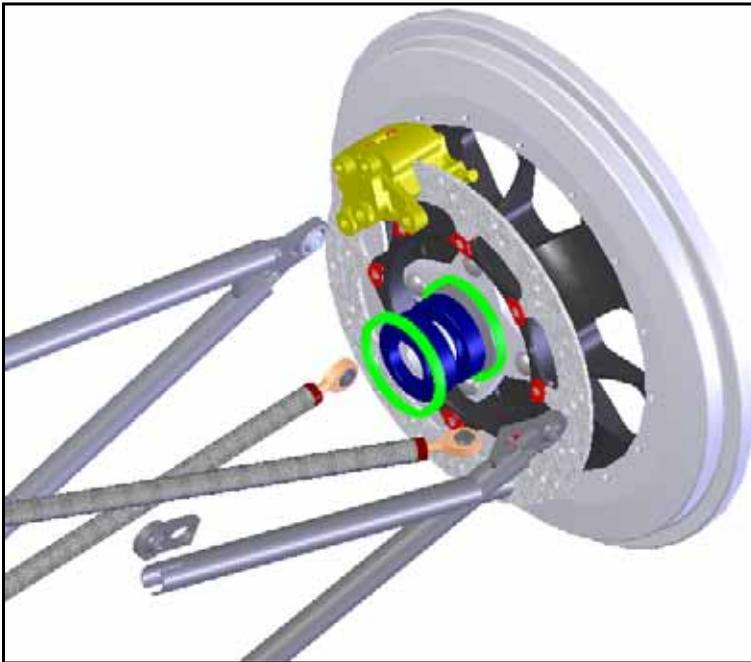




SAE[®] Design and Analysis Project with SolidWorks[®] Software



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Lesson 2

Using Assemblies

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

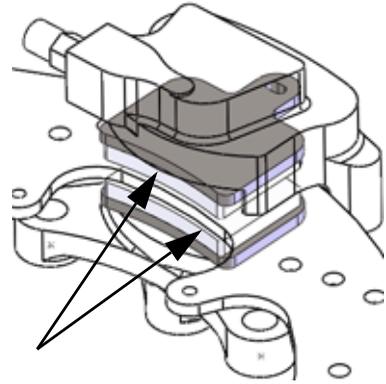
- Understand the difference between edit assembly and edit part modes;
- Create a virtual, in-context part;
- Open a part from the assembly;
- Create a new instance from an existing instance;
- Set the materials in a part;
- Use pack and go to manage the files.

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 make 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.

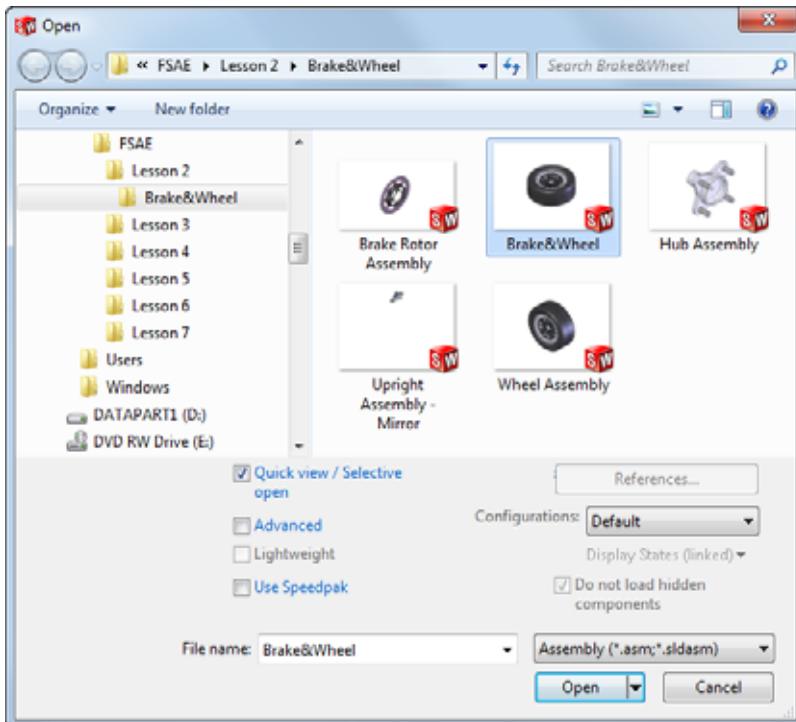


Opening an Assembly with Quick View

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.

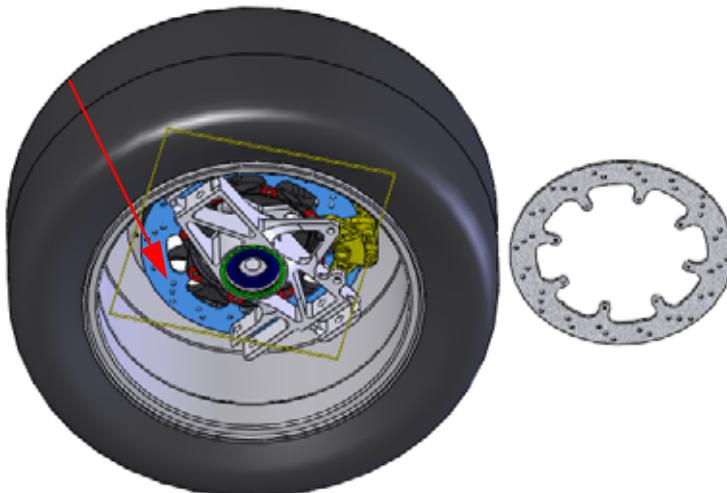
	Hiding Components	Showing Components
Many Components	Hide Others- Hides all visible components but the selected component(s).	Show Hidden Components- Shows all the hidden components for selection.
Single or Few Components	Hide/Show Components- Hides the selected visible components.	Hide/Show Components-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

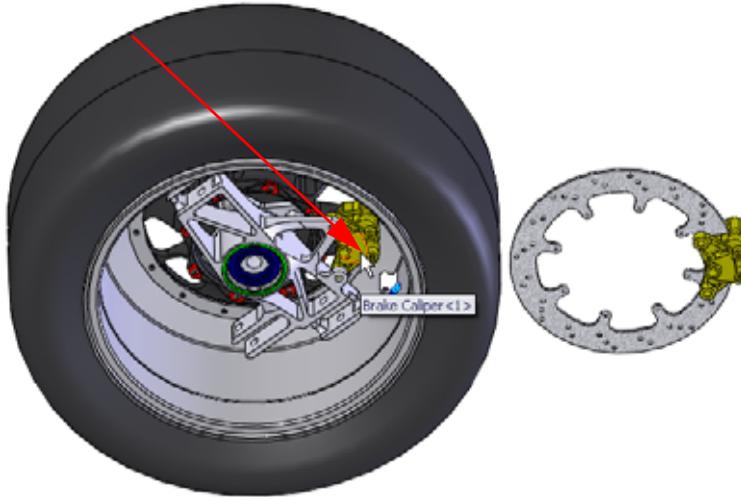
3 Hide others.

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.

