Autodesk Inventor Plastic Features

TECHNOLOGY PREVIEW

Overview

Autodesk® Inventor® Plastic Features Technology Preview is a new opportunity for users to participate early in the evaluation of capabilities that provide intuitive, functional design tools for plastic parts common in consumer electronics.

Autodesk Inventor Plastic Features provides innovative new capabilities to help you create thin wall parts using a new and unique approach where the form of a design can be created using simple shapes and Inventor automatically ensures that the part maintains a constant wall thickness. This removes the often difficult process of determining the right point to create the shell feature for a design. Autodesk Inventor Plastic Features also offers tools to create the complex detailed features common in plastic parts. With Autodesk Inventor Plastic Features you can:

- Capture thin wall part intelligence with new plastic part styles that capture key parameters for thin wall parts
- Use unique shelling technology to create thin wall parts without the need for a shell feature. Extrude, revolve, sweep, loft, fillet, and chamfer all create and modify these thin wall designs
- Create complex mounting faces using the new rest feature
- Fillet your designs with new unique rule based filleting tools
- Create the 3D silhouette curve needed to fine the separation point for top and bottom components of complex organically shaped designs
- Create lips and groves along the mating faces of the top and bottom components
- Create grills and vents
- Create mounting bosses and snaps

Important notes about this Technology Preview

Autodesk is releasing this technology preview to showcase innovative new plastic features technology and to give you an opportunity to influence its future. We invite anyone involved in plastic design to try the Autodesk Inventor Plastic Features Technology Preview and provide your feedback, questions or issues to labs.iv.plastic@autodesk.com. Your feedback helps us make better products and technology, so please tell us what you think!

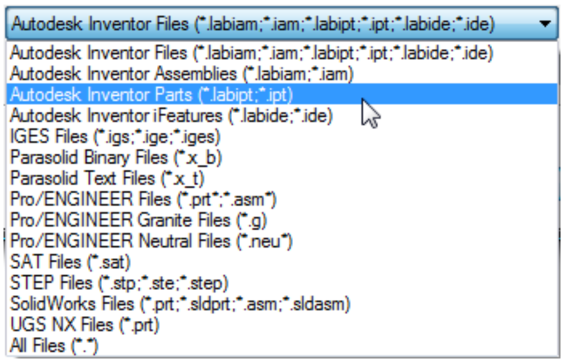
Autodesk Inventor Plastic Features is a full Install based on Autodesk Inventor 2009 Technology. Your use of Autodesk Inventor Plastic Features

Technology Preview is subject to the terms and conditions of Autodesk Labs and the End User License Agreement accompanying the product.

Autodesk Inventor Plastic Features is different from a production release. It has its own special user interface and functionality. The following includes important information about this application:

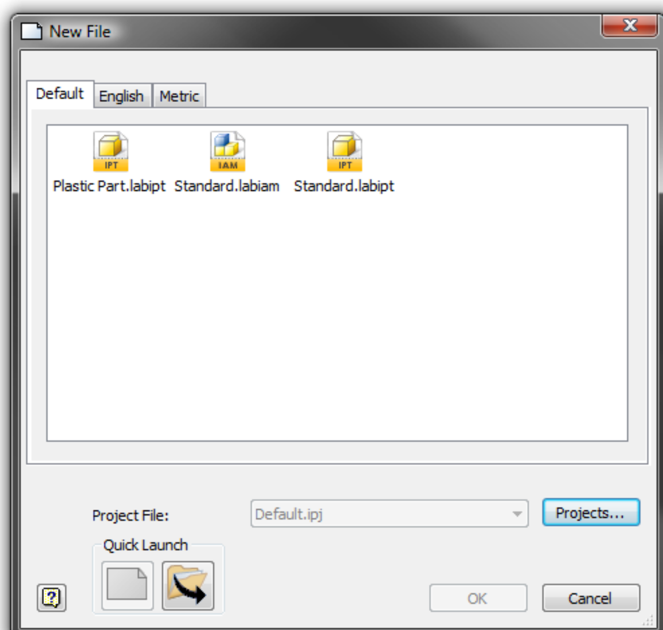
Support new file extensions

To distinguish the files from product data, Autodesk Inventor Plastic Features uses new file extensions for files. For example, we the “xx.labipt” is equivalent to the original file extension “xx.ipt”. New file extensions will be added to the “Open” Dialog.



Because this is an evaluation tool and capabilities here are not part of production releases of Autodesk Inventor, you cannot save production Autodesk Inventor files with this Autodesk Inventor Plastic Features

The “File New” dialog only allows the creation of files for use in Autodesk Inventor Plastic Features



When using existing design, Autodesk Inventor Plastic Features **will copy and convert** the files to the new file extension. This ensures that you do not inadvertently save changes to your product data.

Files will be saved in the directory My Documents/Autodesk Inventor PTP, unless specified by the user in Save As or Save Copy As.

Compatibility with Autodesk Inventor Suite and Professional

Inventor Labs product can open files from AIS (AIP) but, can save the files only to its own file extensions. AIS (AIP) can't open files from Inventor Labs product.

Autodesk Inventor Plastic Features cannot be opened in the released version of Inventor even if the file extension is changed via the file system.

If you need to use a design created with this application in production versions of Autodesk Inventor, please user the STEP translator.

Side by Side Install

You can install Autodesk Inventor Plastic Features with no impact on any existing Inventor product release installation. This includes Autodesk Inventor 2009.

No support the drawing environment.

Drawing files cannot be opened or created.

No support for sheet metal environment

You cannot open or create a sheet metal file in the part environment.

You can open assemblies which contain sheet metal files; however they will be converted to standard parts on open. This will not have any adverse affects on your original part as no production files can be overwritten.

File Time Out

All files saved in a labs release can only be opened for 60 days after the original file creation time.

Limited add in support

Only the Studio add-in will be supported in Autodesk Inventor Plastics Features.

Getting Started

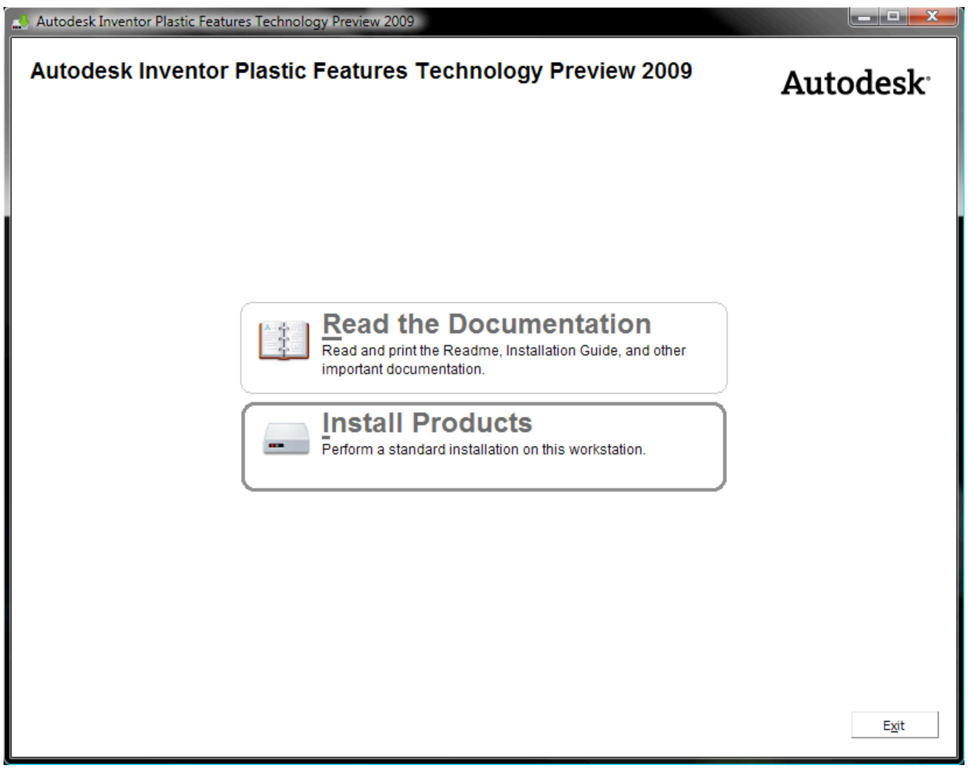
Follow this helpful information to get started with Autodesk Inventor Plastic Features:

System Prerequisites

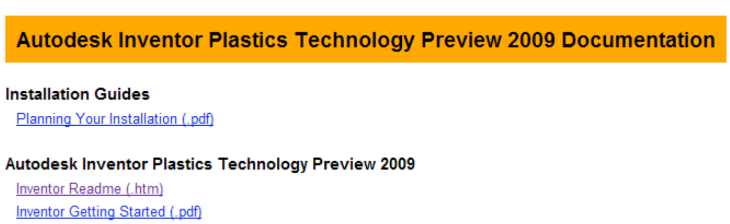
Autodesk Inventor Plastic Features will work on Windows XP and Vista operating systems. It is **not** supported on 64-bit operating systems.

Installation Instructions

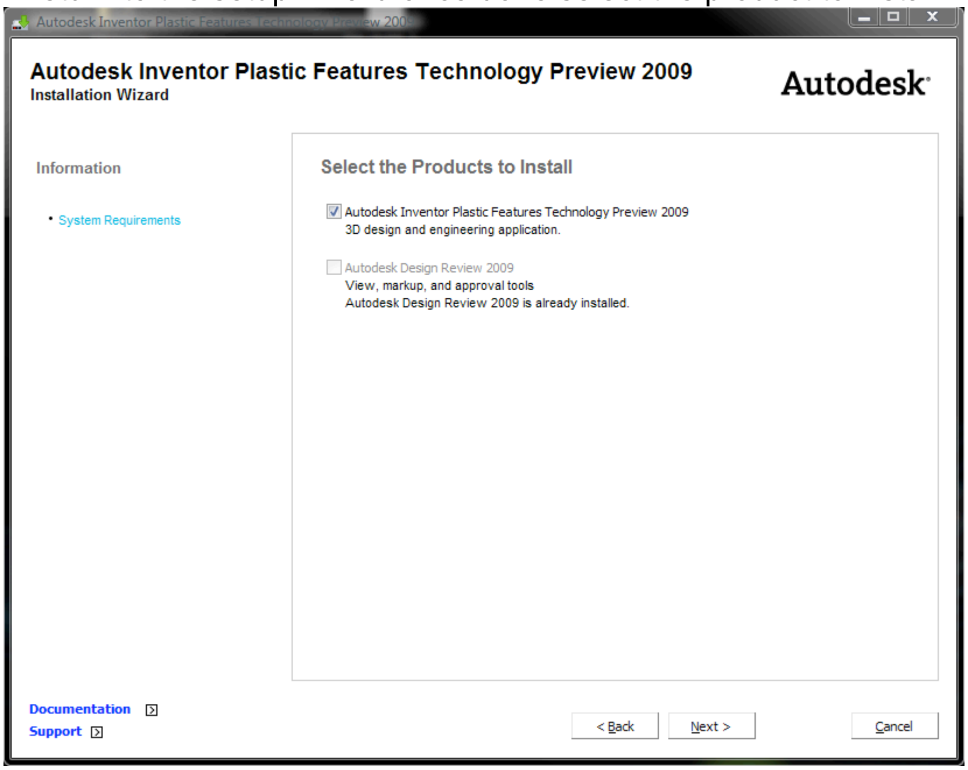
1. Download the Installation for Autodesk Inventor Plastic Features from Autodesk Labs
2. Locate the downloaded archive and unzip it to a local machine. Then run Setup.exe.



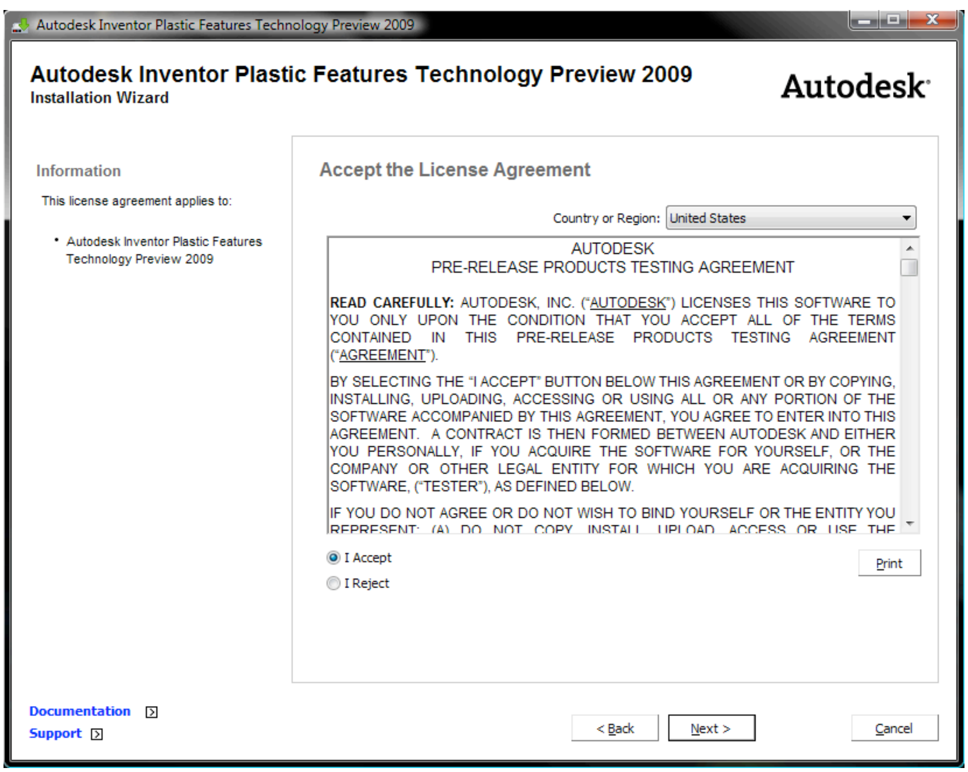
3. It is recommended that you look over the documentation and Readme. Click the Read the Documentation button to get a menu where you can review this important information.



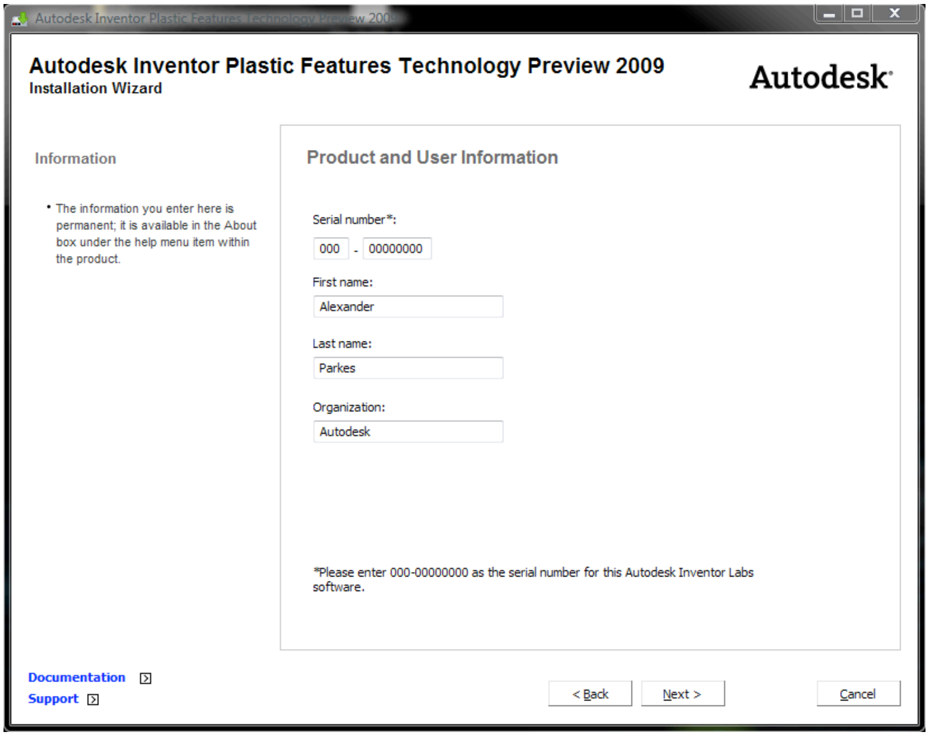
Return to the setup wizard once done select the product to install.



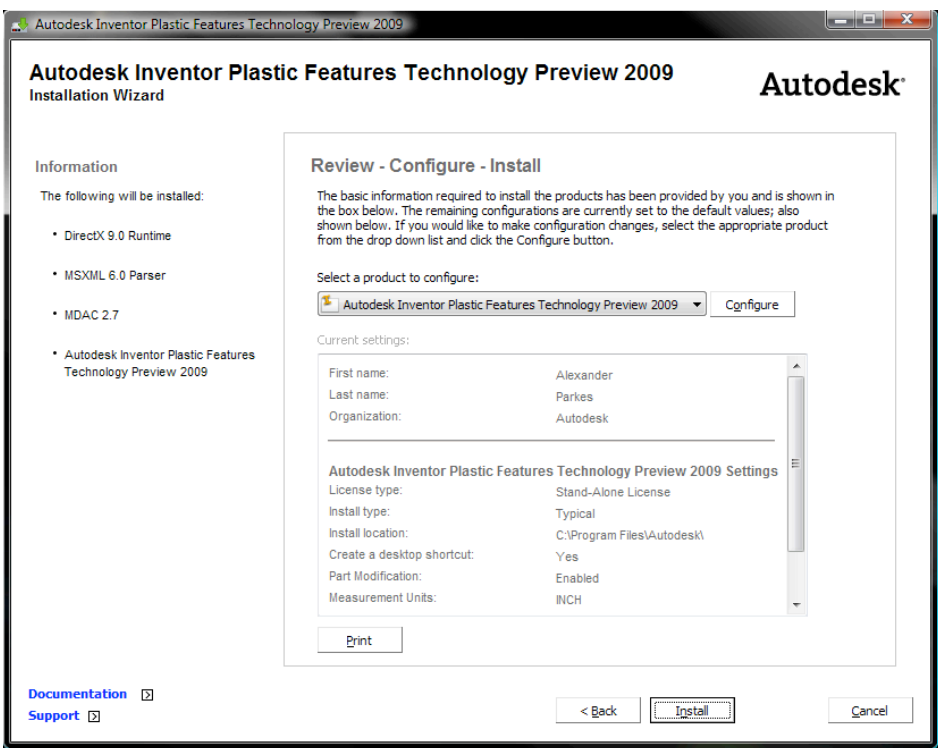
4. Click Next



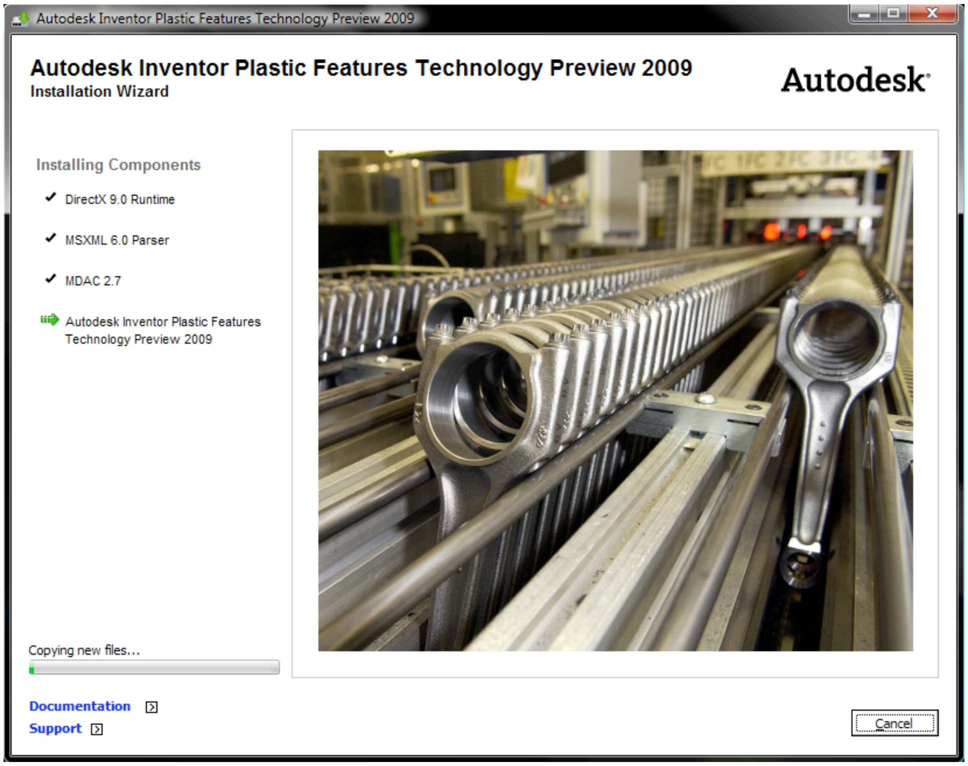
5. Read and Accept the License Agreement. If you try to install Autodesk Inventor Plastic Features on an unsupported language the installer will now quit.
6. Click Next



7. Enter all “000-0000000” for the serial number



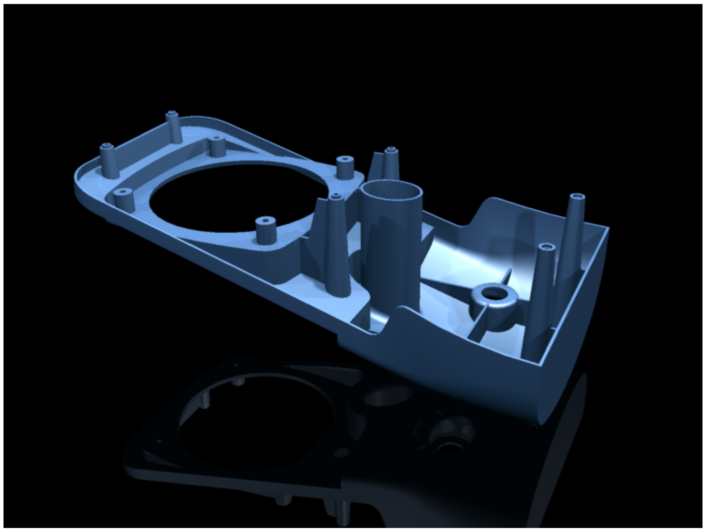
8. If you would like to configure your installation and change default installation options click the configure button. Otherwise click the Install button to begin your installation.



9. After a few minutes time, the application will complete the installation.

Congratulations on installing Autodesk Inventor Plastic Features. If you want to start making parts right way jump to the end of this document where there are two tutorials that walk you through evaluating these exciting and new capabilities.

Autodesk Inventor Plastic Features



Concept

Autodesk Inventor Plastics Features environment is a specialization of the general part modeling application, which is customized for the design of molded, plastic parts. Entering this environment converts a part to a "plastic part", and provides you with commands and features which are not available in the general part environment, as well as additional options to existing commands and features. In general, the Plastic Part Environment will be a superset of the normal Part Modeling Environment. That is, there will not be any tools available in the standard Part Environment that are available in the Plastic Part Environment.

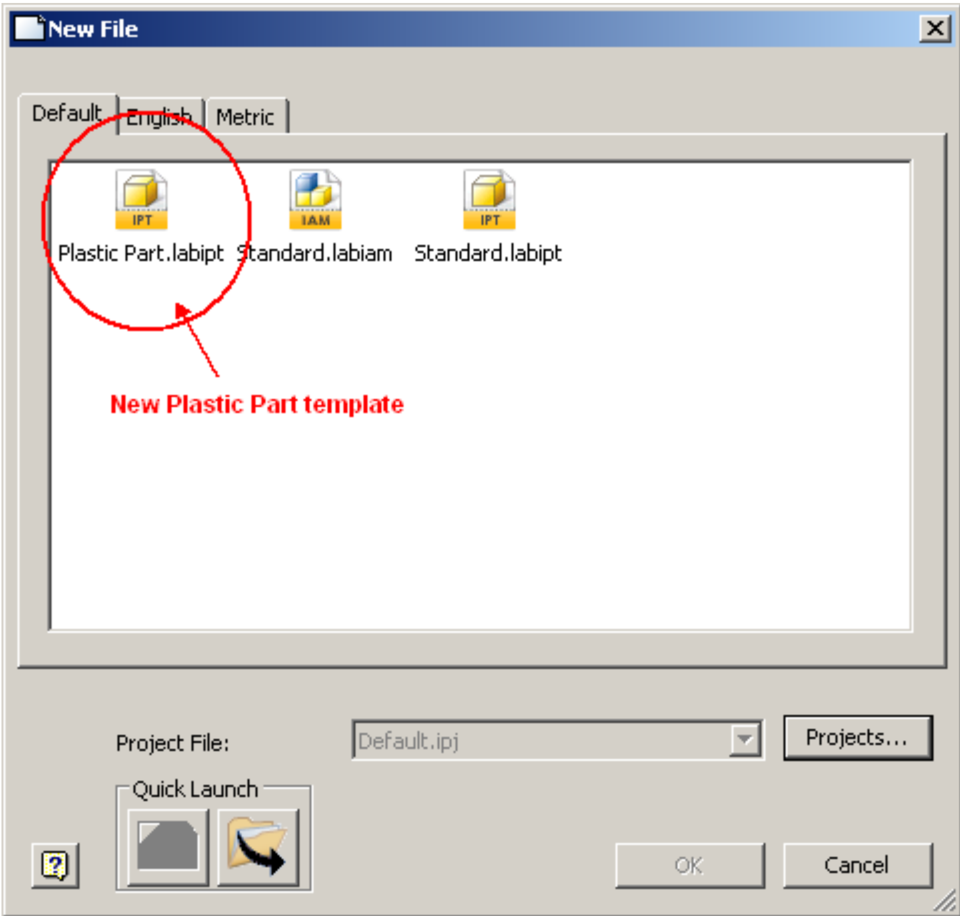
In Plastic Part environment, existing command such as "Extrude", "Revolve", "Sweep", "Loft", "Fillet" and "Chamfer" will have new thin wall options to enable users to create thin-wall-features, where the resulting geometry is of constant thickness, usually uniform across the part. Uniform material thickness is important to the injection-mold process used to fabricate plastic parts. These features are known as "thin features" in Inventor.

In addition, the Plastic Part environment provides many new features which are not available in the normal Modeling Environment, such as "Grill", "Rest", "Boss", "Snap Fit", "Lip" and "Rule Fillet". These new features are intended to provide the ability to easily create complex plastic-specific geometry needed to design molded, plastic parts.

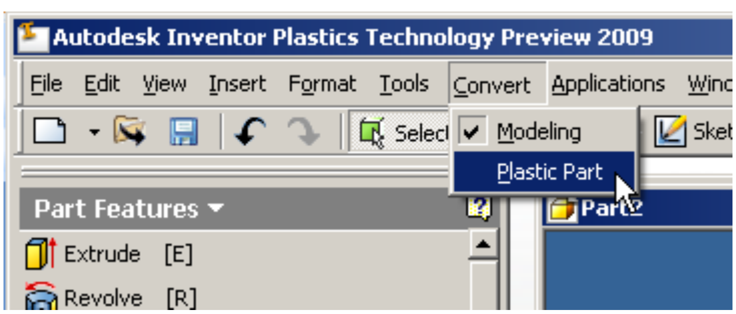
Finally, the Plastic Part environment provides additional Plastic Properties on the part. This property object stores settings and manages special plastic part parameters that are logically associated together. Certain commands in the Plastic Part environment will optionally use the Plastic Properties object as input or default values. Edits to the Plastic Property object will cause the part to re-compute, and use the updated values.

Example

1. To create a new plastic parts file from plastic template Go to the File**->New…** dialog select "Plastic Part.labipt file."



2. To convert an existing part to a plastic part to the Convert menu and select Plastic Part.

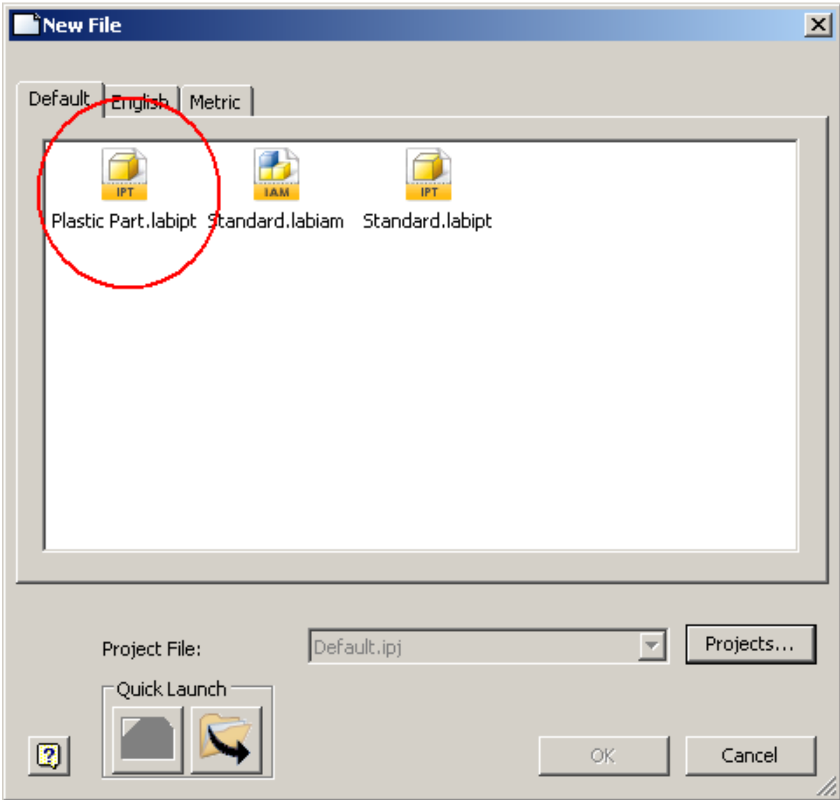


3. This process can be used to either convert a part designed in an earlier release of Inventor to a Plastic Part, or to convert a new part opened with the "Standard.labipt" template into a Plastic Part.

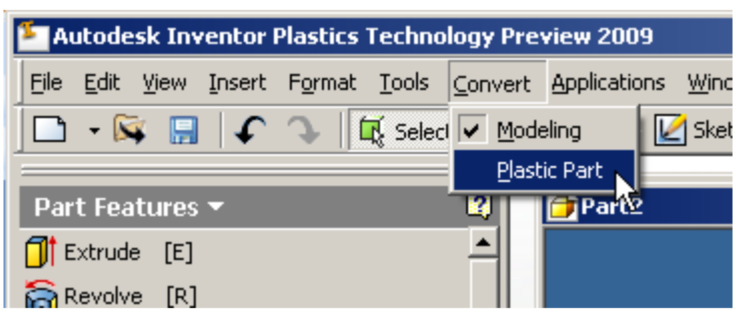
Reference

UI elements of the Plastic Part Environment

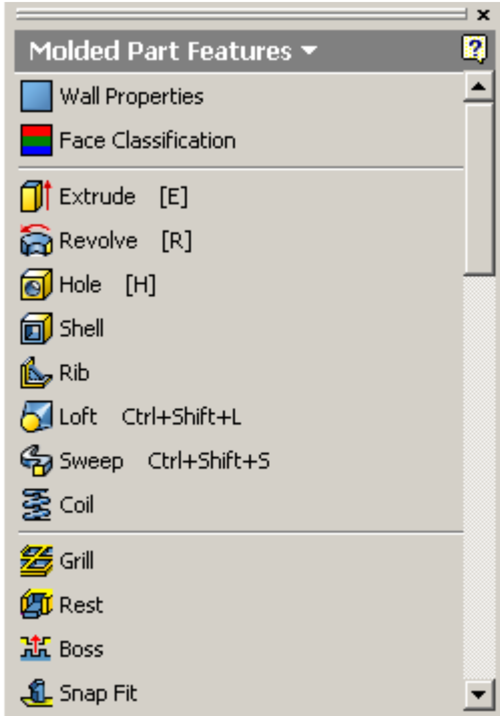
A “Plastic Part.ipt” part template show on the New File Dialog:



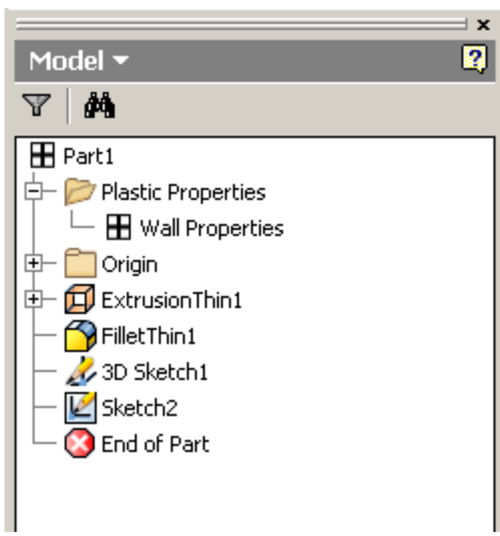
Plastic Part subtype is shown on the “Convert” menu:



Molded Part Features Panel Bar. This Panel Bar is a superset of the part modeling panel bar. All Part Modeling commands as well as new Plastic Part commands appear in the Molded Part Features Panel Bar.



Plastic Part Browser Panel:



The Inventor part browser contains a new Folder node: “Plastic Properties”, which contain the wall properties for the plastic part. Additionally, any “thin wall” features will be shown in the browser using a different icon than standard Inventor.

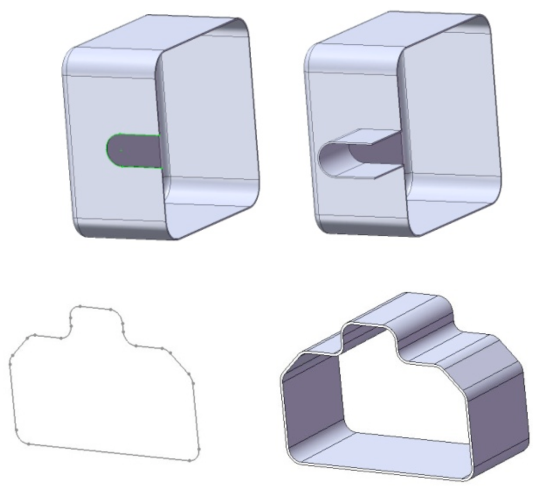
Working with sketched Thin features

Extrude Concept

Thin Extrude supports a “shell-as-you-go” approach to plastic part design. The “Thin” output option allows the designer to indicate whether that feature should affect the part in a thin-wall-aware manner. If “Thin” is selected, Extrude will automatically produce a feature that already understands the thin-walled nature of the part being designed, maintaining the correct part thickness. Using the Cut and Join options, you can control whether Extrude adds a shelled “pocket” or “boss” to the shelled area of the part, and free the user from having to manually create these geometries. Additional user options will allow the designer maximum flexibility to override wall thickness, and add open areas to the material added by Extrude.

In the Plastic Part Environment, Extrude now offers an additional output type to the existing output types (solid and surface). The new output type is “Thin Feature”. When this output type is selected, the result is an addition to the part that has a constant thickness.

Some examples:



Additional Inputs to Extrude in the Plastic Part Environment:

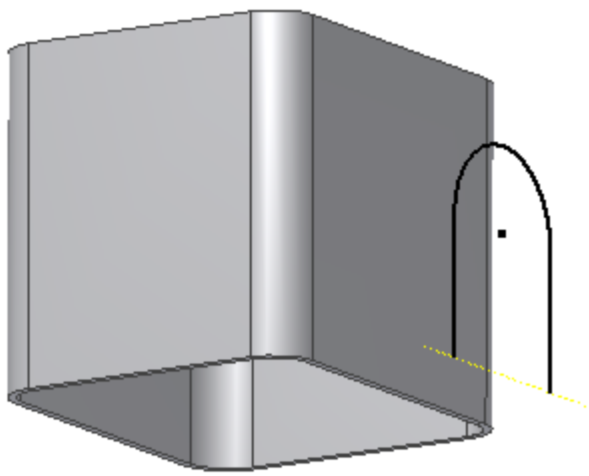
- Thickness. This is a real value which determines the thickness of the feature. If a Wall Property object is defined for the part, its “Wall Thickness” value is the default for this input.
- Offset Direction. This input allows the user to choose between methods for determining how the thickness is derived from the selected profile:
    - Inside – the profile curves determine the outer boundary of the Thin feature
    - Outside – the profile curves determine the inner boundary of the Thin feature

  - Both Ways – the Thin feature is offset both ways from the selected profile, so that the profile curves form the center of the Thin feature

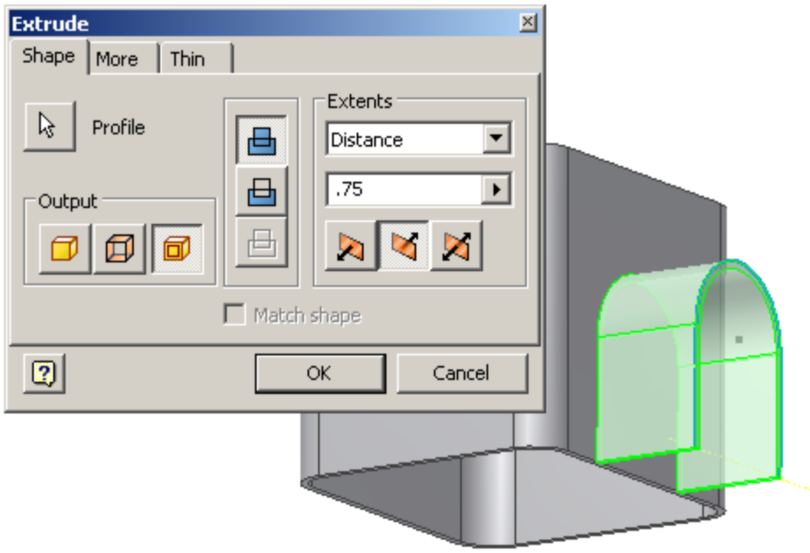
- Join to Shell. This input determines whether the area affected by the new feature is intended to be joined in with the main "shelled area" of the part. It is intended to facilitate the "shell-as-you-go" nature of thin-walled parts.
- End Capping. This input allows the user to create openings in the area of the plastic part affected by the feature, for creating pockets, etc. It allows, for those features with a clear starting and ending section, to have either or both be opened or closed, or to allow selection of profile curves to create "side openings" in the feature.

Extrude Example

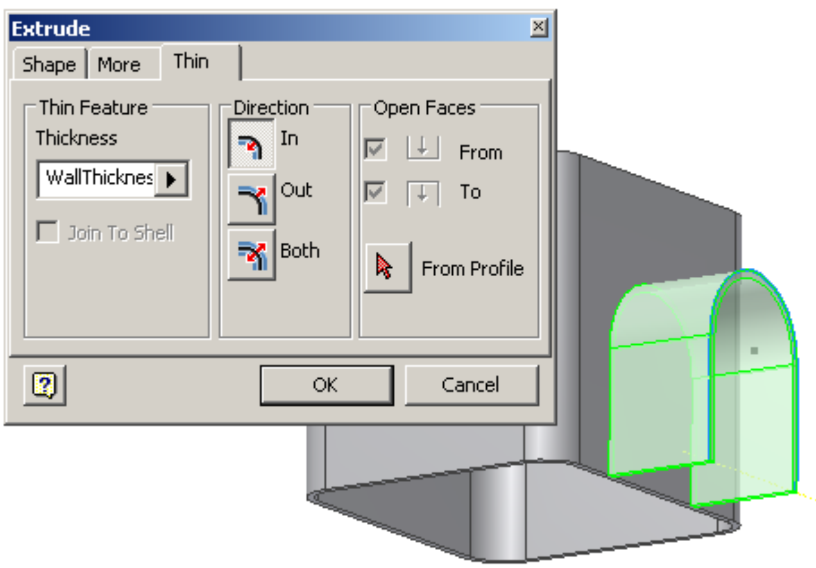
To begin lets create a Thin Wall Extrude with open profile. Start by creating a sketch with an open profile



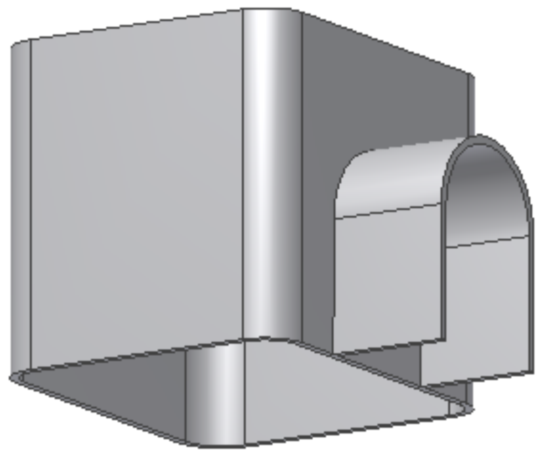
1. Click Extrude, select the profile (if your sketch contains multiple profiles), Inventor shows a preview with default options



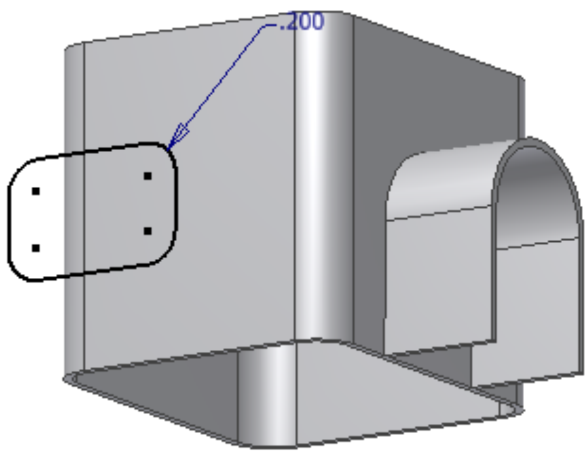
2. Choose standard Extrude options such as operation type (Join/Cut), Extent, etc. Change Thin parameters like distance, Thickness options like In-Out-Both.



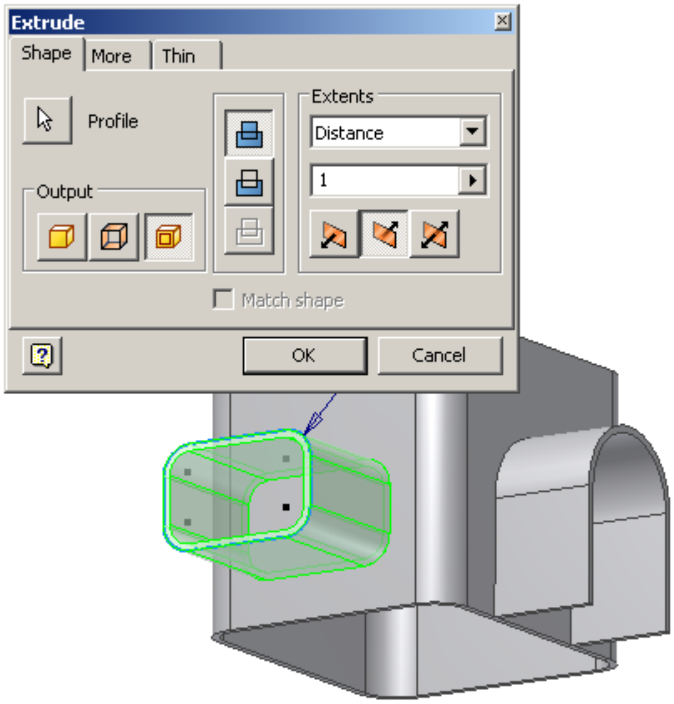
3. Capping and Join to Shell options are disabled for Open profile
4. Press “OK” to create the feature



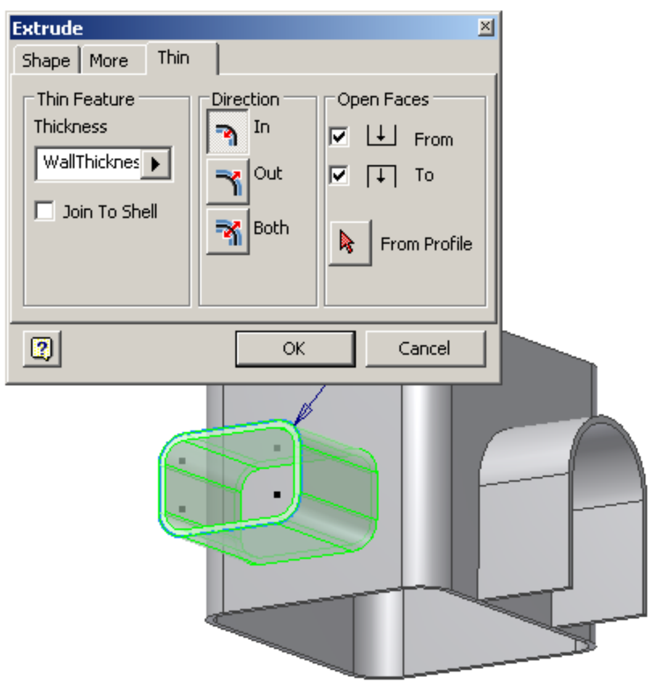
To create a Thin Wall Extrude with Closed profile, begin by creating a sketch with closed profile.



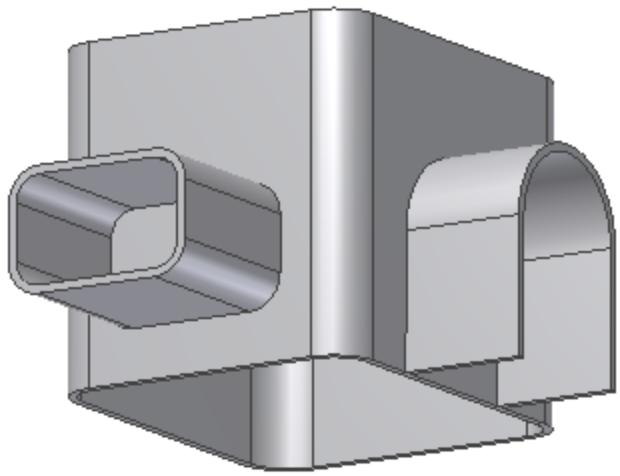
1. Click Extrude, select the profile, Inventor shows preview with default options.



