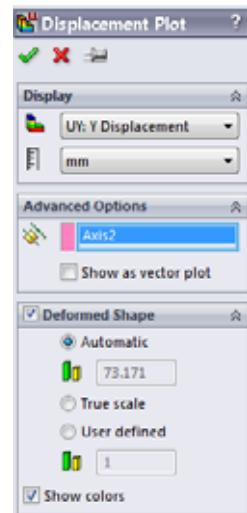
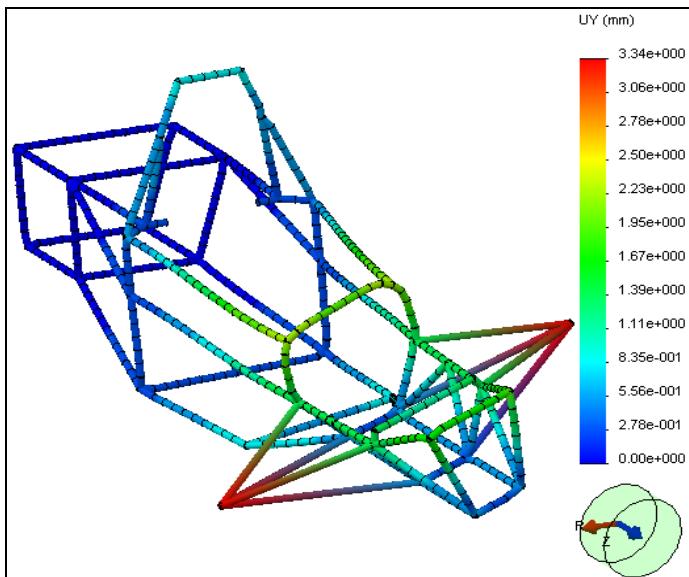
Right-click the Results folder and select **Define Displacement Plot**.

Select **UY: Y Displacement** as the **Component**.

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

Select the **Automatic** scaling.

Click .

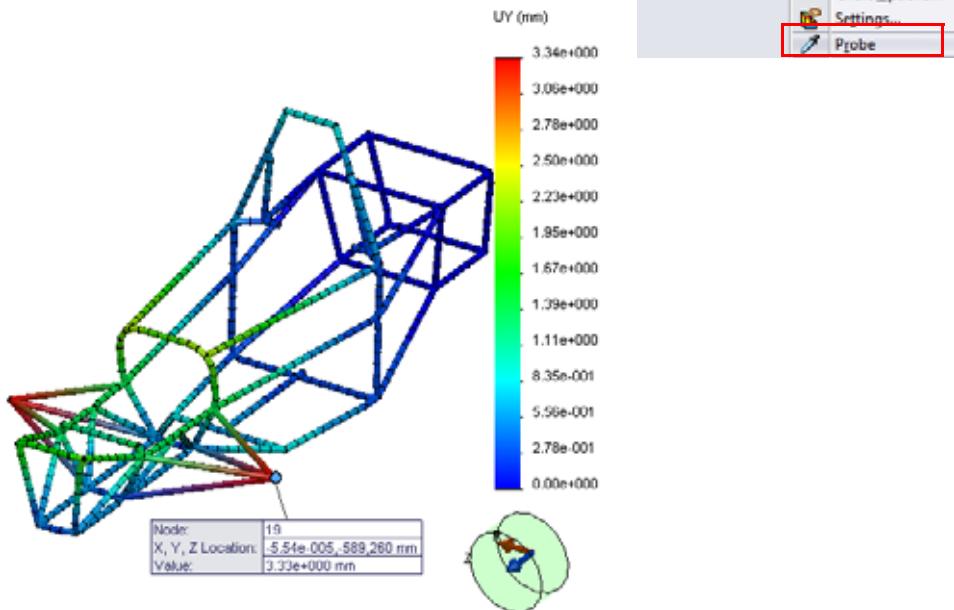


Note: The unit system of this plot is still in mm where we might expect it to have an angular unit since it is a circumferential displacement. In actuality, the software is taking the angular deformation about the axis and multiplying it by the distance away from the axis. To compute the actual angular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

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.

$$\psi = \frac{3.339\text{mm}}{589.2\text{mm}} = 0.00567\text{rad}$$

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.

$$\tau = \frac{100\text{Nm}}{0.00567\text{rad}} = 17646 \frac{\text{Nm}}{\text{rad}}$$