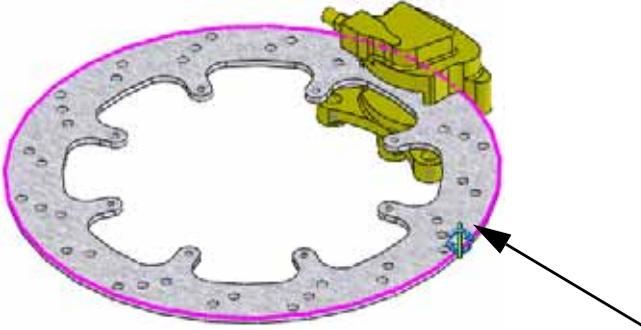


8 Save.

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

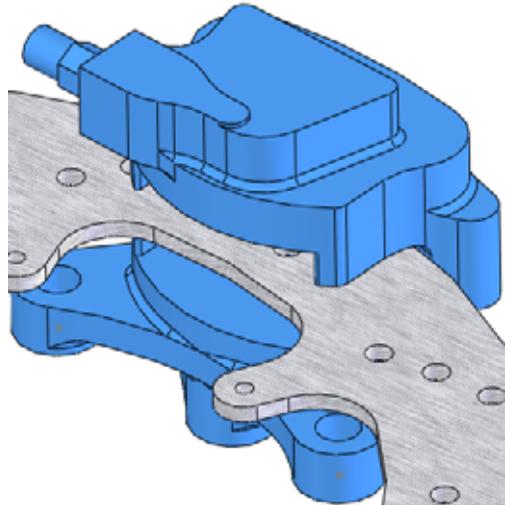
9 Orientation changes.

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.



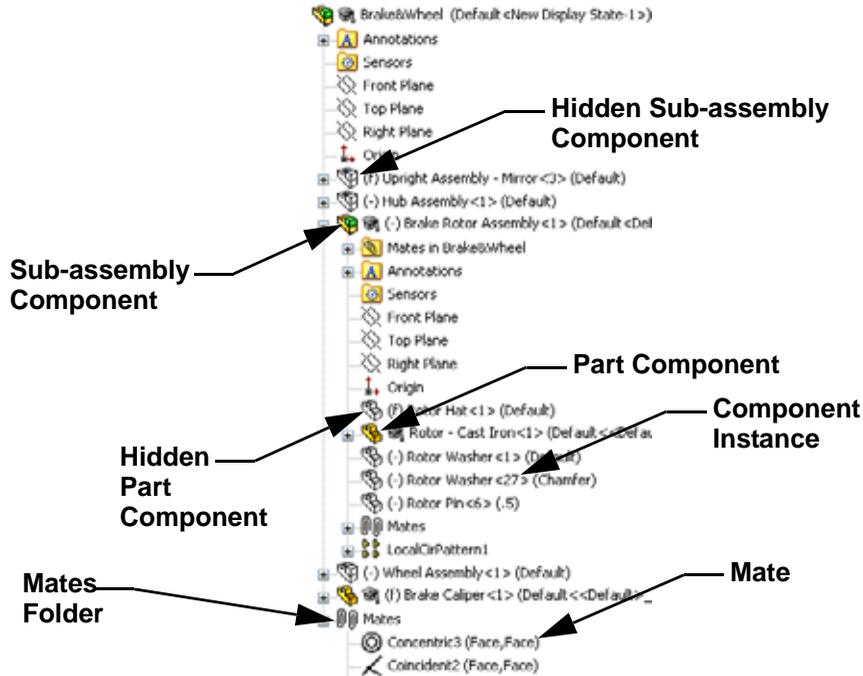
10 Zoom.

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



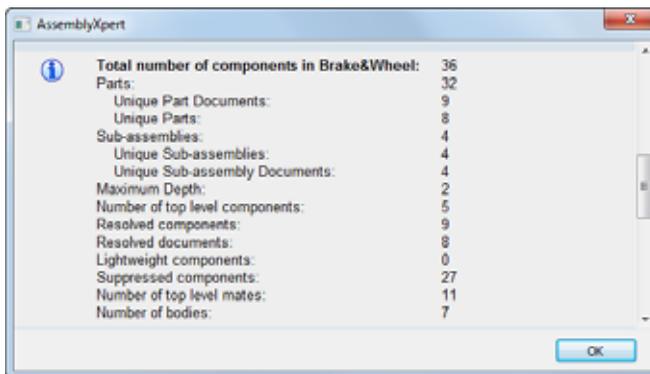
The Assembly FeatureManager design tree

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.

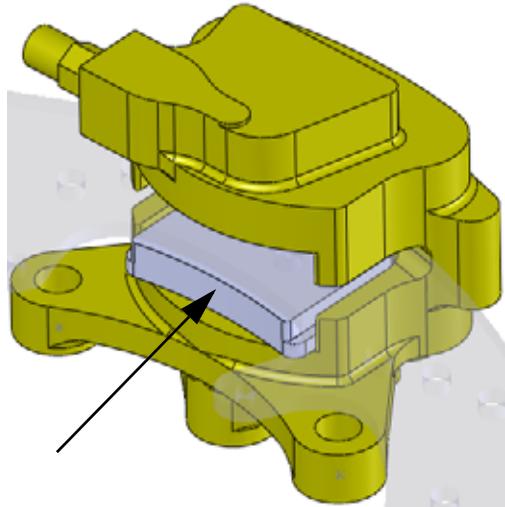
Click **Colors** and click **Use specified color when editing parts in assemblies**. That color is listed under the **Assembly, Edit Part** setting.

Click **OK**.

Creating a New Part

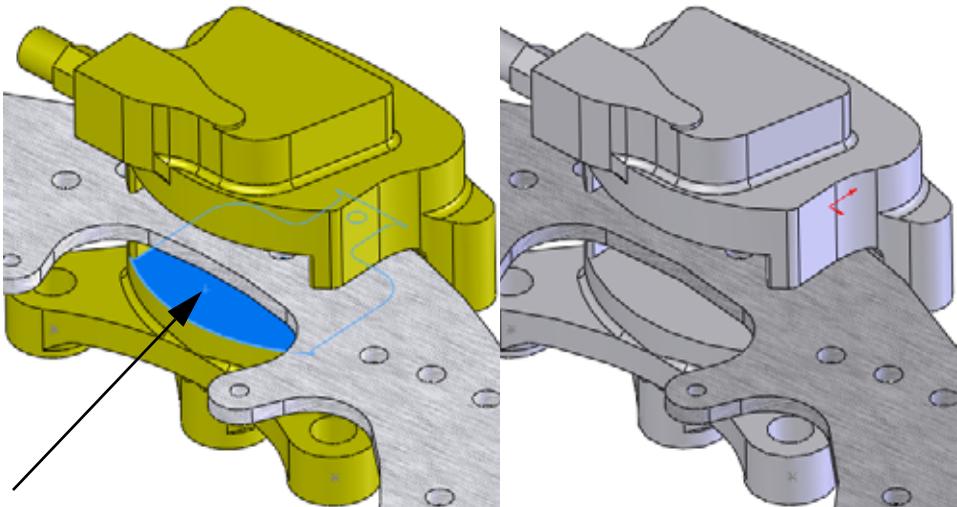
Creating an in-context new part requires a few selections including a planar face or plane for use as a sketch plane.

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.

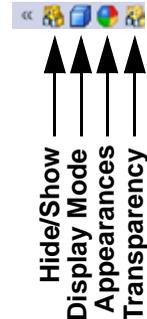
Controlling the Display

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.

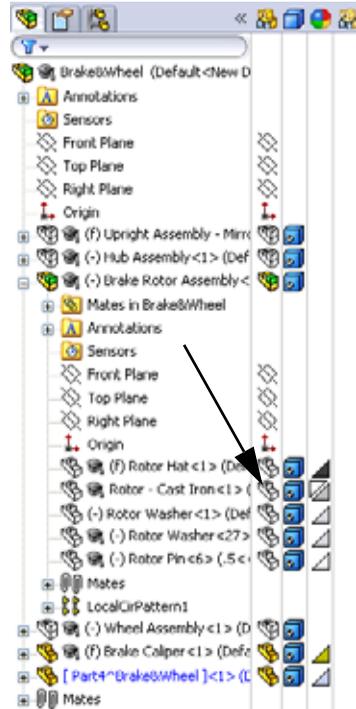
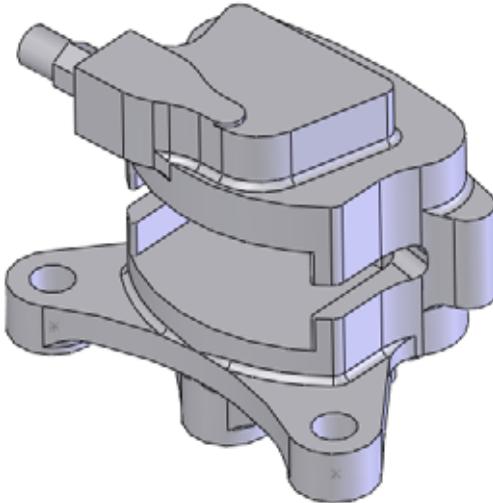


Note: The display pane works independently of the mode.