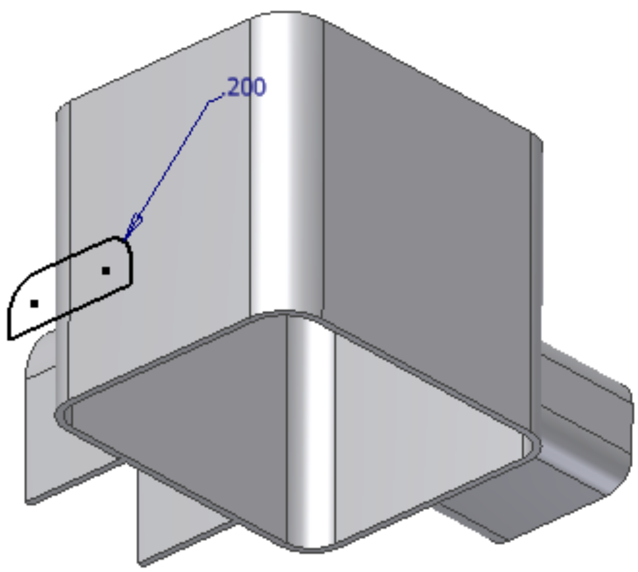
2. Change standard Extrude parameters such as distance, and operation type, and Thin options such as Thickness, Open Faces, and Direction. (Note that capping and Join to Shell options are enabled.



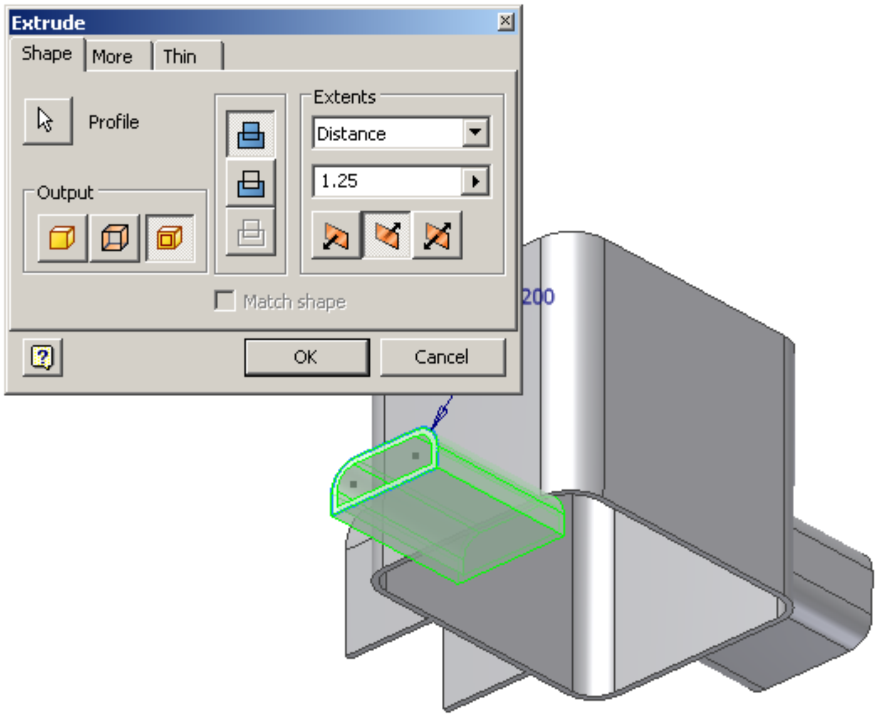
3. Press “OK” to create the feature.



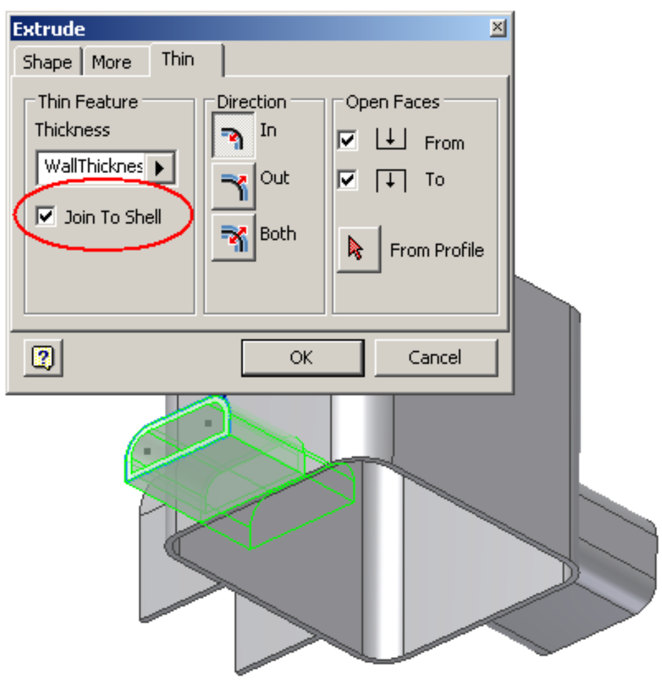
For a thin Wall “Join To Shell” example, start by creating a sketch containing the desired profile



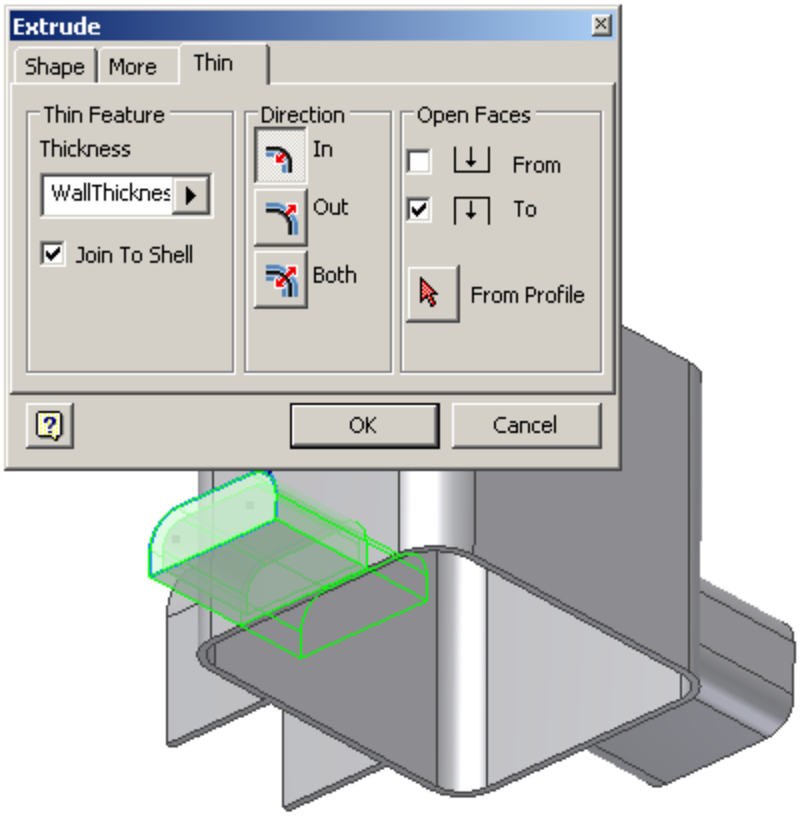
1. Invoke Extrude, and select the profile.



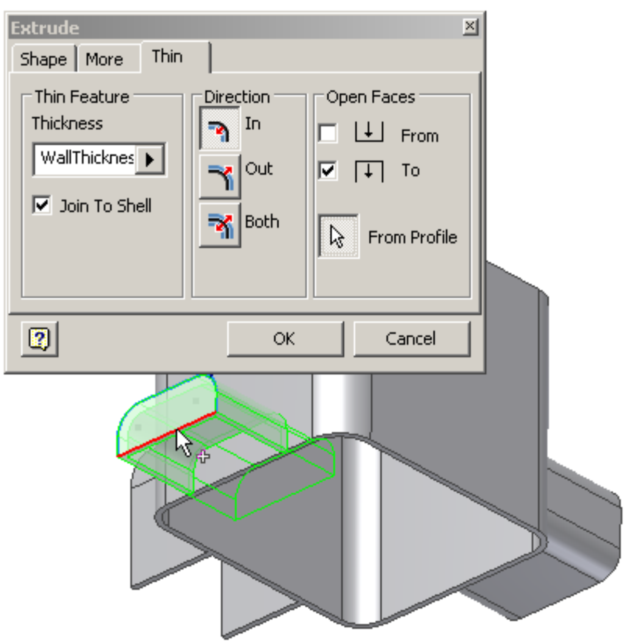
2. Select “Join To Shell” to indicate that the feature should add material to the shelled part



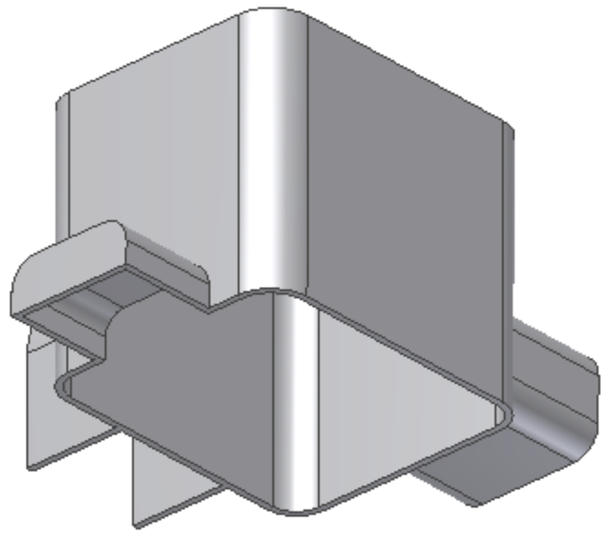
3. Uncheck “From” in “Open Faces” to close the start face



4. Select “From Profile” in “Open Faces”, and select the bottom linear edge of the profile, to open the bottom face of the feature.



5. Click **OK** to create the feature.

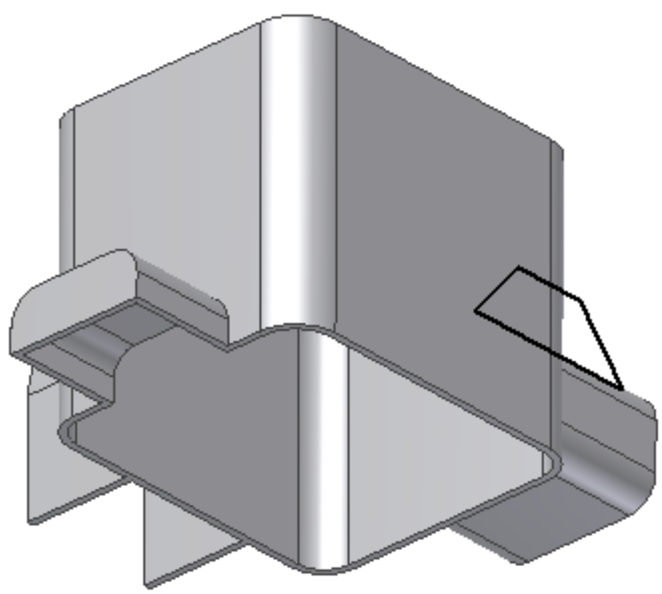


Note that even though the extent of the feature preview extends inside of the shelled area of the part, Join To Shell behavior understands the shelled nature of the part, and does not add material to that area of the part.

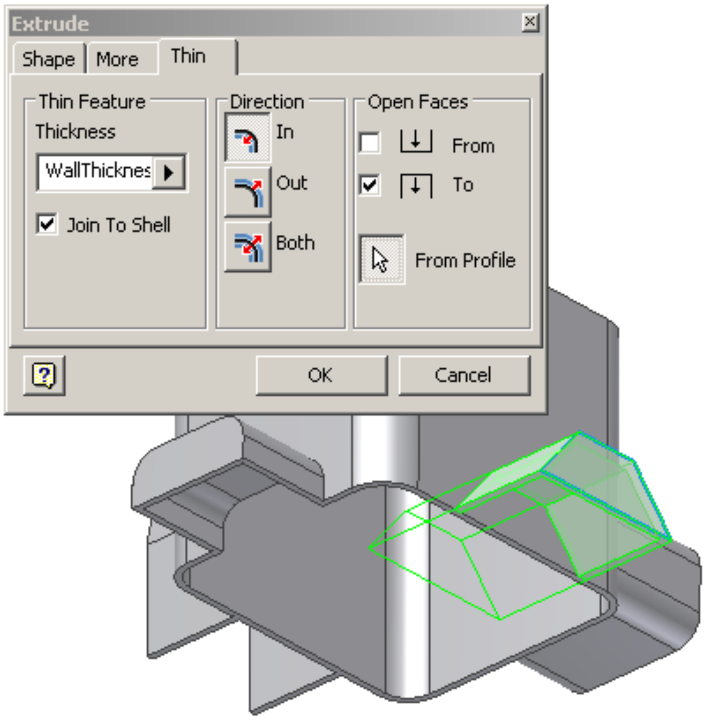
Join To Shell example that removes material from the part

This example is similar to the previous one, with the exception that the user chooses “Cut” to remove material from the plastic part. Again, Inventor, in the Plastic Part environment, understands that the part is a plastic part, so removing material creates a pocket.

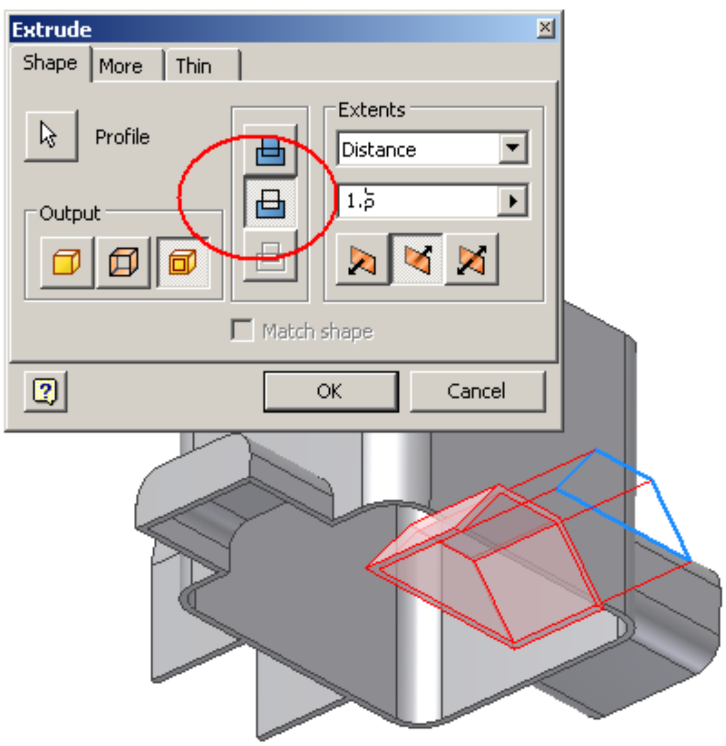
1. Create a sketch containing the desired profile.



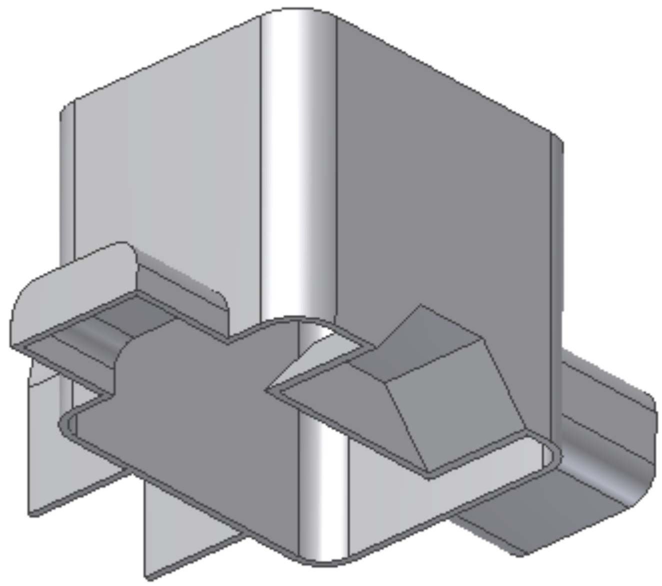
2. As above, invoke Extrude, select the profile, open faces, Join To Shell, etc



3. However, returning to the Shape tab, and selecting “Cut” as the operation, Inventor interprets the Cut operation as creating a pocket in the part.



4. Click **OK** to complete the feature

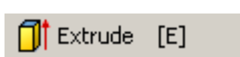


Extrude Reference

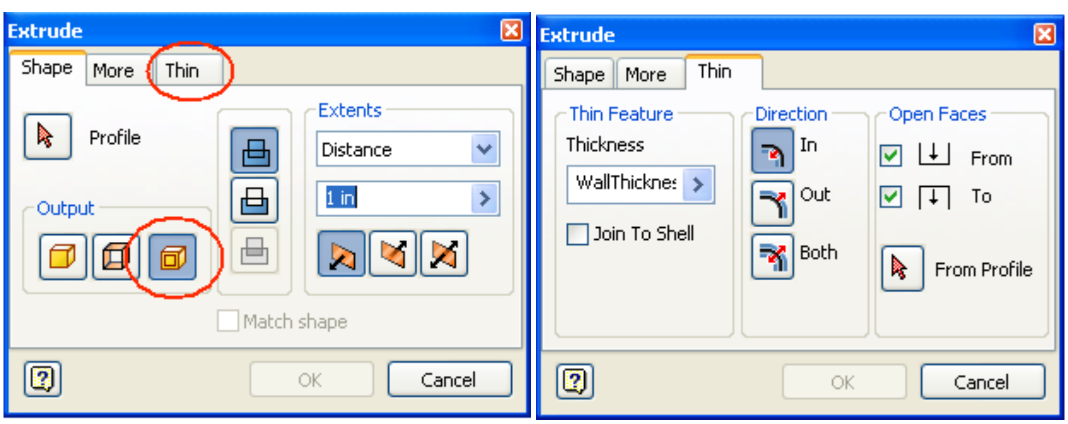
The Extrude feature will support Thin output type. When Thin output type is selected, additional controls, which exist on a new “Thin” tab of the dialog, become enabled.

Access:

Click on the “Extrude” of “Molded Part Features” panel.



Dialog:



Dialog Changes:

The main Extrude dialog supports a third output type, beyond the two supported today (Solid, Surface). The new type is called “Thin”. When you select this output type, a thin feature is created. When this output type is selected, additional controls, which exist on a new “Thin” tab of the dialog, become enabled. Note that if the other output types are selected (Solid, Surface), these controls will be grayed out.

Browser changes:

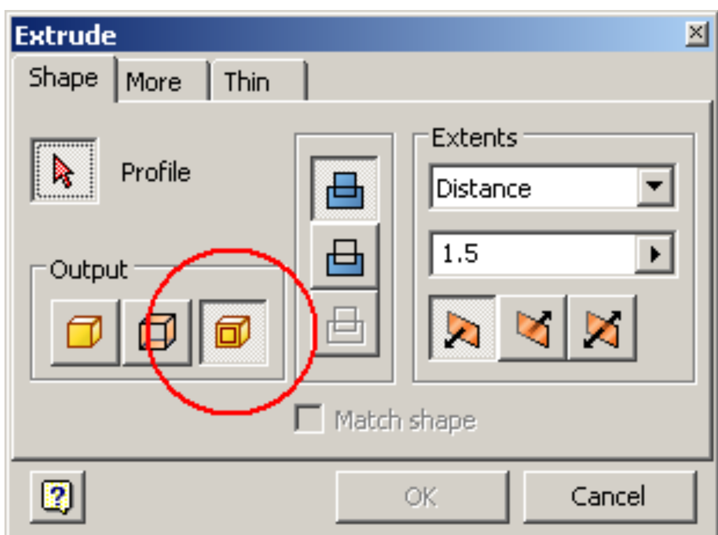
Just as Surface and Solid Extrude features have their own unique browser icon and name space, creating a Thin Extrude will add a new icon type to the browser, and will default to a naming convention that indicates that it is a Thin feature

Additional Extrude Dialog Control Descriptions:

This section describes each control that is new to Extrude in the Plastic Part environment

The Shape tab

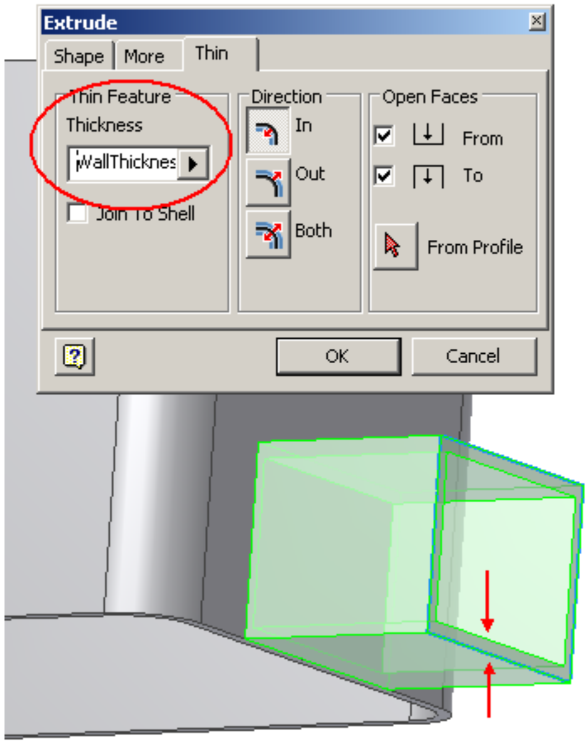
"Thin" output type



If this output option is selected, it enables creation of "thin extrude" features, which is the primary Extrude enhancement in the Plastic Environment. If this option is selected, options on the Thin tab become enabled.

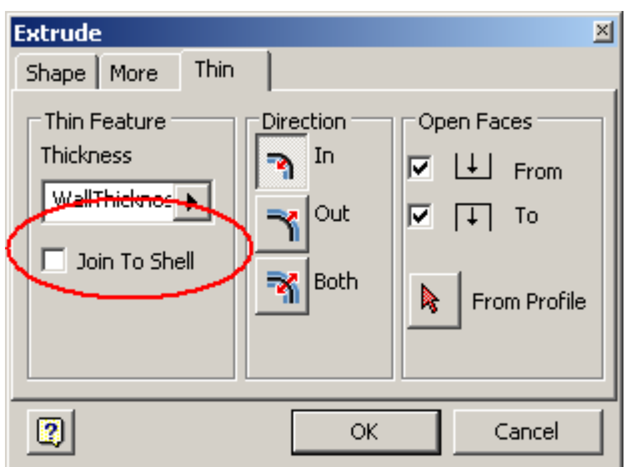
The Thin tab

"Thickness" value



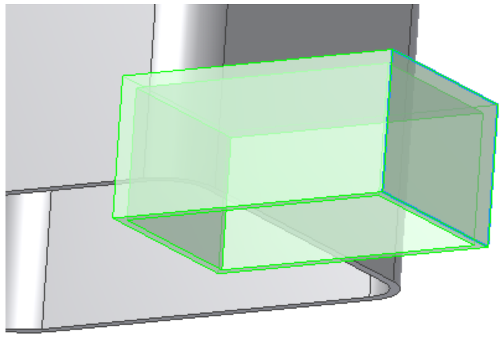
This value controls the thickness of the thin feature being created. The default value is always "WallThickness", which is a parameter that is set in the Wall Properties command for the part. It can be overridden for each feature, if desired.

"Join To Shell" checkbox

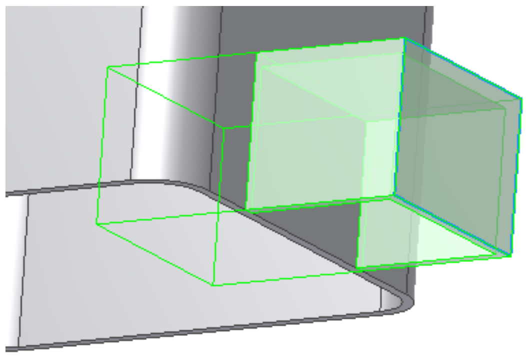


If checked, the resulting feature will honor the "inside" and "outside" of the plastic part. Join and Cut operations will obey Join to Shell differently. For example:

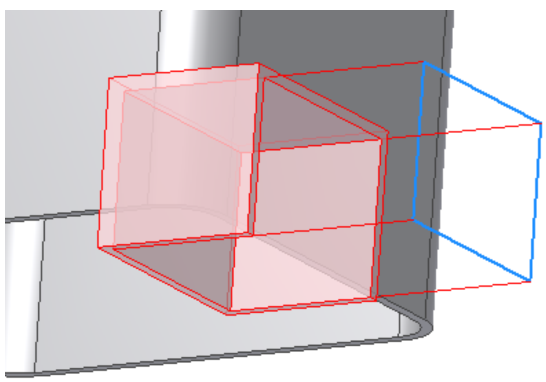
JoinToShell unchecked, with Join operation:



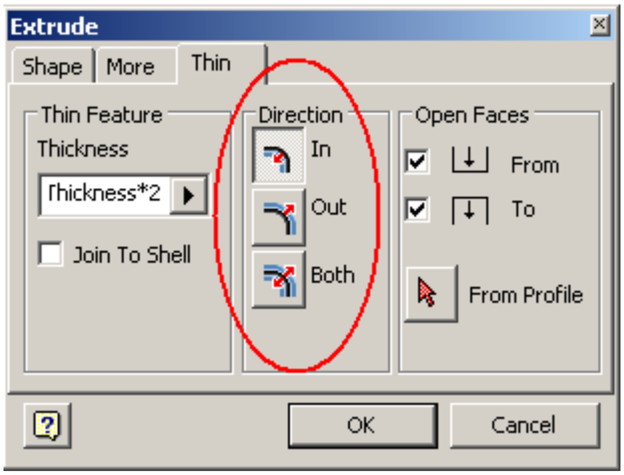
JoinToShell checked with Join operation:



JoinToShell checked with Cut Operation

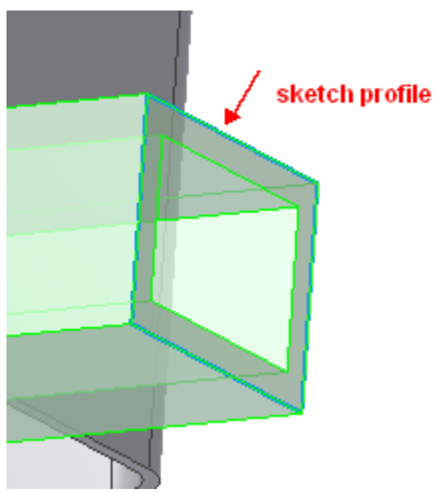


"Direction" choice

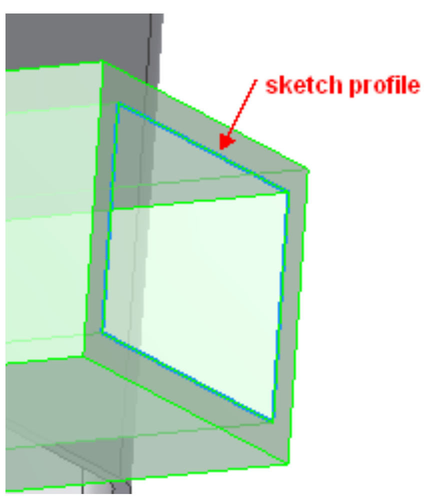


This control adjusts the direction in which the material is offset from the selected profile to achieve the material thickness. Below are some illustrations of the effect of this control:

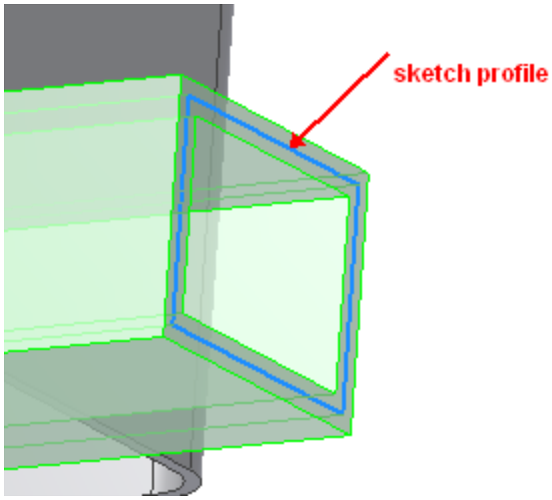
Direction set to "In"



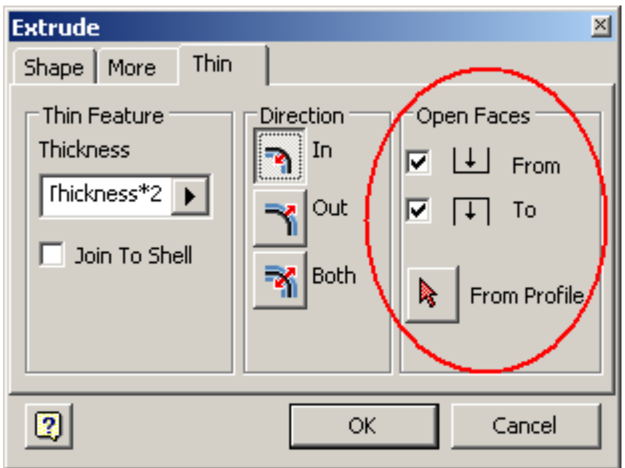
Direction set to "Out"



Direction set to "Both"

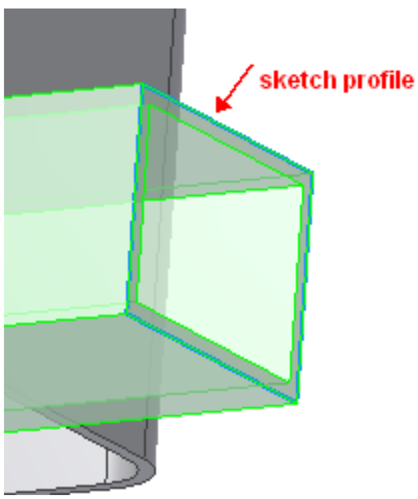


"Open Faces" control

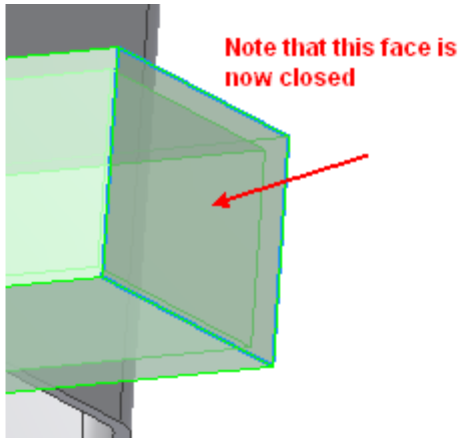


This allows you to control whether end and side faces are open or closed.

The "From" checkbox allows you to open or close the face that corresponds with the sketch face. For example, with "From" checked, the result is



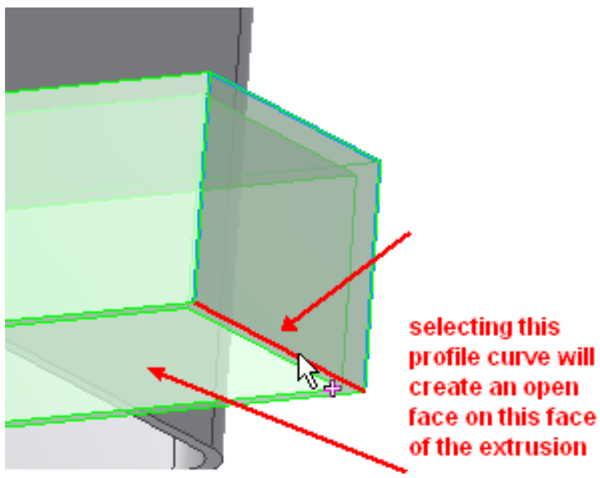
And with "From" unchecked:



The “To” checkbox operates similarly, but on the other end of the extrusion.

The “From Profile” allow you to similarly control open/closed faces for the side faces of the extrusion, by selecting the corresponding sketch profile curves.

For example, if this selection is chosen, the result is:



Revolve Concept

Thin Revolve supports a “Shell-as-you-go” approach to plastic part design. The “Thin” output option allows the designer to indicate whether that feature should affect the part in a thin-wall-aware manner. If “Thin” is selected, Revolve will automatically produce a feature that already understands the thin-walled nature of the part being designed, maintaining the correct part thickness. Using the Cut and Join options, you can control whether Revolve adds a shelled “Pocket” or “Boss” to the shelled area of the part, and free the user from having to manually create these geometries. Additional user options will allow the designer maximum flexibility to override wall thickness, and add open areas to the material added by Revolve.

In the Plastic Part Environment, Revolve now offers an additional output type to the existing output types (solid and surface). The new output type is “Thin Feature”. When this output type is selected, the result is an addition to the part that has a constant thickness.

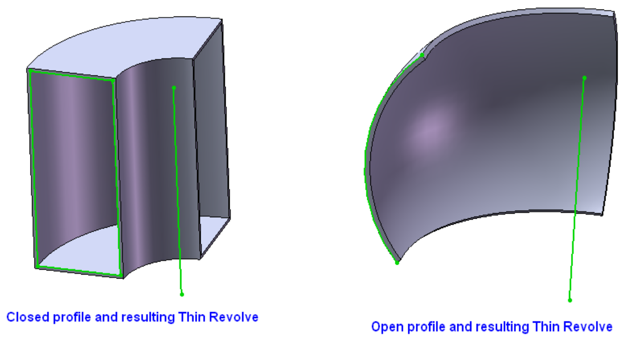
Additional Inputs to Revolve in the Plastic Part Environment:

- Thickness. This is a real value which determines the thickness of the feature. If a Wall Property object is defined for the part, its "Wall Thickness" value is the default for this input.
- Offset Direction. This input allows the user to choose between methods for determining how the thickness is derived from the selected profile:
    - Inside – the profile curves determine the outer boundary of the Thin feature
    - Outside – the profile curves determine the inner boundary of the Thin feature
    - Both Ways – the Thin feature is offset both ways from the selected profile, so that the profile curves form the center of the Thin feature
- Join to Shell. This input determines whether the area affected by the new feature is intended to be joined in with the main "shelled area" of the part. It is intended to facilitate the "shell-as-you-go" nature of thin-walled parts.
- End Capping. This input allows the user to create openings in the area of the plastic part affected by the feature, for creating pockets, etc. It allows, for those features with a clear starting and ending section, to have either or both be opened or closed, or to allow selection of profile curves to create "side openings" in the feature.

Additional Inputs:

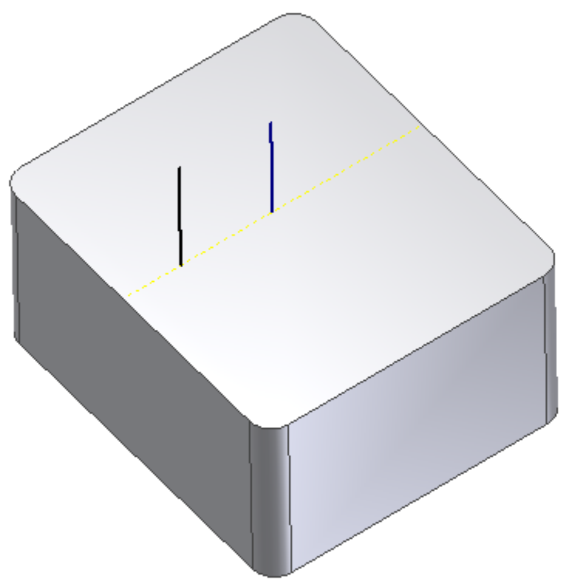
- Thickness. This is a real value which determines the thickness of the feature.
- Offset Direction. This input allows the user to choose between methods for determining how the thickness is derived from the selected profile:
    - Inside – the profile curves determine the outer boundary of the Thin feature
    - Outside – the profile curves determine the inner boundary of the Thin feature
    - Both Ways – the Thin feature is offset both ways from the selected profile, so that the profile curves form the center of the Thin feature
- Join to Shell. This input determines whether the area affected by the new feature is intended to be joined in with the main "shelled area" of the part. It is intended to facilitate the "shell-as-you-go" nature of thin-walled parts.

- End Capping. This input allows the user to create openings in the area of the plastic part affected by the feature, for creating pockets, etc. It allows, for those features with a clear starting and ending section, to have either or both be opened or closed.

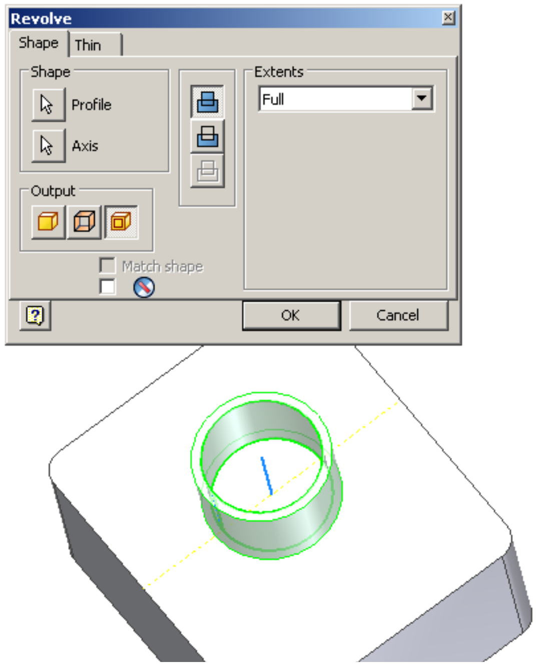


Revolve Example

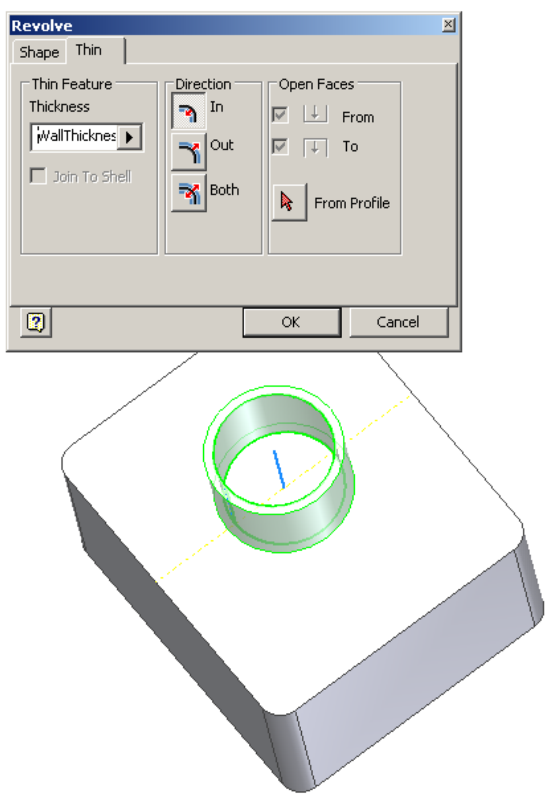
To create a Thin Wall Revolve with open profile, create a sketch containing an open profile.



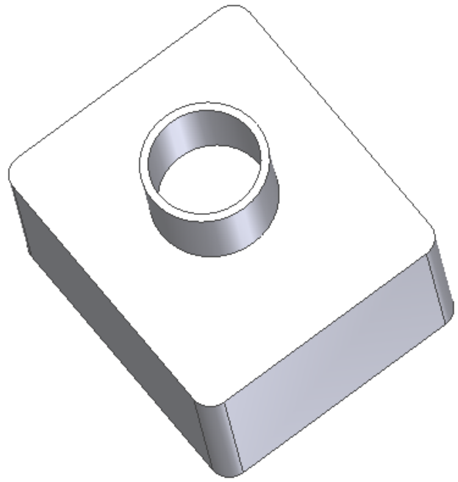
1. Click Revolve, select the profile (if your sketch contains multiple profiles), Inventor shows a preview with default options.



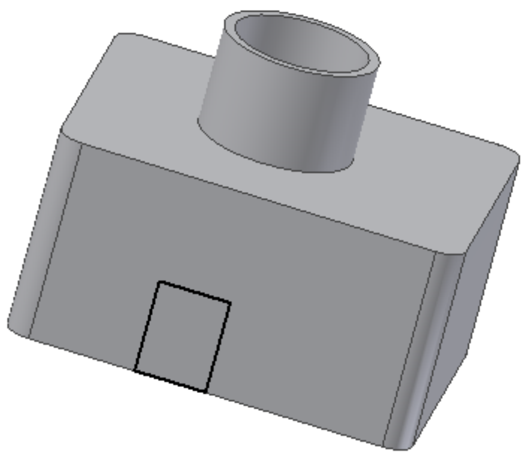
2. Choose standard Revolve options such as operation type (Join/Cut), Extent, etc. Change Thin parameters like distance, Thickness options like In-Out-Both.



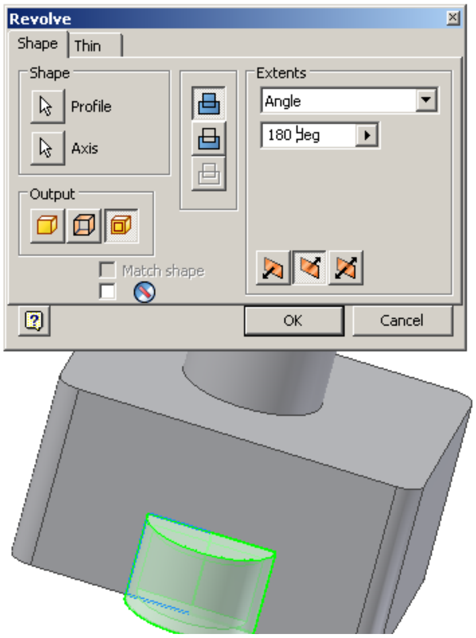
3. Capping and Join to Shell options are disabled for Open profile.
4. Press “OK” to create the feature.



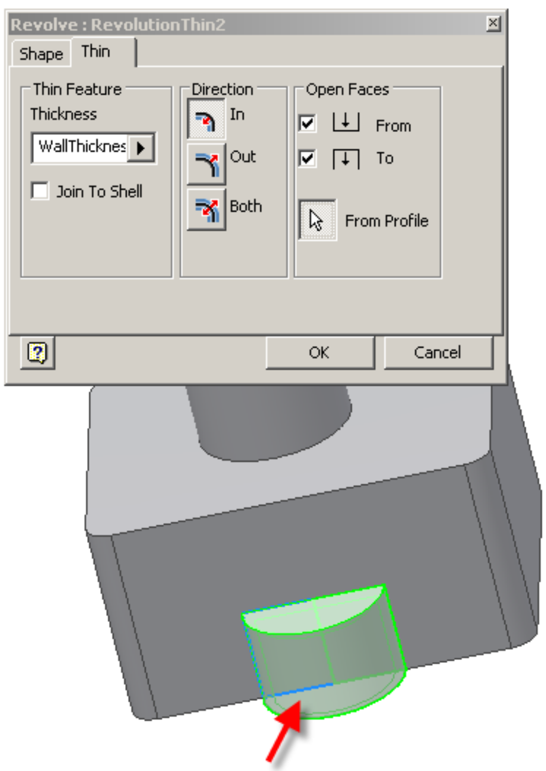
Next we will look at how to create a Thin Wall Revolve with Closed profile.
Start by creating a sketch having closed profile.



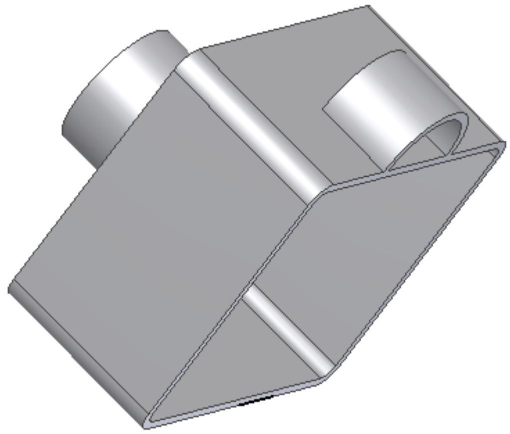
1. Click Revolve, select the profile, Inventor shows preview with default options.



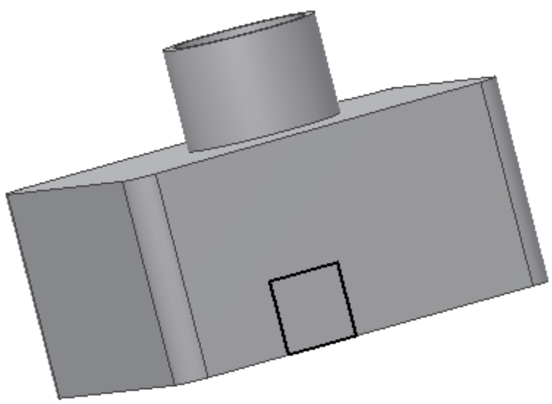
2. Change standard Revolve parameters such as distance, and operation type, and Thin options such as Thickness, Open Faces, and Direction. (Note: that capping and Join to Shell options are enabled. In this example we selected the bottommost line of the profile to remain open.)