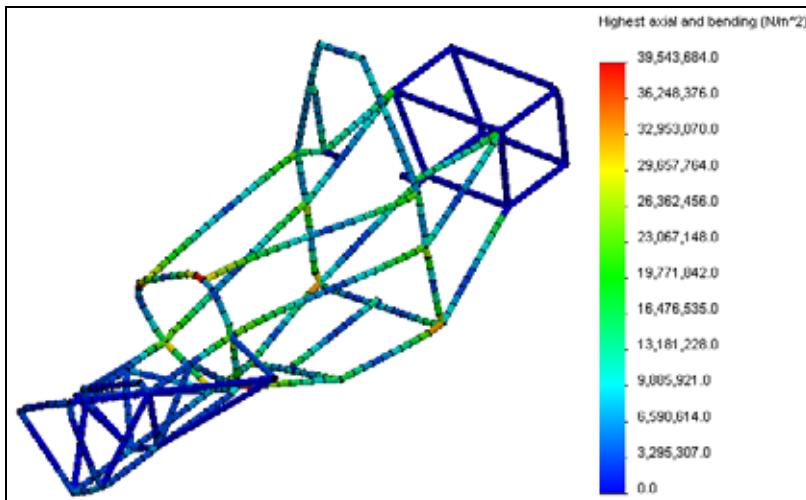
Beam Stresses

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 **Highest 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.

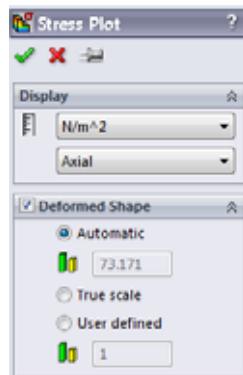
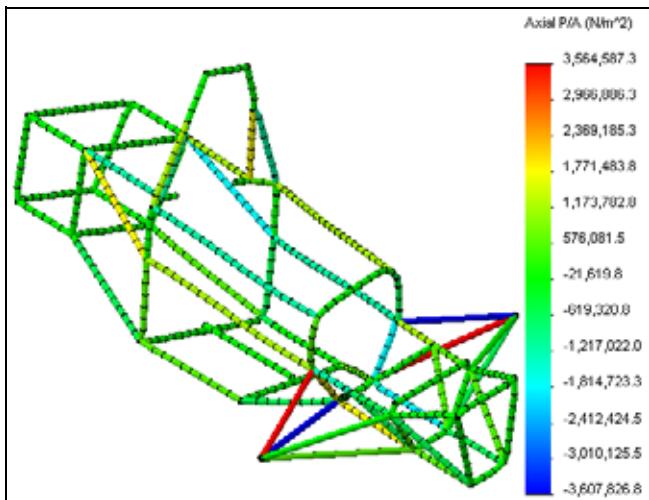
We can view these stress components separately.

19 Plot axial stress.

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

Select **Axial** and **Automatic**.

Click .



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

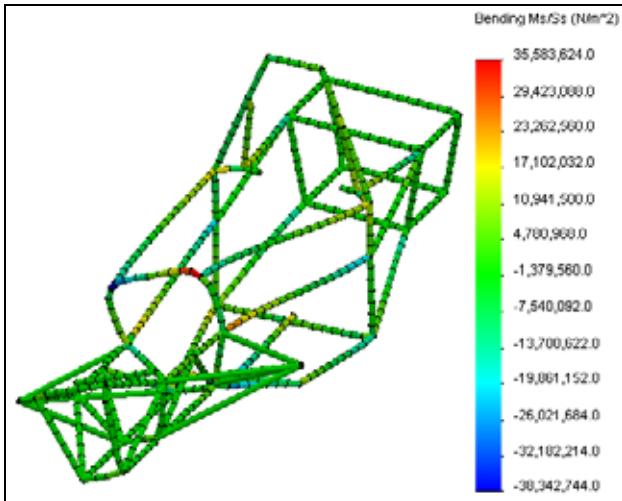
To post process bending and shear stress, the beam directions 1 and 2 must be defined. SolidWorks Simulation defines direction 1 to be along the longest side of the cross section and direction 2 is it's perpendicular.

20 Plot bending stress.

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

Select **Bending in local direction 1** and **Automatic**.

Click .



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.

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.

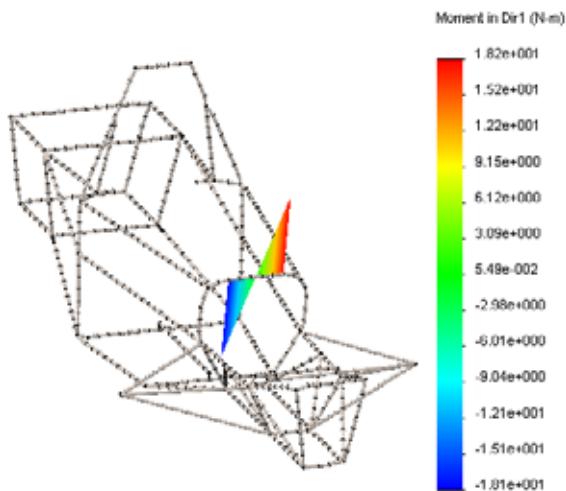
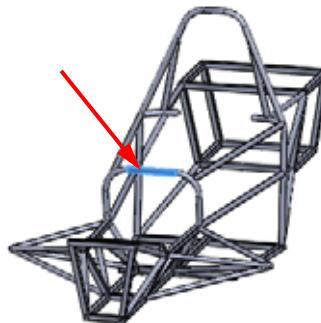
21 Plot bending moment.