16 Display pane.

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

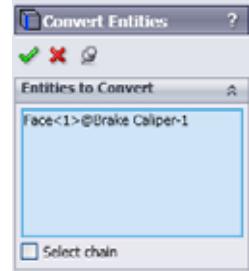
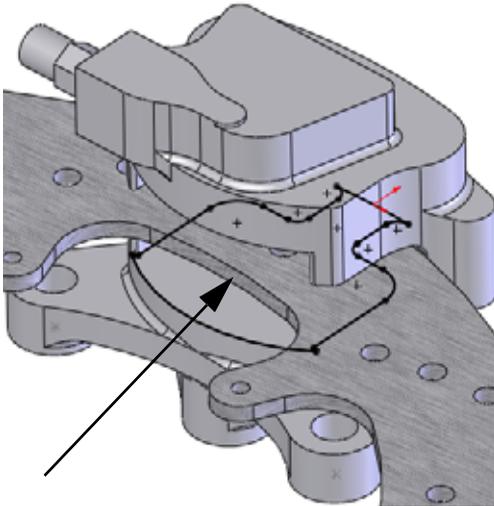
Expand the Brake Rotor Assembly folder. Click the Rotor - Cast Iron component in the **Hide/Show** column as shown to hide it.



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

17 Convert entities.

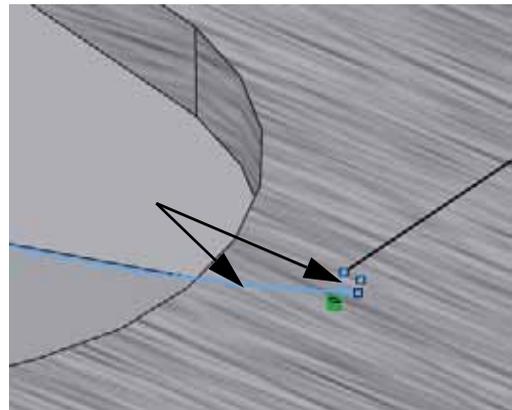
Click **Convert Entities**  and select the face and click  twice.



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.

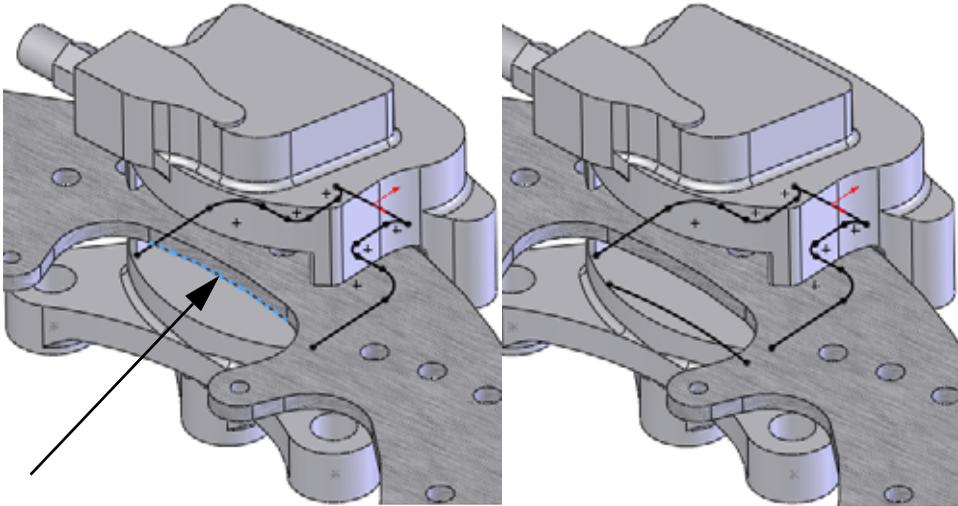
Delete three entities to open the end of the sketch.



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

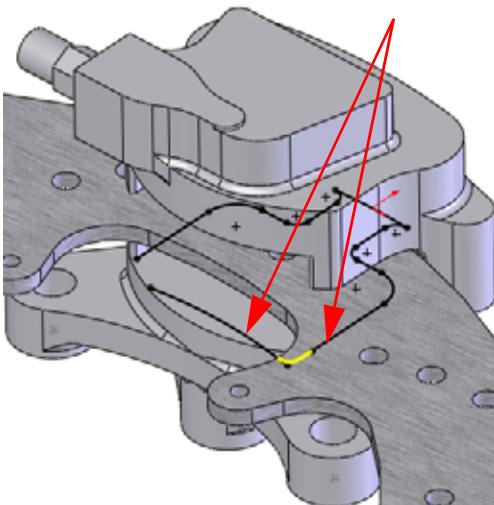
19 Convert edge.

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.

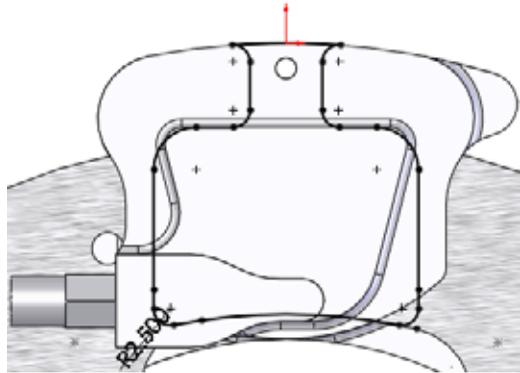
Click **Tools, Sketch Tools, Fillet** , set the **Fillet Radius** to **2.5mm**, and select the first set of geometry by selecting the geometry inside of where they would intersect.



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

21 View normal to.

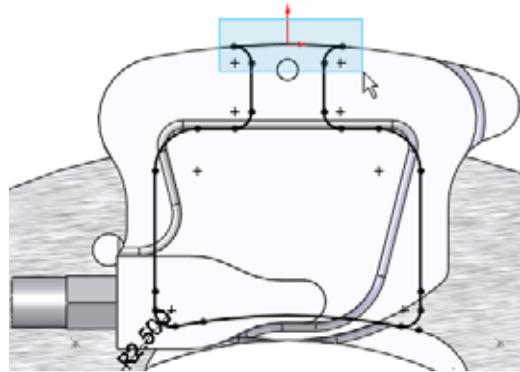
Click **View Normal To**  and zoom in as shown.



22 Delete.

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

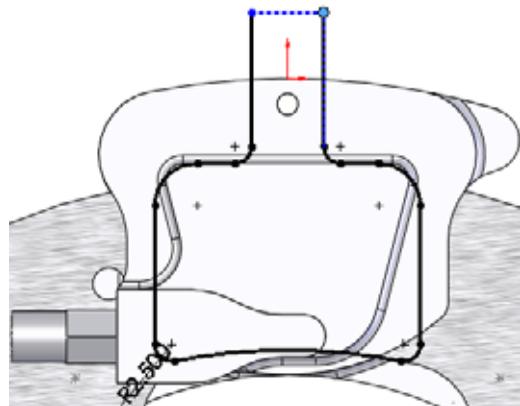
Delete the three selected entities.