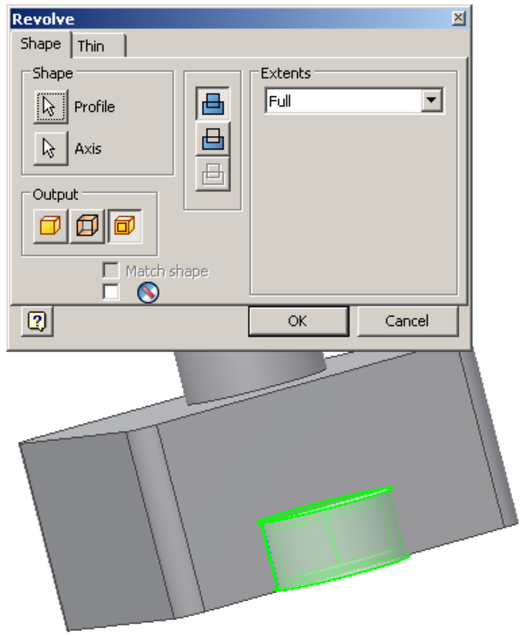
3. Press “OK” to create the feature.



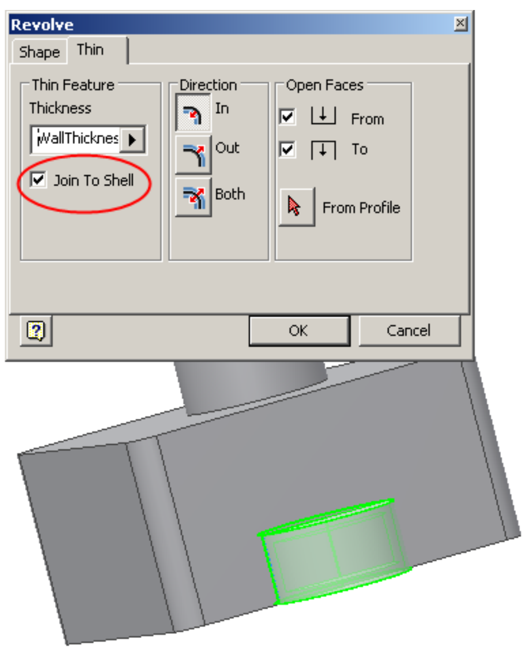
Thin Wall “Join To Shell” example to create a sketch containing the desired profile.



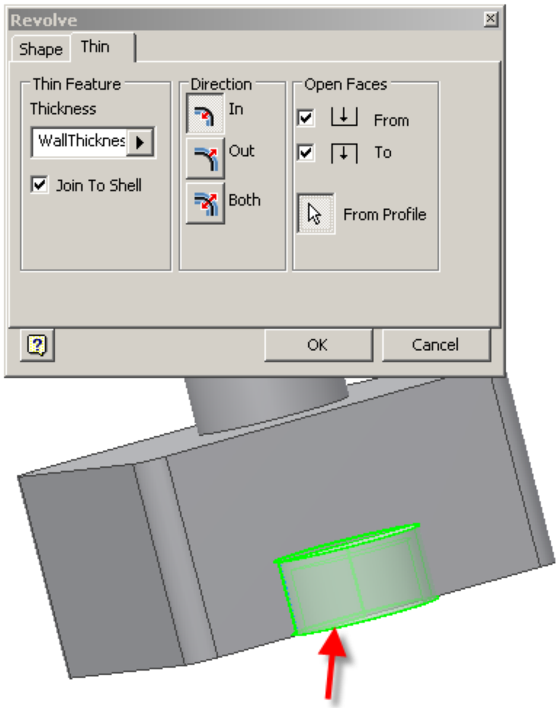
1. Invoke Revolve, and select the profile and axis.



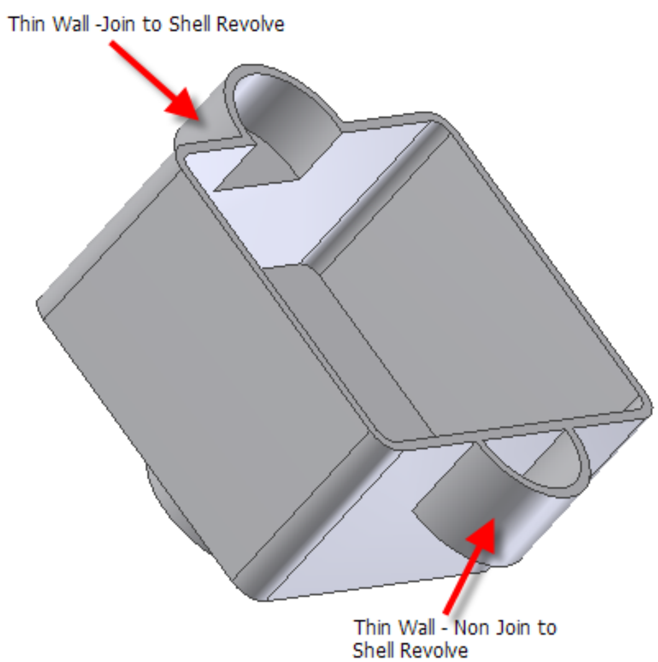
2. Select “Join To Shell” to indicate that the feature should add material to the shelled part.



3. Select “From Profile” in “Open Faces”, and select the bottom linear edge of the profile, to open the bottom face of the feature.



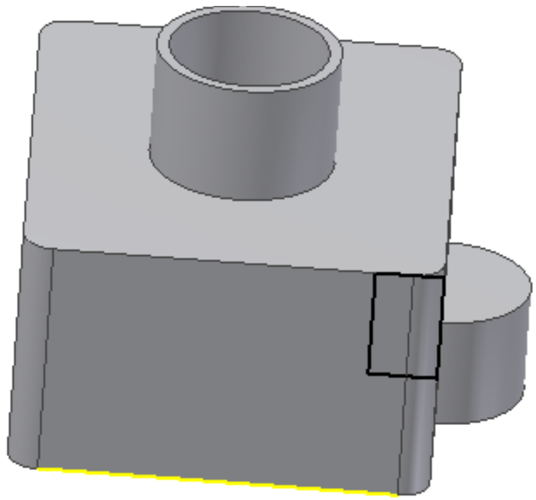
4. Click “OK” to create the feature.

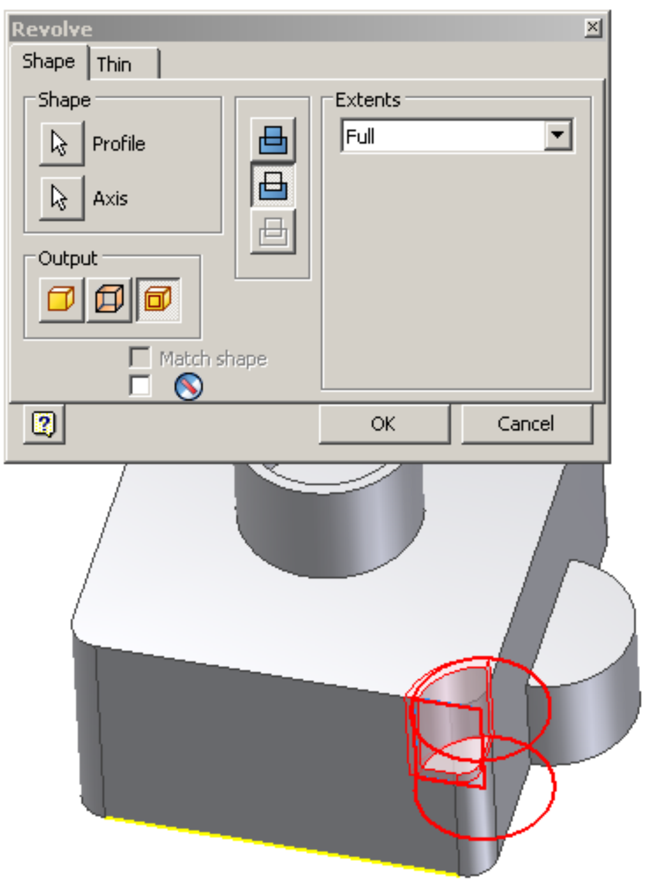


5. Note that even though the extent of the feature preview extends inside of the shelled area of the part, Join To Shell behavior understands the shelled nature of the part, and does not add material to that area of the part.

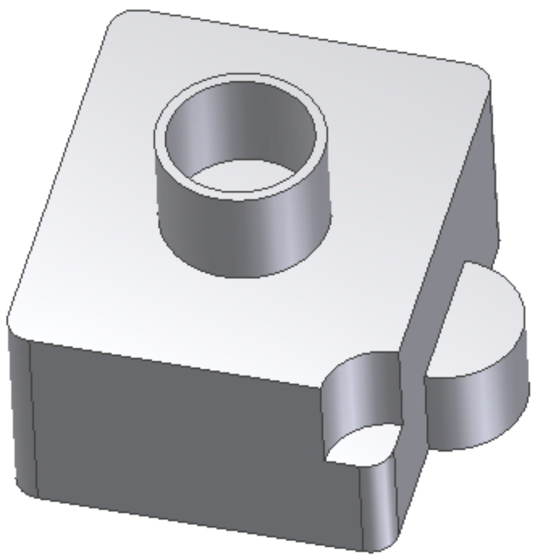
Join to Shell example that removes material from the part

1. This example is similar to the previous one, with the exception that the user chooses “Cut” to remove material from the plastic part. Again, Inventor, in the Plastic Part environment, understands that the part is a plastic part, so removing material creates a pocket.
2. Create a sketch containing the desired profile.





3. Select “Cut” as the operation, Inventor interprets the Cut operation as creating a pocket in the part.
4. As above, invoke Revolve; select the profile, open faces, Join To Shell, etc.
5. Click **OK** to complete the feature



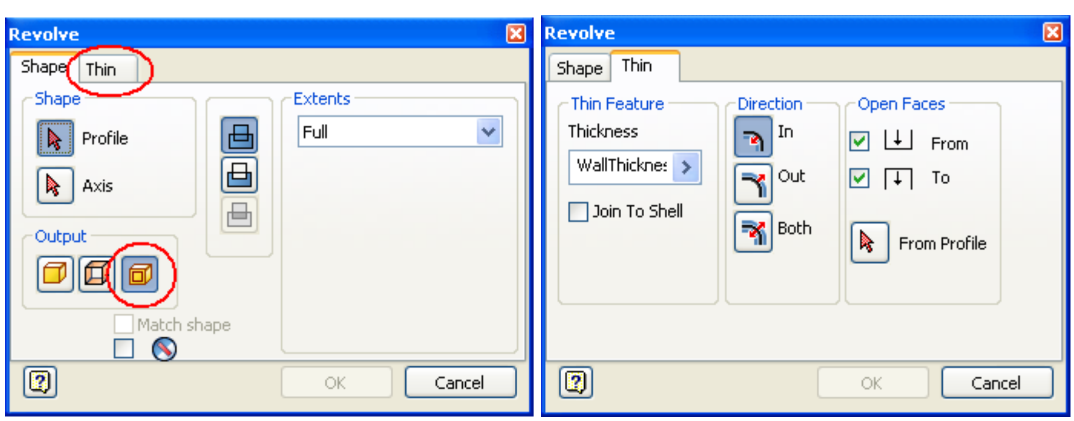
Revolve Reference

The Revolve feature will support Thin output type. When Thin output type is selected, additional controls, which exist on a new “Thin” tab of the dialog, become enabled.

Access:

Click on the “Revolve” of “Molded Part Features” panel.

Dialog:



Dialog Changes

The main Revolve dialog supports a third output type, beyond the two supported today (Solid, Surface). The new type is called "Thin". When you select this output type, a Thin feature is created. When this output type is selected, additional controls, which exist on a new "Thin" tab of the dialog, become enabled. Note that if the other output types are selected (Solid, Surface), these controls will be grayed out.

Browser changes

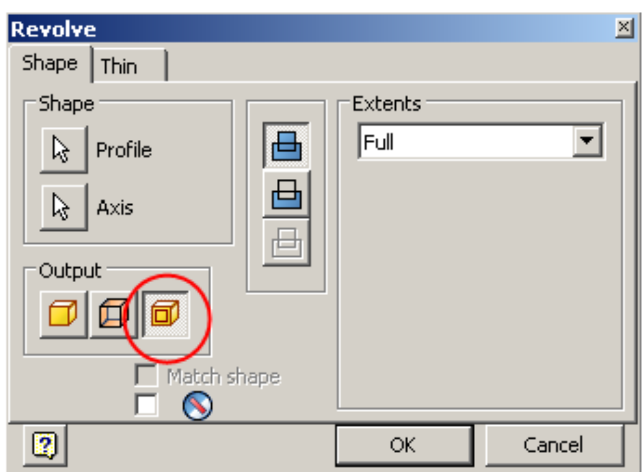
Just as Surface and Solid Revolve features have their own unique browser icon and name space, creating a Thin Revolve will add a new icon type to the browser, and will default to a naming convention that indicates that it is a Thin feature.

Additional Revolve Dialog Control Descriptions

This section describes each control that is new to Revolve in the Plastic Part environment

The Shape tab

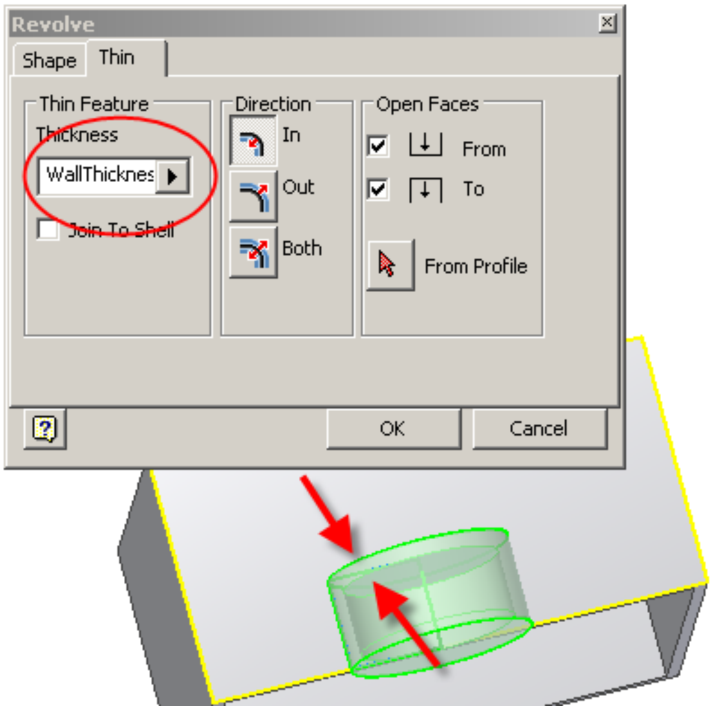
"Thin" output type



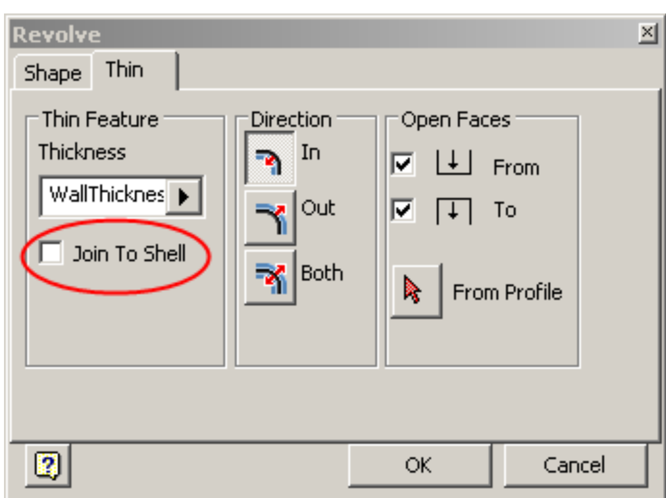
If this output option is selected, it enables creation of “thin revolve” features, which is the primary Revolve enhancement in the Plastic Environment. If this option is selected, options on the thin tab become enabled.

The Thin Tab

“Thickness” value

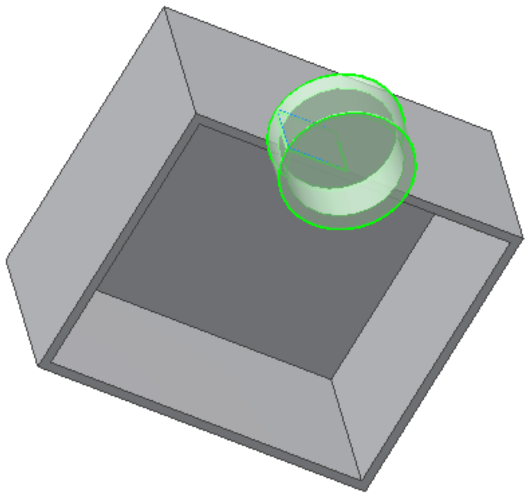


6. This value controls the thickness of the thin feature being created. The default value is always “WallThickness”, which is a parameter that is set in the Wall Properties command for the part. It can be overridden for each feature, if desired.
7. “Join To Shell” checkbox

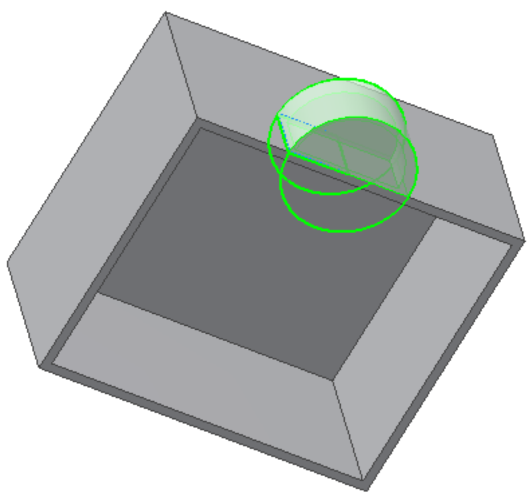


8. If checked, the resulting feature will honor the “inside” and “outside” of the plastic part. Join and Cut operations will obey Join To Shell differently. For example:

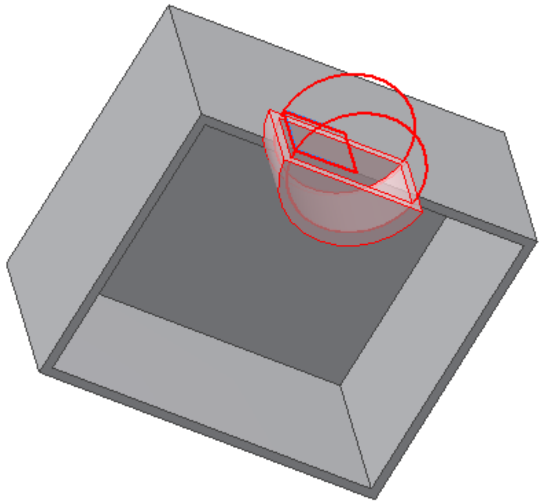
JoinToShell unchecked, with Join operation:



JoinToShell checked with Join operation:



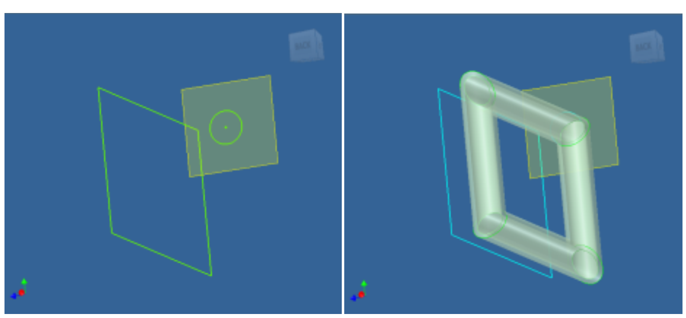
JoinToShell checked with Cut Operation



Sweep Concept

Thin Sweep supports a “shell-as-you-go” approach to plastic part design. Join option allows the designer to indicate whether that feature should add thin-walled material to the part. In this case Sweep will automatically produce a feature that already understands the thin-walled nature of the part being designed, maintaining the correct part thickness. Optionally, using Cut option, Sweep can also attach to the shelled area of the part, and free the user from having to manually create these openings. Additional user options will allow the designer maximum flexibility to override wall thickness, and add open areas to the material added by Sweep.

Sweep now offers an additional output type to the existing output types (solid and surface). The new output type is “Thin Feature”. When this output type is selected, the result is an addition to the part that has a constant thickness. Some examples:



Additional Inputs:

- Thickness. This is a real value which determines the thickness of the feature.
- Offset Direction. This input allows the user to choose between methods for determining how the thickness is derived from the selected profile:

  - Inside – the profile curves determine the outer boundary of the Thin feature
  - Outside – the profile curves determine the inner boundary of the Thin feature
  - Both Ways – the Thin feature is offset both ways from the selected profile, so that the profile curves form the center of the Thin feature

- Join to Shell. This input determines whether the area affected by the new feature is intended to be joined in with the main "shelled area" of the part. It is intended to facilitate the "shell-as-you-go" nature of thin-walled parts.
- End Capping. This input allows the user to create openings in the area of the plastic part affected by the feature, for creating pockets, etc. It allows, for those features with a clear starting and ending section, to have either or both be opened or closed.

Sweep Example

Build Base Thin Wall Sweep with open profile

1. Create a sketch having open profile and path curve
2. Click Sweep, it shows preview with default options
3. Change parameters like Thickness options like In-Out-Both
4. Capping and Join to Shell options are disabled for Open profile
5. If preview looks fine, then press "OK"

Build Base Thin Wall Sweep with Closed profile

6. Create a sketch having closed profile and path curve
7. Click Sweep, it shows preview with default options
8. Change parameters like Thickness options like In-Out-Both, Capping
9. Join to Shell options are disabled for first feature.
10. Use Open Profile to open up solids at picked locations
11. If preview looks fine, then press "OK"

Build non-Base Thin Wall Sweep with open profile

12. Create a sketch having open profile and path curve
13. Click Sweep, it shows preview with default options
14. Change parameters like Thickness options like In-Out-Both
15. Capping and Join to Shell options are disabled for Open profile
16. Use Boolean options like Join-Cut
17. If preview looks fine, then press "OK"

Build non-Base Thin Wall Sweep with Closed profile

18. Create a sketch having closed profile and path curve

19. Click Sweep, it shows preview with default options
20. Change parameters like Thickness options like In-Out-Both
21. Capping and Join to Shell options are enabled
22. If preview looks fine, then press “OK”

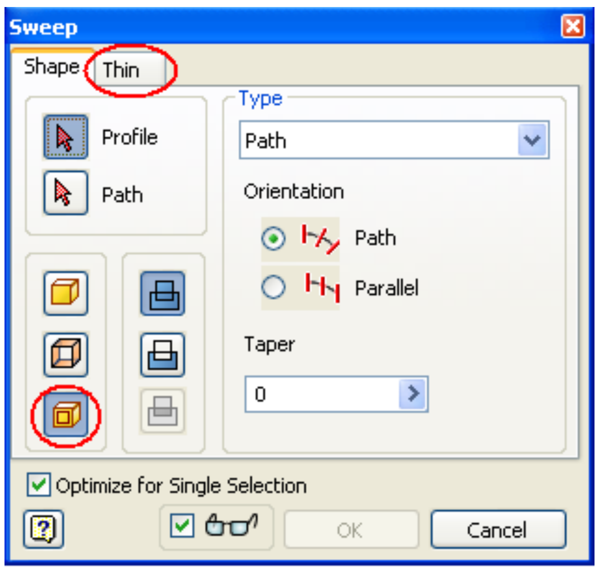
Sweep Reference

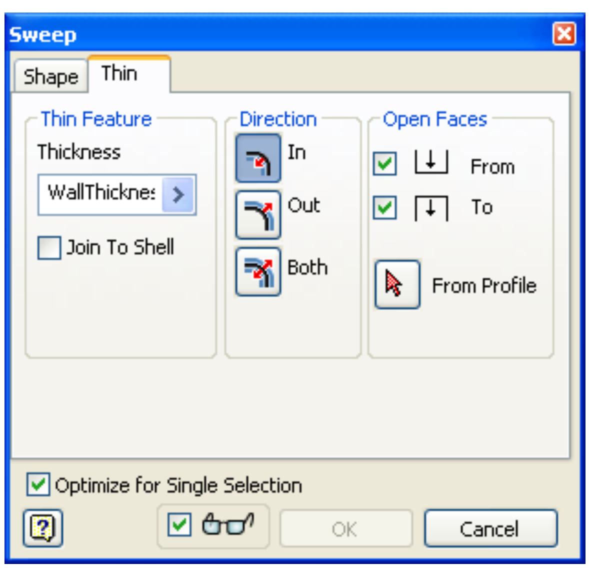
The Sweep feature will support Thin output type. When Thin output type is selected, additional controls, which exist on a new “Thin” tab of the dialog, become enabled.

Access:

1. Click on the “Sweep” of “Molded Part Features” panel.

Dialog:



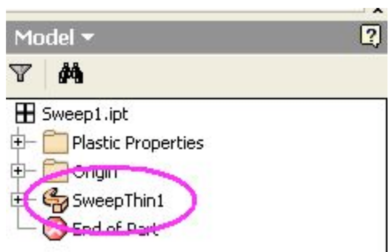


Dialog Changes:

The main Sweep dialog supports a third output type, beyond the two supported today (Solid, Surface). The new type is called “Thin”. When the user selects this output type, a Thin feature is created. When this output type is selected, additional controls, which exist on a new “Thin” tab of the dialog, become enabled. Note that if the other output types are selected (Solid, Surface), these controls will be grayed out.

Browser changes:

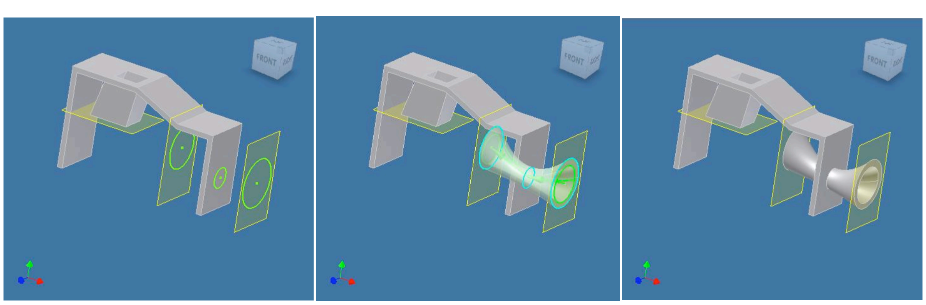
Just as Surface and Solid Sweep features have their own unique browser icon and name space, creating a Thin Sweep will add a new icon type to the browser, and will default to a naming convention that indicates that it is a Thin feature



Loft Concept

Thin Loft supports a “shell-as-you-go” approach to plastic part design. Join option allows the designer to indicate whether that feature should add thin-walled material to the part. In this case Loft will automatically produce a feature that already understands the thin-walled nature of the part being designed, maintaining the correct part thickness. Optionally, using Cut option, Loft can also attach to the shelled area of the part, and free the user from having to manually create these openings. Additional user options will allow the designer maximum flexibility to override wall thickness, and add open areas to the material added by Loft.

Loft now offers an additional output type to the existing output types (solid and surface). The new output type is “Thin Feature”. When this output type is selected, the result is an addition to the part that has a constant thickness. Some examples:



Additional Inputs:

- Thickness. This is a real value which determines the thickness of the feature.
- Offset Direction. This input allows the user to choose between methods for determining how the thickness is derived from the selected profile:
  - Inside – the profile curves determine the outer boundary of the Thin feature
  - Outside – the profile curves determine the inner boundary of the Thin feature
  - Both Ways – the Thin feature is offset both ways from the selected profile, so that the profile curves form the center of the Thin feature
- Join to Shell. This input determines whether the area affected by the new feature is intended to be joined in with the main "shelled area" of the part. It is intended to facilitate the "shell-as-you-go" nature of thin-walled parts.
- End Capping. This input allows the user to create openings in the area of the plastic part affected by the feature, for creating pockets, etc. It allows, for those features with a clear starting and ending section, to have either or both opened or closed.

Loft Example

Build Base Thin Wall Loft with open profile

1. Create multiple sketches having all open profiles
2. Click Loft, it shows preview with default options
3. Change parameters like Thickness options like In-Out-Both
4. Capping and Join to Shell options are disabled for Open profile
5. If preview looks fine, then press "OK"

Build Base Thin Wall Loft with Closed profile

1. Create multiple sketches having all closed profiles
2. Click Loft, it shows preview with default options
3. Change parameters like Thickness options like In-Out-Both, Capping
4. Join to Shell options are disabled for first feature.
5. Use Open Profile to open up solids at picked locations
6. If preview looks fine, then press "OK"

Build non-Base Thin Wall Loft with open profile

1. Create multiple sketches having all open profiles
2. Click Loft, it shows preview with default options
3. Change parameters like Thickness options like In-Out-Both
4. Capping and Join to Shell options are disabled for Open profile

5. Use Boolean options like Join-Cut
6. If preview looks fine, then press “OK”

Build non-Base Thin Wall Loft with Closed profile

1. Create multiple sketches having all closed profiles
2. Click Loft, it shows preview with default options
3. Change parameters like Thickness options like In-Out-Both
4. Capping and Join to Shell options are enabled
5. If preview looks fine, then press “OK”

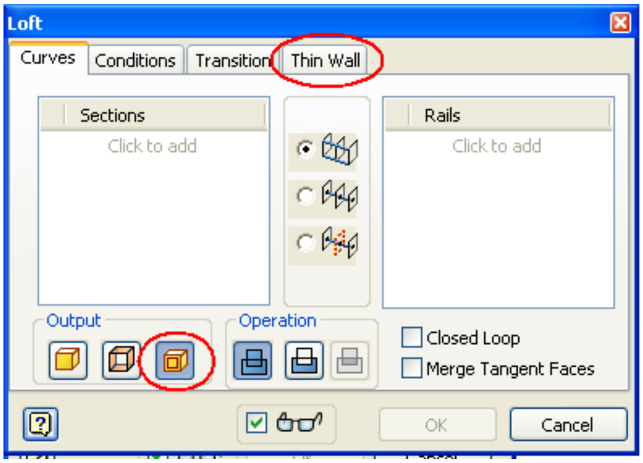
Loft Reference

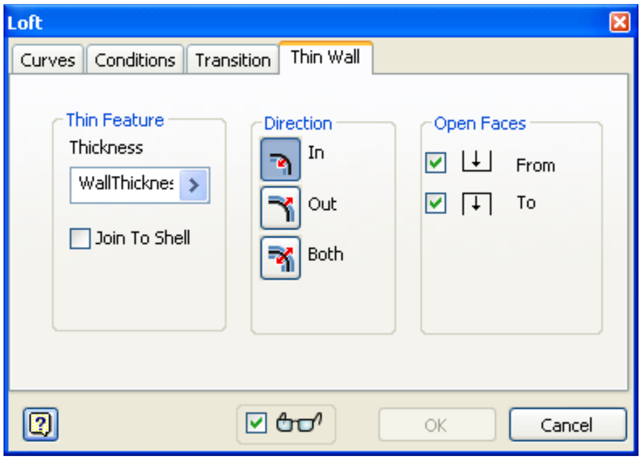
The Loft feature will support Thin output type. When Thin output type is selected, additional controls, which exist on a new “Thin” tab of the dialog, become enabled.

Access:

Click on the “Loft” of “Molded Part Features” panel.

Dialog:



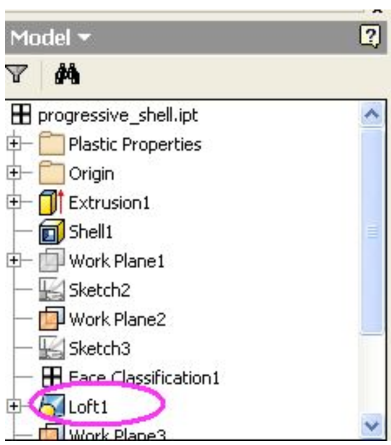


Dialog Changes:

The main Loft dialog supports a third output type, beyond the two supported today (Solid, Surface). The new type is called “Thin”. When the user selects this output type, a Thin feature is created. When this output type is selected, additional controls, which exist on a new “Thin” tab of the dialog, become enabled. Note that if the other output types are selected (Solid, Surface), these controls will be grayed out.

Browser changes:

Just as Surface and Solid Loft features have their own unique browser icon and name space, creating a Thin Loft will add a new icon type to the browser, and will default to a naming convention that indicates that it is a Thin feature



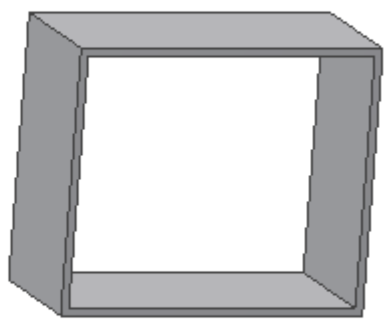
Working with placed Thin features

Fillet Concept

In Thin-wall parts, in order to maintain uniform wall thickness, when an edge is filleted, its corresponding opposite inner/outer edge needs to be filleted. Instead of having to fillet the edges separately with different radius value, the thin-wall option in Fillet allows automatic detection and filleting of corresponding opposite edges. The corresponding opposite edges will be filleted with a radius that will ensure uniform wall thickness. If the edge selected is convex the radius applied to the corresponding opposite edge will be the specified radius minus the wall thickness. If the edge is concave the radius applied to the corresponding opposite edge plus the wall thickness.

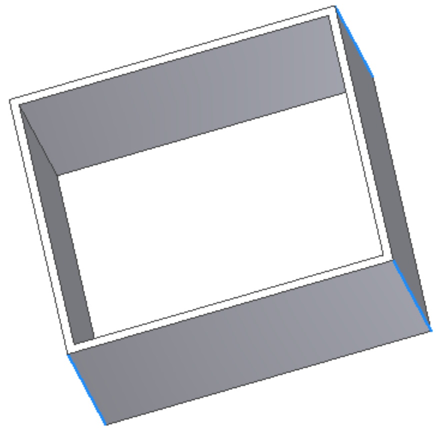
Fillet Example

1. Create or import a Thin-walled solid in Plastics environment

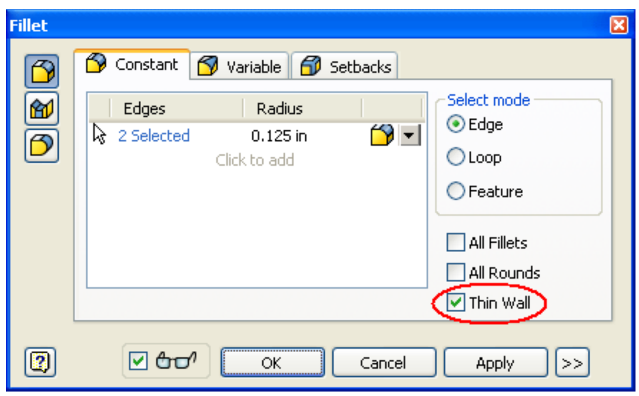


2. Invoke the Fillet command

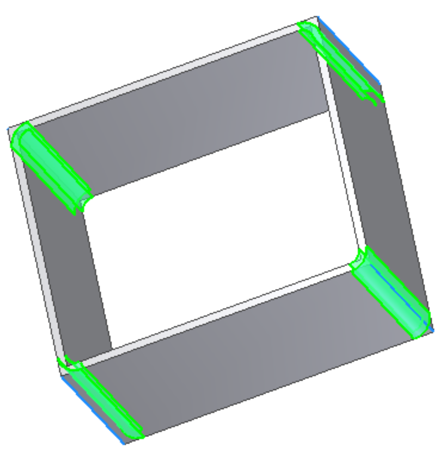
3. Select edge/edges to be filleted



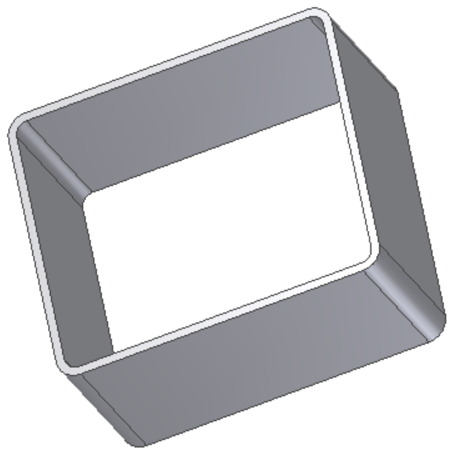
4. By default, Thin-wall option is checked in the dialog



5. Preview shows the original and computed opposite edges



6. Click OK to perform the operation



Fillet Reference

The Fillet feature supports Thin-wall output type

Thin-wall is only applicable to ‘Constant’ type of Fillets, and not on ‘Variable’ and ‘Setback’ types

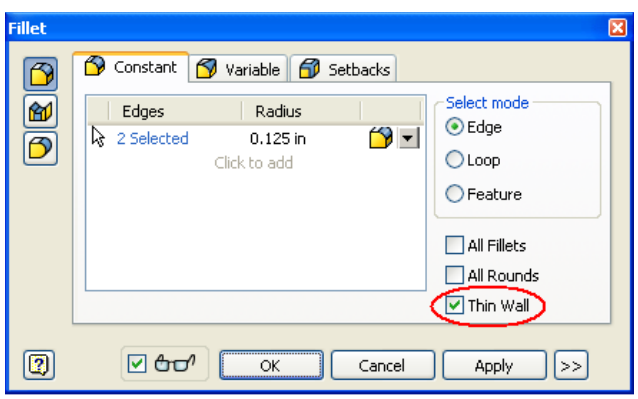
Split Edge type is supported i.e. selected edge has 2 or more corresponding opposite edges. Refer to figure 9 and 10.

Access:

Click on the “Fillet” of “Molded Part Features” panel.

Dialog Changes

The primary UI change to Fillet is in the dialog:



If the ‘Thin Wall’ checkbox is clicked, the resulting fillet will be thin-walled i.e. the corresponding opposite edges will also be filleted.

Wall Thickness Selection

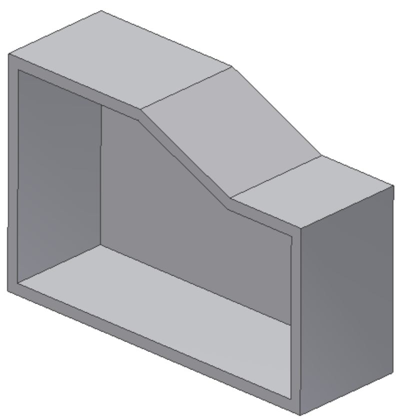
If the actual wall thickness is much greater (more than twice) than Wall Properties Dialog Thickness OR if it is less than the Wall Properties Dialog Thickness, we ignore it and set the Wall Properties Thickness as the actual thickness value.

Chamfer Concept

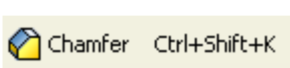
In Thin-wall parts, in order to maintain uniform wall thickness, when an edge is chamfered, its corresponding opposite inner/outer edge needs to be chamfered. Instead of the user chamfering the edges separately with different values, the thin-wall option in Chamfer allows automatic detection and chamfering of corresponding opposite edges. The corresponding opposite edges will be chamfered with a chamfer distance that will ensure uniform wall thickness.   If the edge selected is convex the chamfer distance applied to the corresponding opposite edge will be the specified chamfer distance minus the wall thickness. If the edge is concave the chamfer distance applied to the corresponding opposite edge plus the wall thickness.

Chamfer Example

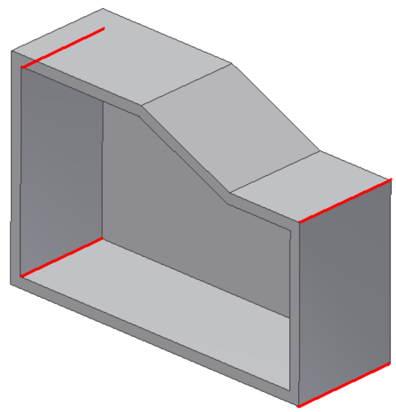
Create or import a Thin-walled solid in Plastics environment



1. Invoke the Chamfer command

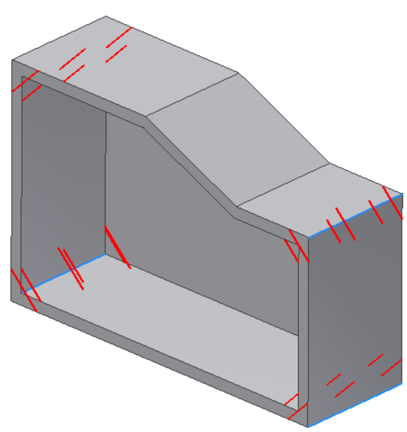


2. Select edge/edges to be chamfered.

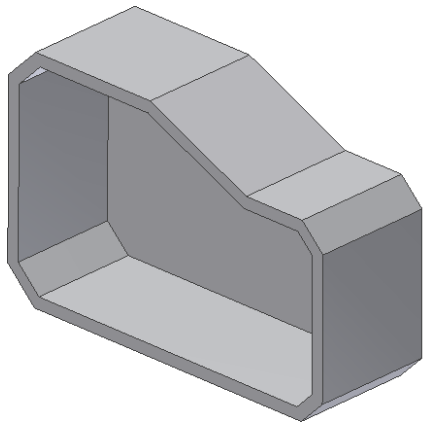


3. By default, Thin-wall option is checked in the dialog

4. Preview shows the original and computed opposite edges



5. Click OK to perform the operation



Chamfer Reference

6. The Chamfer feature supports Thin-wall output type
7. Thin-wall is applicable to all 3 types of chamfers i.e. ‘Distance’, ‘Distance and Angle’ and ‘2 Distances’
8. Split Edge type is supported i.e. selected edge has 2 or more corresponding opposite edges. Refer to figure

Access:

Click on the “Chamfer” of “Molded Part Features” panel.

Dialog Changes

The primary UI change to chamfer is in the dialog:

If the ‘Thin Wall’ checkbox is clicked, the resulting chamfer will be thin-walled i.e. the corresponding opposite edges will also be chamfered.

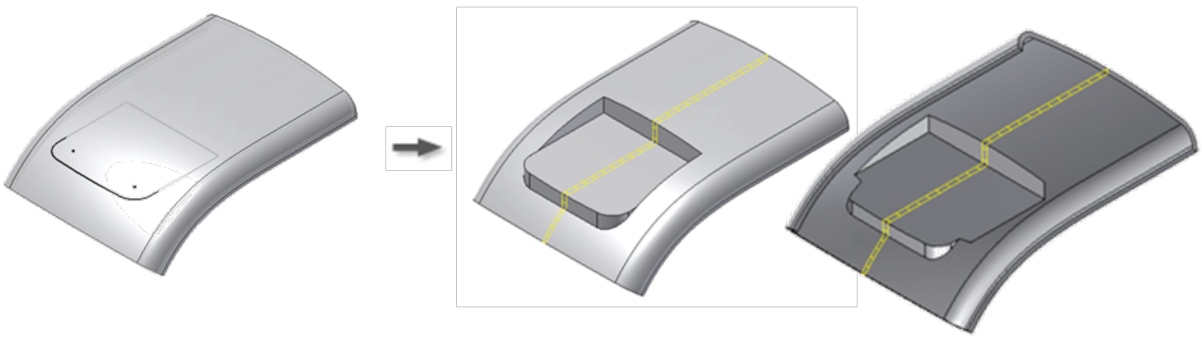
Wall Thickness Selection

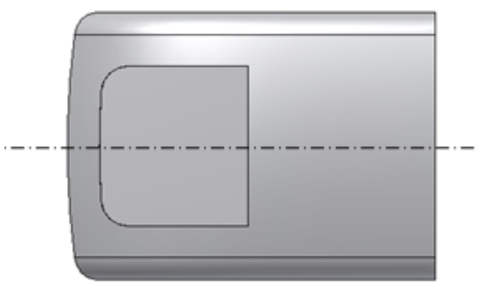
If the actual wall thickness is much greater (more than twice) than Wall Properties Dialog Thickness OR if it is less than the Wall Properties Dialog Thickness, we ignore it and set the Wall Properties Thickness as the actual thickness value.

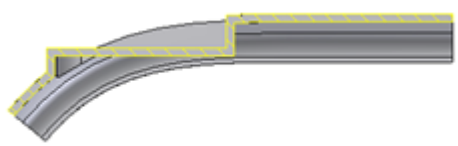
Working with Rest Features

Concept

A Rest is an element of a plastic part that is applied to a slanted or curved wall of a plastic shell to form an area that partly protrudes inside the body and partly outside, this area, that we call landing area, can be used as a placement for another part or provide a surface that has an orientation different than the overall shape.

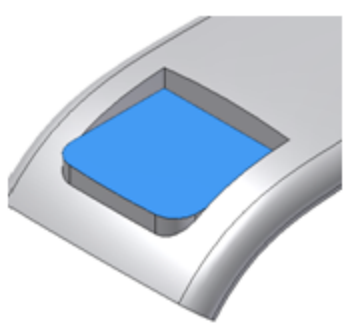






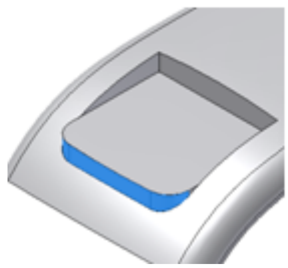
The anatomy of the Rest feature includes:

Landing area

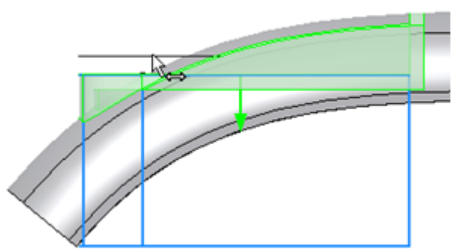


Clearance wall

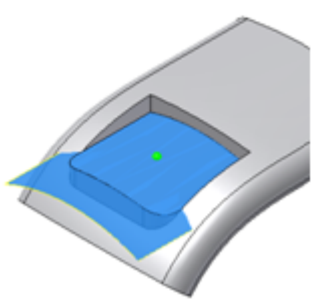
Platform wall



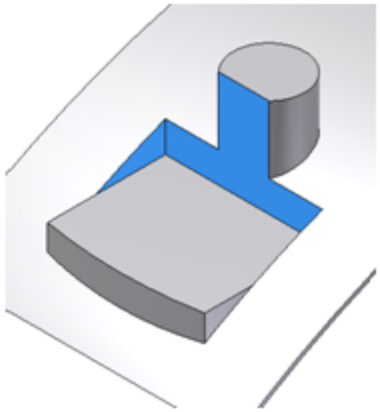
The rest feature is based on a sketch that defines the **landing area** boundary. You can drag the position of the landing at offsets of the sketch plane.