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.

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 stiffer, not one that matches the experiment perfectly.

Lesson 7

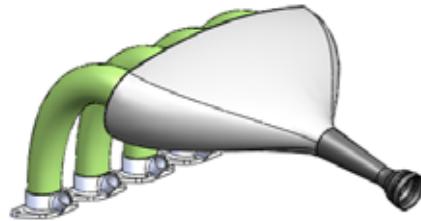
Intake Analysis

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

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

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.



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

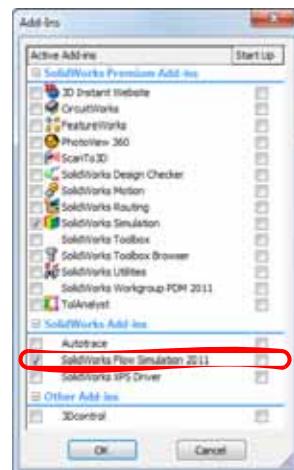
Internal flow analysis deals with the study of flow inside a region bounded by solid geometry. A typical example of an internal flow analysis would be the flow through an HVAC system. In this type of analysis, flow enters the model through some inlet and leaves the model through the outlet. Flow Simulation 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.

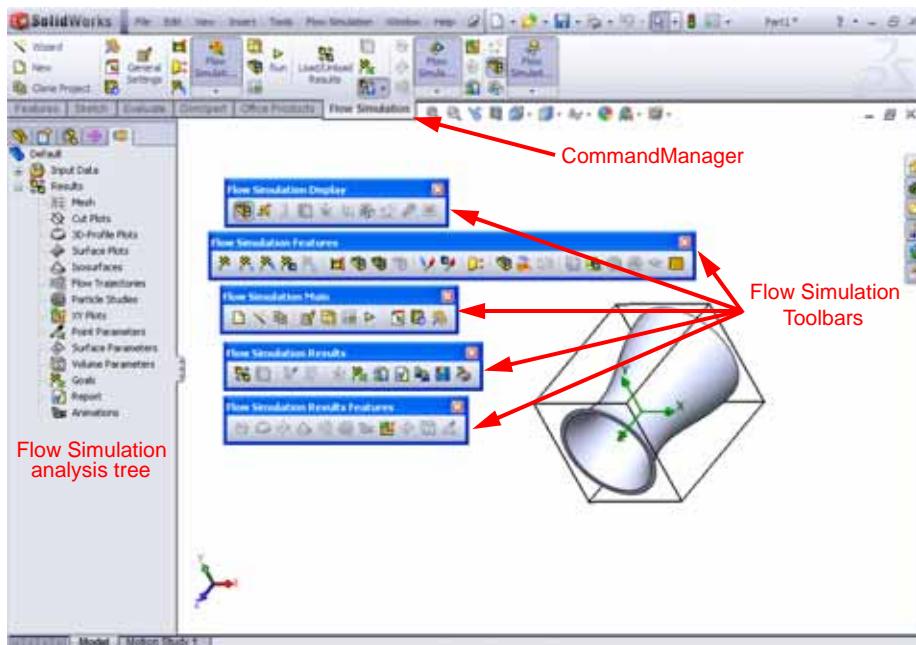
- 2 Start SolidWorks Flow Simulation.**
Click **Tools**, **Add-Ins**. Select **SolidWorks Flow Simulation**.

Click **OK**.



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 right-clicking 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.

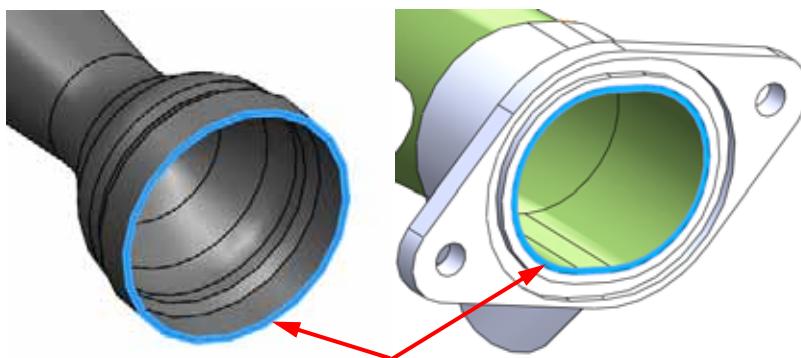
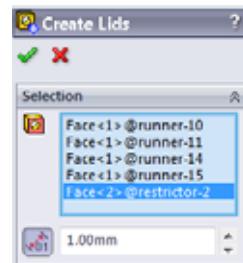
3 Create lids.

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 five 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**.