23 Drag endpoints.

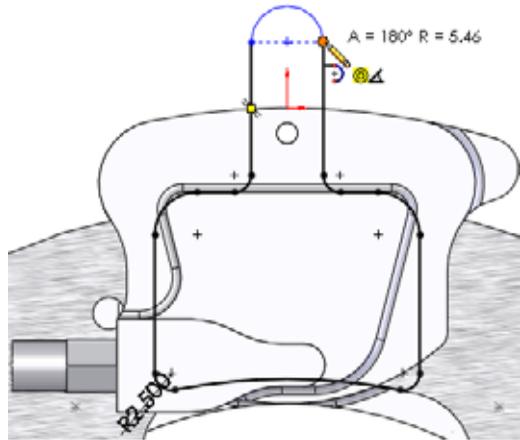
Drag the endpoint of the vertical line outside the geometry as shown.

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



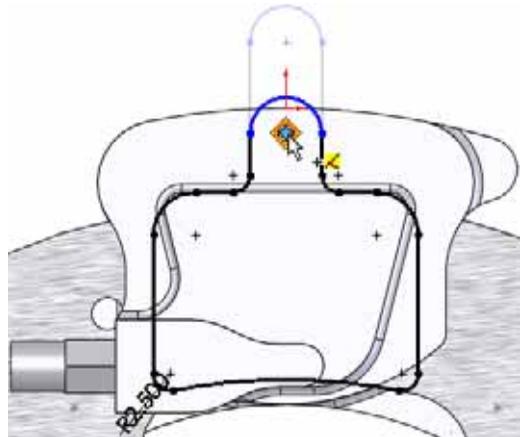
24 Tangent Arc.

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.

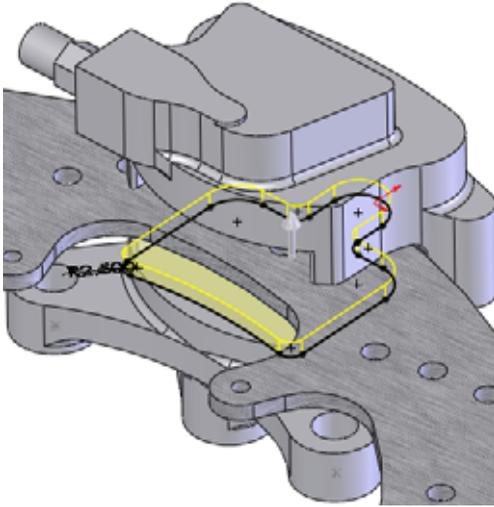


26 Previous view.

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

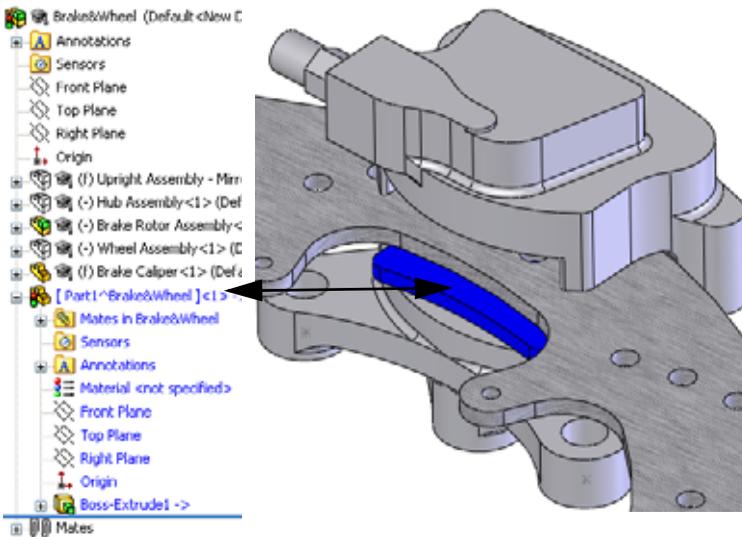
27 Extrude.

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



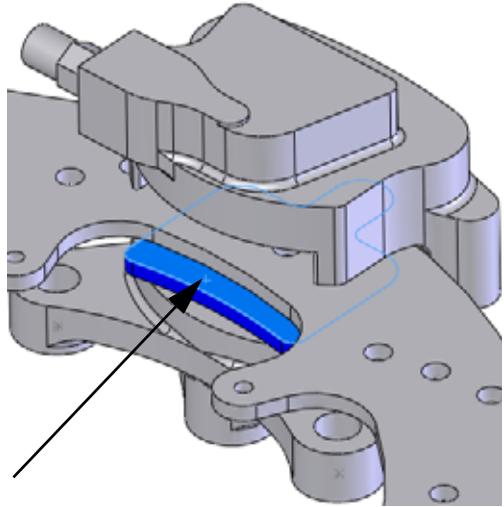
Understanding the Color Coding

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.



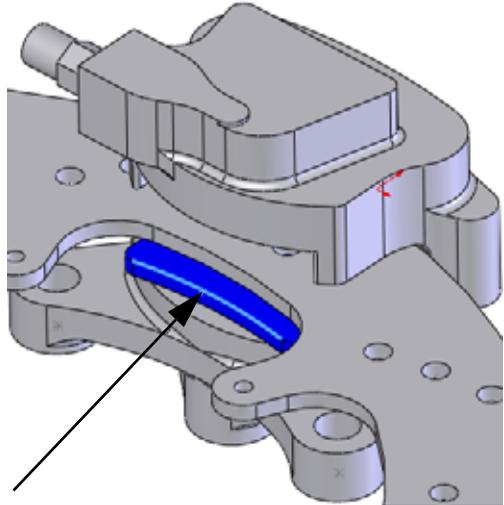
28 New sketch.

Select the face and click **Sketch** .



29 Convert entities.

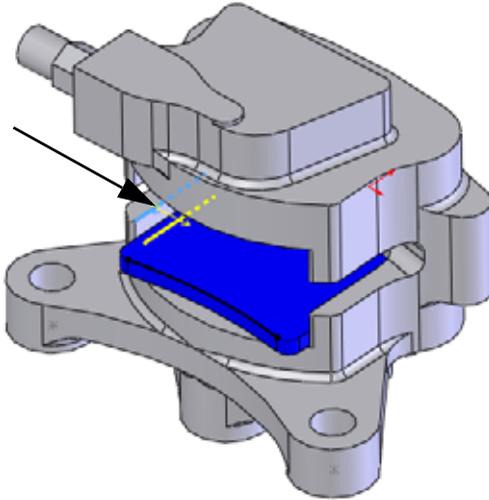
Select the edge and click **Convert Entities**  as shown. Click .



30 Offset entities.

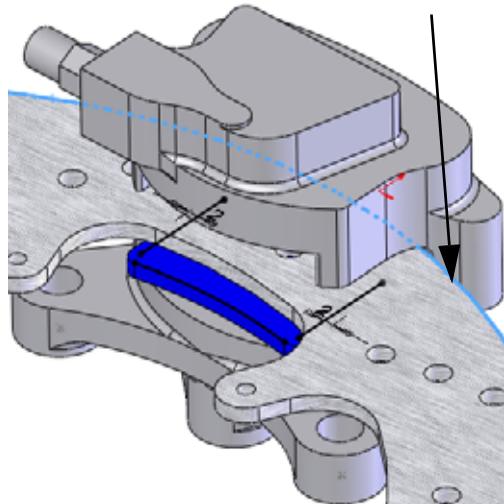
Hide the Rotor - Cast Iron component. Click **Offset Entities**  and set the **Offset Distance** to **2mm**. Select the edge, click **Reverse** and click .

Repeat the procedure for the opposite side.