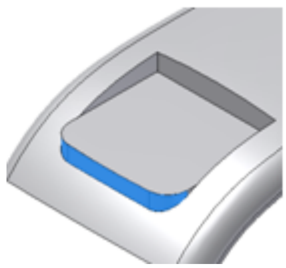
The landing area can also coincide with a surface (“To Surface” option on the More tab).



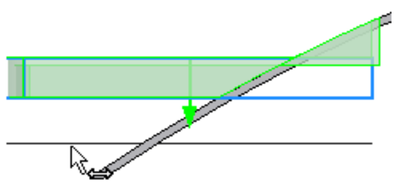
The **clearance** walls extend above the landing area cutting material across the entire target body.



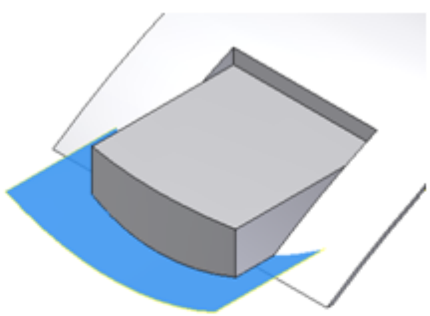
The extent of the **platform** wall is by default “Through all" which means that it is extended to the next face of the target body.



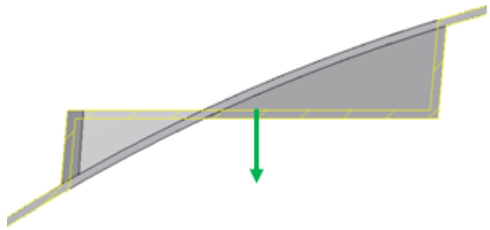
You can drag the platform wall at a distance from the sketch plane.



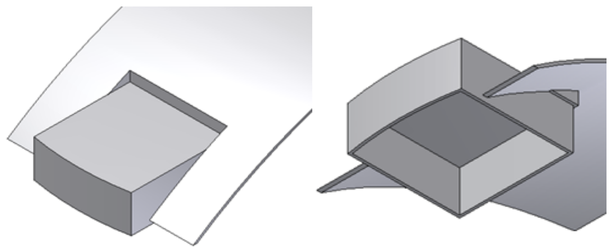
You can also extend the platform wall to a surface.



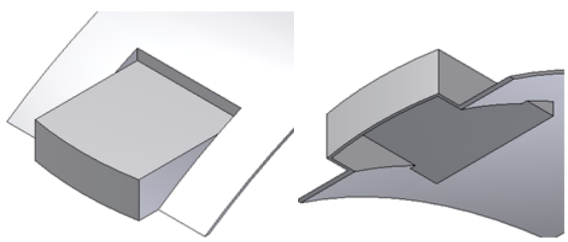
It is possible to specify a draft angle for both the platform and landing walls.



Tip: If the Rest boundary projection along the sketch normal does not stay entirely within the target body, the feature ends up extending the platform walls also inside the body.

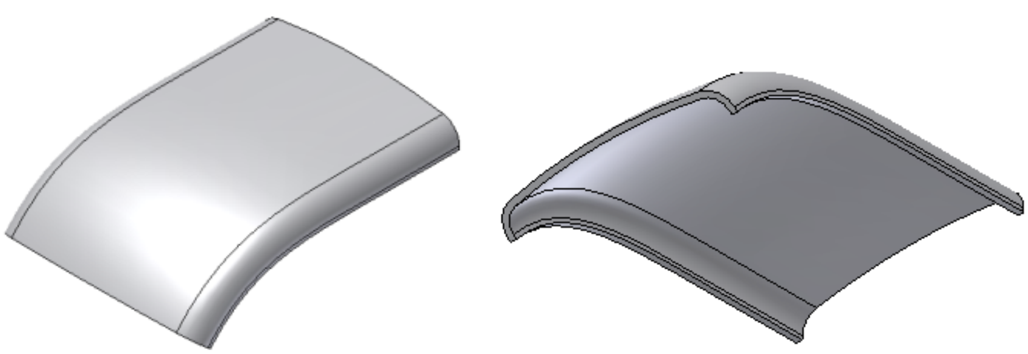


To correct this geometry, use the “To Surface” option on the Shape tab or change the extent parameter for the platform wall to be flush or higher than the body boundary.



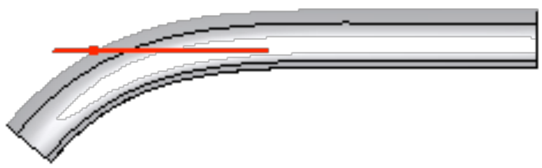
Example

1. Build a Rest feature with a flat landing.



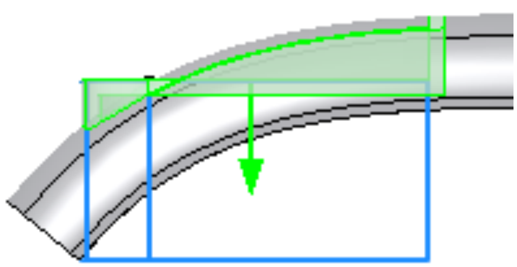
2. Create or import a thin-walled part.

3. Build a 2D sketch that is on either sides of the thin-walled part

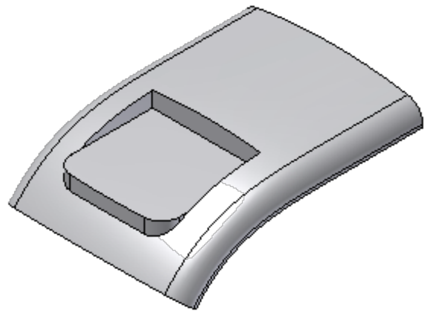


4. Click the Rest tool of the Plastic Features panel.

5. Make sure that the box in the Shape tab is set to **Through All**. The platform direction arrow is similar to what is shown in the image (for example, opposite to the appropriate clearance volume).

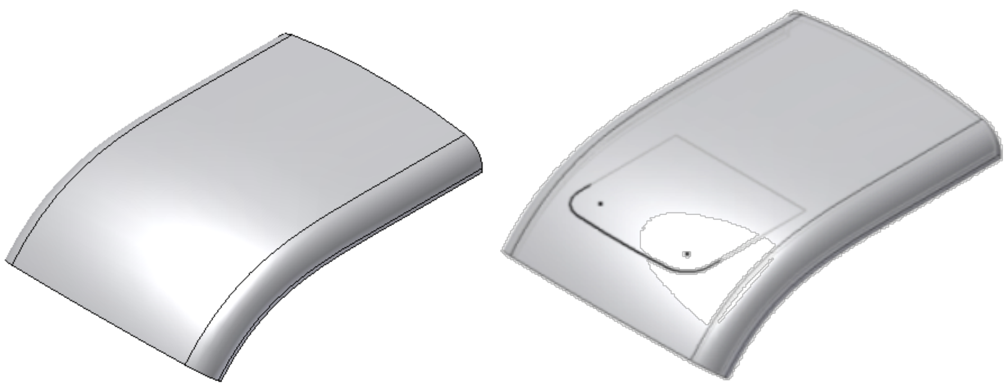


6. You can drag the landing profile or change the landing options to a new distance. Click OK.

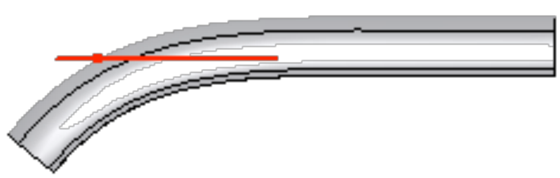


7. The Rest feature is complete.

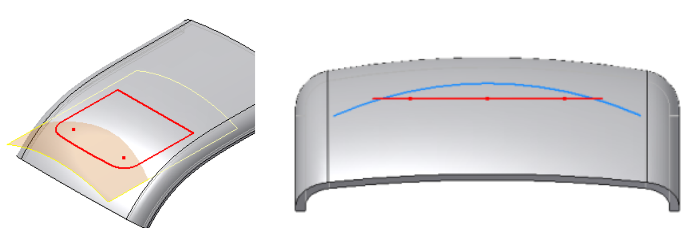
Build a Rest feature with a surface as the landing area.



1. Create or import a thin-walled part.



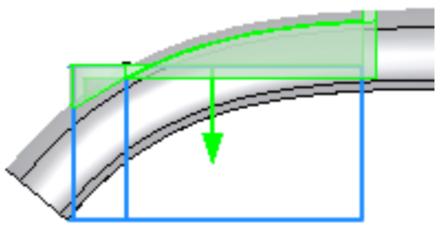
2. Build a 2D sketch that is on either sides of the thin-walled part



3. Build a surface that spans over the entire sketch in the position where the landing surface is needed.

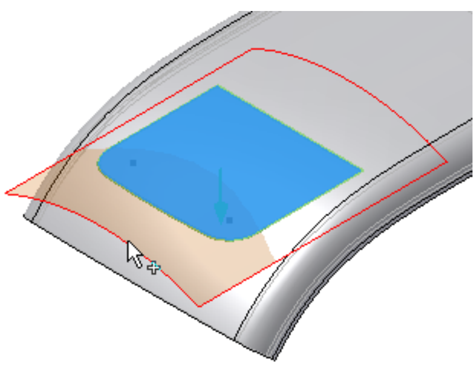
4. Click the Rest tool of the Plastic Features panel.

5. Make sure that the box in the Shape tab is set to **Through All** and that the platform direction arrow sis similar to what is shown in the image (for example, opposite to the appropriate clearance volume).

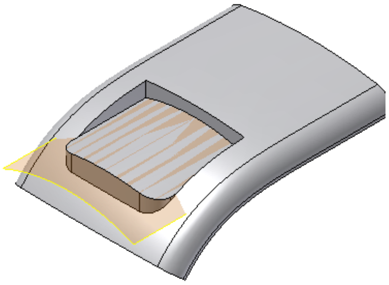
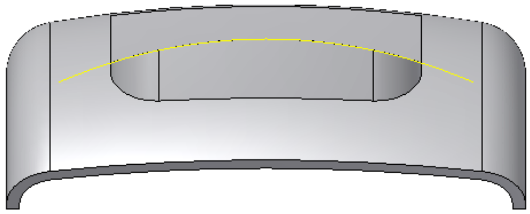


6. Switch to the **More tab**

7. Specify **To Surface** in the **Landing Options** box on the More tab and select the surface.

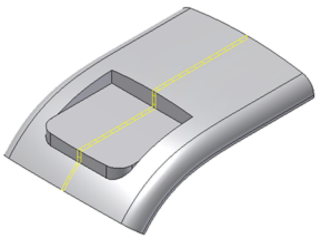
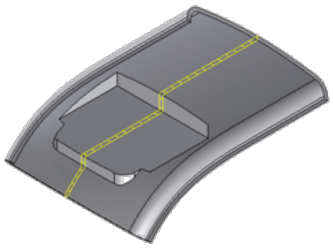


8. Click OK to complete the Rest with a surface-shaped landing.

Reference

Creates a Rest feature using a closed sketch. The Rest feature creates a thin-walled face that cuts across a thin-walled target body. The face is then joined by walls of the same thickness to the body.

Access:

Click the Rest tool on the Plastic Features panel bar.

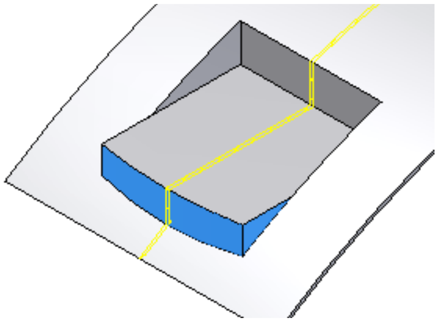
Shape tab

Holds the controls for the Rest placement and the platform options.

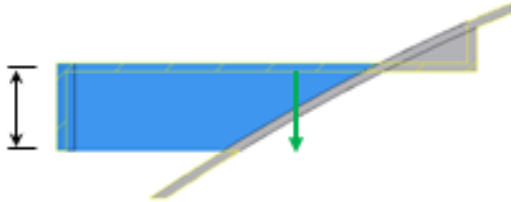
Profile selector to select one or more closed sketch profiles.

Platform extension type

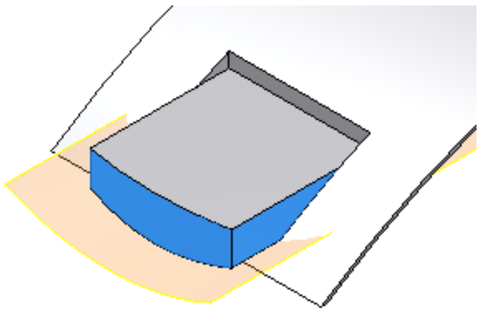
Through All extends the platform to the next face of the target body.



Distance specifies the height of the platform wall.



To Surface specifies the surface to extend the platform walls to.



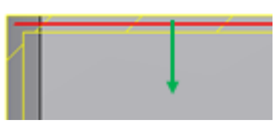
Offset specifies an offset from the reference surface in the To Surface option.

Thickness options specify the side of the offset of the Rest to create the thin wall with respect to the sketch reference and the direction arrow.

Inside

Outside

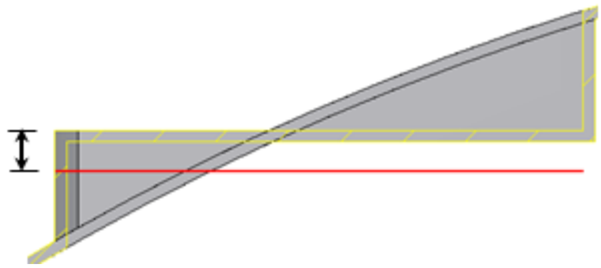
Both Sides



Flip  Flips the direction of the platform/clearance. The correct direction is the one that points to the inside of the target body since the goal is to maintain the landing profile on the outside.

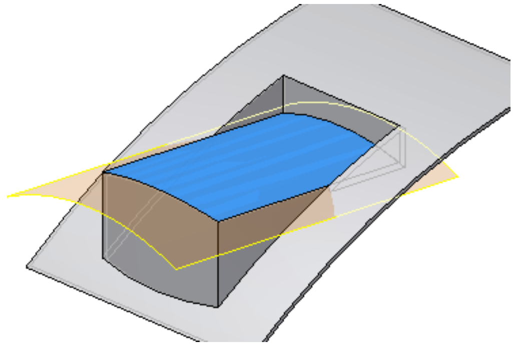
More tab

Specifies the landing parameters and the draft angles of the Rest feature



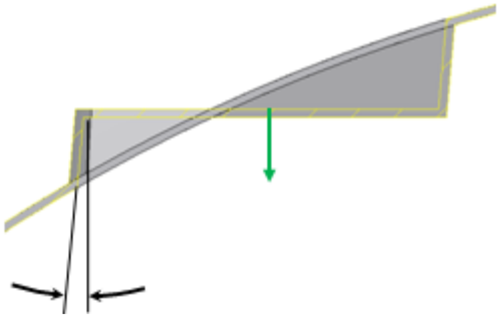
Landing Options

Distance specifies the distance of the landing from the sketch plane

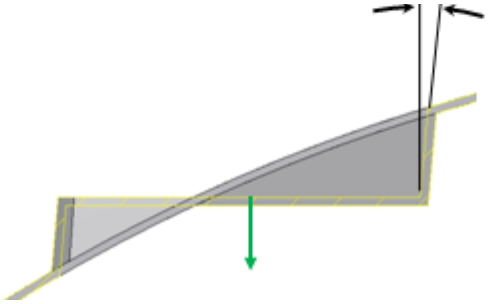


To Surface specifies the surface to place the landing on.

Offset specifies an offset from the reference surface in the To Surface option.



Landing Taper Specifies a draft angle for the platform walls



Clearance Taper Specifies a draft angle for the clearance walls.

Working with Rule Based Fillet Features

Concept

Rule-based Fillets are based on a list of rules that tells the feature how to discover the edges to fillet rather than creating links to specific topologies.

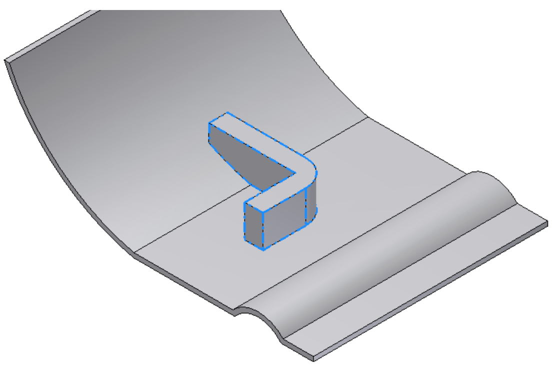
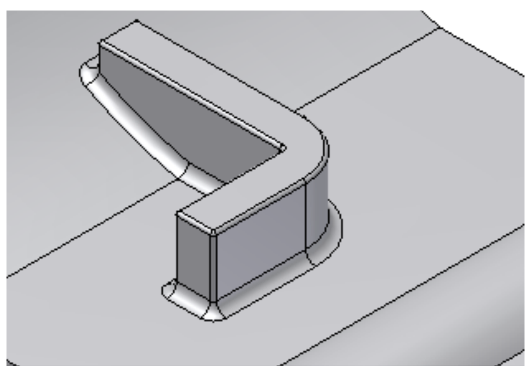
The rules have a source selection of entities (features and faces), and identify edges based on interaction statements. For example "*all the edges that a given feature generates when intersecting with the body*" so such edges are discovered at feature build or update time. There are two advantages:

1. It is faster and easier to identify in one rule-based fillet feature the many edges that otherwise should be selected individually.

Even upon large changes in both the feature and the body topology, the rule likely still applies, and the rule-based fillet easily regenerates itself providing a robust filleting paradigm. Rules that do not match edges for a given body topology are skipped.

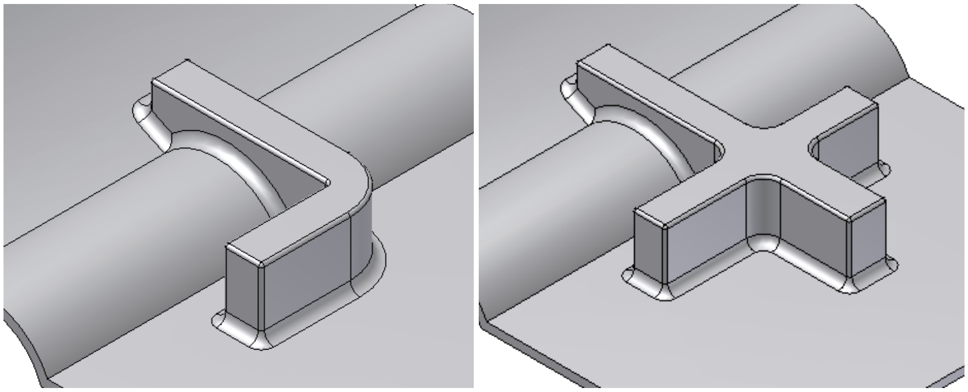
Example

You have a feature like the one in the image (extrusion of a planar profile To Next). You want to fillet all the edges of the feature itself and all the edges at the intersection between the feature and the thin wall of the part.

You can make this fillet with a single rule-based fillet on the highlighted feature with two rules: **Free Edges** (meaning the edges of the feature itself) and **Against Part** (meaning the intersection edges feature-part). Each rule requires only the selection of the feature (so two feature selections total), each rule can have its own radius.

In addition to the simplified filleting operation, there is a greater robustness upon even drastic topology changes of the target body, like the one in the image.



Supported are also drastic topological changes of the feature involved in the rules. For example, a redefinition of the extrusion profile rebuilds the rule-based fillet without any further interaction.

The elements of a rule-based fillet include:

- A **Source selection set** that relies on the features structure rather than on a specific topology (for example, one or more features or one or more faces).
- A **Rule** aimed at identifying the edges (for example, “all the edges created by a feature” or “all the edges created by the intersection between feature A and feature B”).
- Possibly a **Scope selection set** (if the rule requires a secondary selection set, for instance “all the edges created by the intersection between feature A – source set – and features B,C and D – scope set).
- An optional **exclusion set** aimed at limiting the edges identified by the rule (for instance “all the edges created by feature A excluding the edges that belong to a specific face”).
- An optional **convexity filter** to collect only the concave or convex edges (for example, fillets or rounds).This is useful in plastic design to achieve a constant wall at the intersection between two features (for example, all the concave edges formed by the intersection between feature A and feature B).

Each rule has the following input elements:

Features/faces

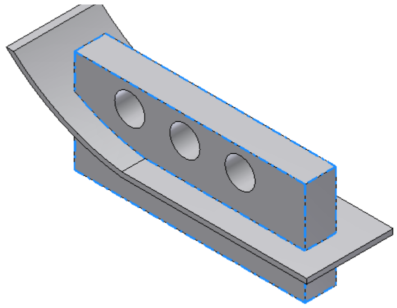
The target features or faces of the rule are picked from the graphics region or from the browser (in case of features), the rules are based on this selection set, also called primary selection set.

Rule type

Depending on the source selection set there are these types of rules:

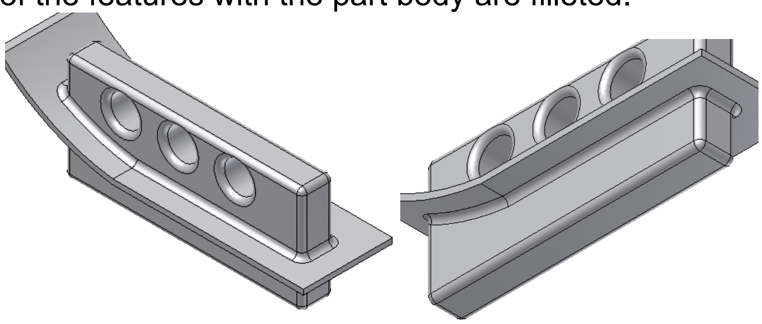
Source selection set: features

Consider the highlighted feature in the source selection set.



Feature Rule types

All edges, all the edges generated by the features themselves and by the intersection of the features with the part body are filleted.

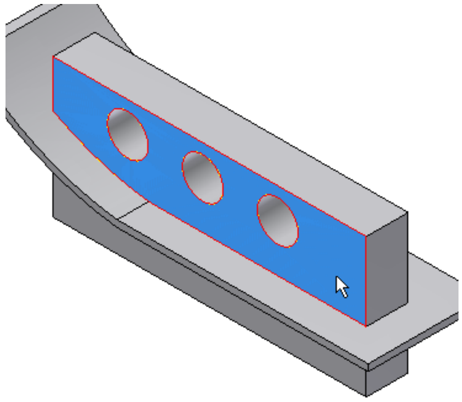


Against Part, only the edges formed by the faces of the features and the faces of the part body are filleted.

Against features, only the edges generated by the intersection of the features of the source selection set and the features in the scope selection set are filleted, for instance consider the three holes to be in the scope selection set.
Free edges, only the edges formed by the faces of the features in the source selection set are filleted.

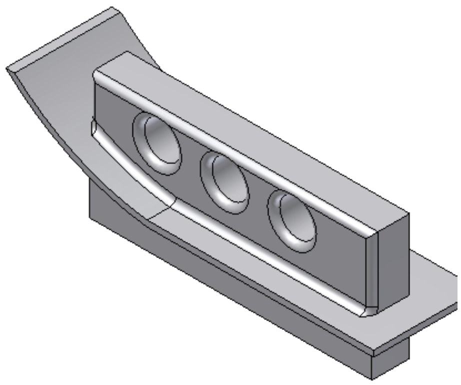
Source selection set: faces

Consider the highlighted face in the source selection set.



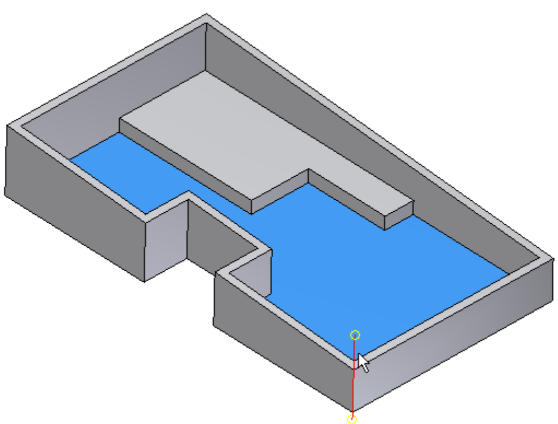
Faces Rule types

All edges, all the edges generated by the selected faces with any other part body faces are filleted.

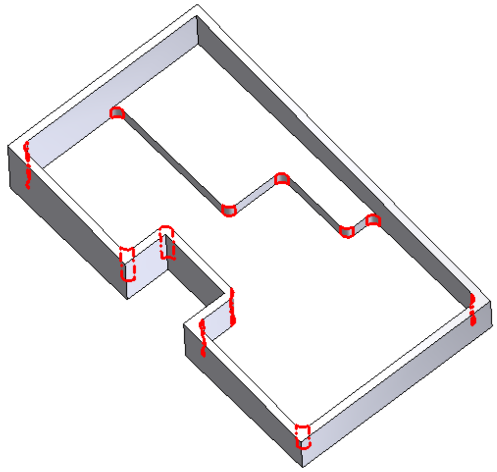


Against features, only the edges generated by the source selected faces and the faces of the features in the scope selection set are filleted, for instance consider the three holes to be in the scope selection set.

Incident edges, in this case the edges that end up on the source faces and are parallel to a selected axis (within a given tolerance) and with the same direction, are filleted. For instance given the indicated face in the source and the vertical axis near the cursor.



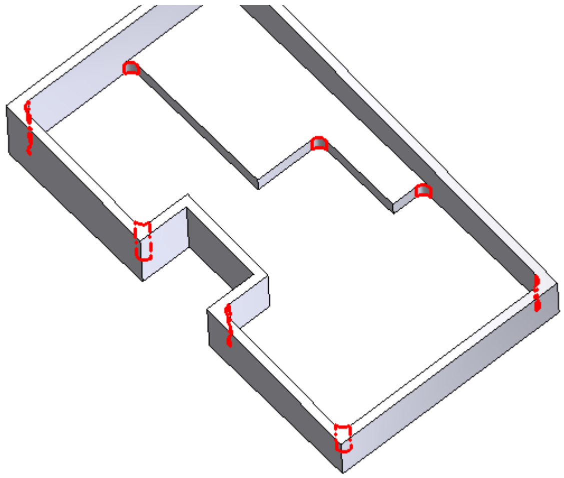
The face rule results in the following fillets as shown in the image.



Convexity options

Among all the edges selected by a given rule-based feature, you can additionally filter by convexity. For example, select only concave edges, convex edges, or both (the default).

All Fillets option: Concave edges. For instance in the incident edges rule above the All Fillets option filters the edges as indicated.



All Rounds option: Convex edges. For example, in the incident edges rule above the All Rounds option would filter the edges as indicated.

Face exclusions

You can pick a set of faces; edges formed by those faces are excluded.

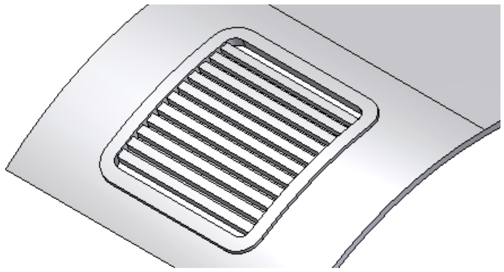
Edge exclusions

You can select a set of edges that are not included in the feature. The rationale for exclusion sets is to improve the flexibility of the rules and also to exclude certain edges that may cause the failure of the rule-based fillet feature.

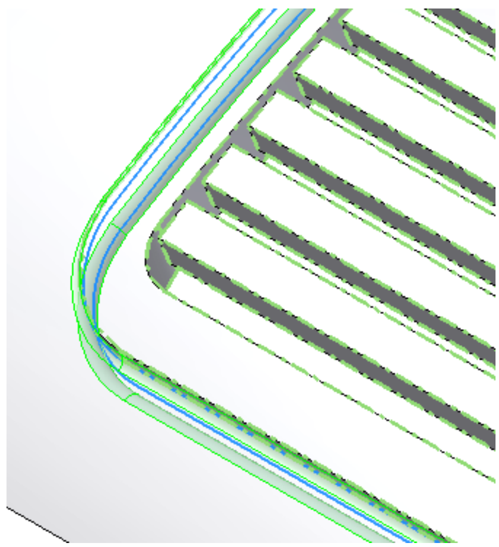
Procedure

How to fillet a grill

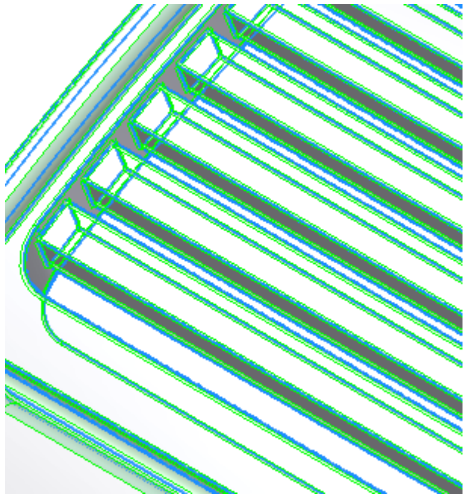
1. To fillet all the edges of the grill with a larger radius around the grill boundary and a smaller radius on the grill ribs



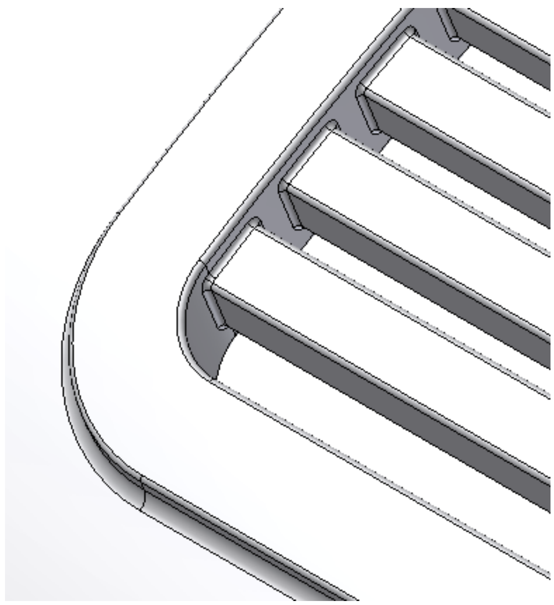
2. Click the Rule based Fillet tool of the Plastic Features panel bar.
3. Make sure the Source is set to **Features** and the Rule is set to **Against Part**. Select the grill and enter an appropriate radius for the edges at the intersection between the grill and the target part body.



4. **Click to Add** another rule.
5. Make sure that the Source is set to Features and the Rule is set to **Free Edges**. Select the grill and enter an appropriate radius for the remaining edges.

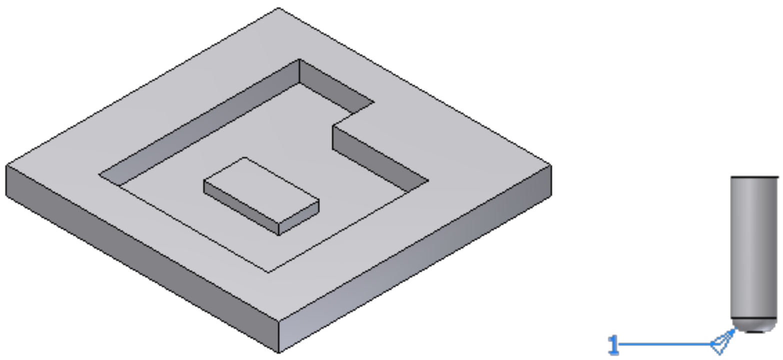


6. Click OK to complete the rule-based fillet.

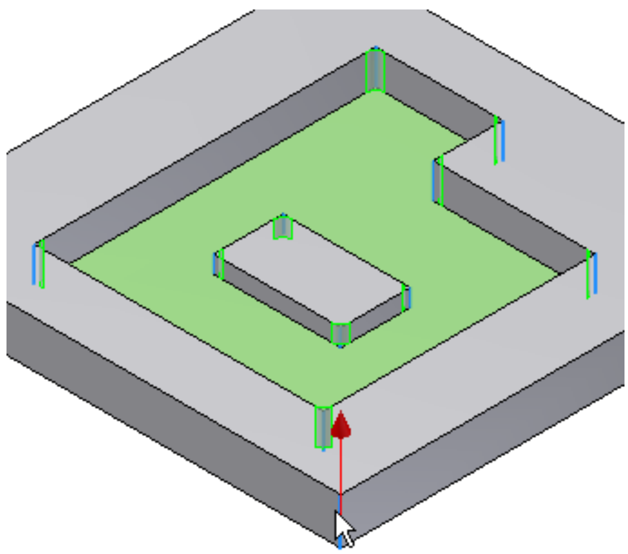


Rule-based filleting of a machined part

1. To fillet the pocket to model a milling process with a tool of3mmdiameter and 1 mm cutting radius. All the concave vertical edges must have a fillet of1.5mmradius (generated by the vertical approach of the tool) while the edges of the bottom face must have a fillet of1mmradius corresponding to the tool cutting radius.



2. Click the Rule based Fillet tool of the Plastic Features panel bar



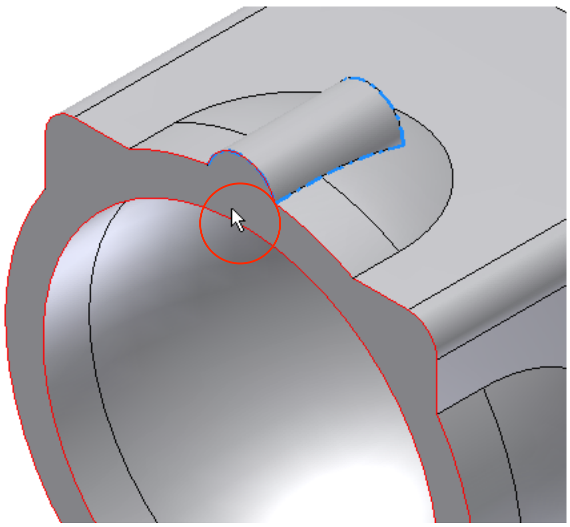
3. Set the Source to “Face” and select the bottom face of the pocket.

Set the rule to “Incident edges” and select a vertical edge to define the axis. Make sure that the direction is pointing up to include all the vertical edges that start from the selected face and go upward (in fact if you try to flip to the opposite direction no edge will be selected). Enter a1.5mmradius.

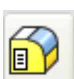
When machining the pocket I do not want to round the convex edges, so I uncheck the “All Rounds” toggle in the Options box.

Now we need to define the1mmradius fillets of the bottom edges.

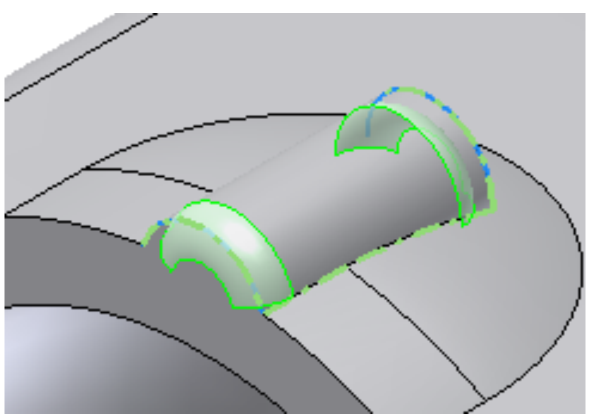
4. “Click to add” a new rule. Set the Source to “Face”. Select the same face as in the previous rule but now set the rule to “All Edges”.
5. Enter a1mmradius and hit OK, the Rule-based fillet is complete.
6. Even upon a change in the pocket shape the Rule-based fillet will rebuild correctly.
7. Exclude merged faces option.
8. I want to fillet the edges of the highlighted boss with different radiuses but not the edges on the selected face.



9. Click on the Rule based Fillet tool of the Plastic Features panel.

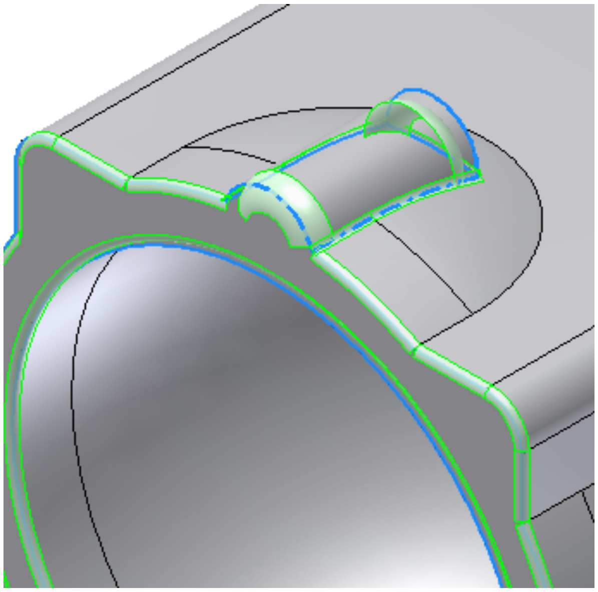


10. Make sure that the “Source“combo indicates “Feature” and the “Rule” indicates “Free Edges”. Select the feature and enter a radius value.

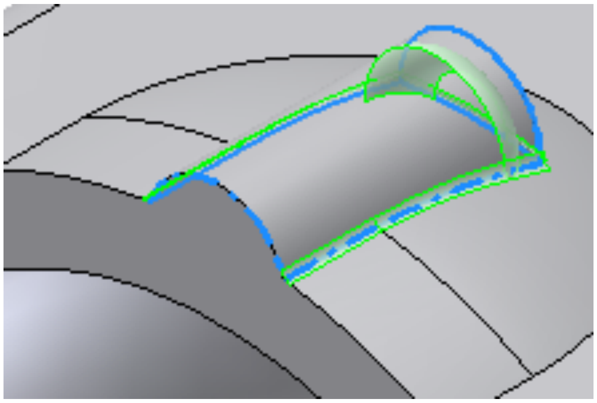


11. Add another rule to the same Rule-based fillet, Source: “Feature”, rule: “Against Part”.

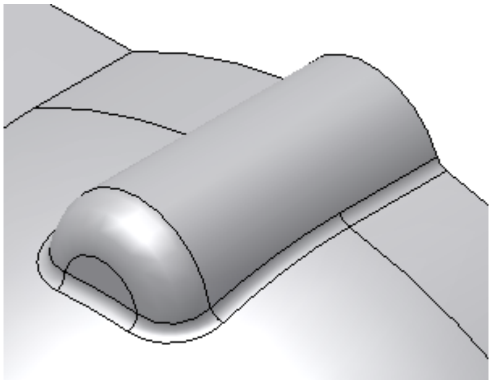
12. Select the same feature and enter a radius value. Note that also the edges of the planar face get fillets; this is because the face of the boss is *merged* with the face of the body...

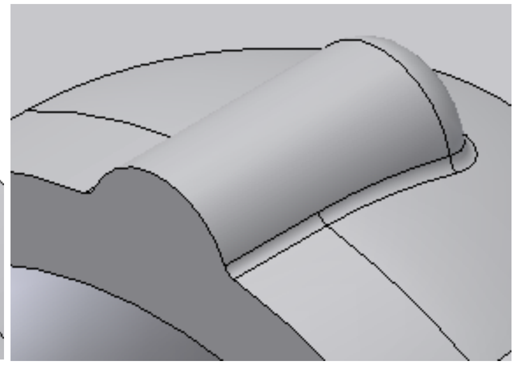


13. ... to avoid the merged faces to be included in the edges selection click on the “More” button and check the “Remove Merged Faces” toggle. The preview fillets on the planar face disappear.



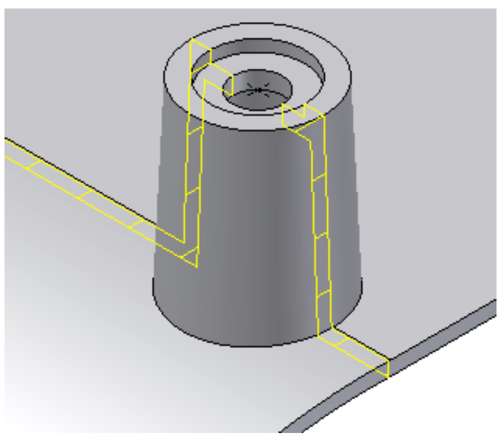
14. Click OK, the Rule-based fillet is done.





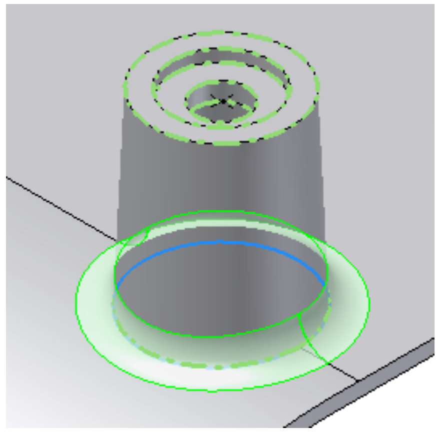
How to model constant wall thickness between a Boss and a part body.

1. In this example we show how to achieve a constant thickness wall at the intersection between a feature and the part. The new thin fillet is not always possible and using a rile fillet is a great way to build a constant thickness wall.

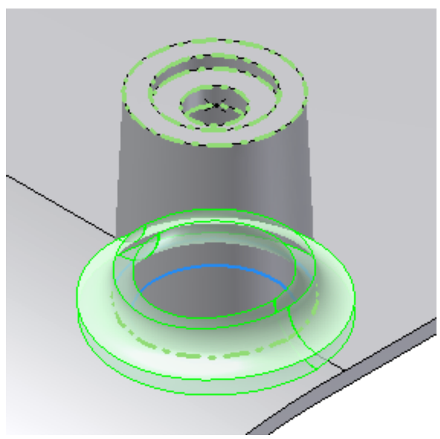


2. Click on the Rule based Fillet tool of the Plastic Features panel.

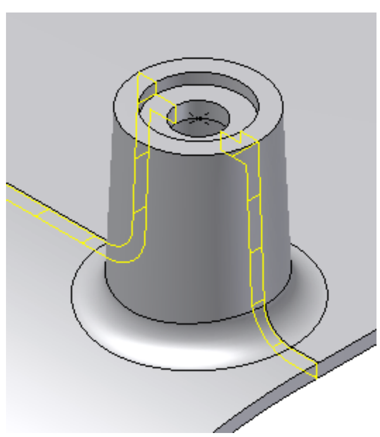
3. Set the Source to “Feature” and select the Boss. Set the Rule to “Against Part”. Uncheck the “All Rounds” toggle. Enter a suitable fillet for the outside of the part.



4. “Click to add” another rule. Set the Source to “Feature” and select the same Boss. Set the Rule to “Against Part”. This time uncheck the “All Fillets” toggle. Enter a radius which is equal to the one entered in the rule above augmented by the part thickness.

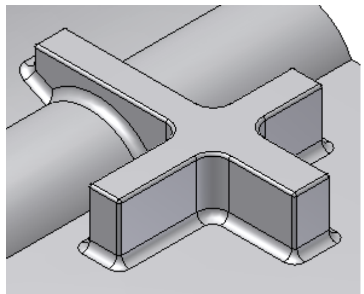


5. Click OK, the Rule fillet is done and the part has a constant thickness at the intersection Boss-part body.



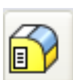
Reference

Creates constant radius fillets to the edges matched by rules. The rules have a source selection of entities (features and faces), and identify edges based on interaction statements, for instance "*all the edges that a given feature generates when intersecting with the body*" so such edges are discovered at feature build/update time.



Access:

Click the Rule based Fillet tool on the Plastic Features panel bar.



The user interface is organized as a table; each row contains one rule and the related source selection set.

Source

Specifies a base selection set tobeusedby a rule,canbeset to:

- **Feature** To select one or more features of the target body to be used by the rule of that row.
- **Face-** To select one or more faces of the target body.
- **Selection indicator** Click on the pen to activate the selection of the Source elements. When the pen turns into a pointer the selection is active.
- **Selection list** Indicates the number of selected items of each row Source.
- **Radius** Indicates the radius of the constant radius fillets of each row.
- **Rule Indicates** the rule of the active row, the available rules depend on the Source setting:

Source: **feature** rule types**:**

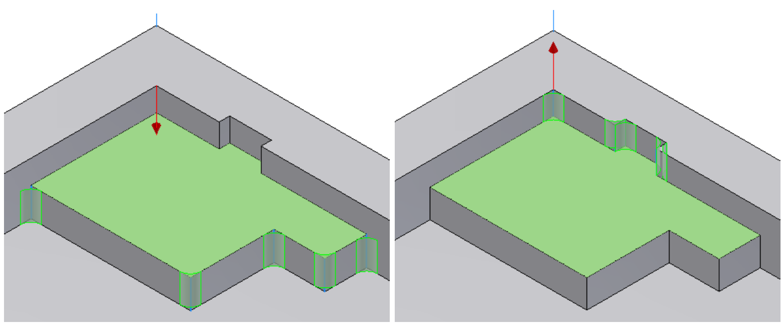
- **All edges** *All* the edges generated by the features themselves and by the intersection of the features with the part body will be filleted.
- **Against Part** Only the edges formed by the faces of the features and the faces of the part body will be filleted.
- **Against features** When this rule is set a **Scope features selector** pops up in the dialog, only the edges generated by the intersection

between the features of the source set and the features in the scope set will be filleted.