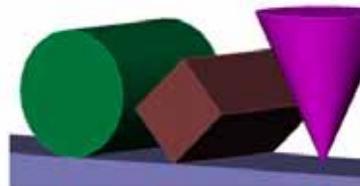
Note: The thickness of the lid for an internal analysis is usually not important however, the lid should not be so thick that the flow pattern is affected down stream. For most cases, the lid thickness could be the same thickness used to create the neighboring walls.

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.

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.

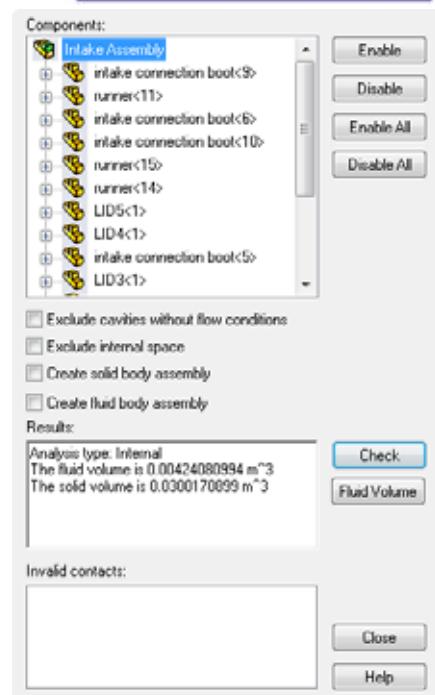
Make sure the **Exclude Cavities without flow conditions** check box is cleared.

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.

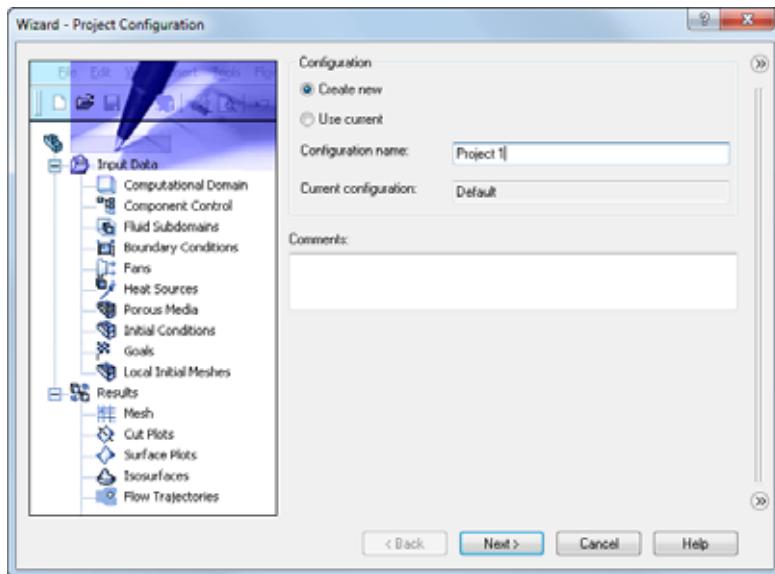
5 Create a project.

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

6 Create new project.

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

In the **Configuration name** box, enter **Project 1**.

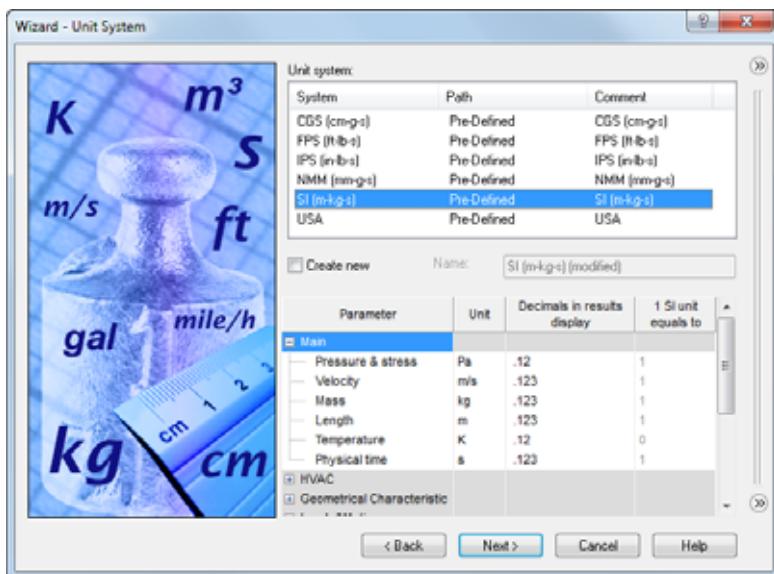


Click **Next**.