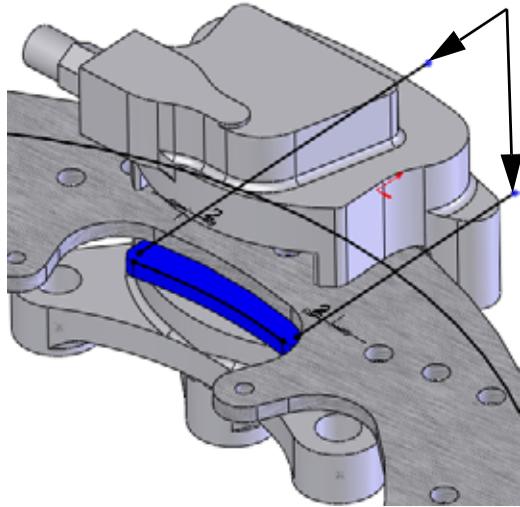
31 Convert.

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



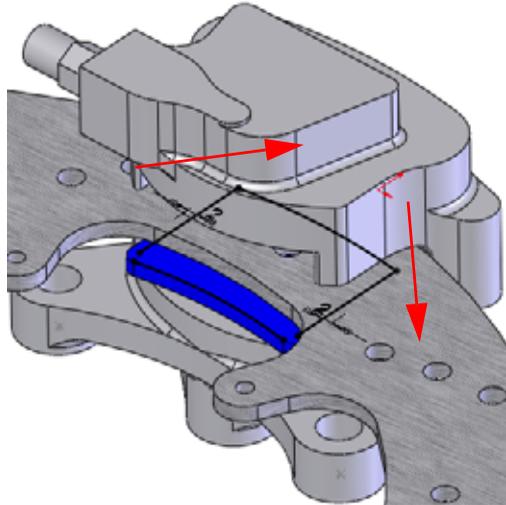
32 Drag.

Drag the open endpoints out beyond the converted edge.



33 Trim.

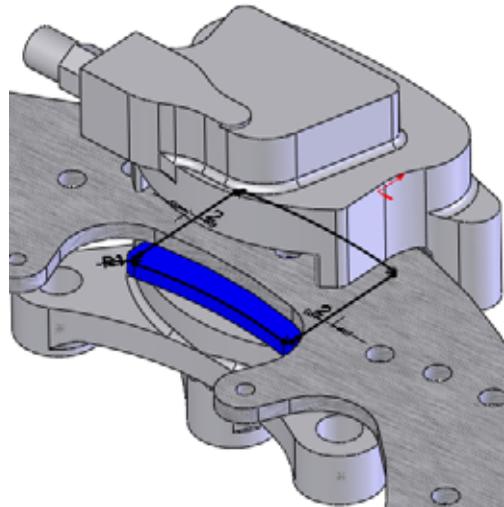
Click **Trim Entities** ,
Power trim . **Click+Drag** across the sections of geometry using the paths shown to trim away excess geometry.



34 Sketch fillets.

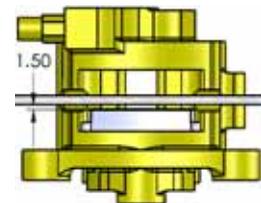
Add sketch fillets **Radius 1mm** in four places as shown.

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



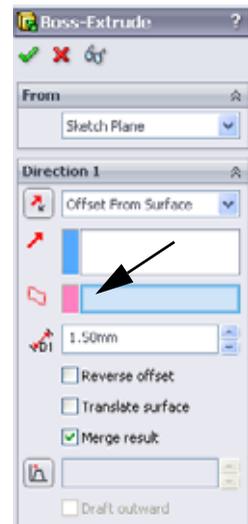
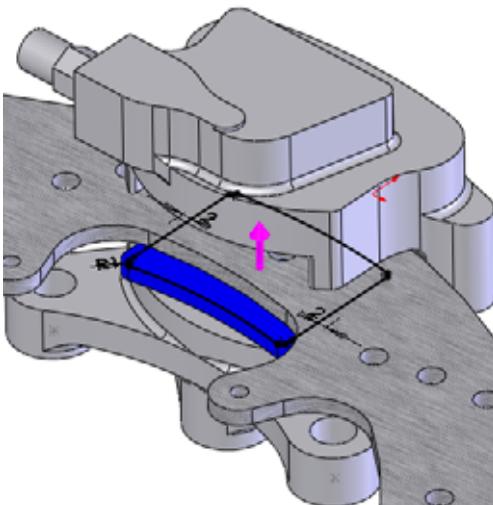
Extrusions In-Context

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.



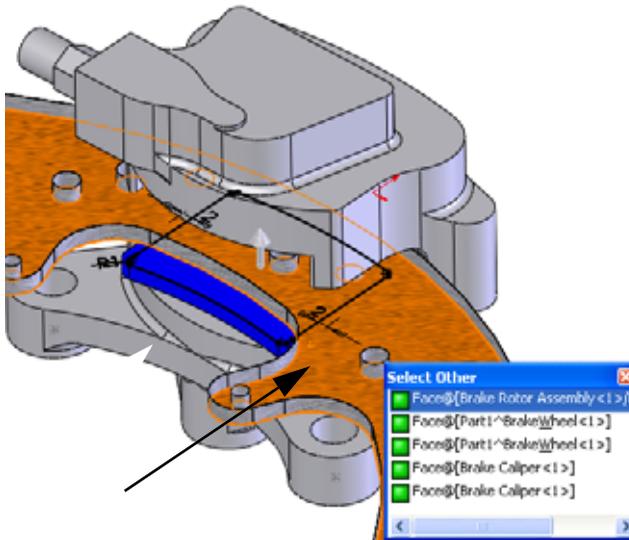
35 Extrude.

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.

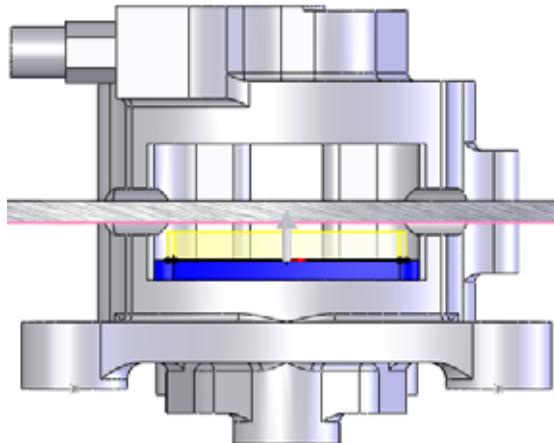
Right-click the face as shown and select **Select Other**. Click the top selection, Face@[Brake Rotor Assembly<1>/R



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

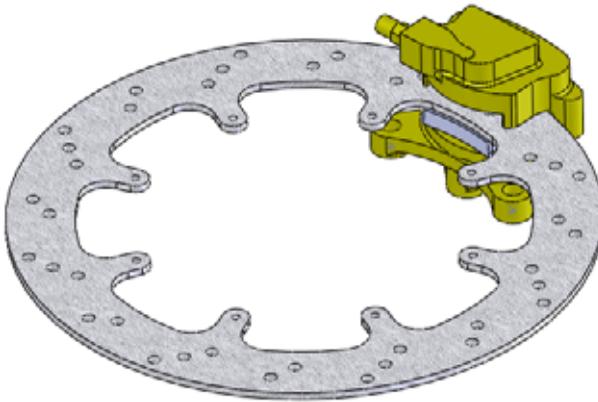
Edit Assembly Mode is the opposite toggle of **Edit Part Mode** and is the default state of the assembly where you can add components and mates. It is triggered by exiting the editing of a part in the assembly or opening of an assembly file.



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.