- **Free edges** Only the edges formed by the faces of the features in the source selection set are filleted.

Rule types

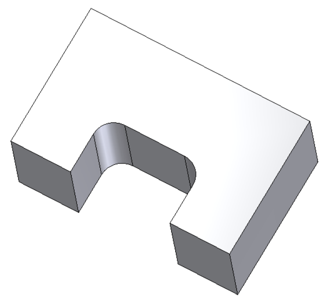
- **All edges** *All* the edges generated by the selected faces with any other part body faces will be filleted.
- **Against features** *When* this rule is set a **Scope features selector** pops up in the dialog, only the edges generated by the source-selected faces and the faces of the features in the scope set will be filleted.
- **Incident edges** When this rule is set an **Incident Edges** box pops up in the dialog. The **Direction** selector is to select the axis; the **Flip** button is to set the direction of the edges that will be matched by the rule. In this case the edges that end up on the source faces and are parallel to a selected axis (within a given tolerance) and with the same direction, will be filleted.
- **Tolerance** the tolerance allows capturing edges that diverge from the direction axis by the indicated angle; this is for instance useful to capture all the lateral edges of a pocket with a draft angle.
- **Tip:** the direction of reference for the incident edges filters the edges as indicated in the picture. When the direction is pointing upward the edges selected are those that “leave” the face in the up direction, on the contrary when the arrow points downward the edges selected are those that “leave” the face pointing downward.



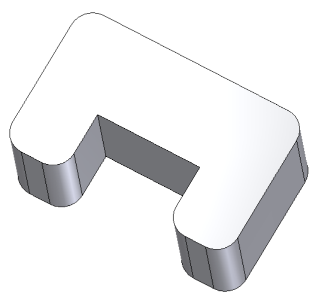
Convexity options

Among all the edges selected by a given Rule-based feature users can additionally filter by *convexity*, i.e. select only concave edges, convex edges, or both (the default). The convexity options are contextual to each row of the Rule-based fillet and the toggle show the actual setting of each active row.

- **All Fillets** option: selects only concave edges.



- **All Rounds** option: selects only convex edges.



More

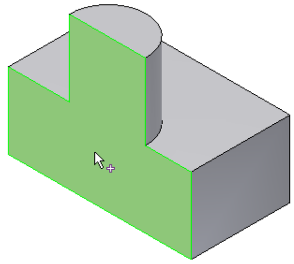
Sets the options for the Rule-based fillets. The default settings are correct for most features.

The following options are the same as the Part Fillet:

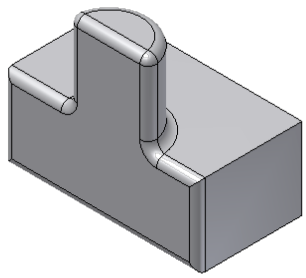
- Roll along sharp edges
- Rolling ball where possible
- Automatic edge chain
- Preserve All Features

Remove Merged Faces Removes from the matched edges all those edges that lie on faces that share a common geometry with other faces of the selection sets (called “merged faces”).

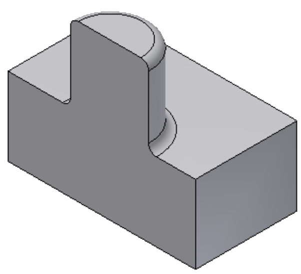
For instance the face indicated in the picture is in common between the box and the cylinder extrudes features.



When doing a Rule-based fillet, “Free Edges” rule, with the cylinder in the Source set you get this result for **Remove Merged Faces OFF:**



And this result for **Remove Merged Faces ON**:



Exclude Options

The rationale for exclusion sets is to improve the flexibility of the rules by filtering out the edges lying on selected faces or some edges directly selected that would be otherwise matched by the rules. Exclusions are common to the entire feature.

Faces selector   the users can pick a set of faces, edges formed by those faces will be excluded.

Edges selector the user can select a set of edges that will not be included in the Rule-based fillet feature.

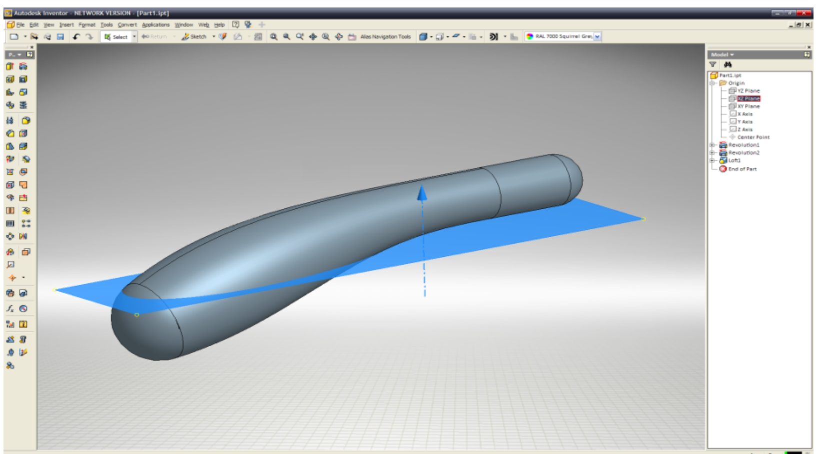
Working with 3D Silhouette Curves

Concept

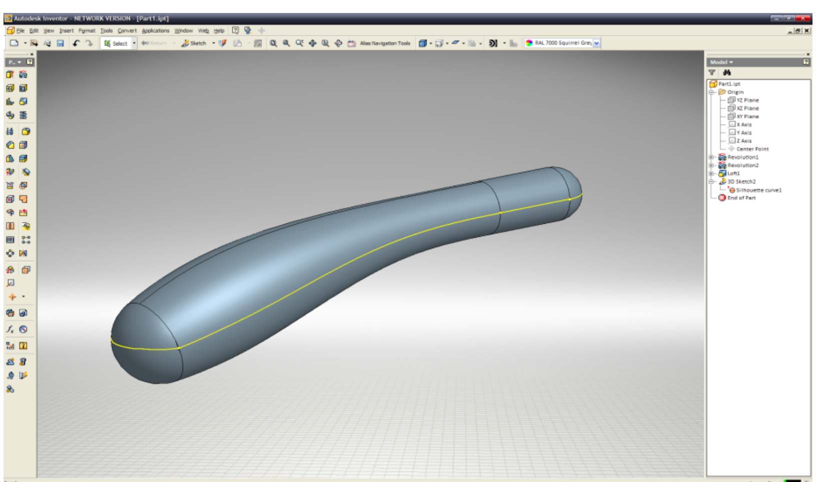
A silhouette curve is a 3D curve created in a 3D sketch that represents the outer boundary of the surface along the direction vector.

Example

1. Create or import a solid in a part
2. Create a new 3D sketch
3. While in the 3D sketch environment, invoke the Silhouette Curve command.
4. Select a direction for the silhouette curve



5. An associative silhouette curve is created; faces that lie in the pull direction are ignored. They can be filled in with lines or splines.

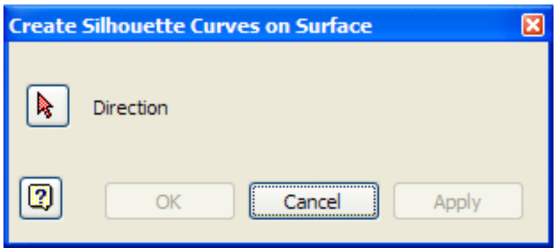


Reference

Access:

Click on the “Silhouette” of “3D sketch” panel.

Dialog



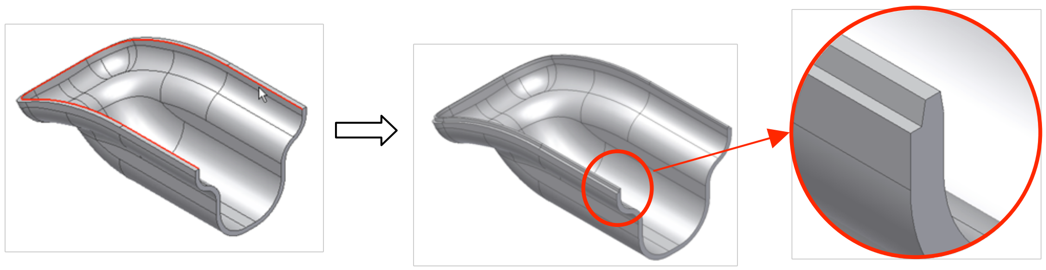
Direction

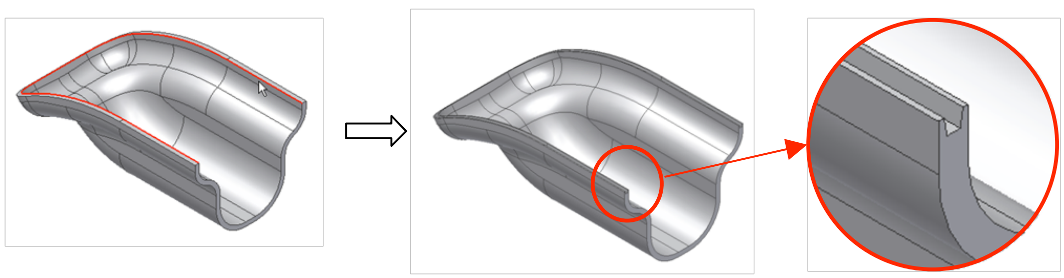
Direction vector along which the outer boundary (silhouette curves) of the surface will be created.

Working with Lip Features

Concept

Lips and grooves features are often part of a plastic product design. They are used to join two parts precisely at the parting line along the walls. The feature usually consists of two separate elements that precisely mate: the lip itself and the groove. The Lip feature allows the creation of alternatively both elements based on the path along the wall and the geometric parameters.



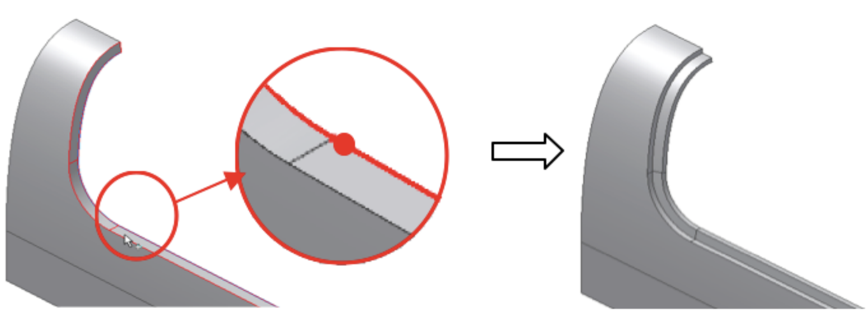


Path guidelines

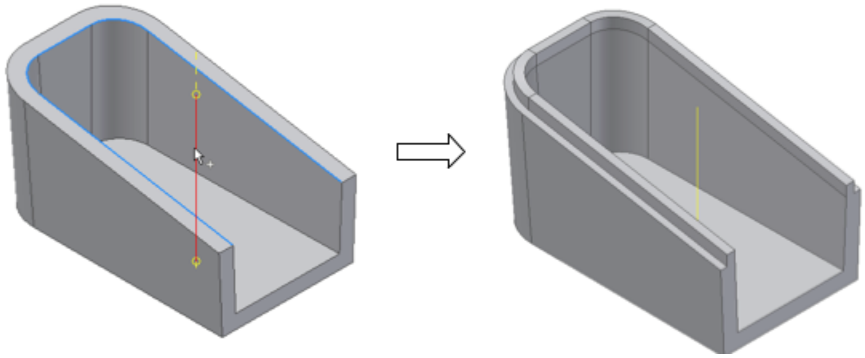
The path consists of a set of boundary edges smoothly connected (tangent continuous).It is possible to select more than one path in the same Lip/Groove feature as long as the Lip face (for example, the face of the thin wall where the Lip/Groove lies) is unique and tangent continuous. You can also select the lip face as the guide face.

Guide face vs. pulling direction

The Lip/Groove may be referred to the Guide Face (the face on the thin wall section). In this case, the Lip/groove cross section stays at a constant angle with the Guide face along the entire path.

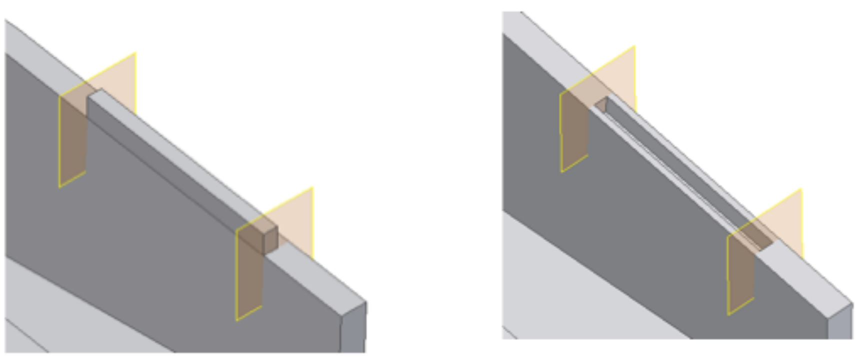


Alternatively, the Lip/Groove may be referred to a Pulling direction. In this case, the Lip/Groove cross-section stays parallel to the selected pulling direction along the entire path.



Path Extents

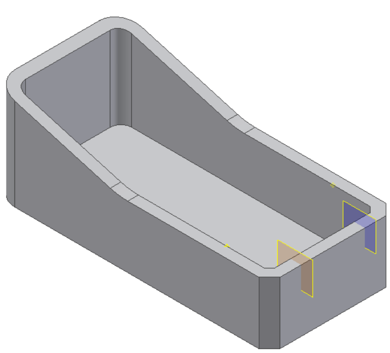
It is also possible to limit the extension of the Lip/Groove to one or more portions of the path (note: this is available only on open paths). You must select two or more limiting elements (points and planes) and the sections to keep or discard.



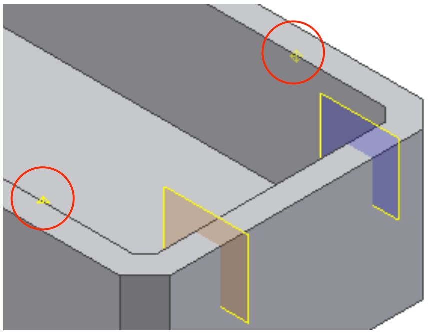
Example

Build a lip feature on multiple paths. Limit the lip extension on one of the paths to two trimming planes.

1. Create or import a thin-walled part like the one in the picture. Note: The open face is tangent continuous.

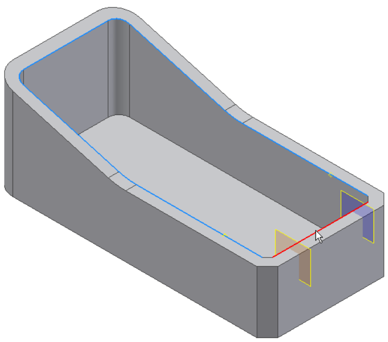


2. Define two work planes and two work points (see the red circles) along the path.

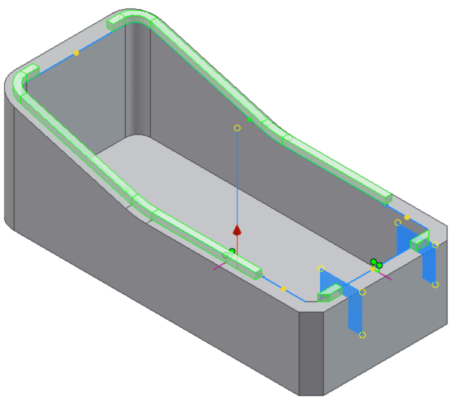


3. Select the Lip feature of the Plastic Features panel.

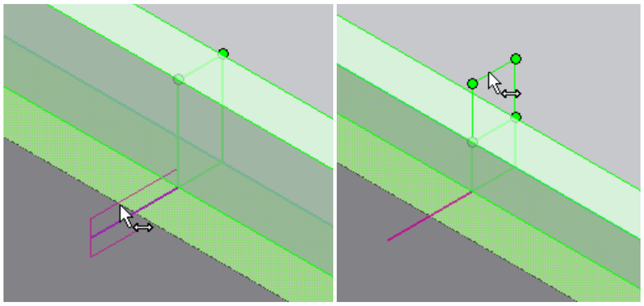
4. Select the two paths (blue and red in the picture). Note: Do not select the chamfer edges, leave the path open.



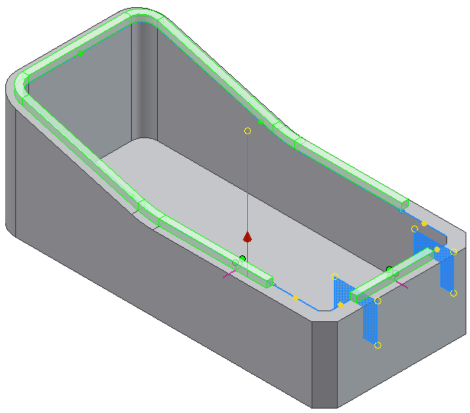
5. Select the Z axis as the Pull direction.



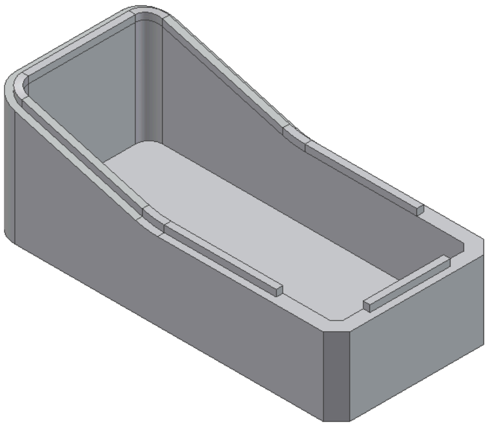
6. Use the manipulators on the preview to adjust the Lip cross-section and the clearance volume. Note: You can drag the cross section to a more convenient location along the path.



7. Expand the Path Extents selector and select the two planes and the two points.At first the Lip algorithm makes a tentative choice of sectors. Use the green and yellow dots to customize the sections to keep or remove

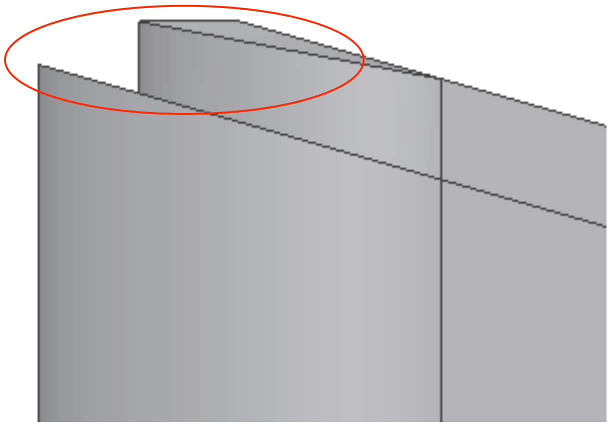


8. Click the green or yellow dots until the configuration is like the one in the picture.

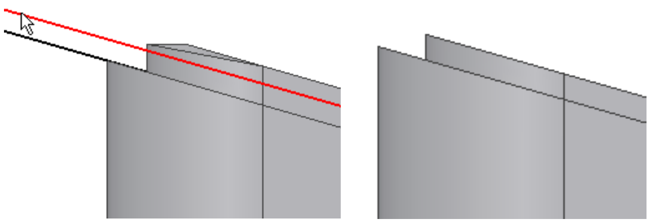


9. Click OK.

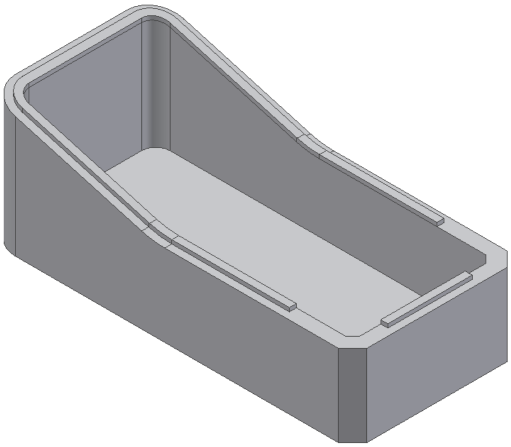
10. One drawback of the Lip with Pull direction as a reference is that the lip top face may swerve at the rounds of the path.



11. If necessary, you can correct it by cutting the part at an offset of the parting element (See the red profile).

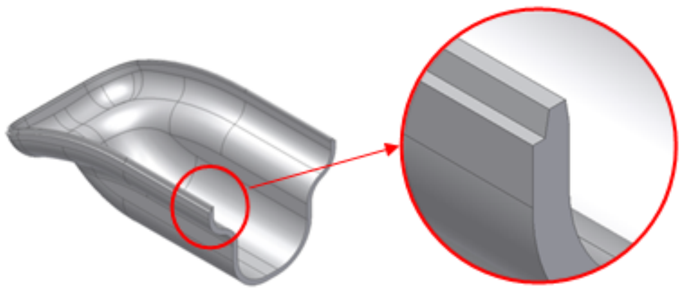


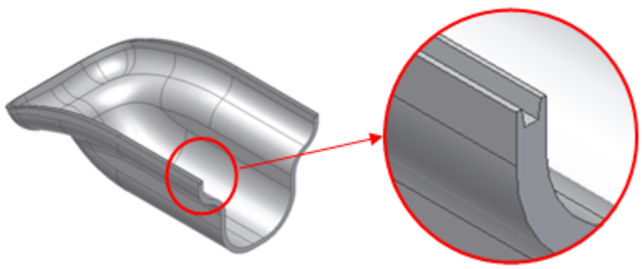
12. You can use the split command and select this profile as the split tool to cut the part.



Reference

Creates a Lip or Groove feature on the thin wall of a part.





Access:

Click the Lip button on the Plastic Features panel bar.

Lip/Groove toggle. Determines the type of feature to be either Lip or Groove.

Shape

Path Edges. Button to select one or more paths. Each path must be tangent continuous. All the paths of the same Lip/Groove must stay on a tangent continuous face.

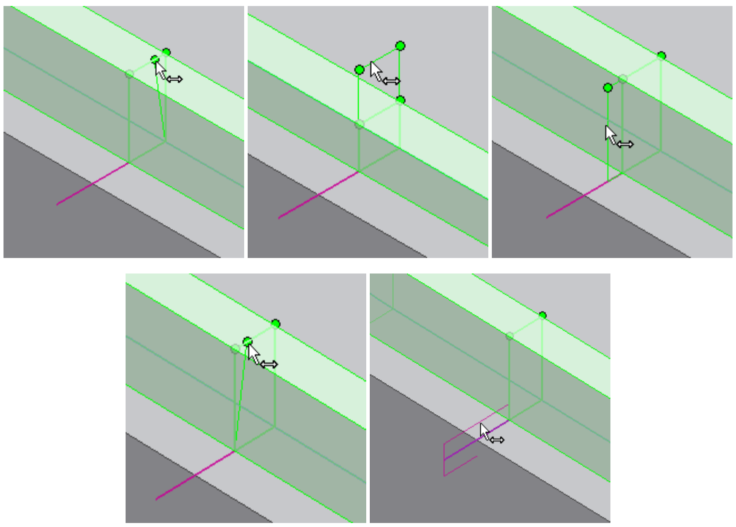
Guide Face. Button to select the Guide face. The Guide face has the path edges on its neighborhood. If selected, keeps the cross section of the Lip/Groove at a constant angle along the path.

Pull Direction. Expands the button to select the Pull direction of the Lip/Groove. It is alternative to the Guide Face. When selected ensure that the Lip/Groove cross section stays parallel to it along the entire path.

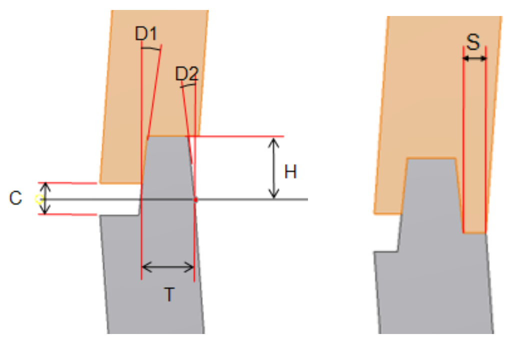
Path Extents. Expands to the button to select the elements to trim the Lip/Groove against. The same selector allows the selection of both the trimming elements and the sections of the path to keep or remove. Use the green and yellow dots to determine which portions of the path to keep or remove. Note: the Path extents functionality works only for open paths and is disabled when the path is closed.

Lip

The Lip tab holds the geometric parameters input for the Lip type. In alternative to a precise input in the text box, use the manipulators that allow the interactive dimensioning of all the parameters.



The Lip tab has the input text boxes for:



Lip draft angles D1, D2

Lip height H

Lip thickness T

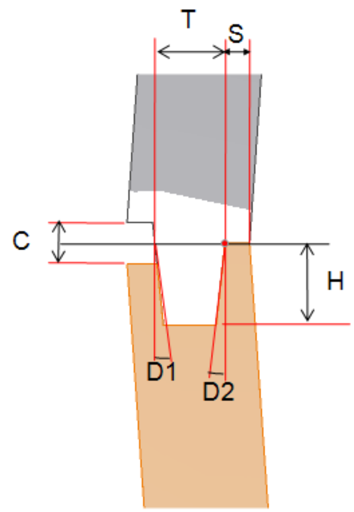
Shoulder width S

Clearance height C

Groove

The Groove tab holds the geometric parameters input boxes for the Groove type. In alternative to a precise input in the text box, use the manipulators that allow the interactive dimensioning of all the parameters.

The Groove tab has the input text boxes for:



Groove draft angles D1, D2

Groove height H

Groove thickness T

Shoulder width S

Clearance height C

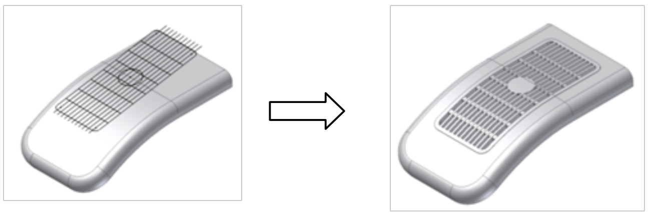
Working with Gill Features

Concept

Grill features are used to create vents or openings on the thin walls of a part to provide air flow for internal components.

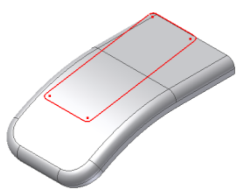
Placement

Grill features are created by projecting the patterns of one or more 2D sketches on the surface of the part.

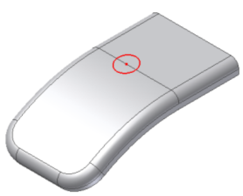


The patterns that are used to define the Grill geometry are:

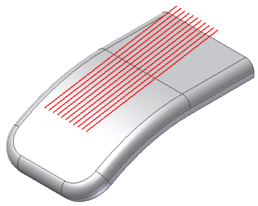
1. **Boundary**: must be a closed sketch. It limits the extent of the grill. Often the boundary is raised at an offset of the part surface.



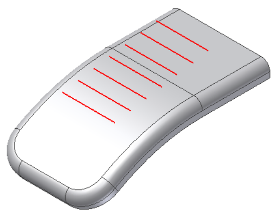
2. **Island**: an area filled with material usually at the center of the grill. The island often has the same thickness as the shell. It must be a closed sketch.



3. **Ribs**: one pattern that fills the grill area. The external faces of the ribs are flush or slightly recessed with respect to the boundary external face.

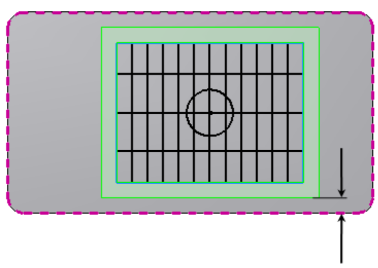


4. **Spars**: a secondary pattern, usually added to improve the stiffness of the ribs.

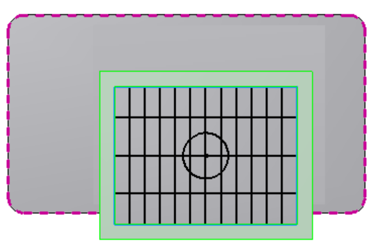


For the Grill feature to be successful the boundary external contour projection (plus a 10% tolerance band around the external contour) must stay entirely within the part surface and no material gaps (holes) are allowed inside the boundary projection.   Examples:

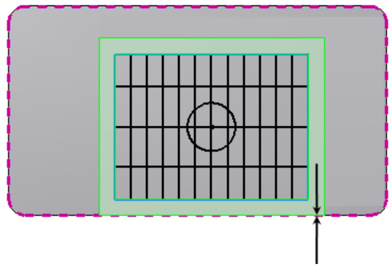
Grill ok, the external contour of the Grill boundary projection stays within the target body with a 10% tolerance.



Grill will not compute, the external contour of the Grill boundary projection is not entirely included within the target body.



Grill will not compute, the external contour of the Grill boundary projection is too close to the margin of the target body (the distance tolerance is below the 10% threshold).

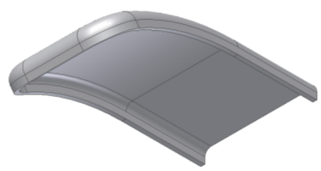


The boundary sketch is the only mandatory element to build a Grill. The area of the openings of a Grill feature is reported in the Flow area section of the dialog box and in a Reference parameter. The flow area does not take into considerations the optional draft angles.

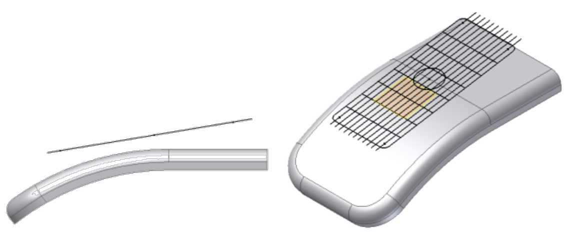
Example

Build a Grill feature on a thin walled part starting from a 2D sketch.

1. Create or import a thin-walled part.

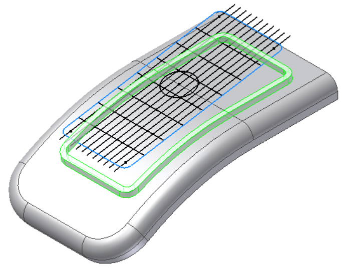


2. Define the various Grill outlines on a 2D sketch on a plane external to the surface where the Grill is desired.

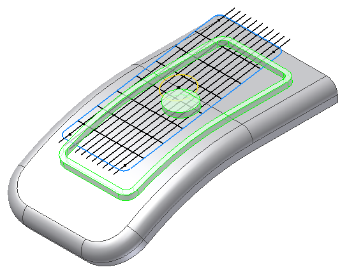


3. Click the Grill tool of the Plastic Features panel.

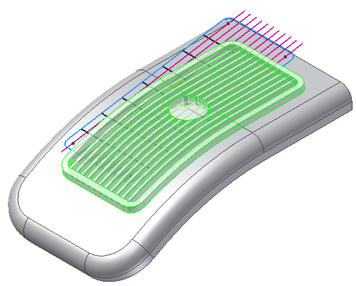
4. Select the Boundary profile, and enter the Thickness, Height, Outside Height (i.e. offset from the body face) parameters. The preview shows the corresponding geometry.



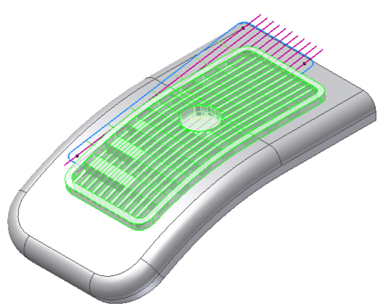
5. Select the Island profile and enter the Island wall thickness parameter.



6. Select the Rib profiles. You can use window selection (by dragging the cursor left to right you select only what is entirely included in the window, by dragging the cursor from right to left the window selects whatever entities are intersected by the window boundaries). Enter the Rib Thickness, Height, Top Offset (from the Grill Boundary face) parameters.



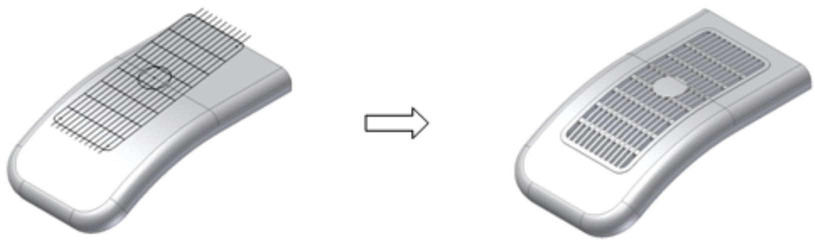
7. Select the Spar profiles. You can use window selection here also. Enter the Spar Thickness and Offsets (from the Rib faces) parameters.



8. Enter a draft angle.

Reference

Creates a Grill feature using the various patterns included in one or more 2D sketches.



Access:

Click the Grill button on the Plastic Features panel bar.

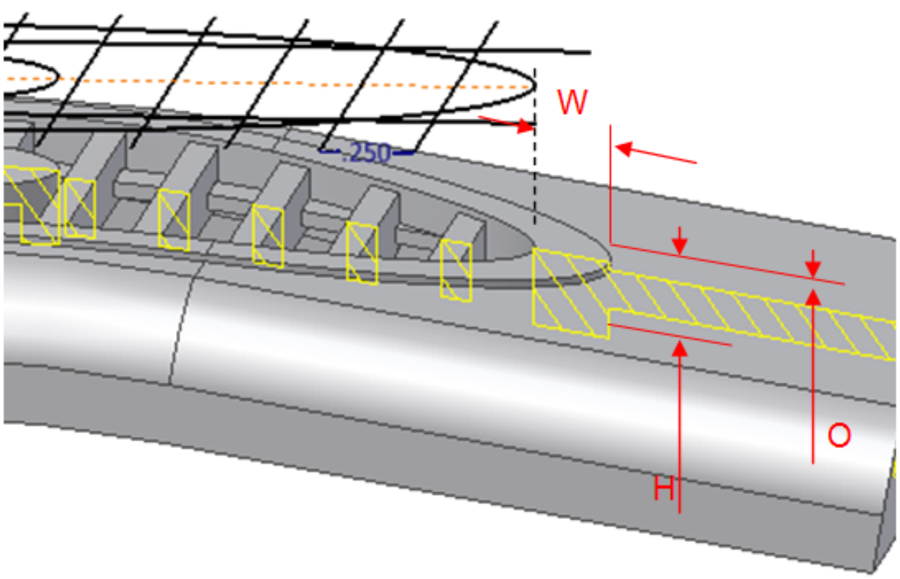
Boundary

Specifies the closed extent of the Grill.

Profile. One planar *closed* profile. The selection of the Boundary profile implicitly determines the Grill *direction* as the direction from the profile centroid to the closest point of the part along the profile plane normal. The Boundary profile is mandatory to build the Grill.

The parameters are:

W: Boundary **Thickness**, H: Boundary **Height**, O: **Outside Height offset** of the Boundary top face with respect to the part surface.



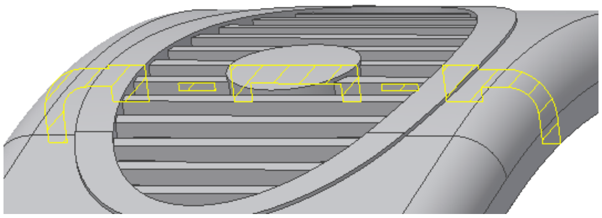
Island

An area filled with material usually at the center of the grill. The island often has the same thickness as the shell.

Profile. One planar, *closed* profile. The plane must be parallel to the Boundary plane (profile can belong to the same sketch as the Boundary). If no profile is selected, the Island is not present. It is not necessary to trim the Island profile at the boundary profile. This operation is automatically executed by the feature itself.

The Island has the thickness as the only parameter. Its top and bottom limits correspond to the Boundary top and bottom faces. Ribs and Spars are trimmed at the profile of the island.

Note: A zero thickness means that the island does not have a wall and is filled with material.



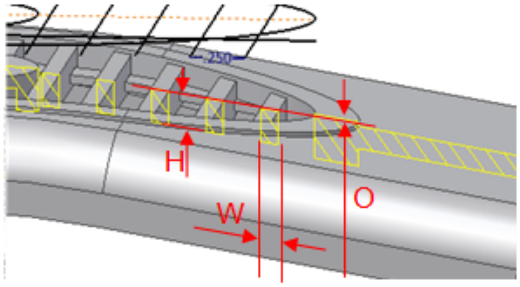
Ribs

One set of curves that fill the Grill area. The external faces of the ribs are flush or slightly recessed with respect to the boundary external face.

Profile. One or more *open or closed* profiles belonging to the same sketch on a plane that is parallel to the boundary profile (can belong to the same sketch as the boundary). The profiles are the centered outlines of the ribs. It is not necessary to trim them at the boundary profile. This operation is automatically executed by the feature itself.

The parameters are:

W: Rib **Thickness**, H: Rib **Height**, O: Rib **Top Offset** from the external face of the Boundary. Ribs can extend below the internal face of the target body.



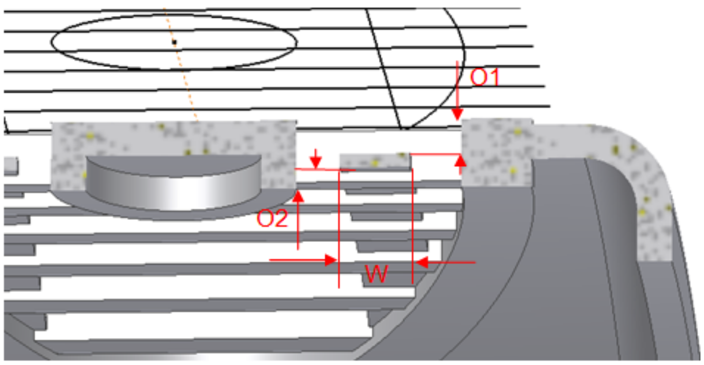
Spars

A secondary set of curves, usually added to improve the stiffness of the ribs.

Profile. One or more *open or closed* profiles belonging to the same sketch on a plane that is parallel to the boundary profile (can belong to the same sketch as the boundary). The profiles are the centered outlines of the Spars. It is not necessary to trim them at the boundary profile. This operation is automatically executed by the feature itself.

The parameters are:

O1: **Top Offset** from the top face of the ribs, O2: **bottom Offset** from the bottom face of the ribs, W: **Thickness**



Draft

The draft angle for the grill requires only two parameters:

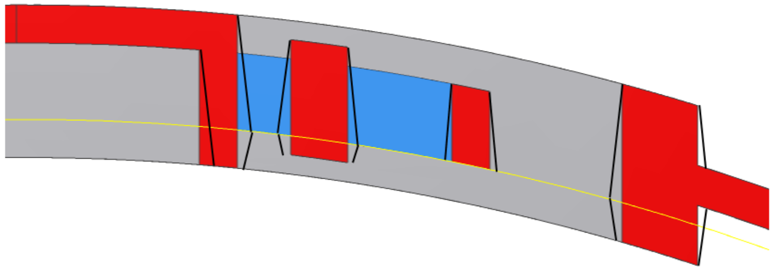
9. A draft angle (no sign).
10. (Optionally) the offset from the *Boundary Outside* of the parting surface.

If you input the draft angle, the software chooses "the best practices" draft configuration. If you decide to define the parting element (as the offset from the *Boundary external face* of the parting surface), then the draft is modeled accordingly without any check on the manufacturability.

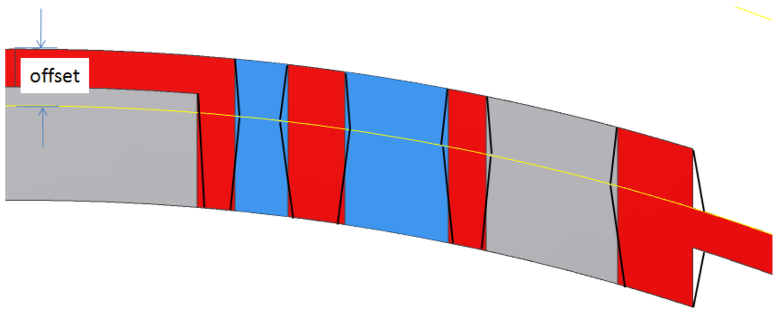
The "best practice" draft is defined:

- The parting as far as possible from the "Outside" defined by the position of the Boundary profile (to minimize the esthetical damage of the parting lines).
- Minimize the double draft (to simplify manufacturing).
- The neutral always to coincide with the "Outside" face of each element (to have the nominal dimensions on the visible part of the Grill).

In practical terms, the parting element is either the inside faces of the Boundary or the Spar.



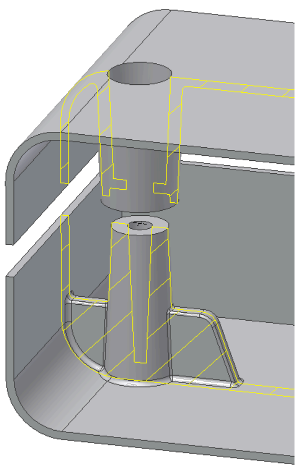
Alternatively, you can define the parting element as the part surface (before the application of the Grill) at a given Offset from the Boundary outside face. By changing the offset amount, you can position the parting element where deemed necessary. The outside faces remain neutral.



Working with Boss Features

Concept

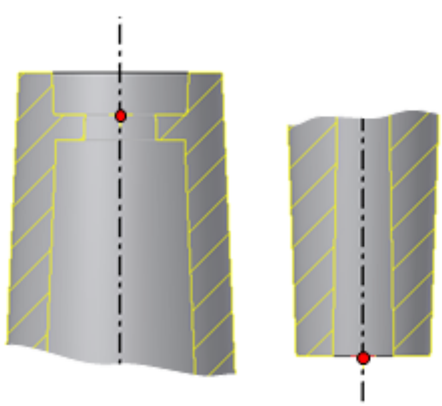
Fasteners are a common joining mechanism in plastic parts. To place and hold fasteners in place usually a couple of paired geometries must be modeled on both parts: a boss for the head of the fastener and a post for the fastener thread to bite into.



The Boss feature provides for the design of both components of a fastener placement, called **head** and **thread**, in addition it allows the design of fastening ribs that often occur especially when the length of the boss is greater than three times the diameter.

Placement

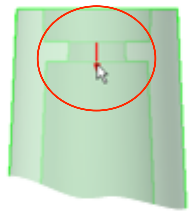
The *placement* of a Boss feature is referred to a point centered on the mating face of, respectively, the head and the thread. The Boss is then extended to the next face of the target part body. Note that to design the two mating components in the two opposite bodies it is enough to use the same placement elements and opposite directions.



The user may select alternatively the following placement elements:

- ***Points From sketch*** ("From sketch" option), select placement points from the points defined in the same sketch, the sketch normal provides the *boss direction*, click on the boss direction preview arrow to switch

- ***3D Work points*** *or sketch points*("On Point" option), select placement points from different sketches or work points, in this case the sketch normal is not used, the boss direction must be defined by selecting geometric or work elements.



It has to be noted that the feature can only exist if it can be extended to a face of the target part body that is opposite to the placement point in the boss direction. The feature determines automatically the closest face of the part body it can extend to, users can select a different target body and flip the direction as well.

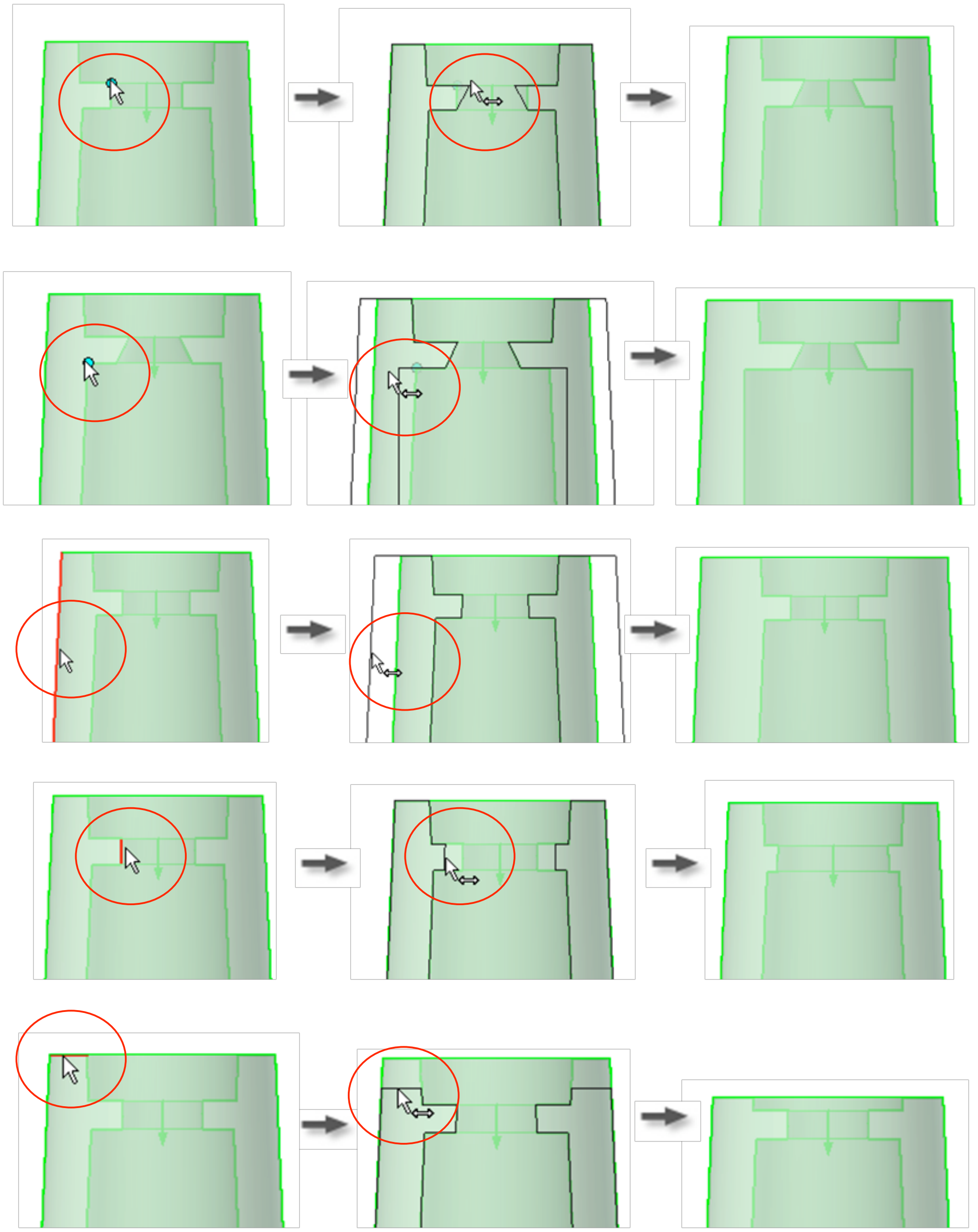
The *dimensions* of the Boss feature can effectively be set by use of *manipulators* that are available on the preview to interactively define the geometric parameters when precise dimension input is not required.