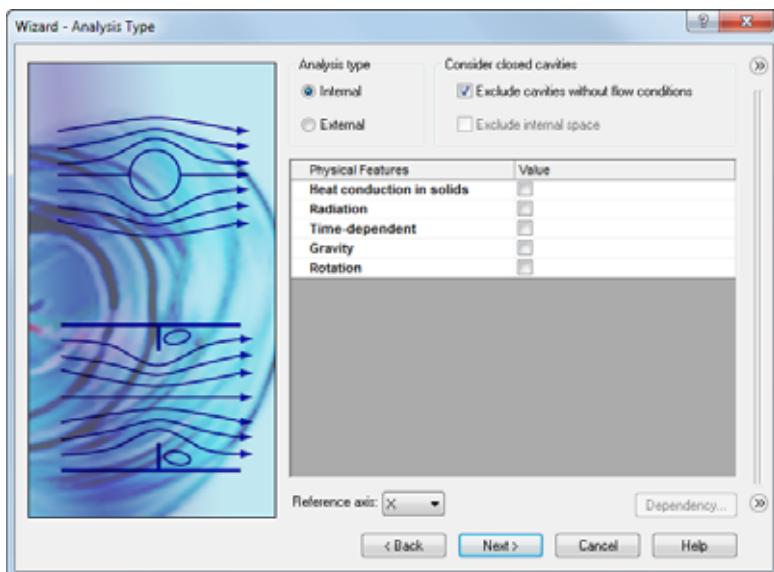
Note: SolidWorks Flow Simulation will create and activate a new configuration called Project 1 when the wizard is finished. All data associated with running the analysis will be stored in a separate folder in the model directory numbered sequentially, i.e. “1”, “2”, “3”... etc. based on how many projects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

7 Select units.

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



8 Select analysis type.



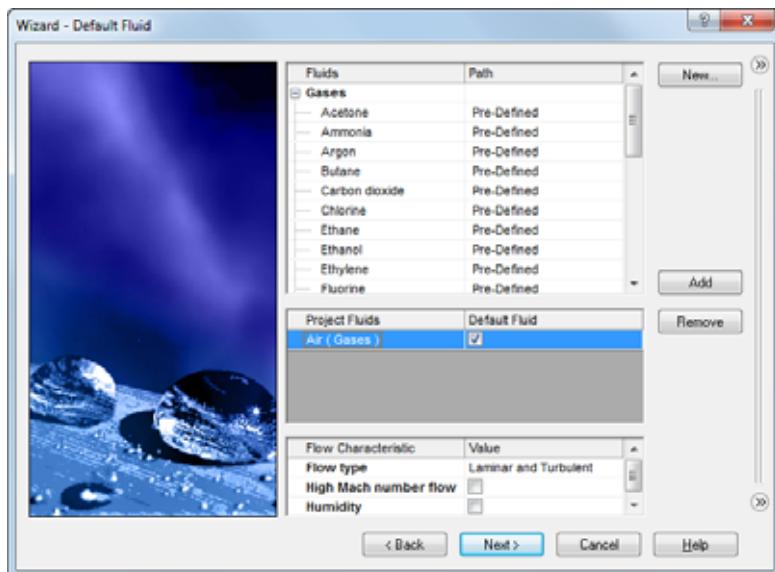
Select **Internal** under **Analysis Type**.

Click **Next**.

9 Select fluid.

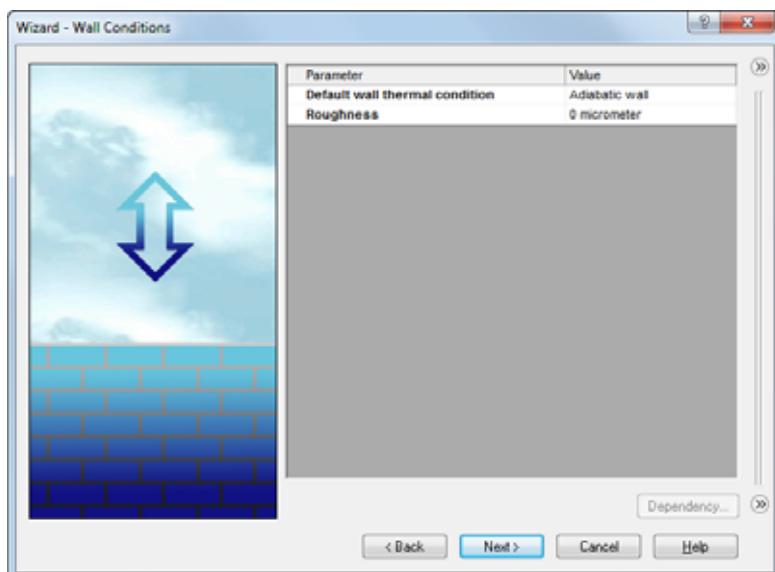
Expand the **Gases** tree and double-click **Air** to add it to the **Project Fluids** list.

Click **Next**.