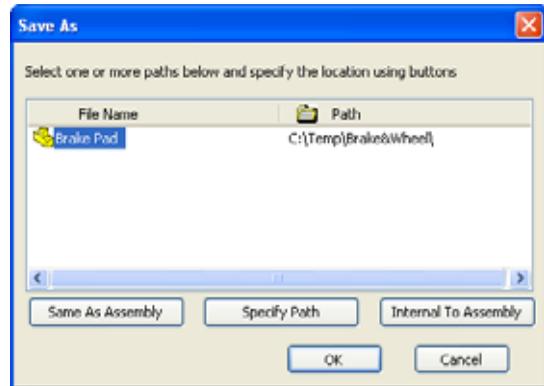
Saving a Virtual Part as an External Part

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**.



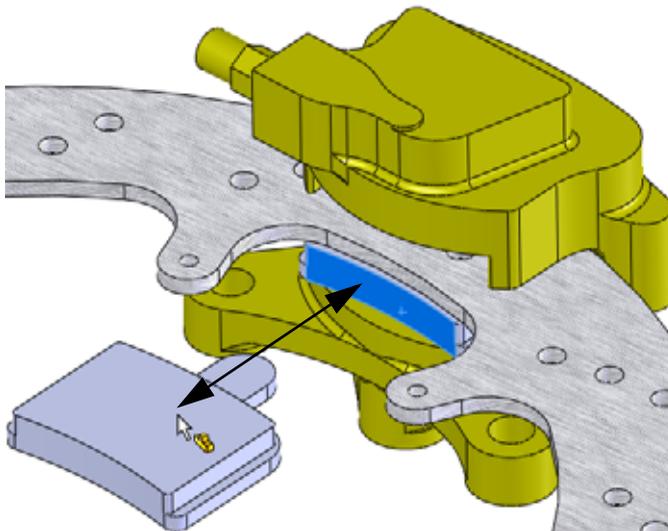
Tip: The part name appears without the square brackets.

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.

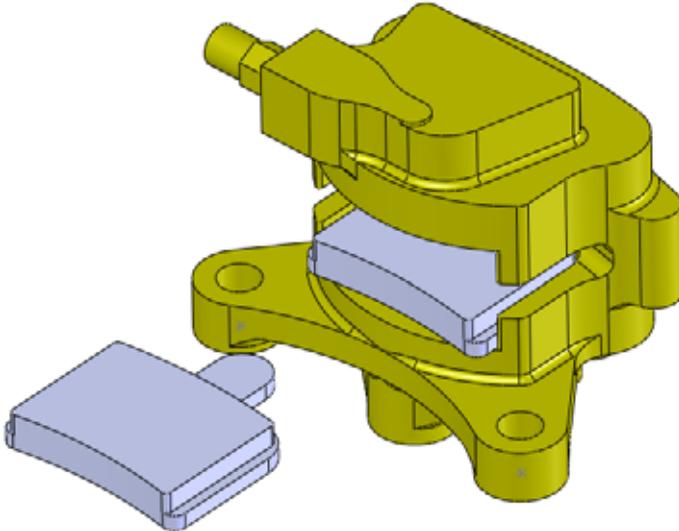
4 Copy an instance.

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



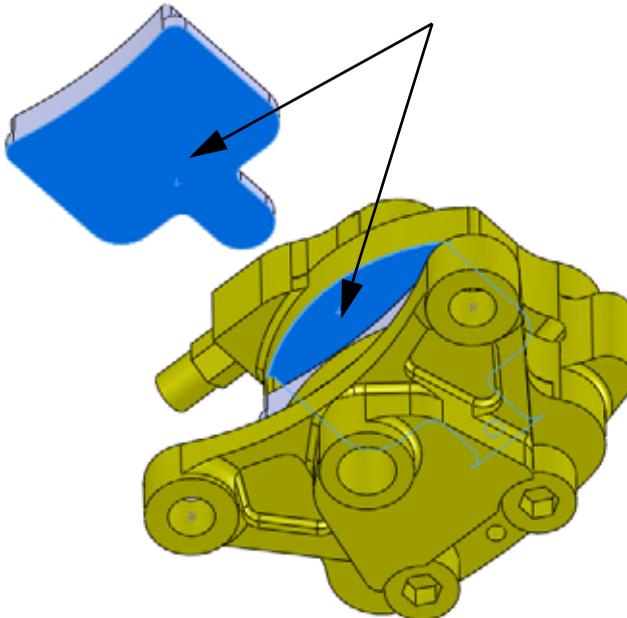
5 Hide component.

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



6 First mate.

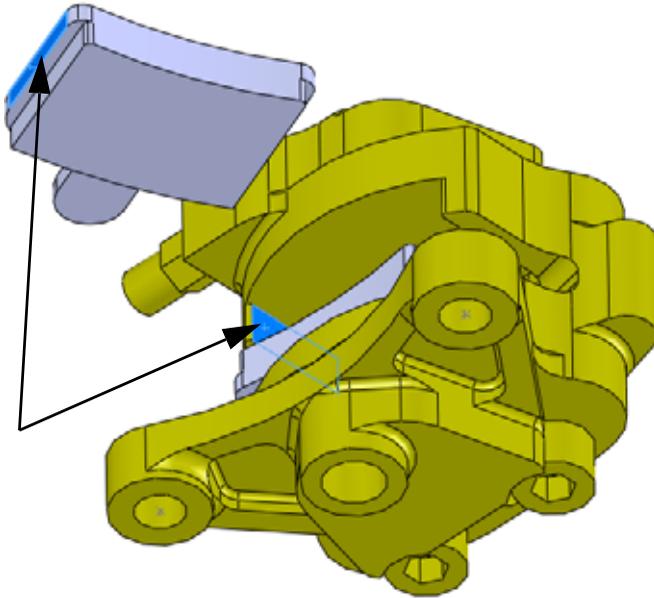
Press **Shift+Up Arrow** key. Click **Insert, Mate**  and select the faces as shown.



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

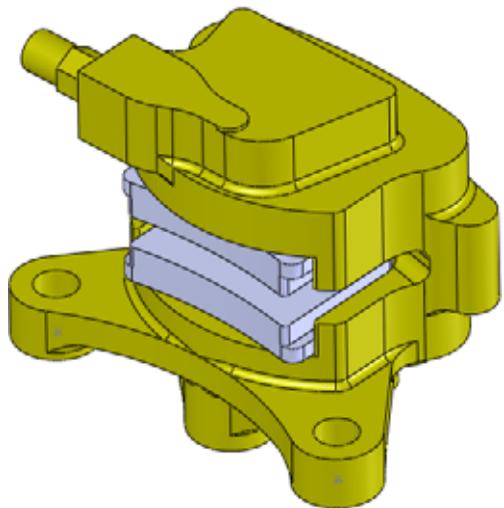
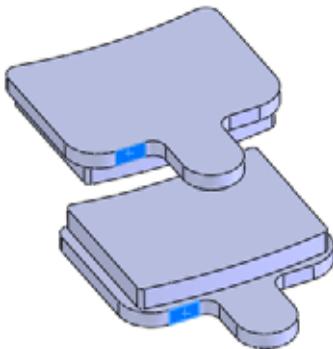
7 Second mate.

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.