Preview manipulators

At preview the cross section of the Boss is shown in a plane close to the viewing plane. Hovering over the cross section highlights the segments that can be dragged; hovering on key points will show additional dots that can be dragged. The reason why some dots remain hidden is to avoid excessive cluttering of the preview. So the manipulators interaction may require some discovery action from the user, mostly done by hovering the pointer over the preview, but it is very simple and intuitive and fast to get used to.

There are two types of manipulators:

- *Dot manipulators*, they define some of the angular and linear parameters of the Boss, in the pictures below there are some examples of how they work.
- P*rofile manipulators*, to define the linear parameters of the Boss, examples below.

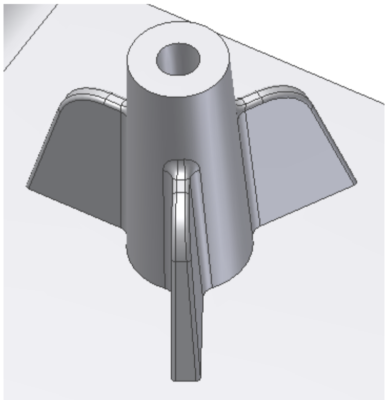


Stiffening ribs.

When adding stiffening ribs on a placement of type “From sketch” the first rib alignment angle is measured from the sketch Y axis.

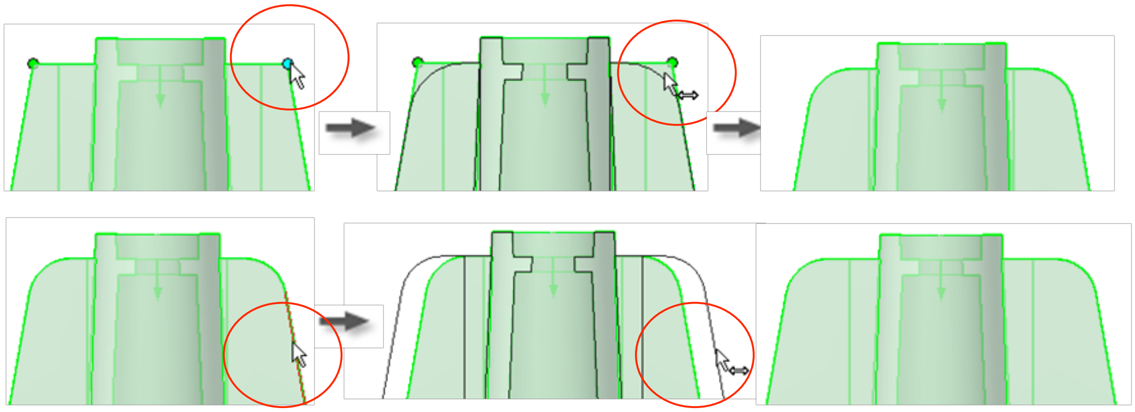
If the placement is of type “On Point” the user may select a direction for the first rib alignment by picking a linear geometric element that will be projected on the plane normal to the Boss direction.

The user can select the number of ribs that will be evenly distributed all around the Boss starting at the first rib alignment.

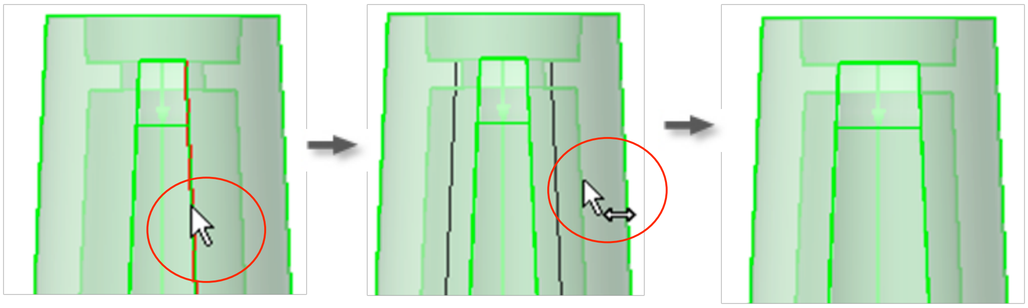


Also for the stiffening ribs there are convenient manipulators to interactively determine the dimensions.

Front view:



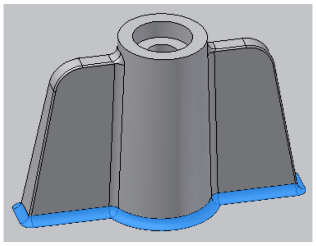
Side view:



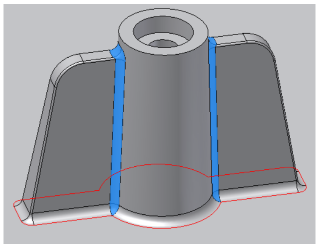
Boss Fillets

The Boss feature has also the option to define constant radius fillets at three different locations as indicated in the pictures below:

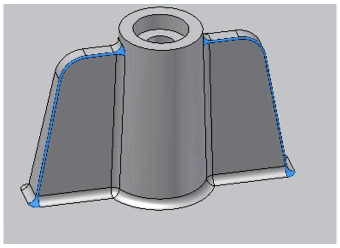
1. At the intersection edges of the entire Boss feature with the part body target,



2. At the intersection edges between the stiffening ribs and the boss,



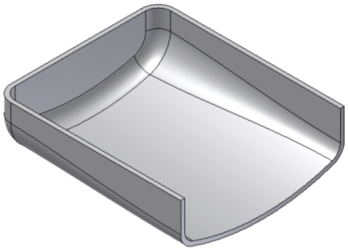
3. On the edges of the ribs.



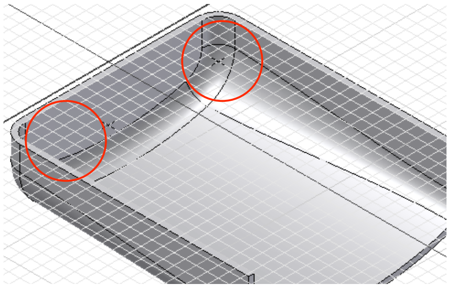
Example

Build a Boss feature (Head component) on a part using points of a 2D sketch.

1. Create or import a thin walled part.



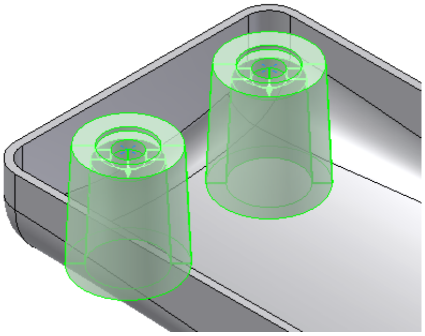
2. Build a 2D sketch with a couple of points corresponding to the Boss placements.



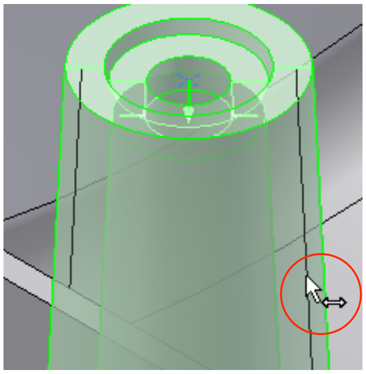
3. Click on the Boss tool of the Plastic Features panel.

4. Make sure that the Placement method is “From Sketch” and the Boss type is “Head”.

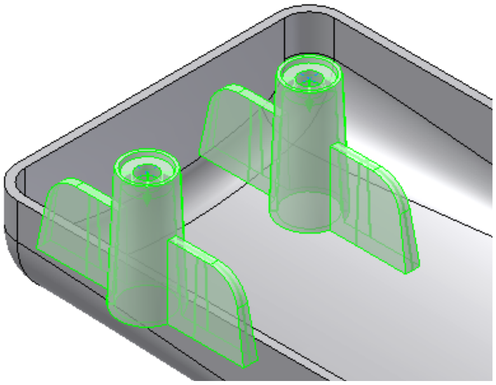
5. Select the points for the placement (ctrl + click to unselect). The closest part body in the direction of the sketch normal (both ways) will be chosen as the target body, if you want a different target just use the body selector on the main tab.



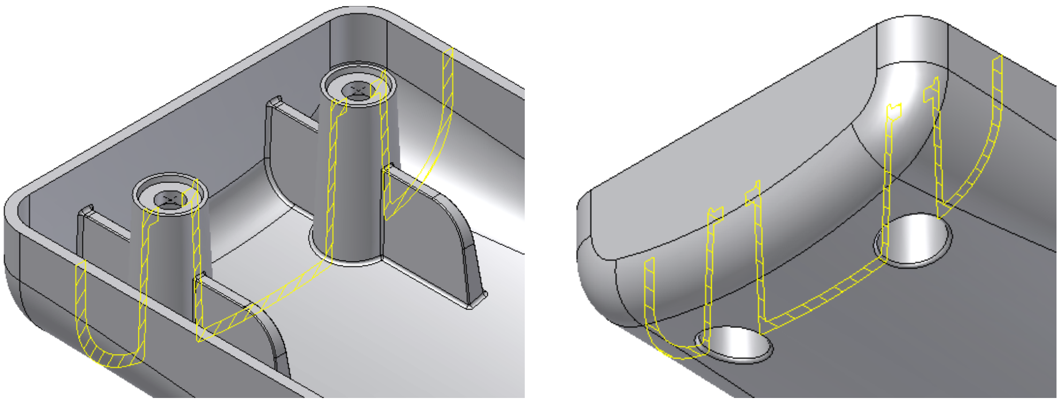
6. Adjust the Boss head parameters or use the manipulators until the desired dimensions are reached.



7. Add 2 stiffening ribs, make sure they are oriented like in the picture (it may be necessary to enter 90^{o} angle depending on the sketch Y axis orientation). Adjust the ribs dimensions using the manipulators or the dialgo input fields.



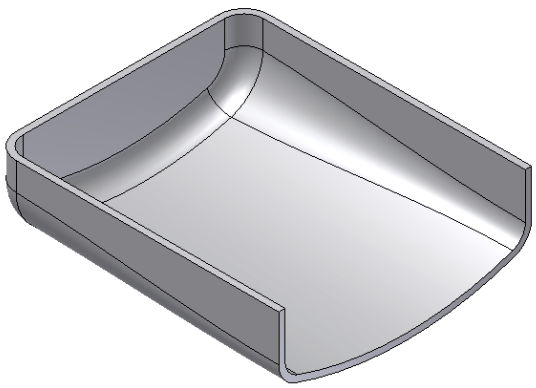
8. Add fillets at the intersection boss-body, at the ribs edges and at the intersection ribs-boss. Hit OK, the mounting Boss is done.



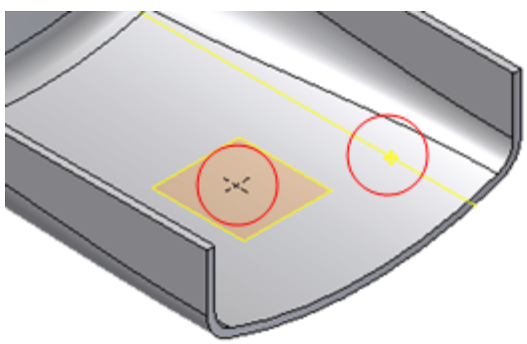
Build a Boss feature (thread component) on a part using On Point placement.

Here the goal is to create a couple of thread bosses using one point from a sketch and one 3D work point, for instance in order to have different boss heights.

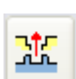
1. Create or import a thin walled part. The goal here is to place a Boss feature on the circular edge at a position that is not convenient for the "on sketch" positioning style, for instance at a point on the edge where the hook direction is not likely to be aligned with a sketch X or Y axis.



2. Build the construction elements: one sketch point and one 3D workpoint where the Bosses are desired.

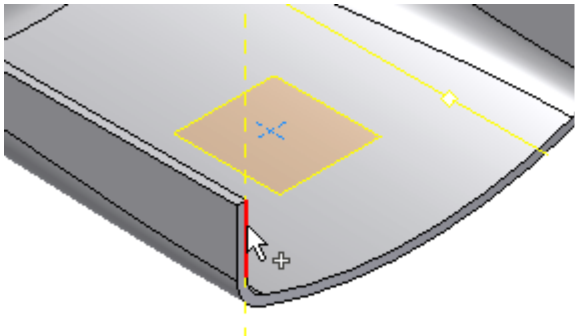


3. Click on the Boss tool of the Plastic Features panel

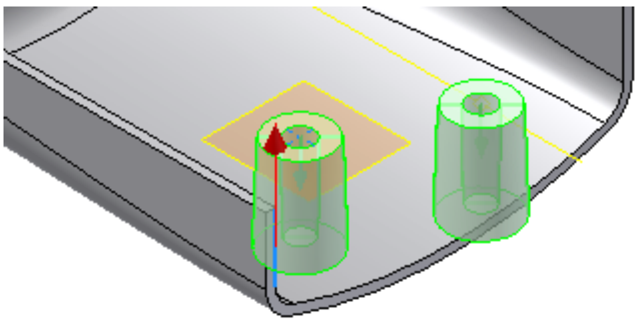


4. Make sure that the Boss type is "Thread" and that Placement method is "On Point".

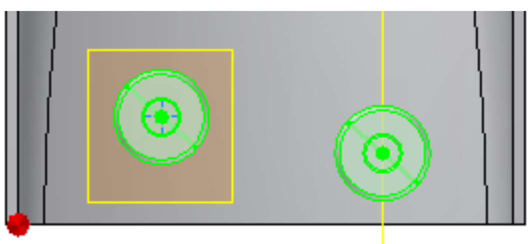
5. Select the points defined in step 2 as the *Centers*, select a Vertical edge like the one in the picture as the *Direction*.



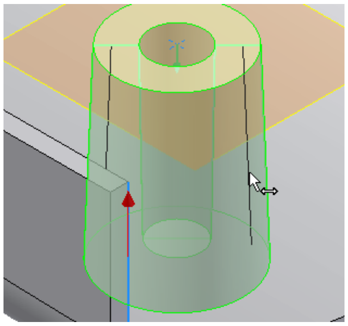
6. As soon as the direction is defined (it's the same for all the bosses in the same feature) the preview will show up.



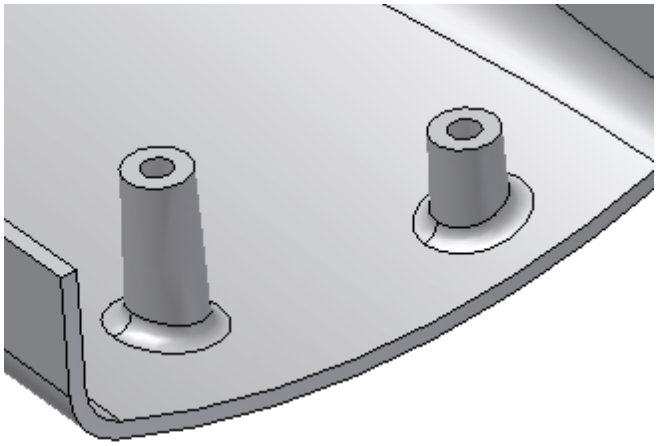
7. Make sure that the bosses can extend to the target body, i.e. their projection will be completely within the part body boundary.



8. Adjust the dimensions as usual, and assign a fillet for the intersection boss-part body.



9. Stiffening ribs are not necessary here, hit OK, the Boss is completed.



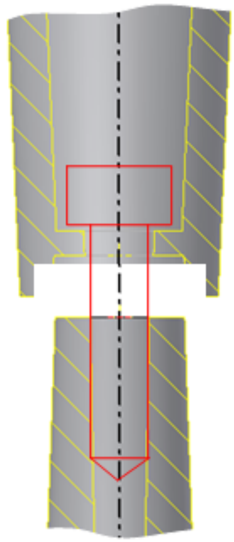
Reference

Creates a Boss feature using points of 2D sketches or 3D work points and direction elements.

Access:

Click the Boss tool on the Plastic Features panel bar.

Type specification



Head type. This is where the head of the fastener will stay

Thread type. This is where the thread of the fastener will engage.

Shape tab

Common to both types. It holds the controls for the Boss placement.

Placement

From sketch placement requires 2D points on the same sketch.

Centers selector, to select the points.

Flip, flips the direction. Note that the direction is initially automatically set as the one to the closest distance with the part bodies along the sketch plane normal axis. Users can also flip the direction by clicking on the direction preview arrow.

On Point placement requires 3D work points or sketch points and one direction.

Centers selector, to select the points.

Direction selector defines the boss direction.

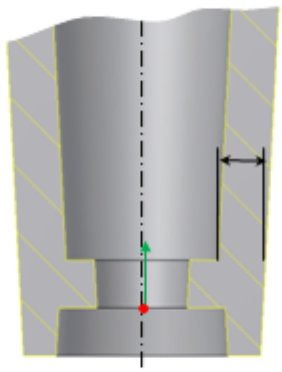
Flip, flips the boss to the opposite direction.

 **Fillet.** Defines the constant fillet radius at the intersection between the boss (and ribs) with the target part body.

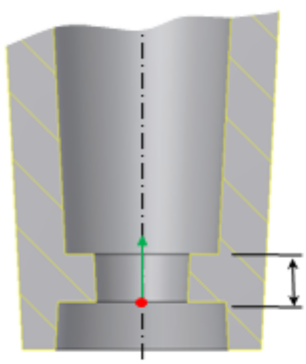
Head tab (Head type, counter bore style)

Specifies the head parameters of the Boss feature for a counter bore style of fastener.

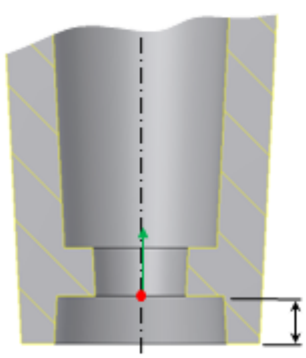
Wall thickness



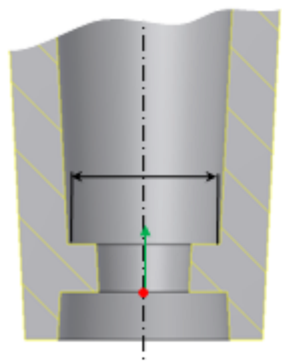
Shank height



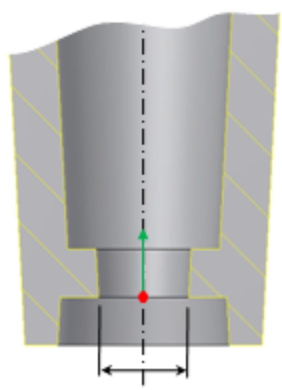
Clamp height



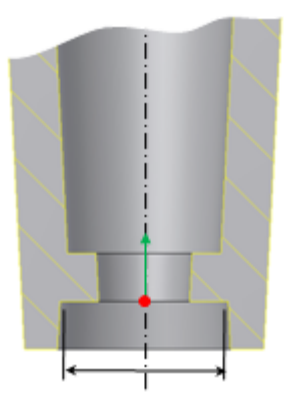
Head diameter



Shank diameter

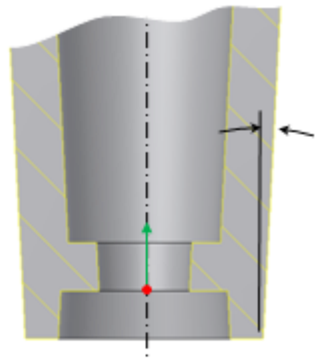


Clamp diameter

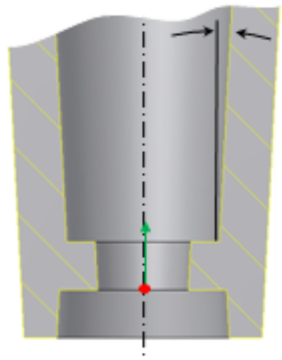


Draft options

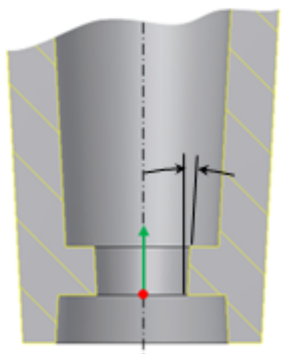
Outer draft angle



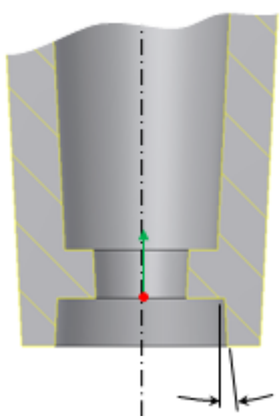
Inner draft angle



Shank draft angle



Clamp draft angle

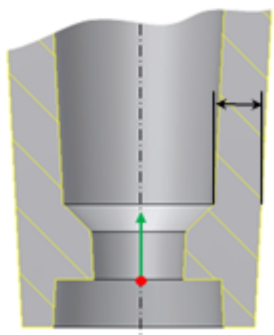


Head tab (Head type, countersink style)

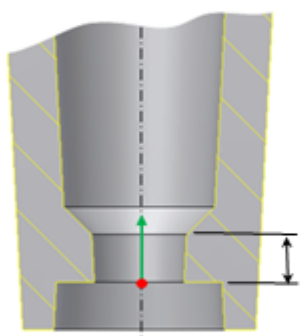
Specifies the head parameters of the Boss feature for a countersink style of fastener.

Countersink angle

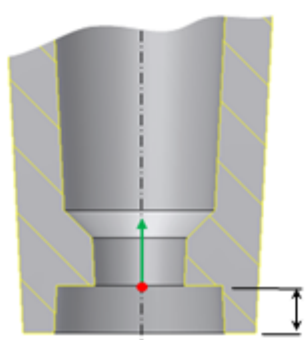
Wall thickness



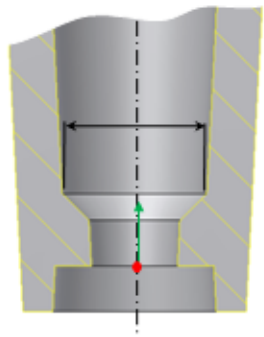
Shank height



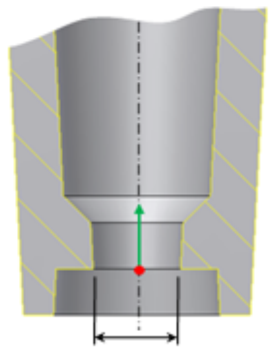
Clamp height



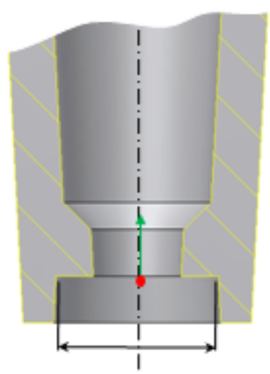
Head diameter



Shank diameter

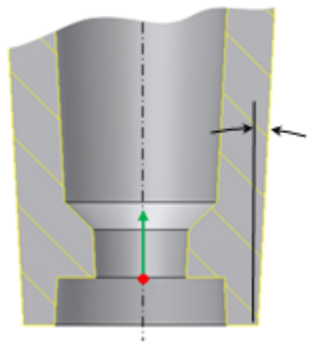


Clamp diameter

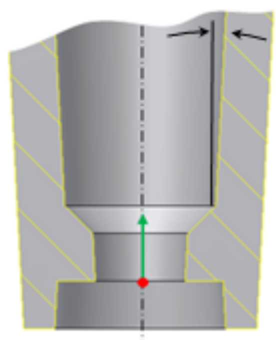


Draft options

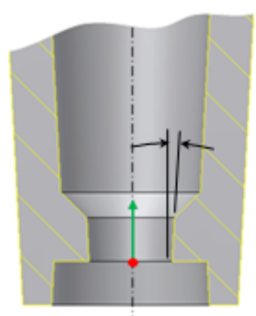
Outer draft angle



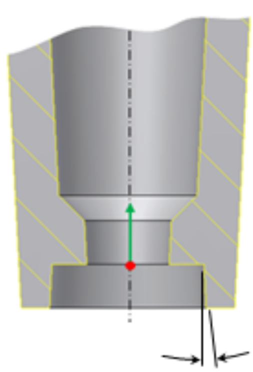
Inner draft angle



Shank draft angle



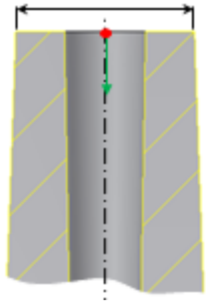
Clamp draft angle



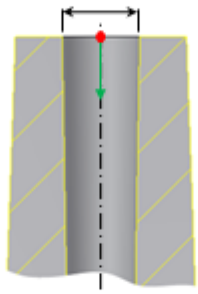
Thread tab (Thread type)

Specifies the thread boss parameters of the Boss feature.

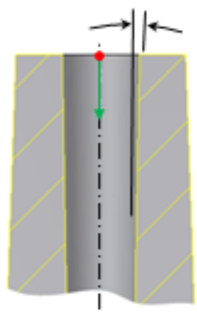
Thread boss diameter



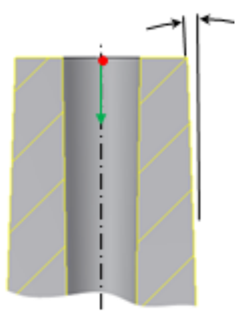
Thread boss hole diameter



Thread boss inner draft angle

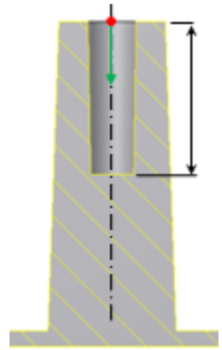


Thread boss outer draft angle

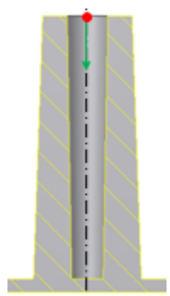


The thread boss can have three types of hole (**hole check mark “On”**):

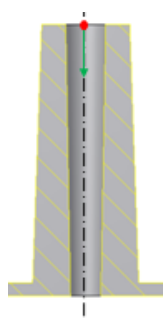
Depth the user is required to enter the hole depth



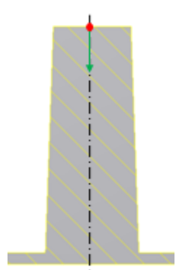
Full Depth the hole is automatically extended to the internal wall face, this is the best practice to avoid sink marks in molded plastic parts.



Through the hole cuts the entire part



If the **hole check mark** is **off** the thread boss can model an ejector pin



Ribs tab

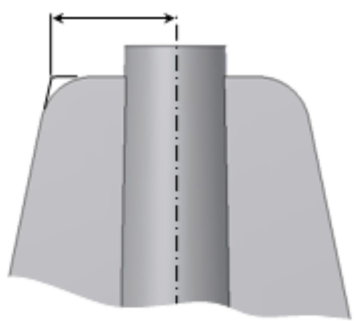
Stiffening ribs are optional, if the **Stiffening Ribs** check mark is on all the controls to input the Ribs parameters become active.

Number of stiffening ribs indicates the number of ribs surrounding each Boss.

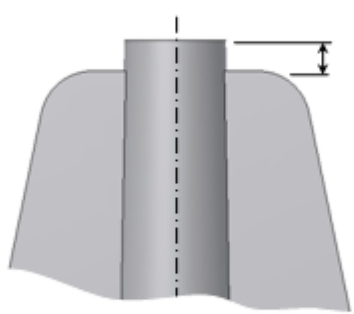
Rib thickness

Rib draft

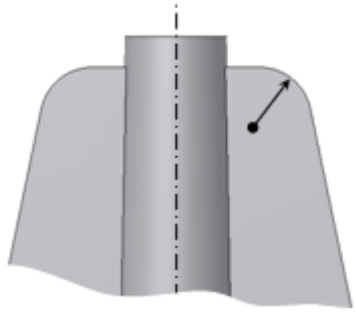
Shoulder length



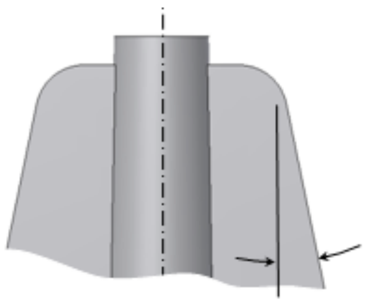
Shoulder top offset



Shoulder radius



Shoulder flare angle



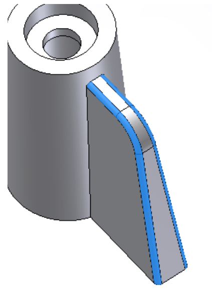
Start angle it's the first rib inclination angle with respect to the sketch Y axis (when Placement is On Sketch), or with respect to the projection of the start direction on a plane normal to the Boss direction (when the Placement is On Point).

Start direction the selector is active when the Placement is On Point; use this selector to define the reference direction of the first rib.

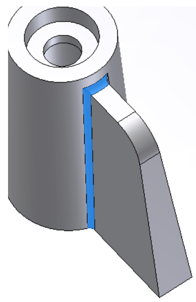
Flip to flip the direction of the first rib reference axis.

Rib Fillet options

Rib fillet radius on the edges of the rib.



Rib blend radius at the intersection between the ribs and the boss.

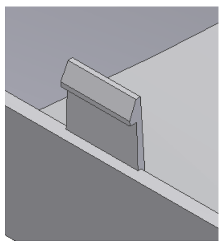


Working with Snap Features

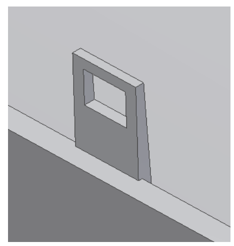
Concept

A common joining mechanism among plastic parts is the Snap fit. The feature models the most common shapes for a snap fit: the hook and loop styles. It is based on a placement datum (a point on a sketch or a 3D point), two directions and a set of geometric parameters to determine the shape that can also be directly manipulated on the graphics preview.

Hook style:



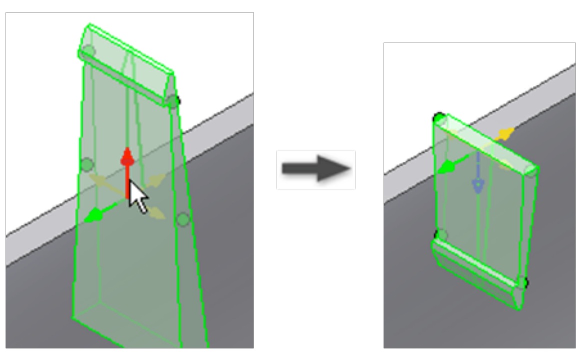
Loop style:



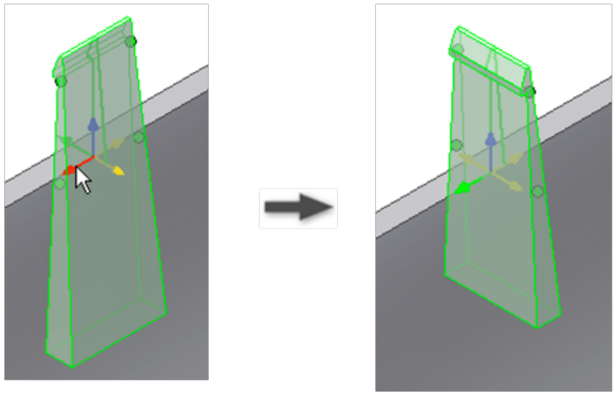
Placement

The *placement* of a Snap fit feature is determined alternatively by the following elements:

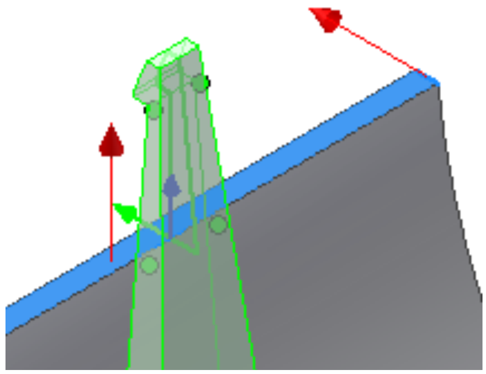
- ***Points on a sketch***, the sketch normal provides the *beam direction*, click on the direction preview arrow to toggle the beam in the opposite direction.



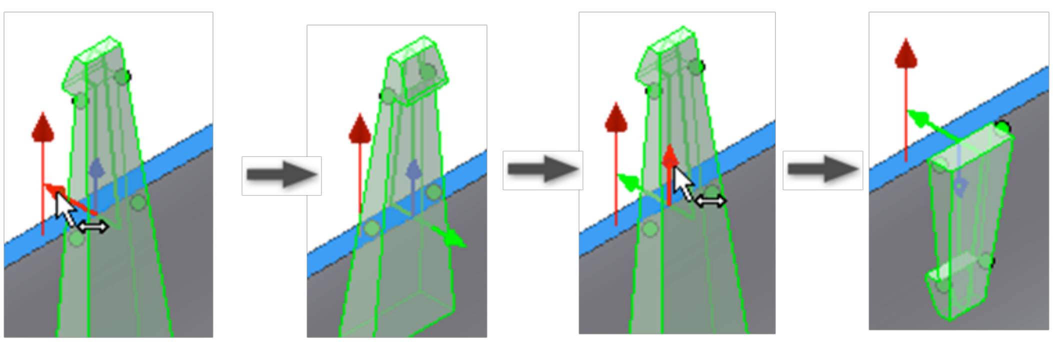
- The hook/catch direction is normal to the beam direction; click the direction preview arrows to switch to the 4 quadrants at 90o.



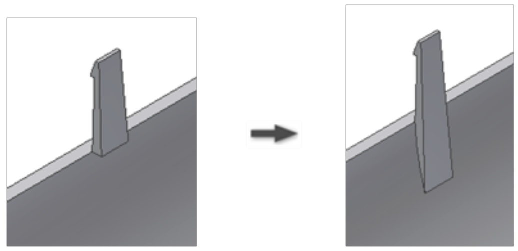
***Work points** or sketch points*, select placement points from different sketches or work points, in this case the sketch normal is not used, and the beams direction and hook/catch direction must be defined by selecting geometric or work elements.



Once the beam and hook/catch directions are chosen the user can still flip the orientation of both by selecting and dragging the arrows.



Finally the user has the option to automatically extend the shape of the beam until the next face.

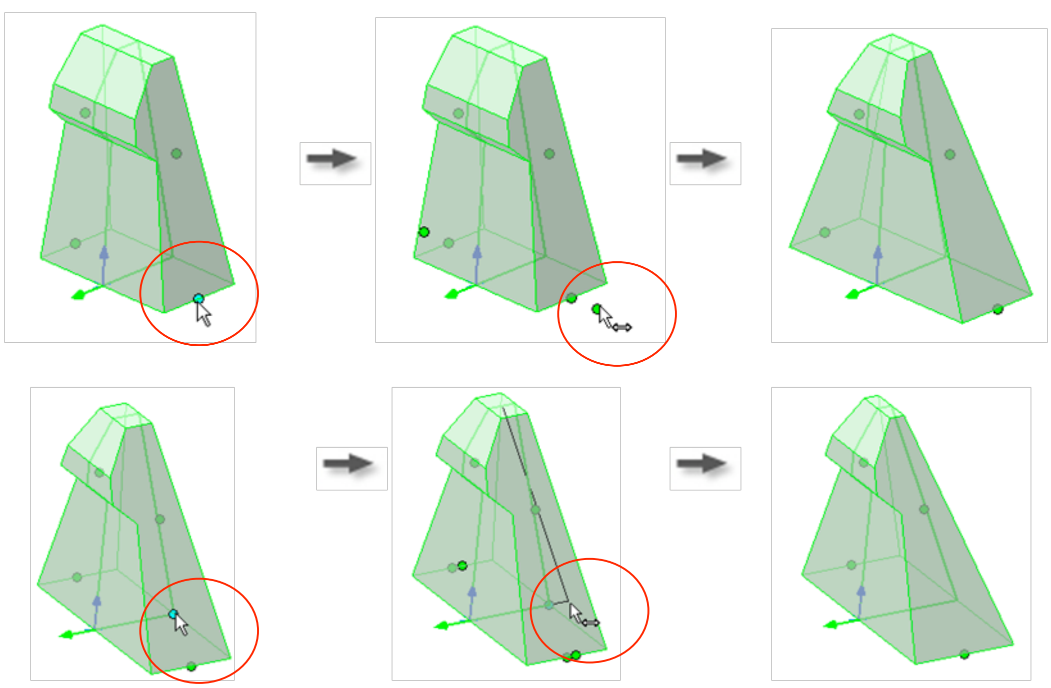


The *dimensions* of the Snap fit feature can effectively be set by use of *manipulators* that are available on the preview to interactively define the geometric parameters when precise dimension input is not used.

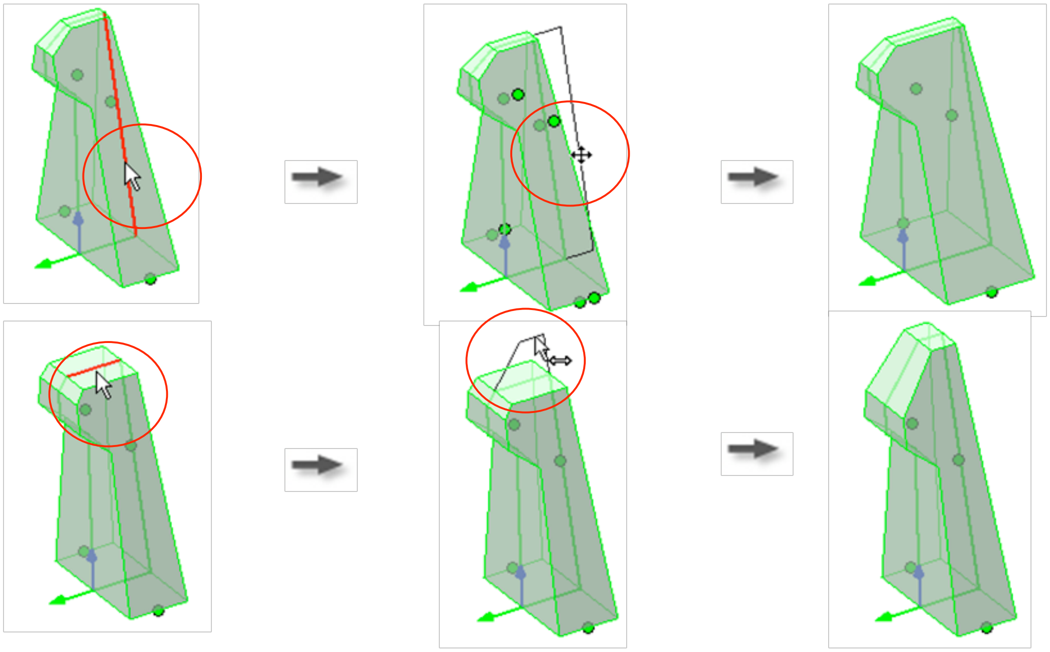
At preview the cross section of the Snap-fit and a few dots are shown, hovering over the cross section highlights the segments that can be dragged, hovering on key points will show additional dots that can be dragged, the reason why some dots remain hidden is to avoid excessive cluttering of the preview. So the manipulators interaction may require some discovery action from the user, mostly done by hovering the pointer over the preview, but is very simple and intuitive and fast to get used to.

There are two types of manipulators:

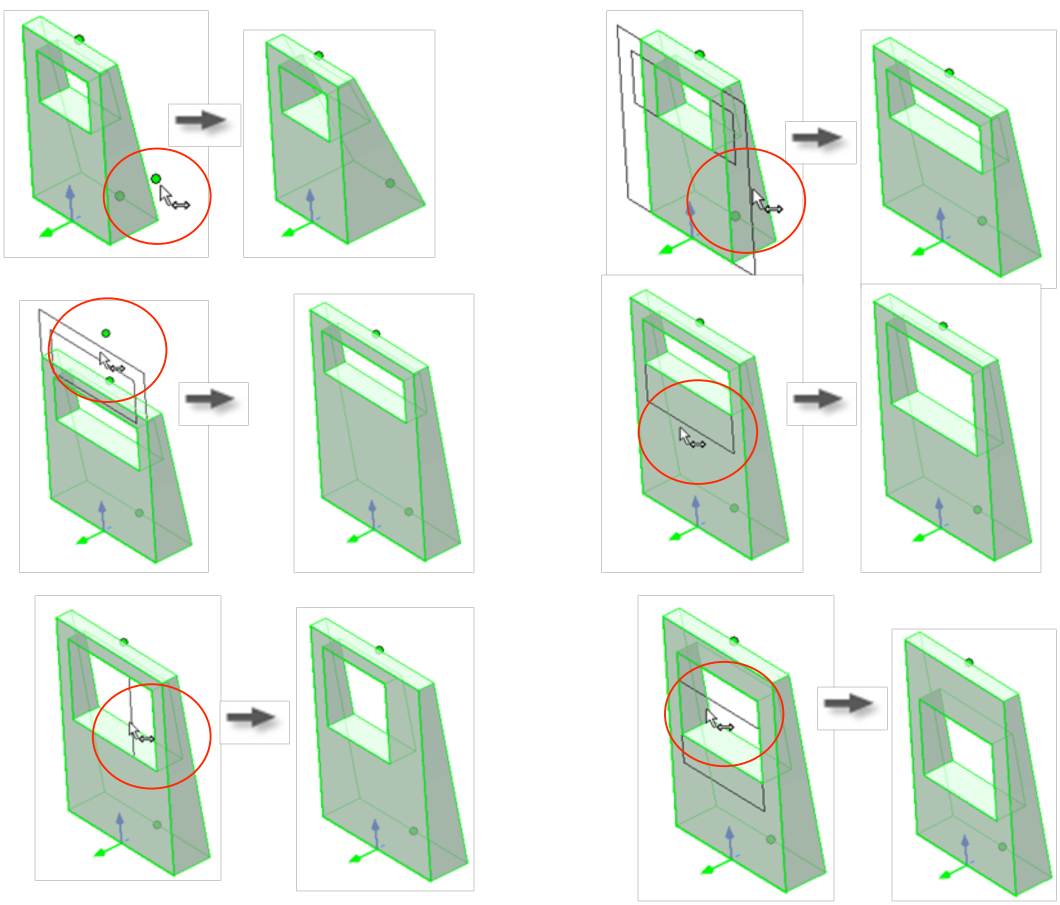
- *Dot manipulators*, they define some of the angular and linear parameters of the snap-fit, in the pictures below there are some examples of how they work.



1. Profile manipulators, to define the linear parameters of the snap-fit.



2. The loop style has similar manipulators.

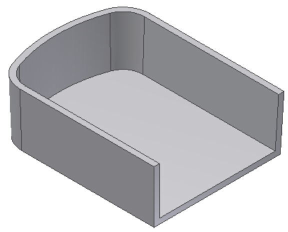


Preview can be turned on/off with the usual check box control.

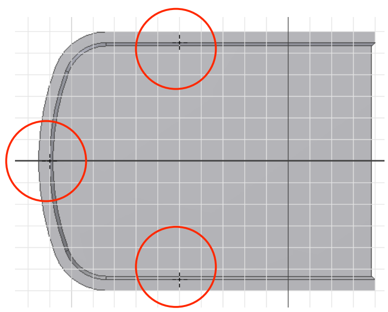
Example

Build a Snap fit feature on a part starting from points on a 2D sketch.

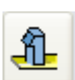
1. Create or import a thin walled part.



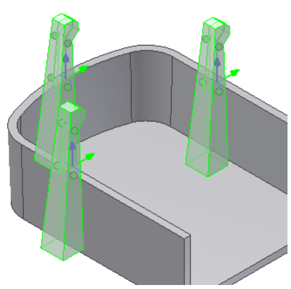
2. Build a 2D sketch with points corresponding to the Snap fit placements.