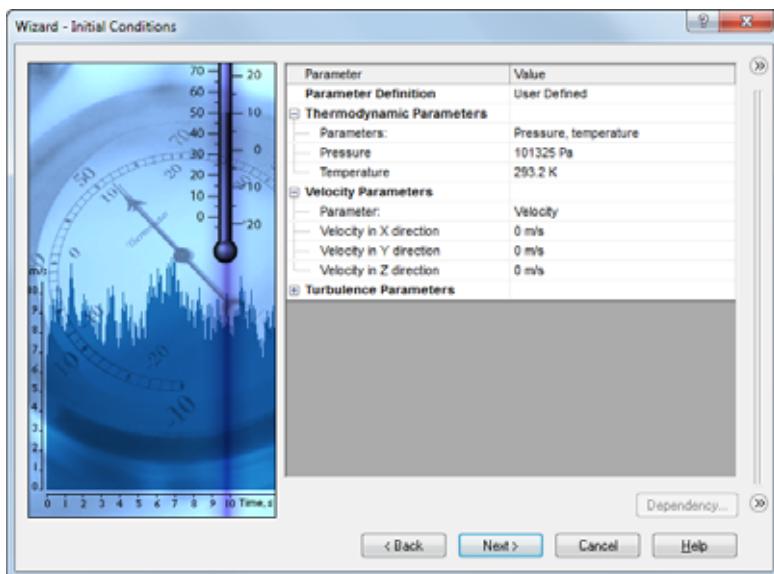
10 Select wall conditions.

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



11 Initial conditions.

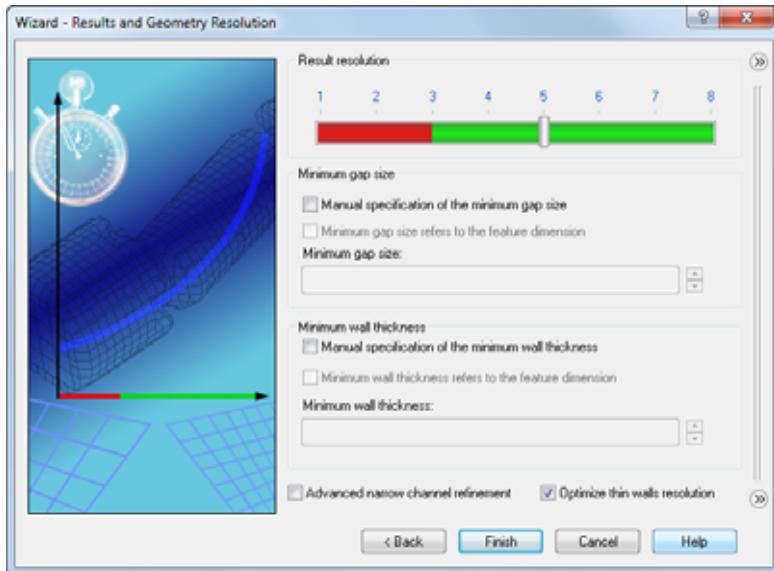
Accept the default conditions and click **Next**.



12 Results and Geometry Resolution.

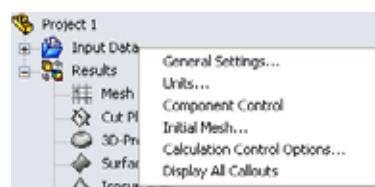
Set the **Result Resolution** to 5.

Click **Finish**.



13 Review Flow Simulation analysis tree.

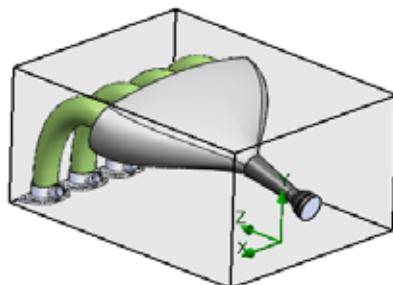
SolidWorks Flow Simulation has created and activated an additional configuration called Project 1. In addition, the Flow Simulation analysis tree tab has been created in the FeatureManager.



Select the Flow Simulation analysis tree tab .

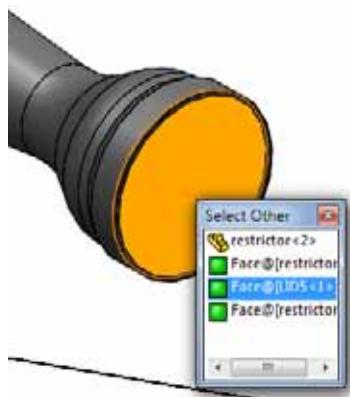
If changes need to be made to the settings of the project, you can right click the Input Data folder and select the appropriate option.

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.

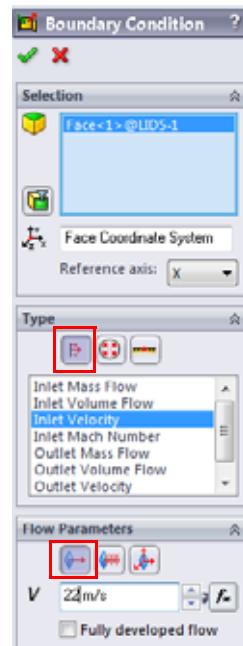
15 Set up boundary condition.