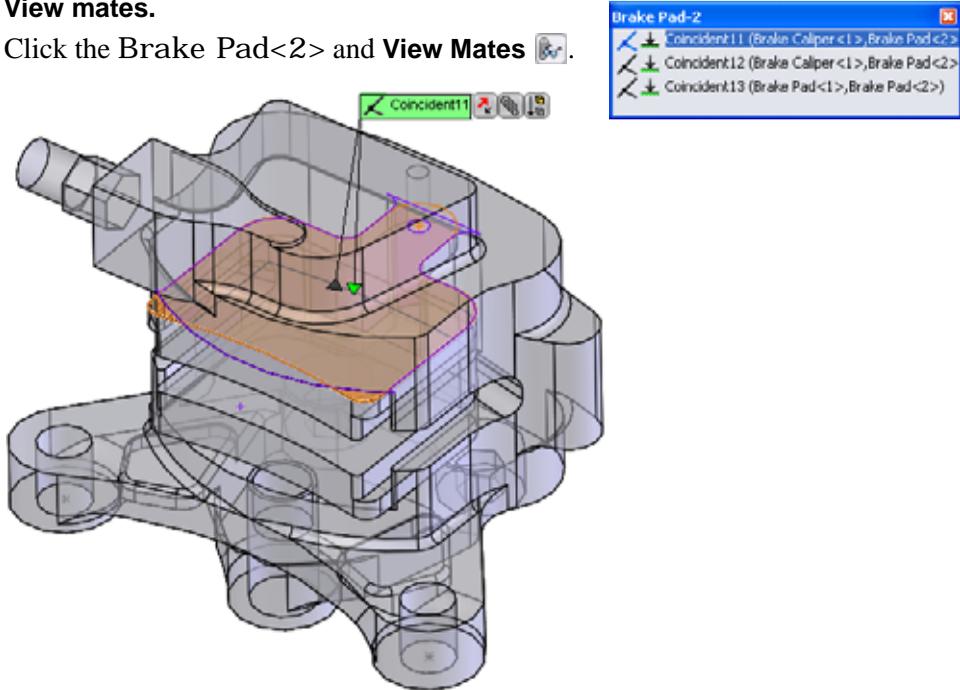
Viewing the Mates of a Component

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

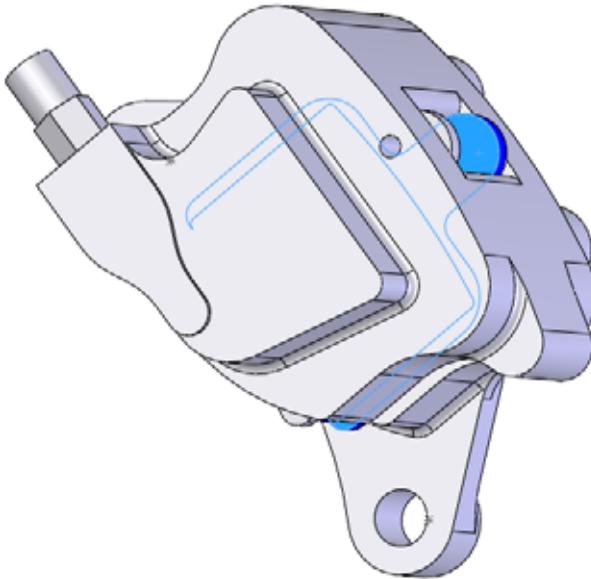
Any component part can be edited in the assembly whether it was created in the context of the assembly or not. Toggling back to edit part mode uses the same command; **Edit Part**.

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.

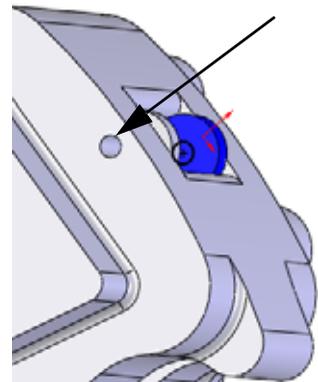


12 Convert edge.

Click the circular edge and click

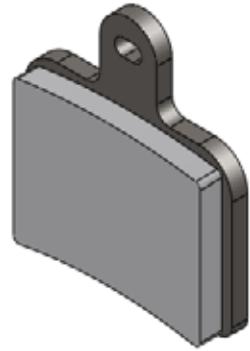
Convert Entities . Click .

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.



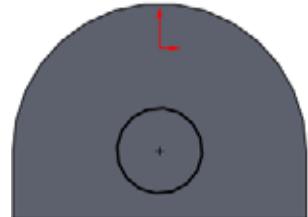
1 Open Brake Pad.

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** .

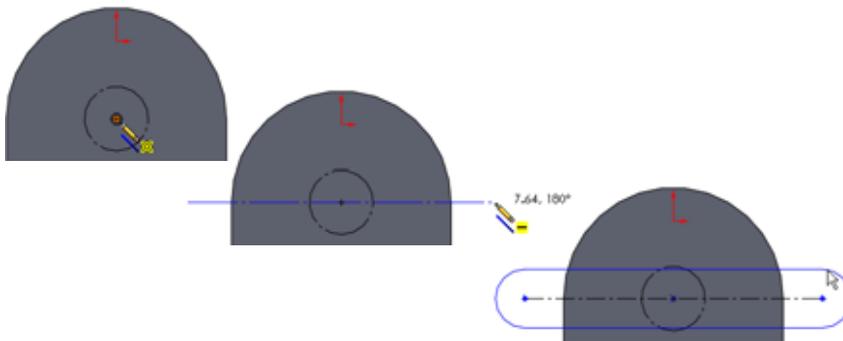


3 Construction.

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

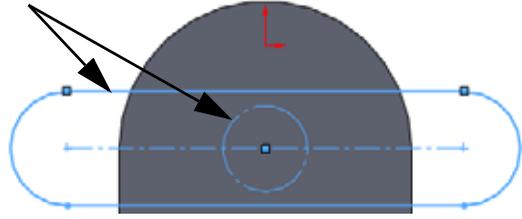
4 Slot.

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 .



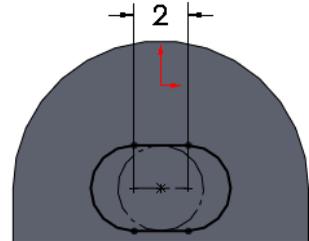
5 Relations.

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



6 Dimension and cut.

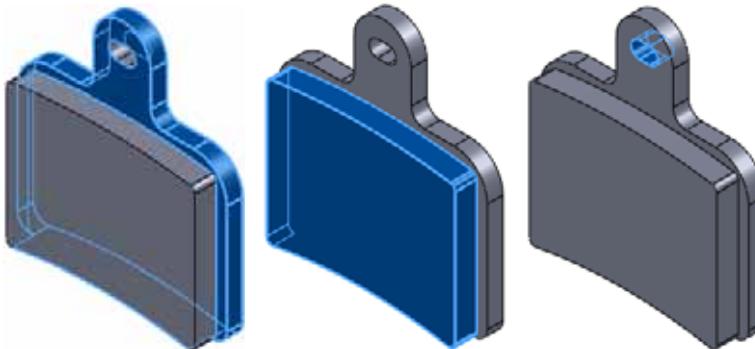
Add a **2mm** dimension as shown to fully define the sketch. Create a cut using a **Through All** end condition.



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.



Boss-Extrude1->

Boss-Extrude2->

Cut-Extrude1->

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.

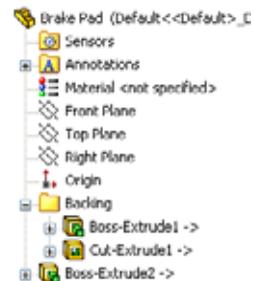
8 Reorder.

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



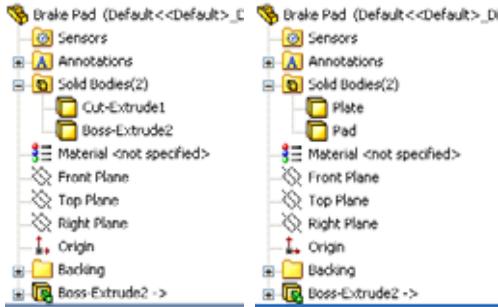
9 Folder.

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.



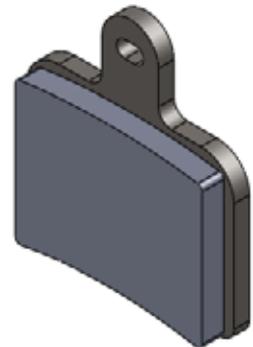
Tip: The default names were taken from the last feature that was applied to the body.

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.