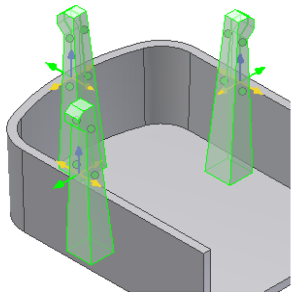
3. Click on the Snap fit tool of the Plastic Features panel and make sure that the Placement method is “From sketch” and the Snap fit style is “Cantilever Snap Hook Join”.



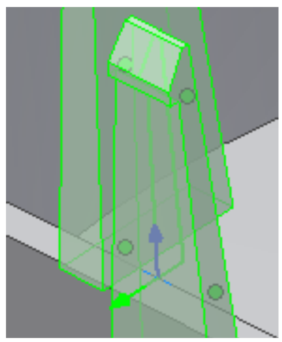
4. Select the points for the placement (ctrl + click to unselect).



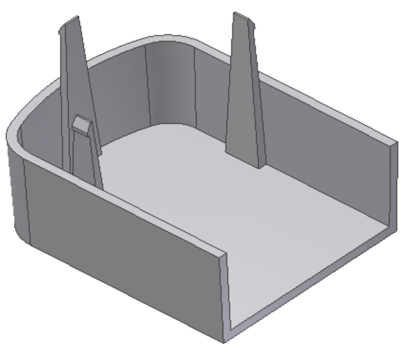
5. Notice that the four possible directions for each Snap fit are aligned with the X Y axis of the sketch plane. Click on the hook direction arrow to properly orient each hook.



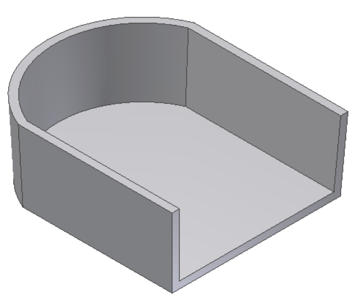
6. Adjust the beam and hook parameters or use the manipulators until the desired shape is reached. Note that the shape manipulators are not active when the Hook direction is down (to facilitate the selection of the directions) .



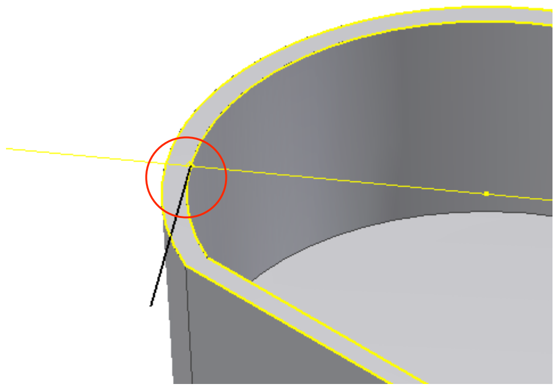
7. Make sure that the Extend toggle is checked and click OK.



Build a Snap fit feature on a part starting from a 3D point.



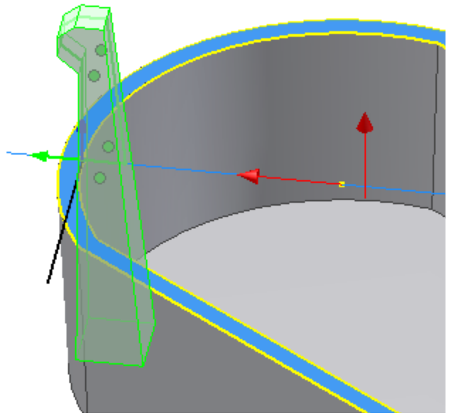
1. Create or import a thin walled part. The goal here is to place a Snap fit feature on the circular edge at a position that is not convenient for the “on sketch” positioning style, for instance at a point on the edge where the hook direction is not likely to be aligned with a sketch X or Y axis.



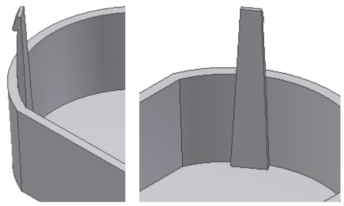
2. Build the construction elements and a 3D work point on the internal wall edge at the position where the Snap fit is desired, for instance the *3D work point* can be the end of a line that is tangent to the internal edge and the hook direction can be a work axis defined by the 3D work point and the center of the circular edge.

3. Click on the Snap fit tool of the Plastic Features panel and make sure that the Placement method is “On Point”.

4. Select the work point defined in step 2 as the *Center*, select the blue face in the picture as the *Direction* and the work axis defined in step 2 as the hook direction.



5. Adjust the shape parameters as desired and click OK.



Reference

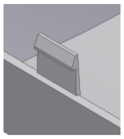
Creates a Snap fit feature using points of 2D sketches or 3D work points and direction elements.

Access:

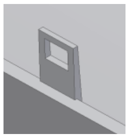
Click the Snap fit tool on the Plastic Features panel bar.

Style specification

Cantilever Snap Hook style



Cantilever Snap Loop style



Shape tab

Common to both styles. Specifies the placement of the Snap fit.

Placement

From sketch placement requires points on a sketch plane (**Centers**).

 **Beam Direction** flips the direction (that initially corresponds to the sketch plane normal).

Hook/Catch Direction shows 4 arrows to choose from at 90^{o}.

On Point placement requires 3D work points or sketch points (**Centers**), and two directions.

Direction defines the beam direction.

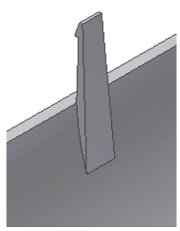
 **Beam Direction** flips the direction opposite to the vector defined by the Direction selection.

Hook/Catch direction selector specifies the direction of the hook/catch

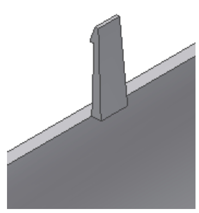
 **Hook direction** shows 4 arrows to choose from at 90^{o}.

Extend the check box specifies whether the beam should:

Extend to next:



Stop at the point:



Beam tab (Cantilever Snap Hook style)

Specifies the beam parameters of the Snap fit feature.

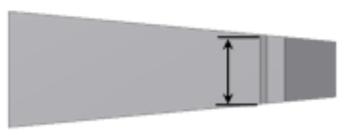
Beam thickness at the wall

Beam length

Beam thickness at the hook

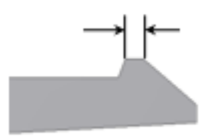
Beam width at the wall

Beam width at the hook

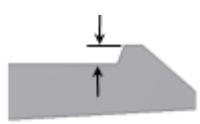


Hook tab (Cantilever Snap Hook style)

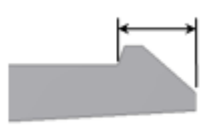
Hook tip length



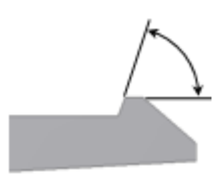
Hook undercut depth



Hook length



Hook retention face angle



Hook insertion face angle

Clip tab (Cantilever Snap Loop style)

Clip length

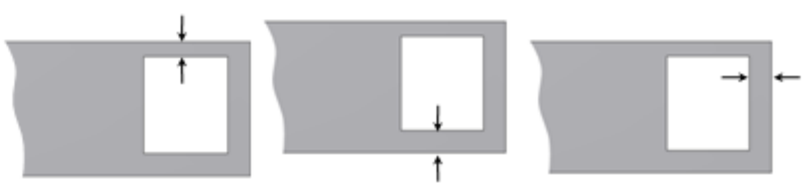
Clip width

Clip thickness at the wall

Clip thickness at top

Clip tab (Cantilever Snap Loop style)

Catch width at the sides and top



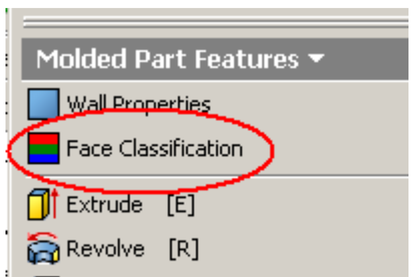
Catch opening length

Working with Face Classification Command

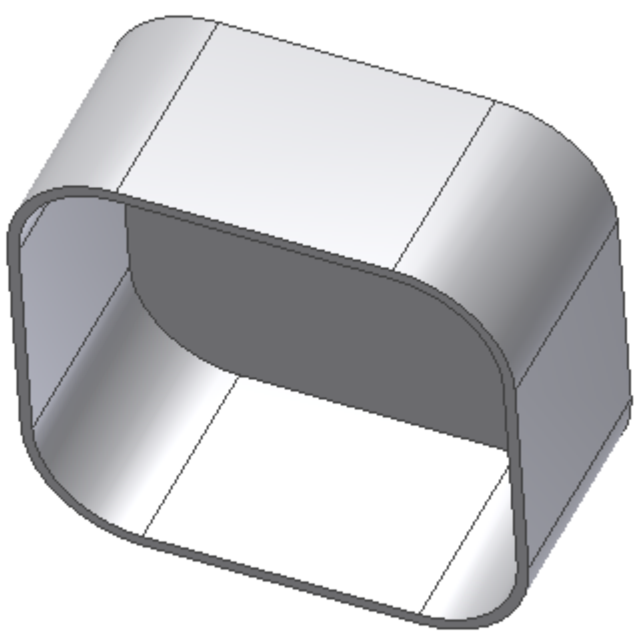
Concept

The Join to Shell functionality in the Plastic Part Environment in Inventor in general is very robust. However, you may encounter cases where unexpected results occur. In most cases, those are caused by complex geometry where Inventor is unable to correctly classify the "inside" and "outside" of the plastic part. As a result, the Plastic Part Environment contains a command that allows you to assist Inventor in determining which faces are "outside" (called "Material Side" in Inventor) and "inside" (called "Material Inside" in Inventor) of the part. Most users should never have to manually override the standard Face Classification for plastic parts.

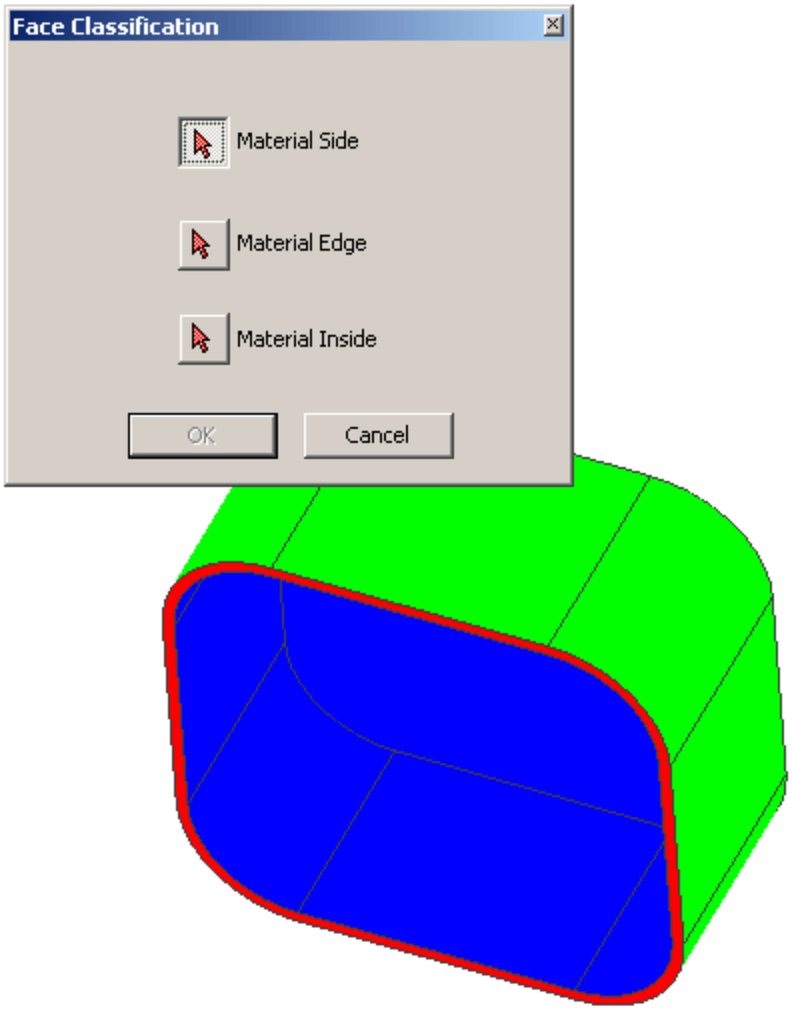
Note that this tool can also be used as an informational assistant – to allow you to examine what Inventor thinks is the inside, outside, and "edge" faces of the part. For instance, if you invoke the Face Classification command, from the Molded Part Features panel bar:



On the simple part below:



You will see the following results:



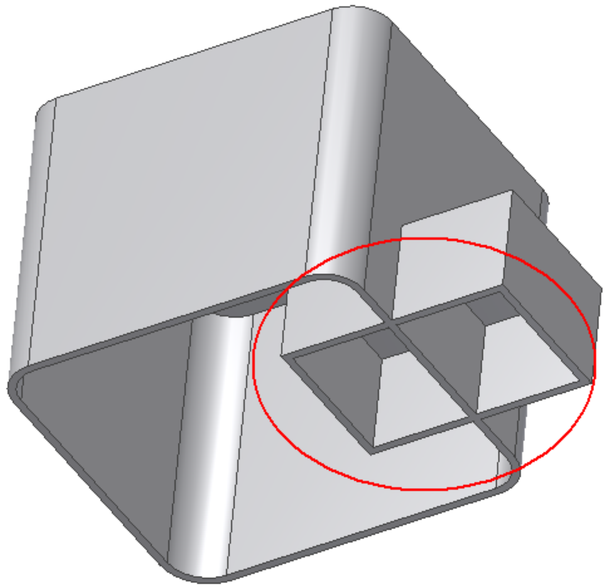
In this display:

Green faces are considered “Material Side” (or the “outside” of the part)

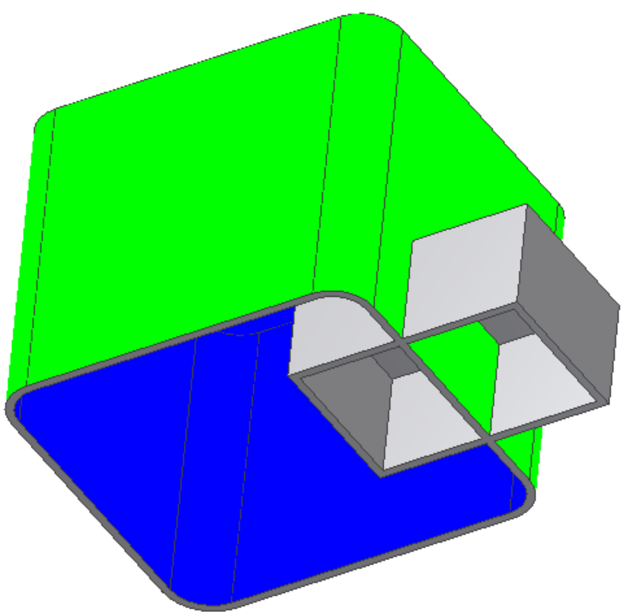
Blue faces are considered “Material Inside” (the “inside” of the part)

Red faces are considered “Material Edge” faces

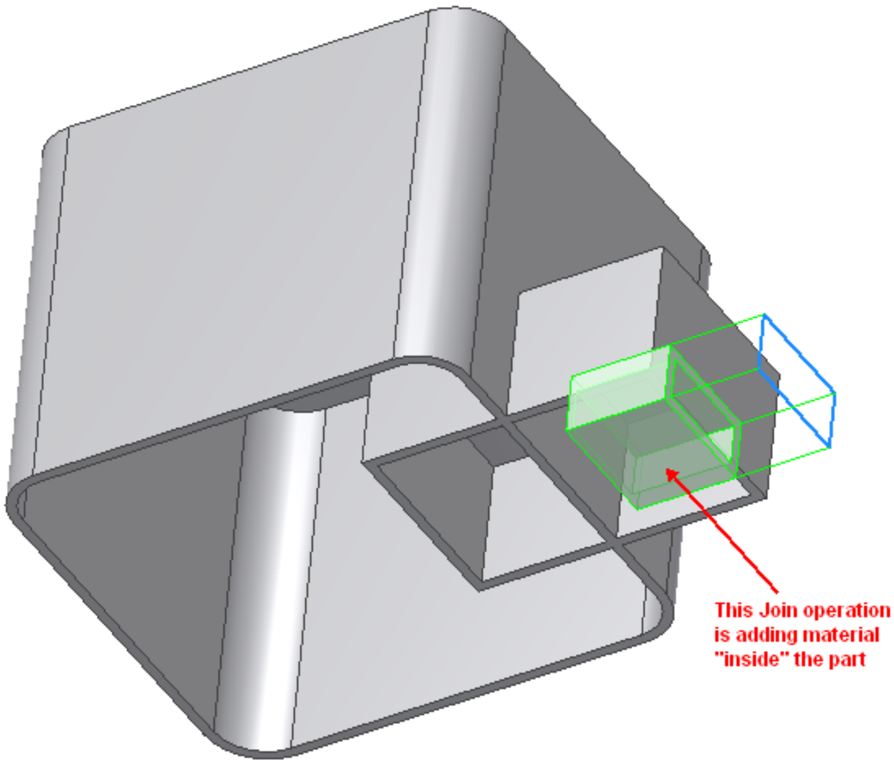
Next, consider an artificial example where Inventor is unable to correctly categorize the faces automatically. This case is not a realistic plastic part, but is show for illustrational purposes:



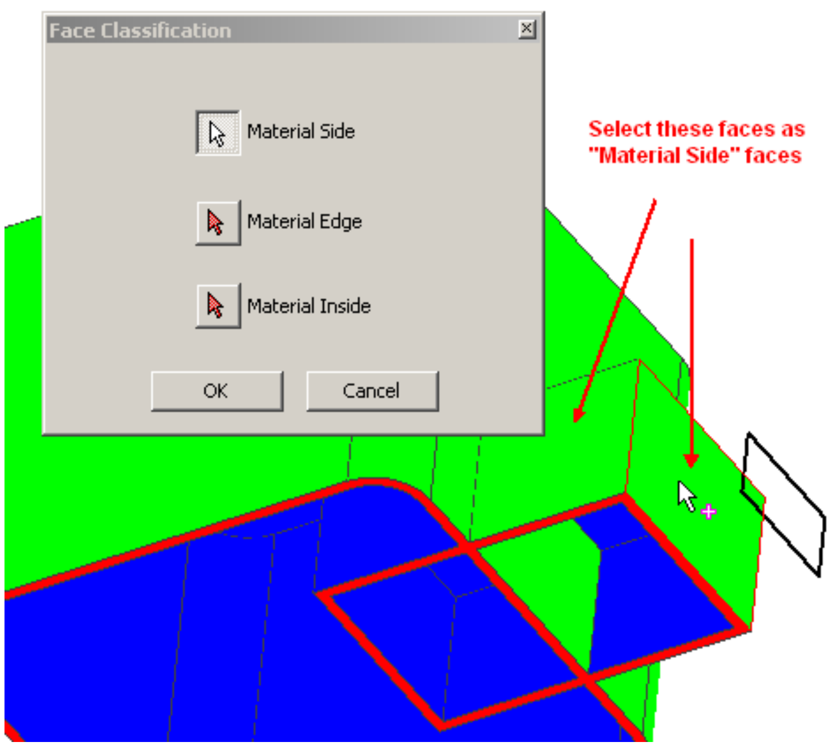
The circled area is ambiguous. It's not clear, even to human observers, which faces are inside, and which are outside. If you bring up the Face Classification command on this part you will see the faces that are ambiguous:



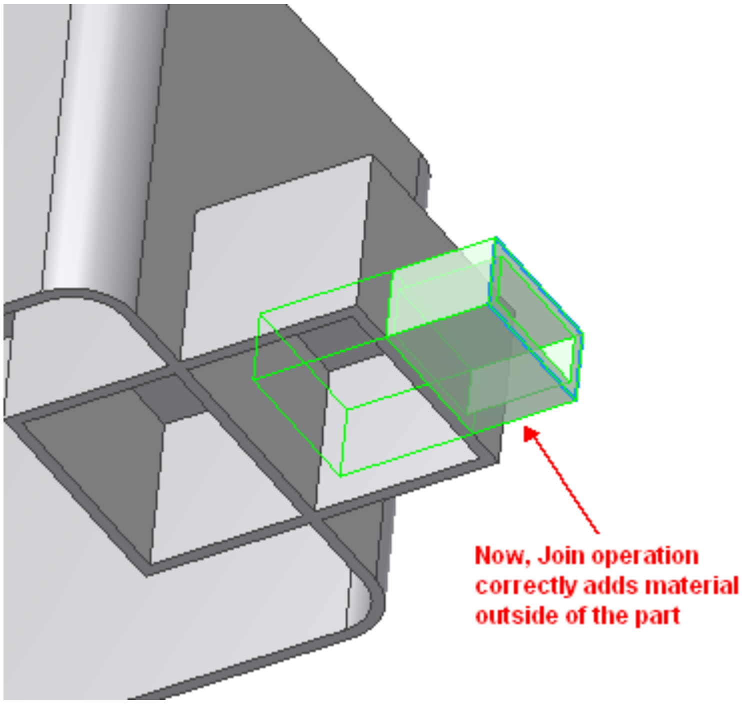
As a result, if you try to use Join To Shell in this area, you will get unexpected results. For example:



However, if you re-classify faces as follows, using the Face Classification command:



The, the Join To Shell extrude produces the expected result:

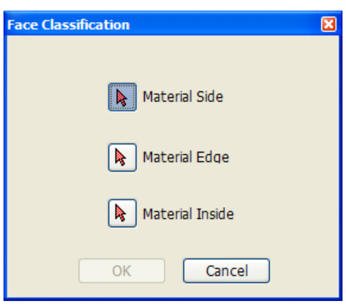


Example

Create “Face Classification” from command bar.

Click on the “Face Classification” of “Molded Part Features” panel.

Click “Material Side” to select outside faces, click “Material Inside” to select inside faces, click Material Edge” to select rib faces



If you think the classification result is correct, click OK.

Reference

Creates a Face Classification feature using three sets of faces. The Face Classification Feature will add property wall properties to the body faces.

Access:

Click on the “Face Classification” of “Molded Part Features” panel.

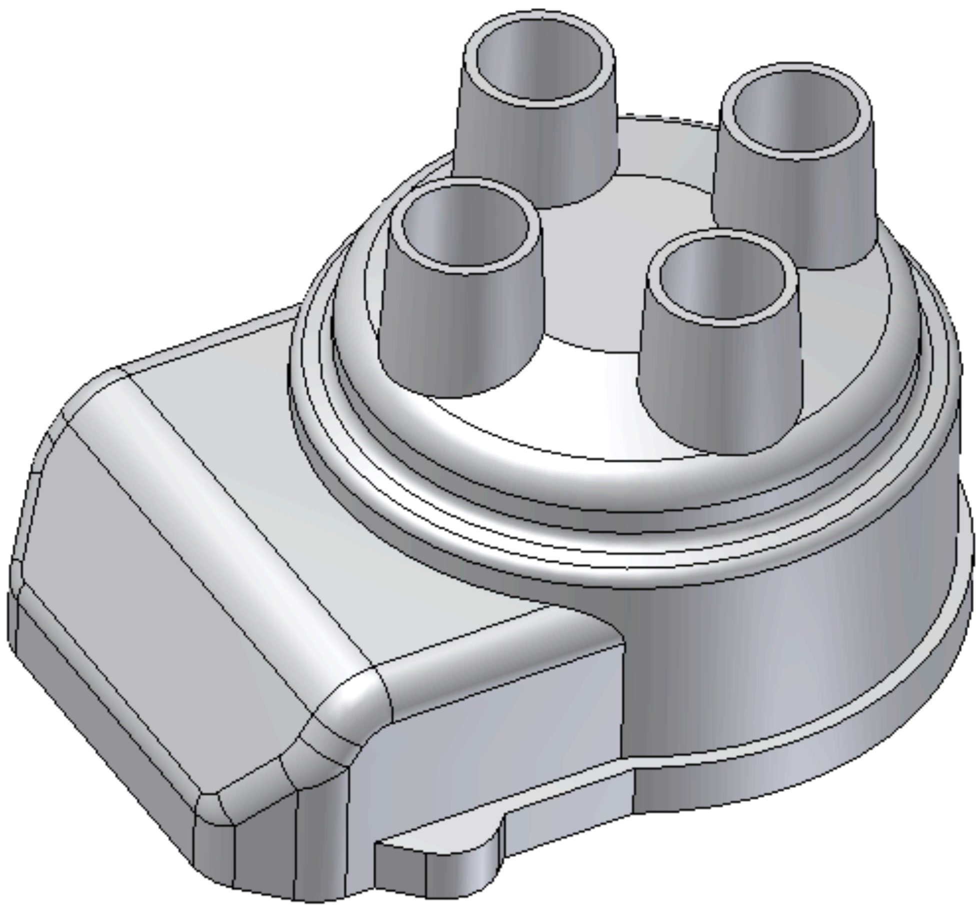
Dialog:

Material Side	To select one or more faces as outside faces.
Material Edge	To select one or more faces as rib faces.
Material Inside	To select one or more faces as inside faces.

Thin Tutorial

Objective

1. The goal of the tutorial is to introduce the new Thin features and “shell as you go” technology in Autodesk Inventor Plastic Features. We will model a simple plastic distributor cap.



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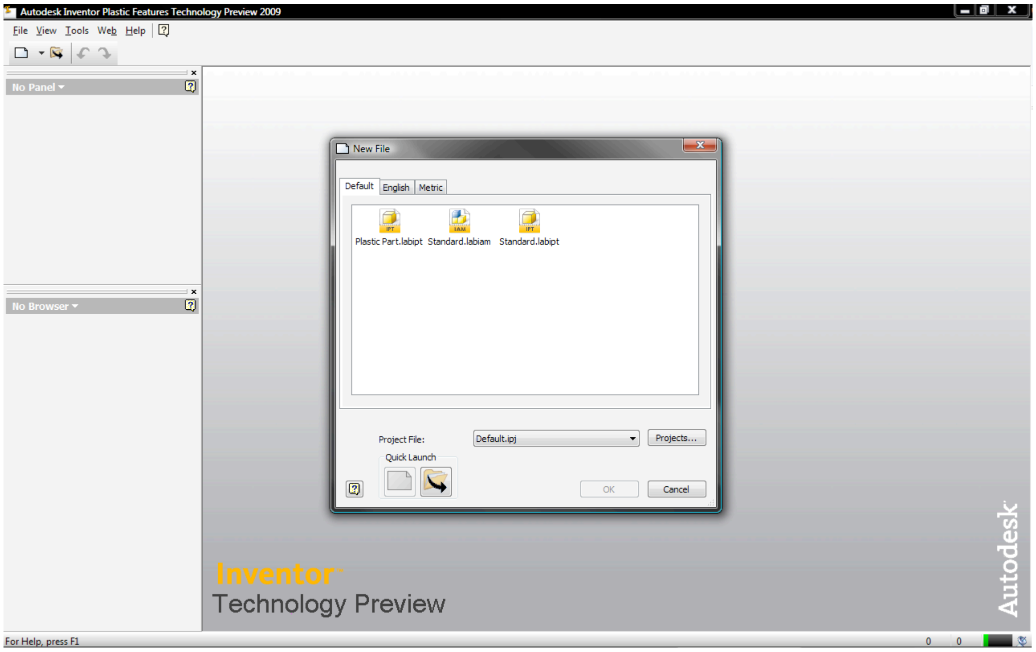
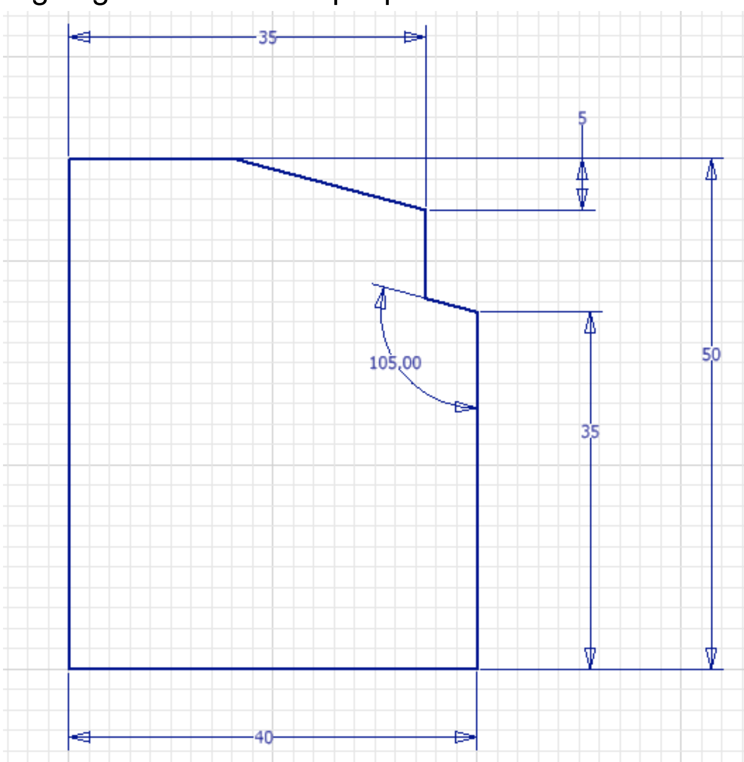
Building the first feature

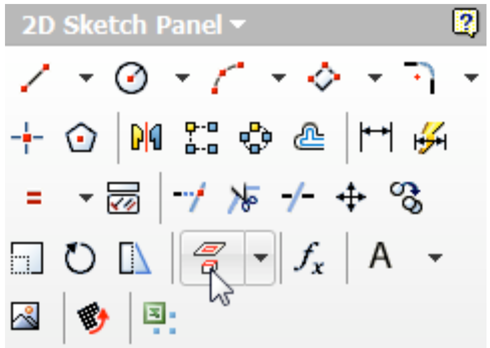
2. Start a new Plastic Part file by using the **File -> New…** menu command. From New File Dialog select the “Plastic Part.labipt” template file.



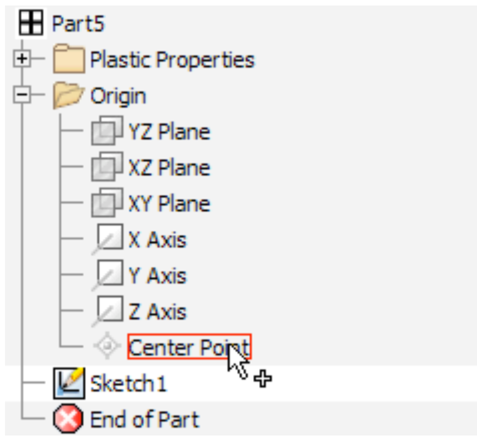
3. We are going to sketch a simple profile for our first feature. A revolve.



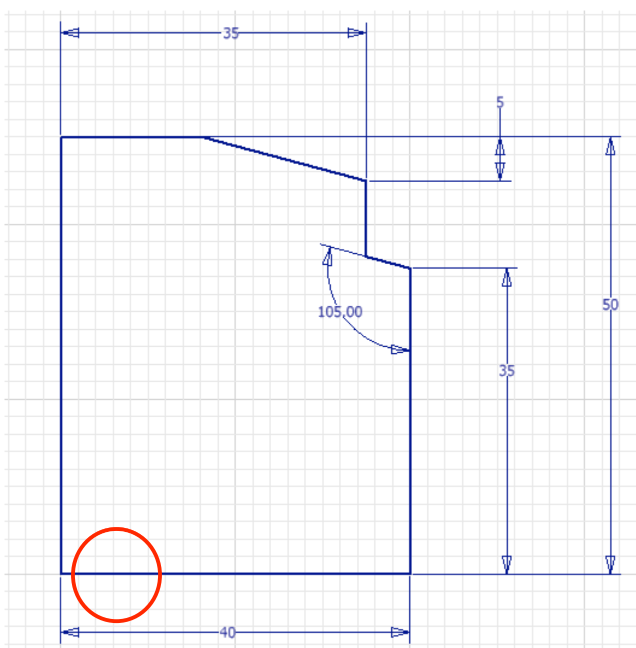
4. To fix our sketch to the origin we need to project it. From the 2D sketch panel select the **Project** command.



5. Select the **origin point** in the browser.

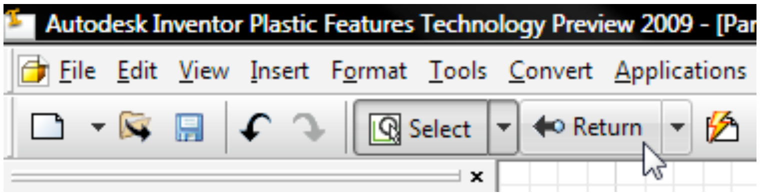


6. Drag the lower left corner of your sketch until it snaps to the project origin point. The sketch should change color to let you know that it is fully constrained.

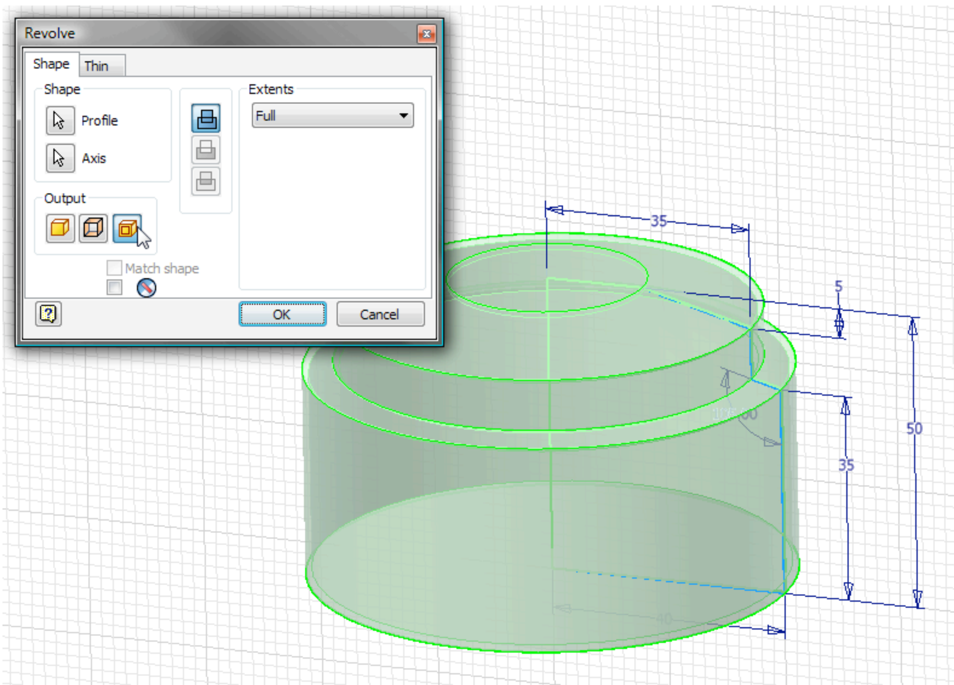


6.

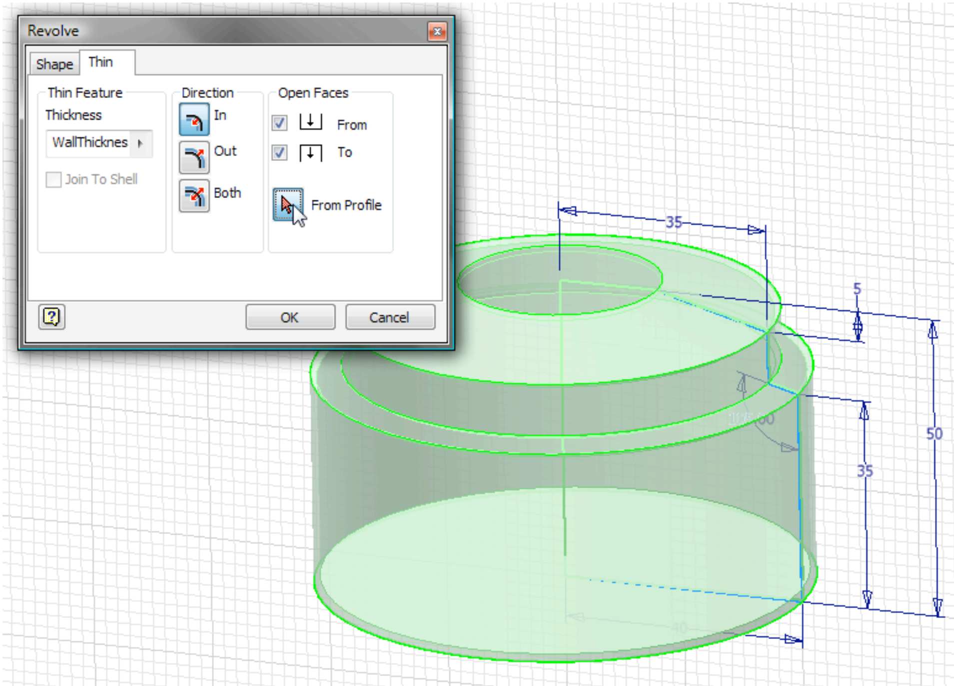
7. We are done sketching so we can pres the **Return** button to exit our first sketch.



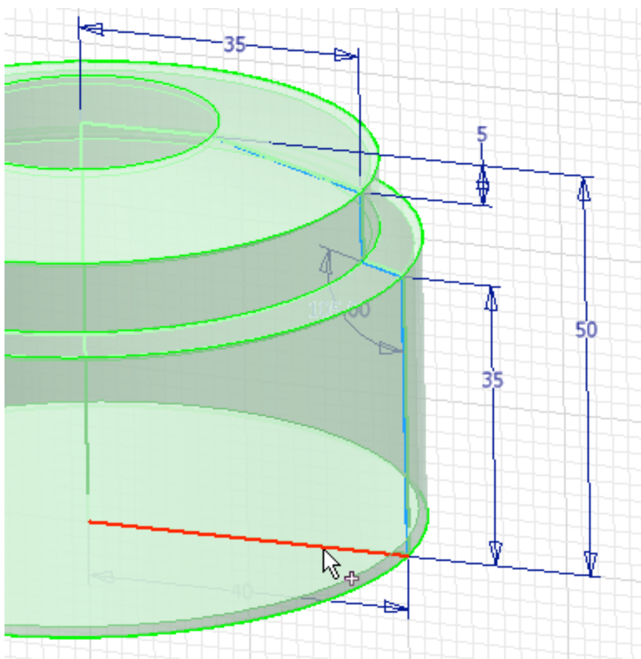
8. From the Molded Part Features panel bar select the revolve feature command. Notice the new Output type for Thin features. (Note: You may want to rotate your model view to get a better view of the revolve preview).



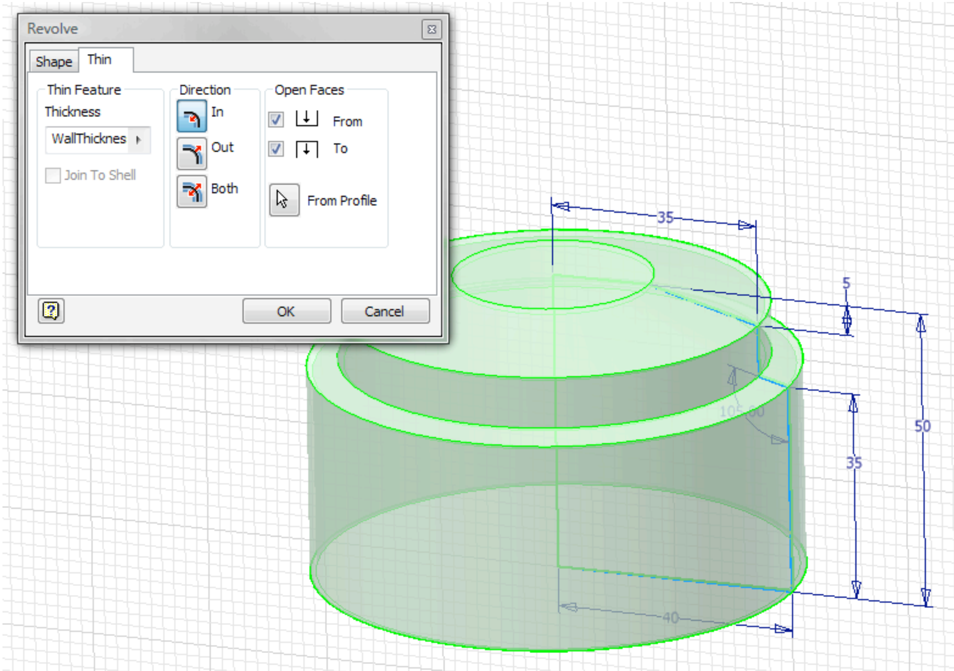
9. In the Revolve dialog switch to the **Thin** tab. These are all the new options for thin features. Notice that the thickness of the Thin feature is automatically tied to a **"WallThickness"** parameter that is set in the Plastic Part template file. We want to create a Thin feature but need to open the bottom of the revolve feature. This is possible using the **From Profile** selector to select the sketch element that represents the face(s) we want to leave open.



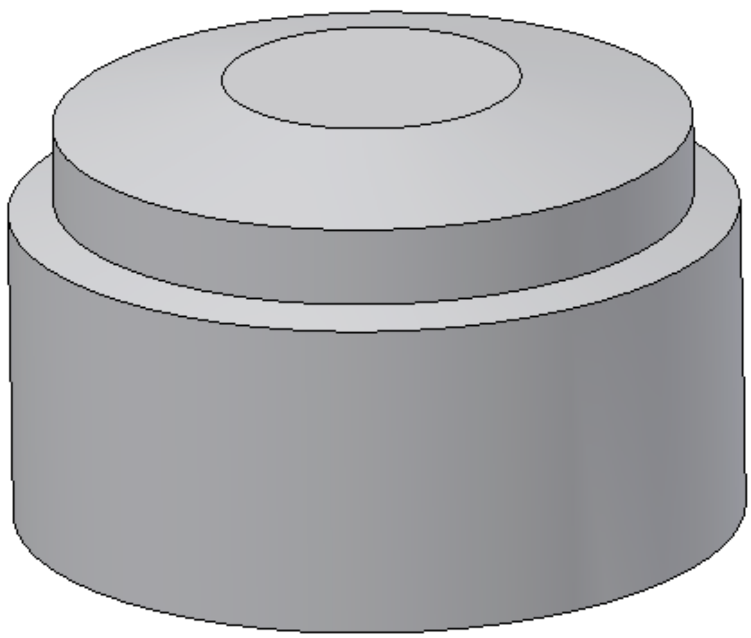
10. Select the bottom line.