Under Type select **Flow Openings**.

Select **Inlet Velocity**.

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

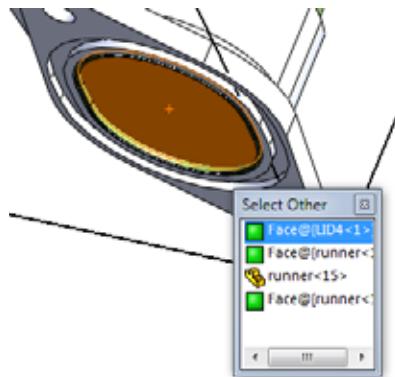
Click .



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.

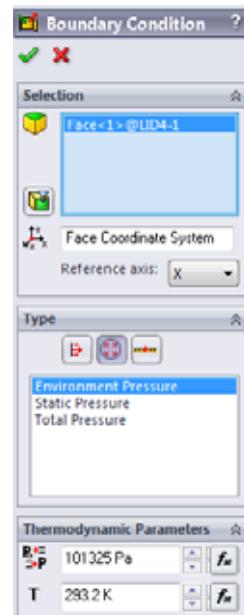


17 Set up boundary condition.

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.

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.

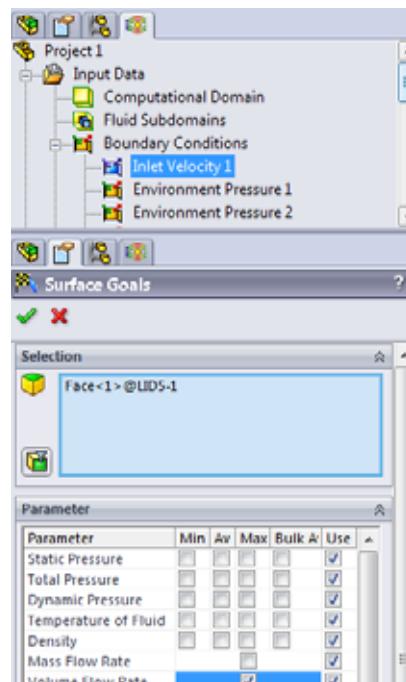
19 Surface goal.

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.

21 Surface goal.

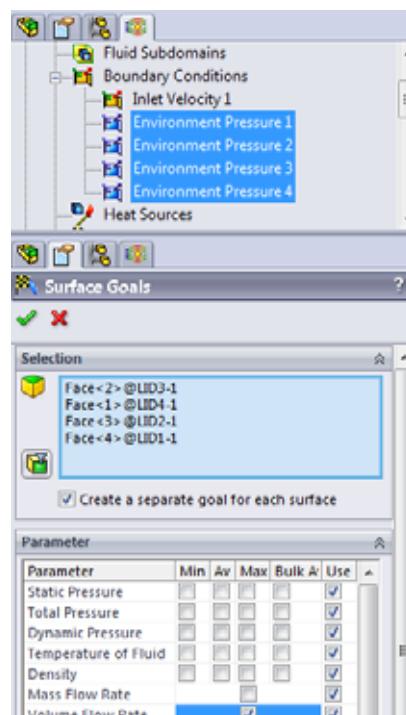
Repeat the previous steps to apply a surface goal for the volume flow rate at the outlets.

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.

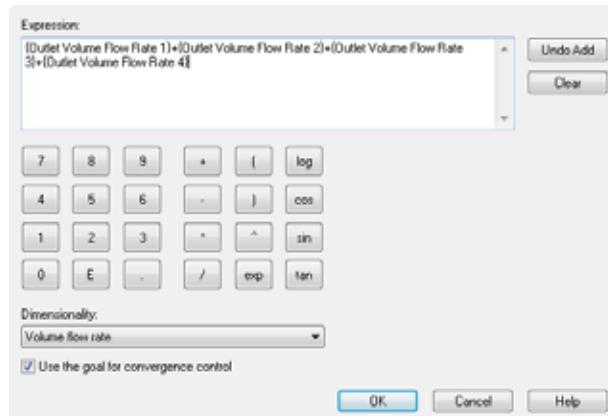
Right-click Goals and select **Insert Equation Goal**.

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**.



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.