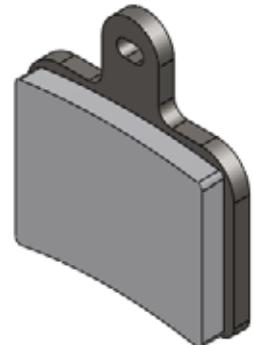
Right-click the body Plate and select **Material, Edit Material**. Under **Steel**, select **1023 Carbon Steel Sheet (SS)**. Click **Apply** and **Close**.



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.

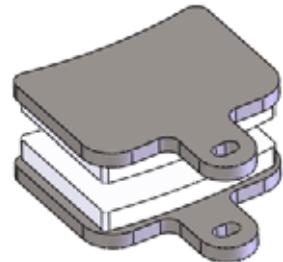


This message appears:

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.



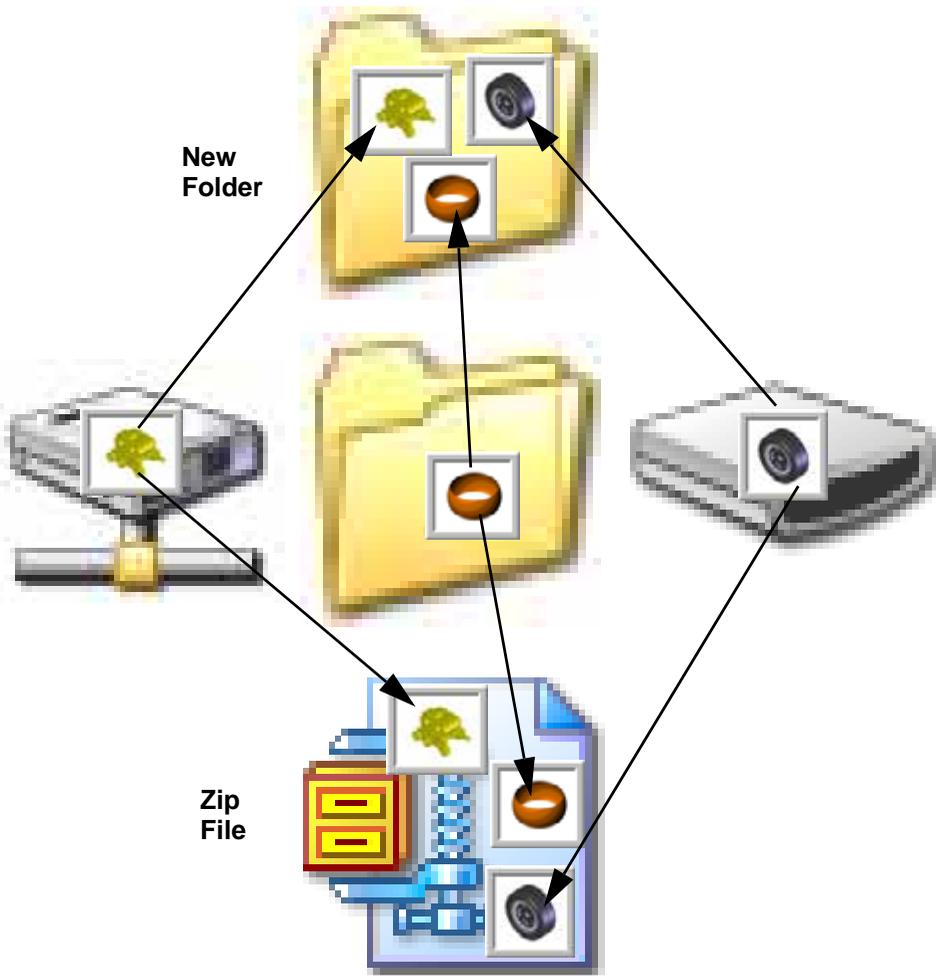
15 Display state.

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

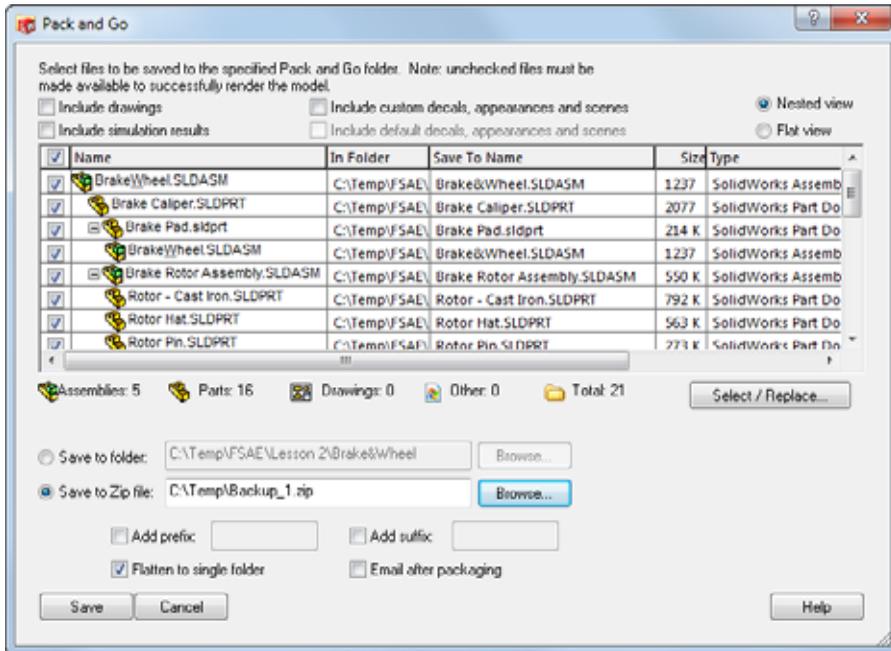
The work flow using Pack and Go creates multiple backups, using the last backup to start the next work session.

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

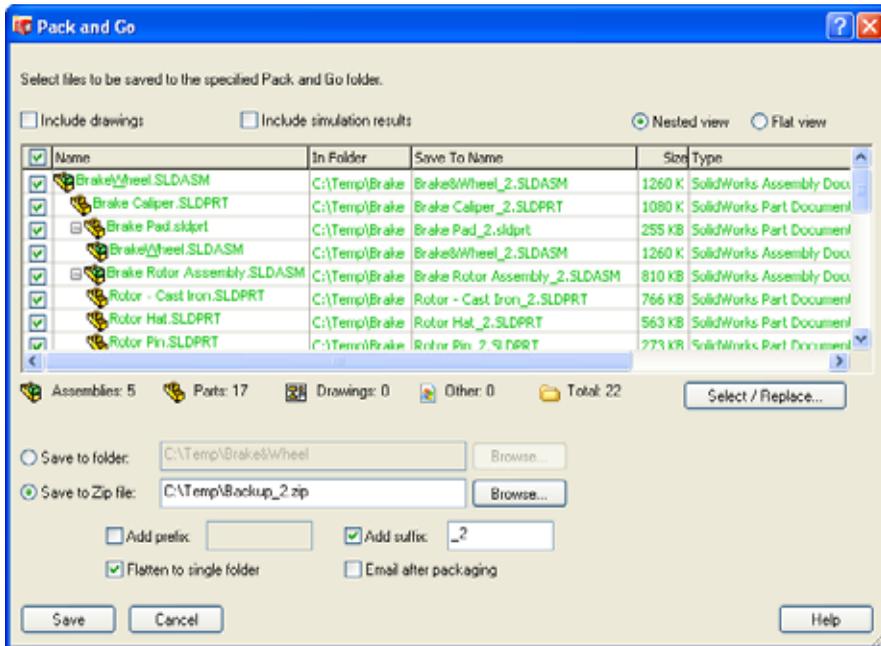
If you want to rename the files with each new backup, the **Add prefix** and **Add suffix** options can be used.

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.