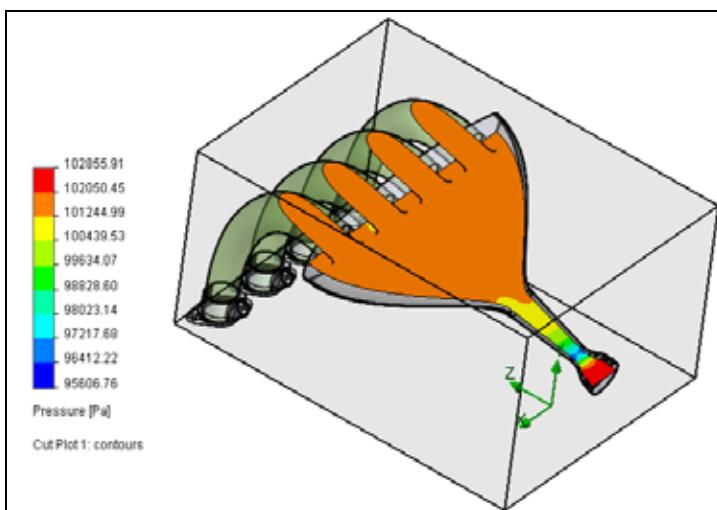
Note: The transparency can also be changed by right-click the parts in the FeatureManager design tree.

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 .



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.

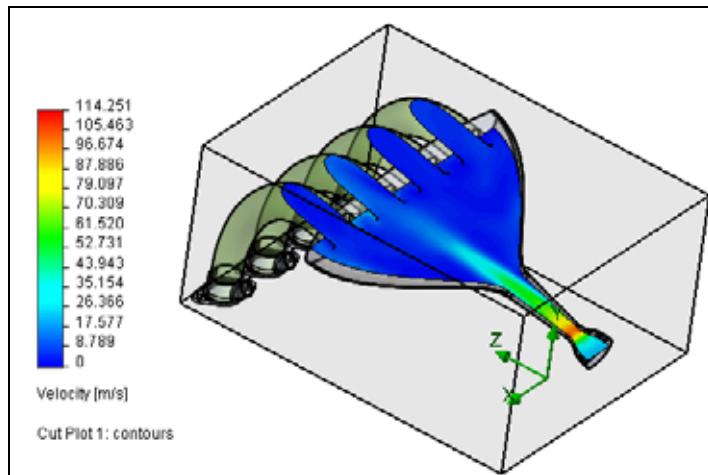
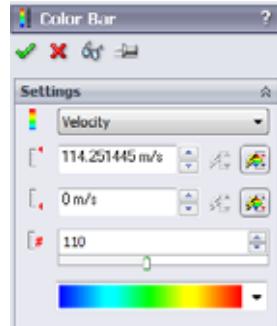
27 Adjust view settings.

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**.



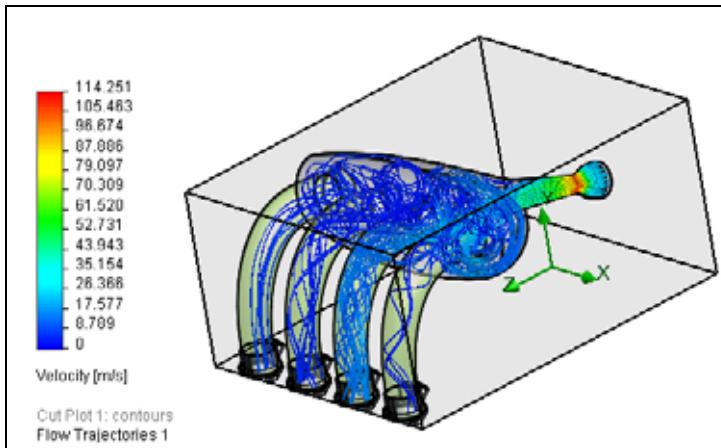
Hide the Cut Plot when you are through.

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.

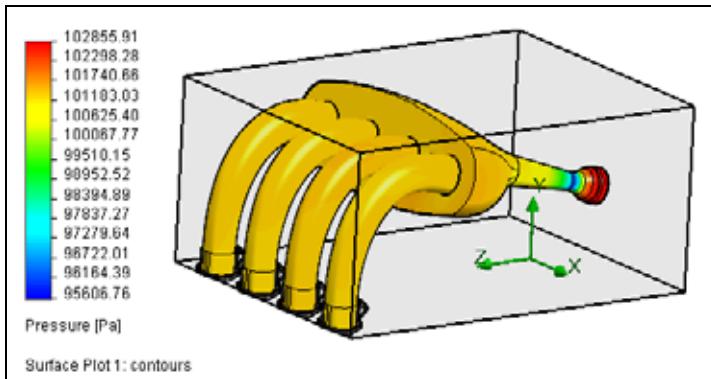
29 Surface plot.

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 .



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	Min/Max
Inlet Volume Flow Rate	[m^3/s]	0.028436893	0.028437589	
Outlet Volume Flow Rate 3	[m^3/s]	-0.013188415	-0.012900775	
Outlet Volume Flow Rate 1	[m^3/s]	-0.007475256	-0.007463144	
Outlet Volume Flow Rate 4	[m^3/s]	-0.003294207	-0.003300252	
Outlet Volume Flow Rate 2	[m^3/s]	-0.004089407	-0.005109606	
Sum of Outlet Volume Flow	[m^3/s]	-0.028851948	-0.028853857	

Iterations: 414
Analysis Interval: 52

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 your 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.