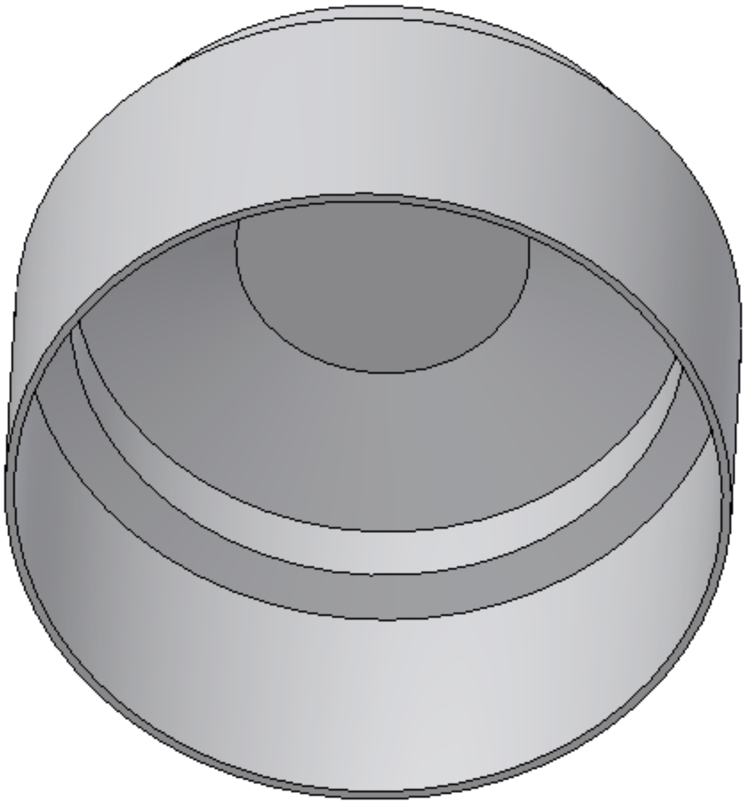


11. The preview updates to show the results.

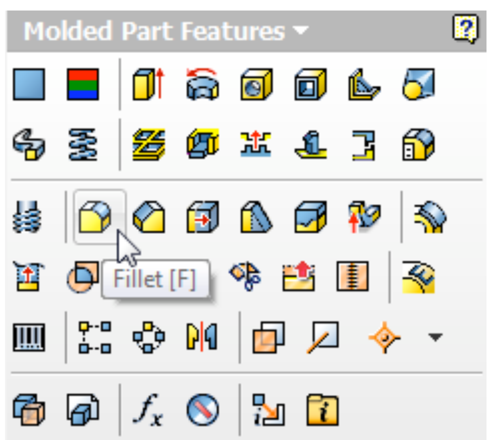


12. Hit **OK** to create your first thin feature

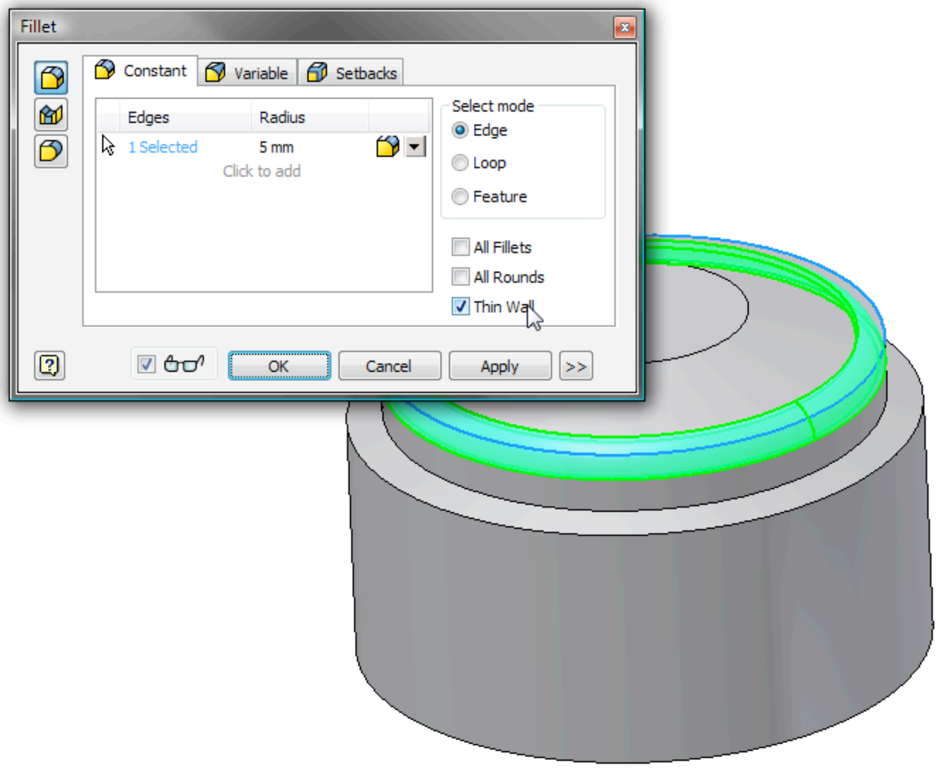




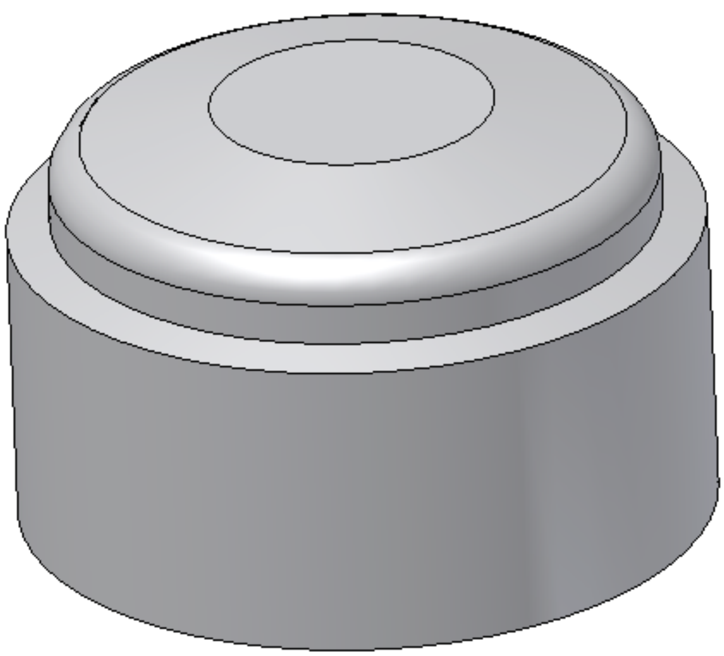
13. Next we need to add some fillets. From the Molded Part Features panel bar, select the **Fillet** Feature Command.

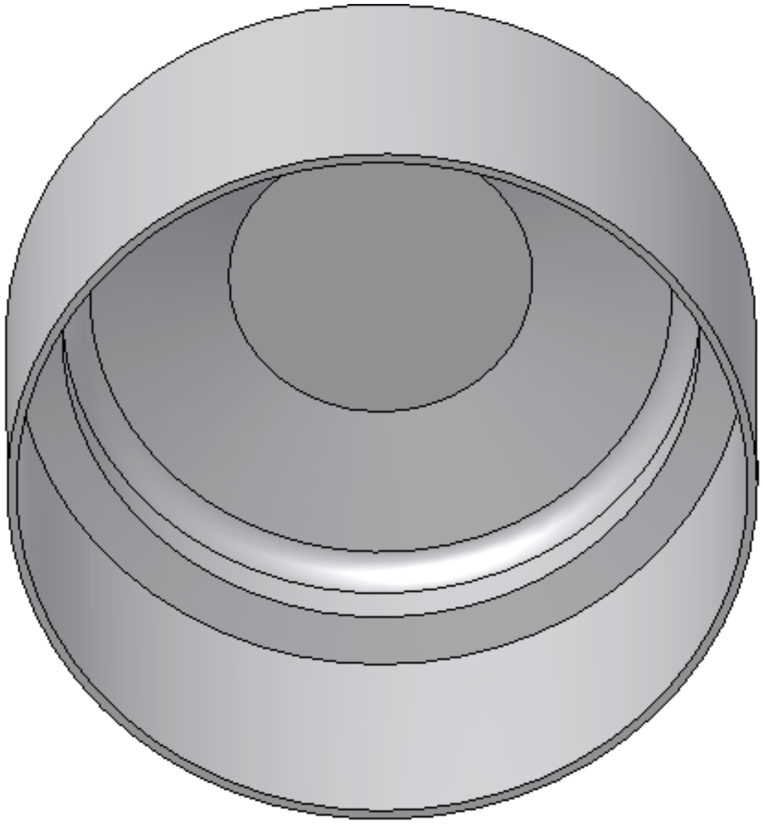


14. Notice the new thin wall check box. Select the edge shown and enter a fillet radius value of **5mm**. Hit **OK**.

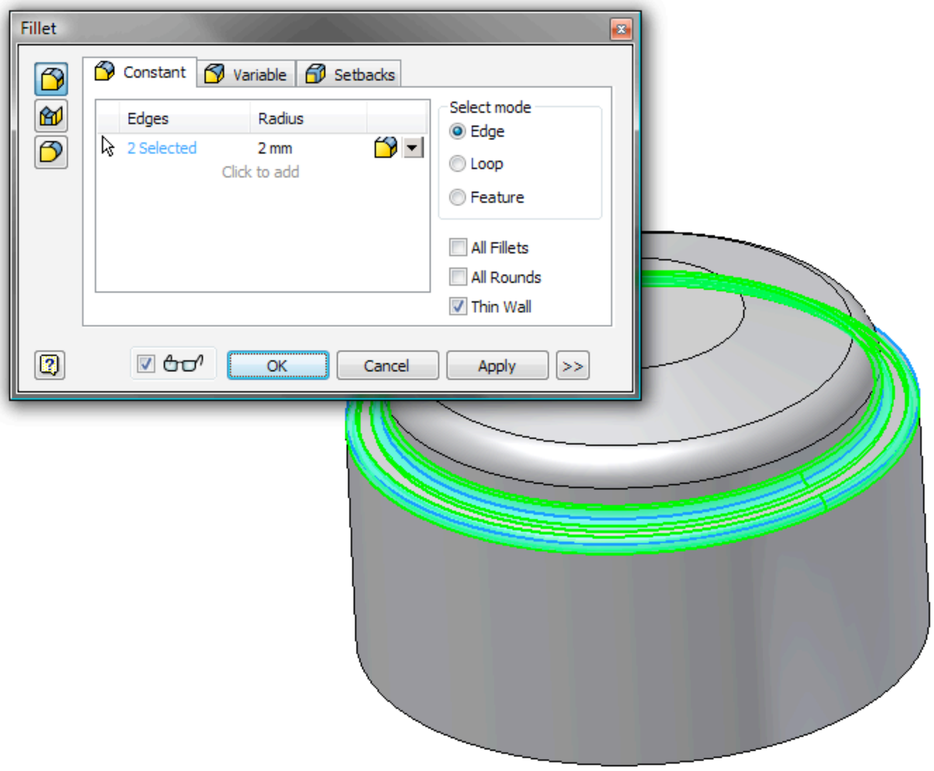


15. The fillet is created on the edge selected with a radius of **5mm a**nd… the matching inside edge is filleted automatically to a radius of 5mm – “WallThichness.” (Note: Wall properties also have radius values for Outer Radius and Inner radius. You could use these named parameters as well).

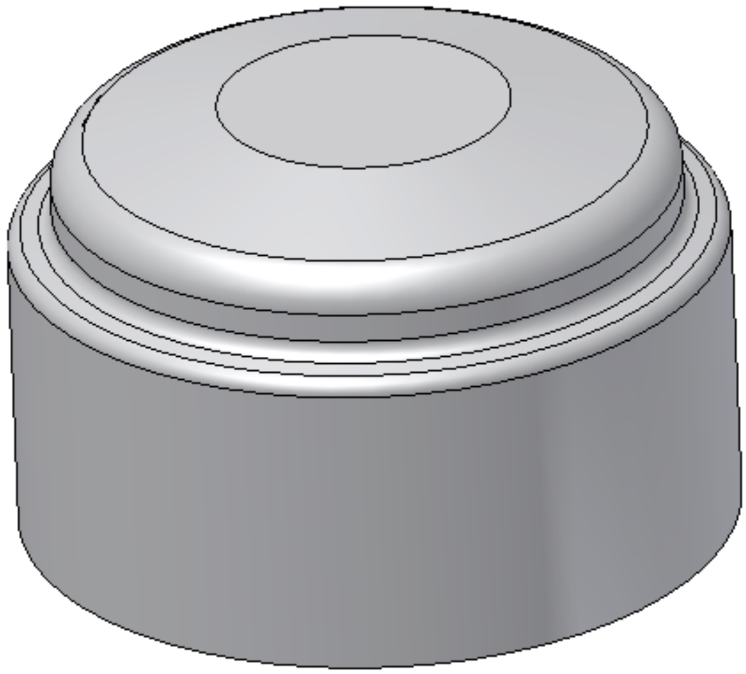




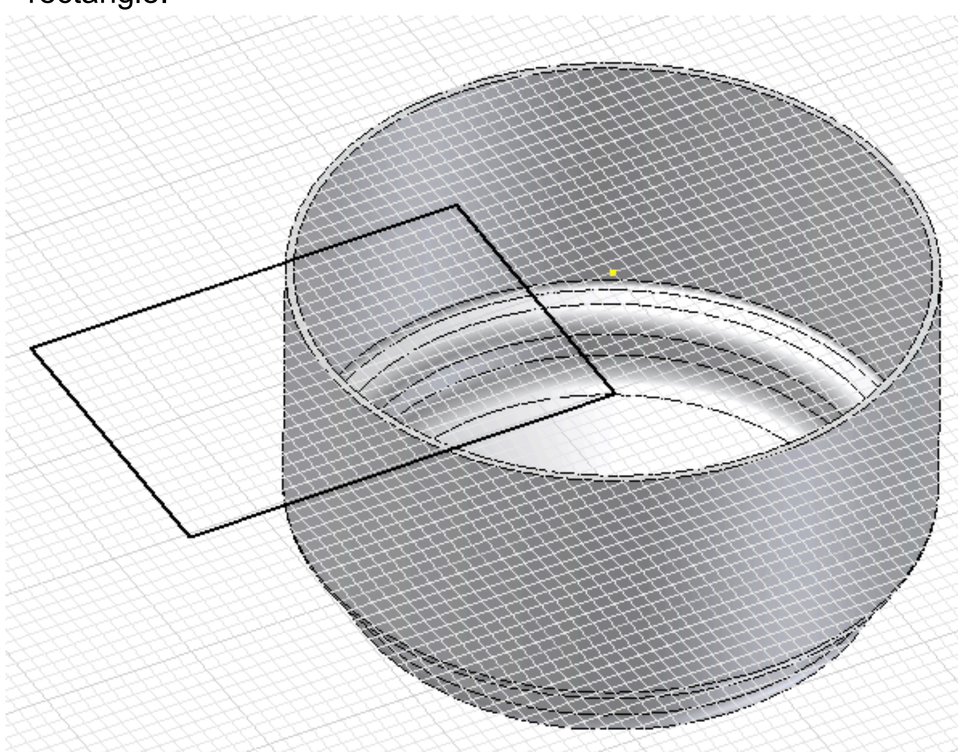
16. Create two additional fillets on the edges shown with a radius of **2mm**.



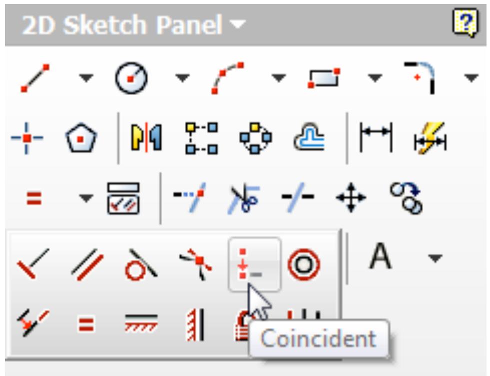
17. Again, the inside and outside fillets are created automatically.



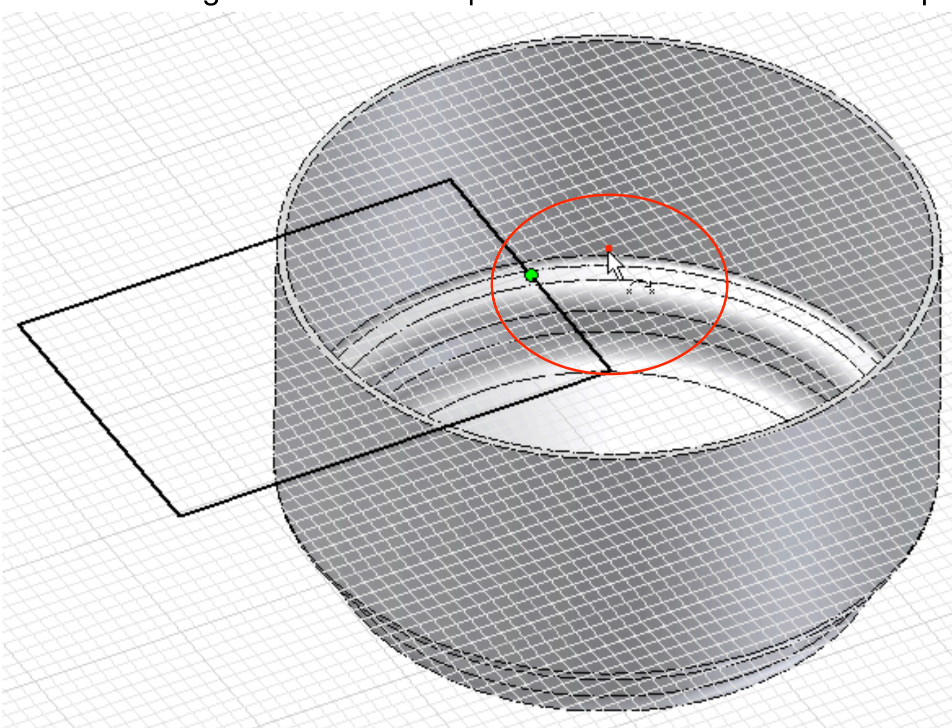
18. Create a sketch on the bottom face of the part as shown. Draw a rectangle.



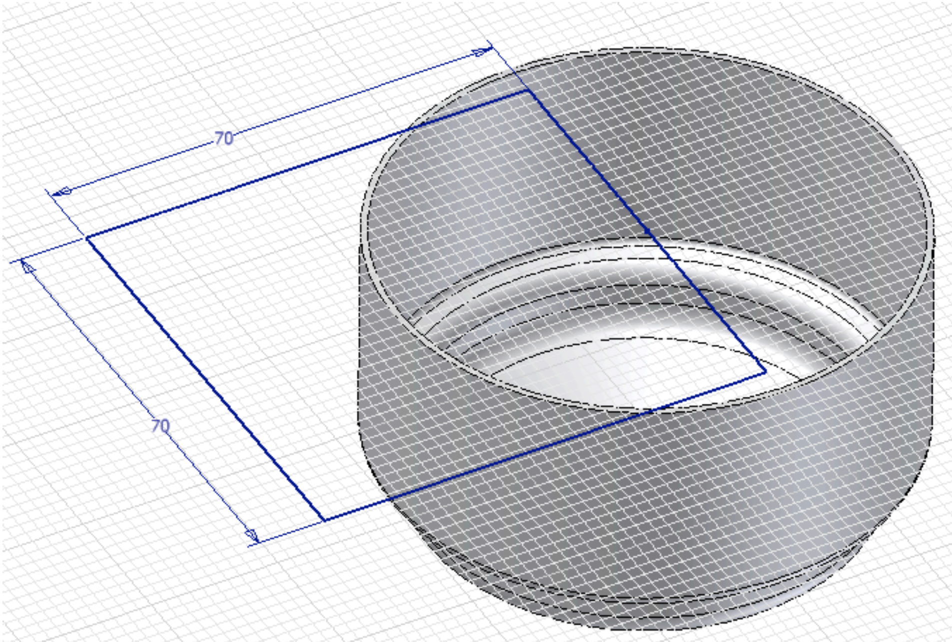
19. From the 2D Sketch panel bar select the **Coincident** constraint from the constraint fly out.



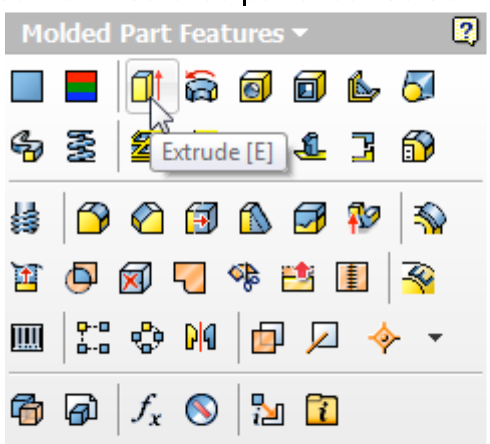
20. Create a coincident constraint between the midpoint of the short edge of the rectangle and the center point of the circular faces of the part.



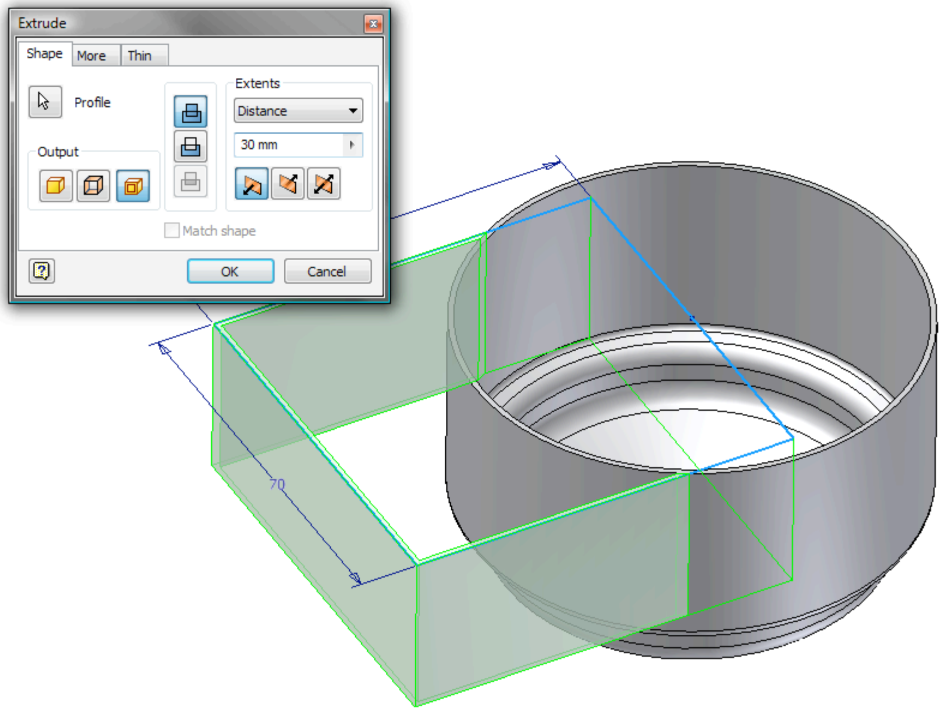
21. Dimension the Rectangle as shown.



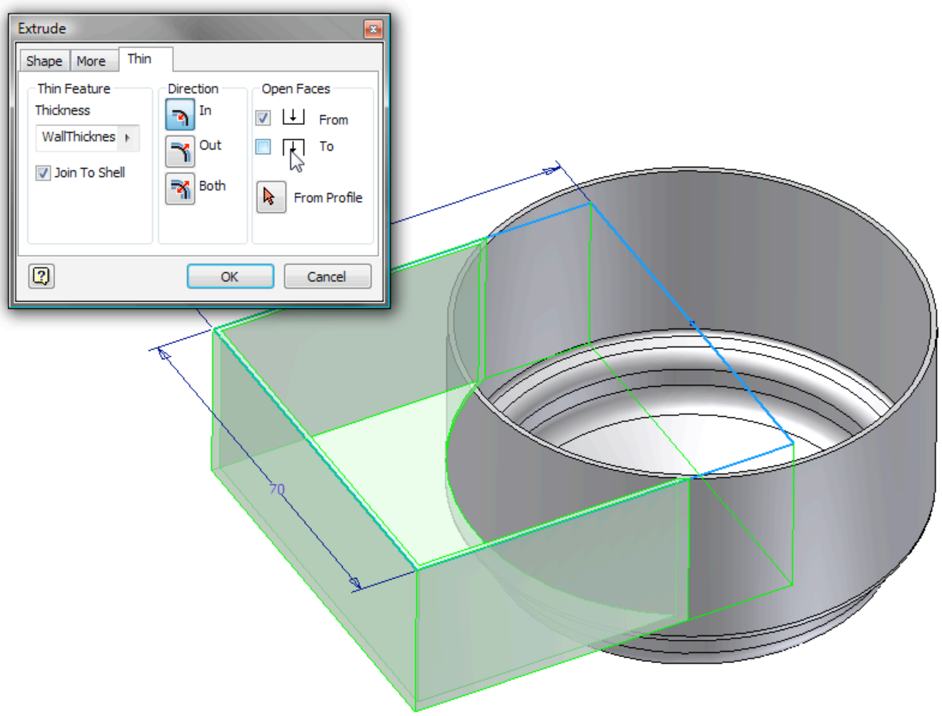
22. From the Molded Part Features panel bar select the **Extrude** feature.



23. Immediately you will see a new preview showing the unique capabilities of thin features. Enter an extrude distance of **30mm**

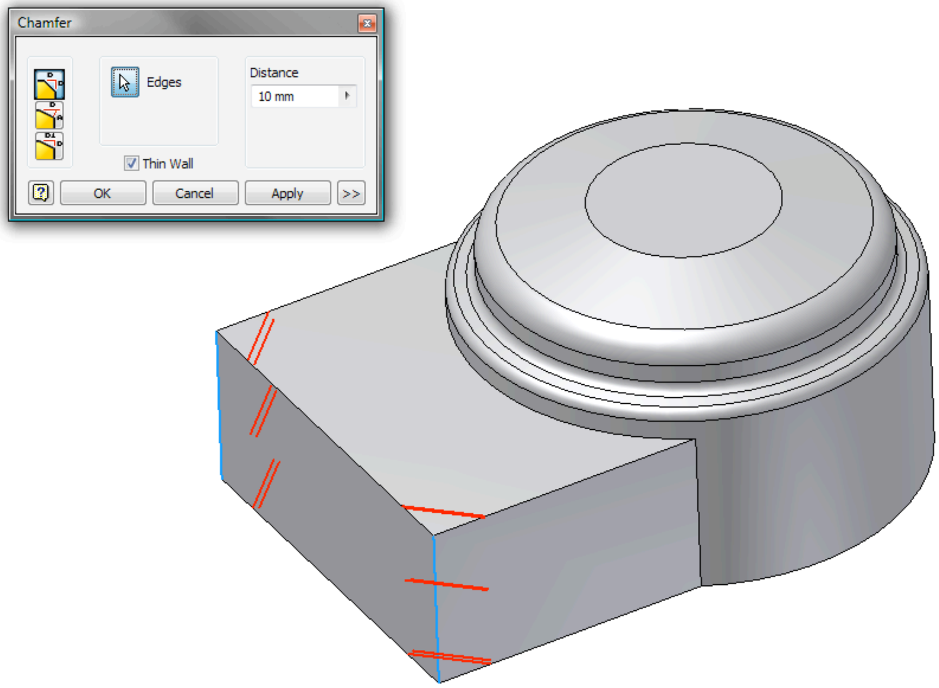


24. Switch to the **Thin** tab and uncheck the "**To**" check box for Open faces. Experiment with these settings to see the results of the preview. Also notice that in the thin features frame there is a new option for Join to shell. This allows you to shell as you go! Be sure to set your options shown before proceeding, and then hit **OK**.

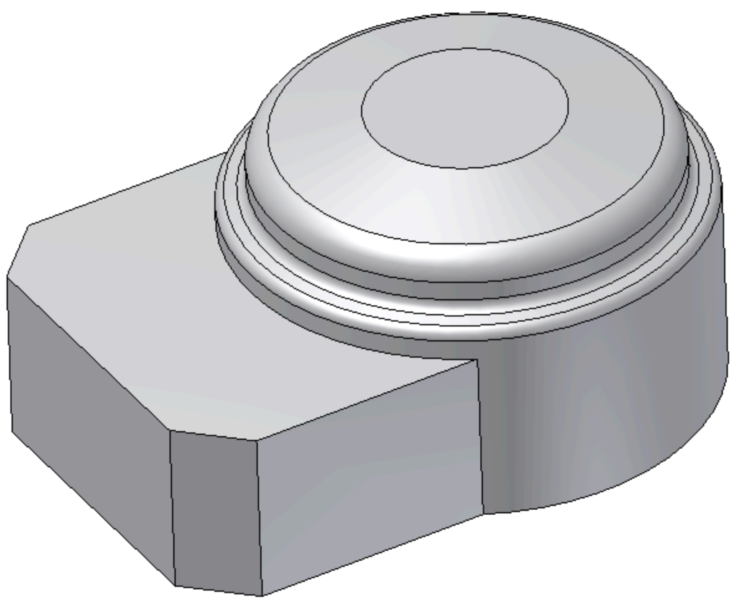


7.

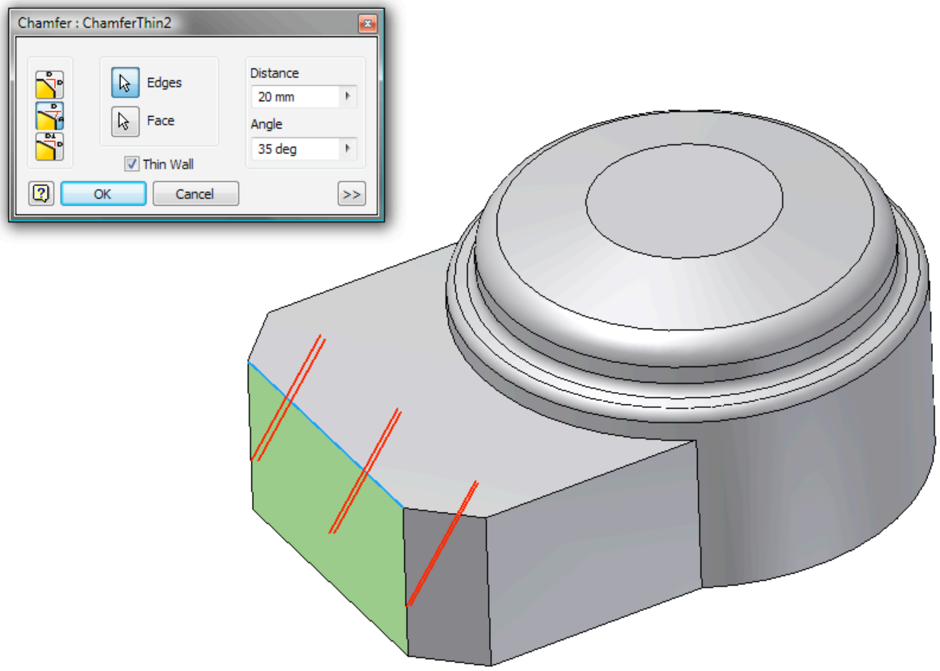
25. Let’s finish shaping the box by using a chamfer feature. Select the two edges shown and enter a chamfer size of **10mm**.



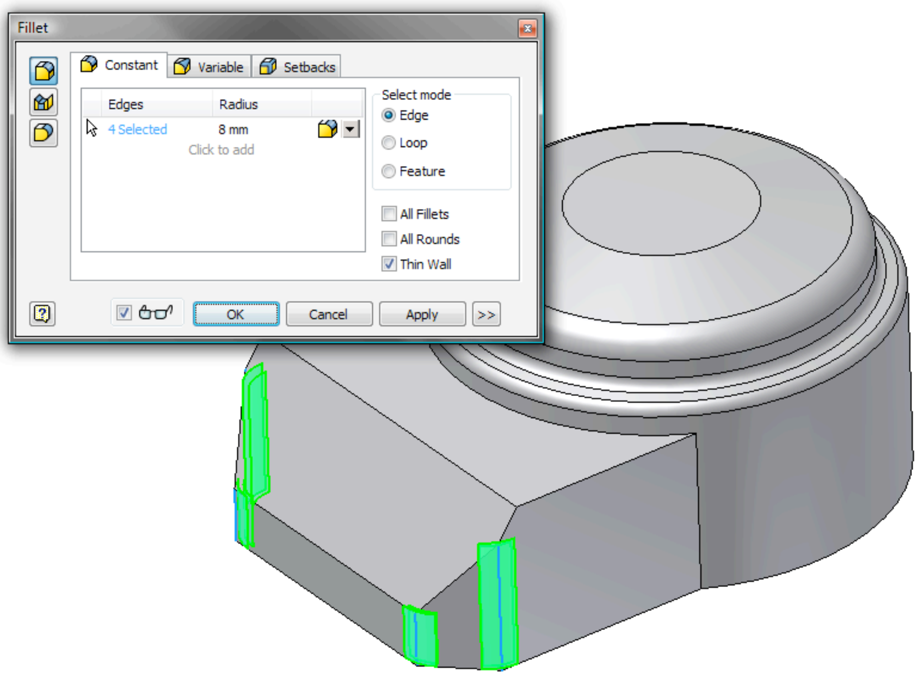
26. Notice that the chamfer is automatically created “thin” like the fillet.



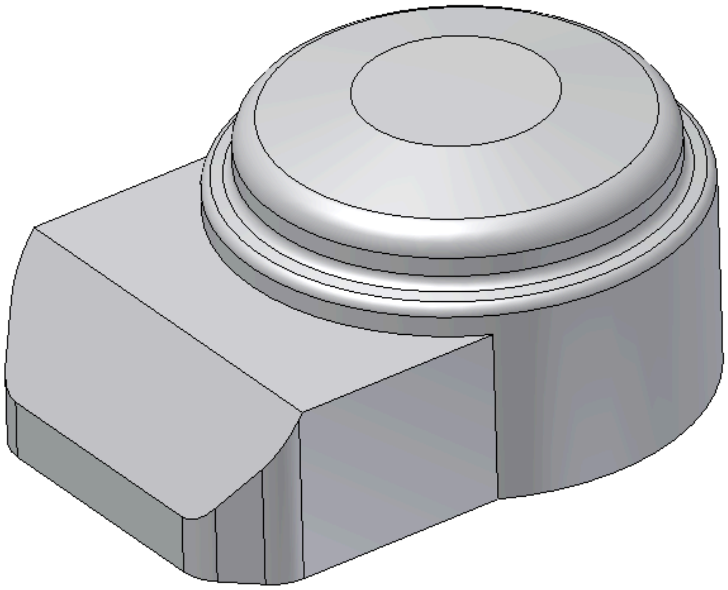
27. Add a second chamfer. Use the “Distance angle” construction option and enter a distance of **20mm** and an angle of **35**. Hit **OK** when ready.



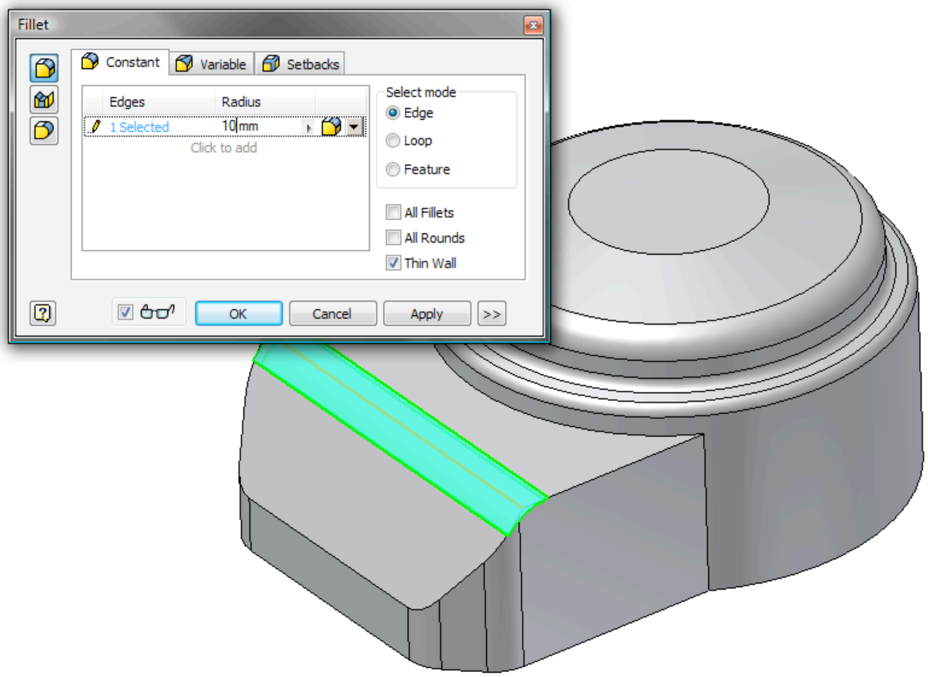
28. We will finish with another set of rounds. The first fillets are **8mm** radius.



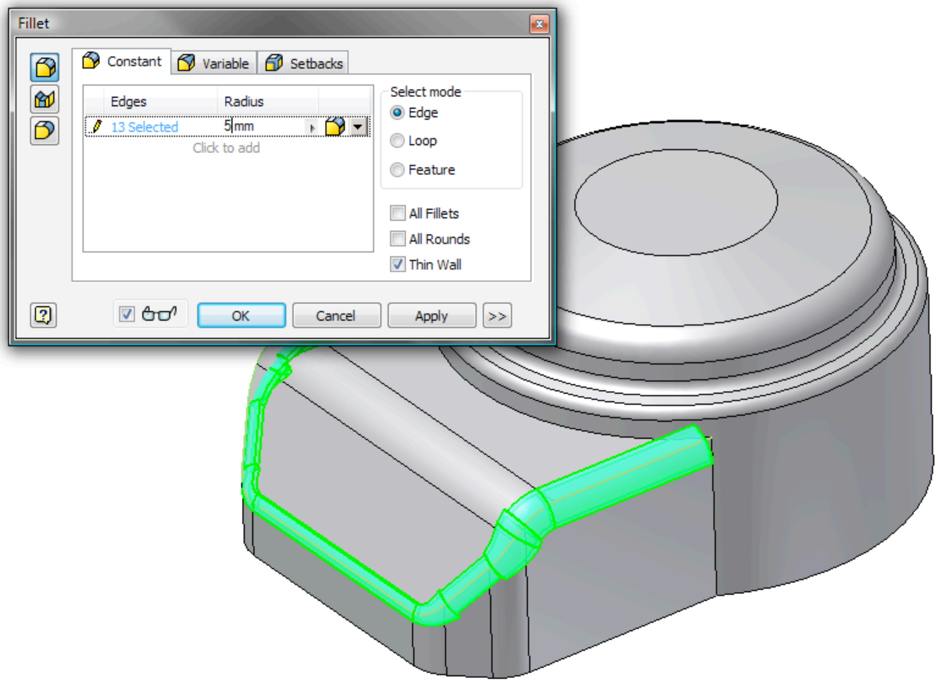
29. Notice all the thin features keep constant wall thickness even thought the geometry is getting more and more complex.



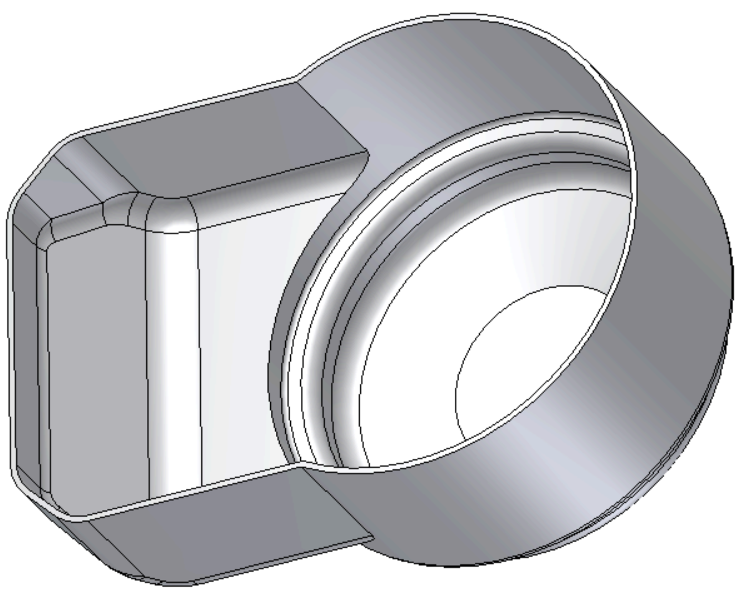
30. The next fillet is **10mm**.



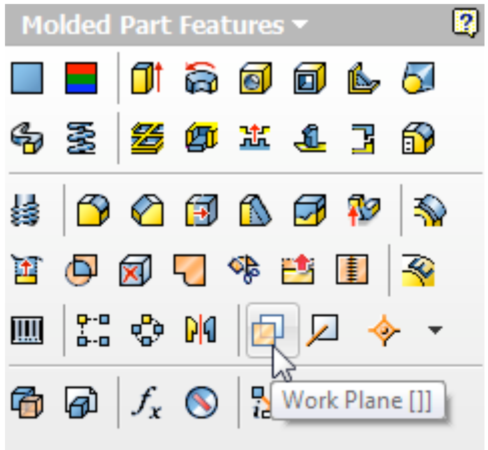
31. And the final fillet is **5mm**.



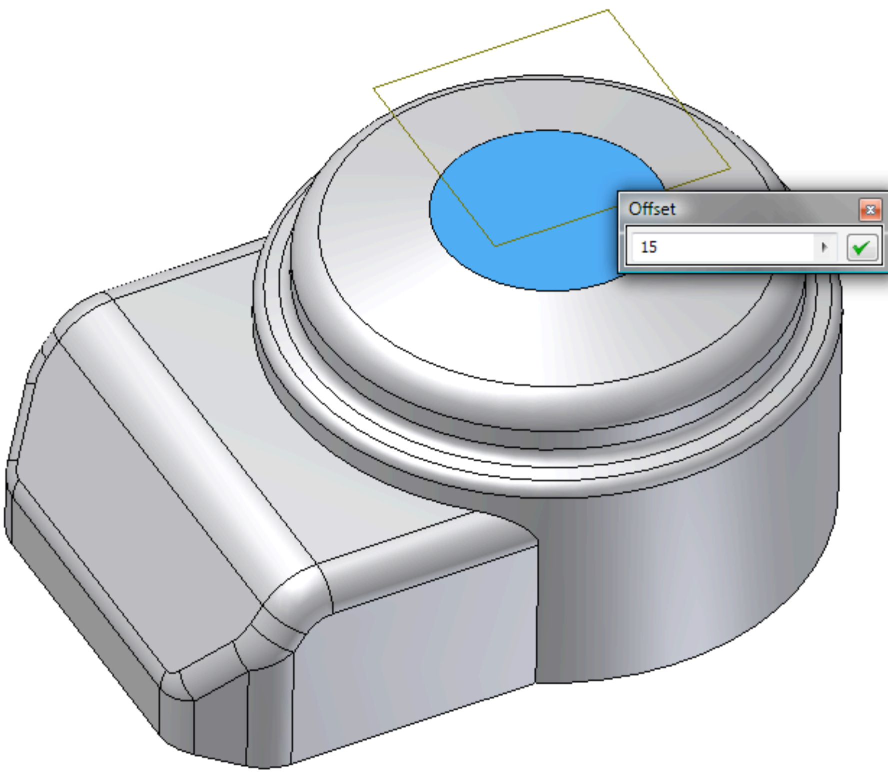
32. We have completed the filleting of the box end.



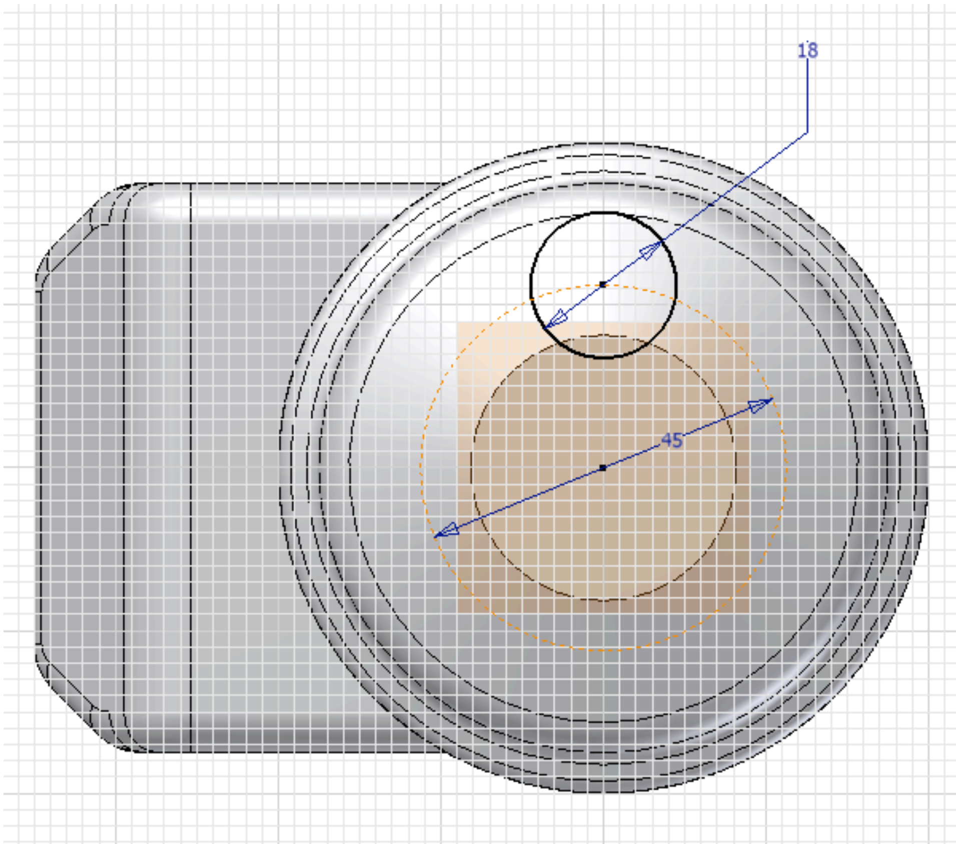
33. To continue we need a construction plane. From the Molded Features panel bar select the **Work plane** command.



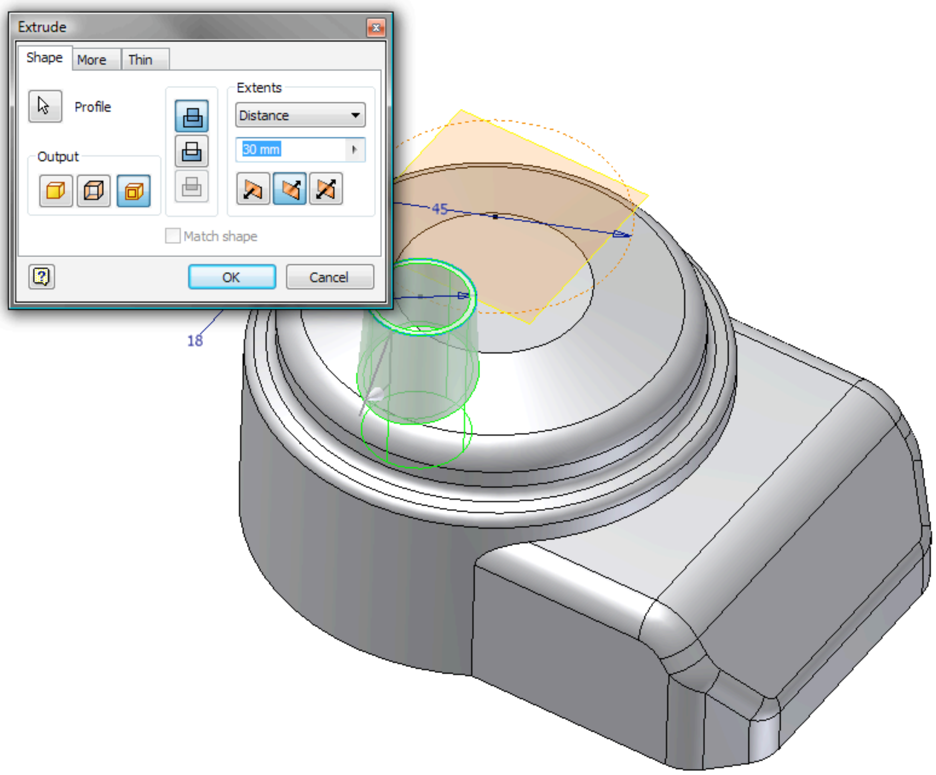
34. Select the top face and ender an offset of **15mm**.



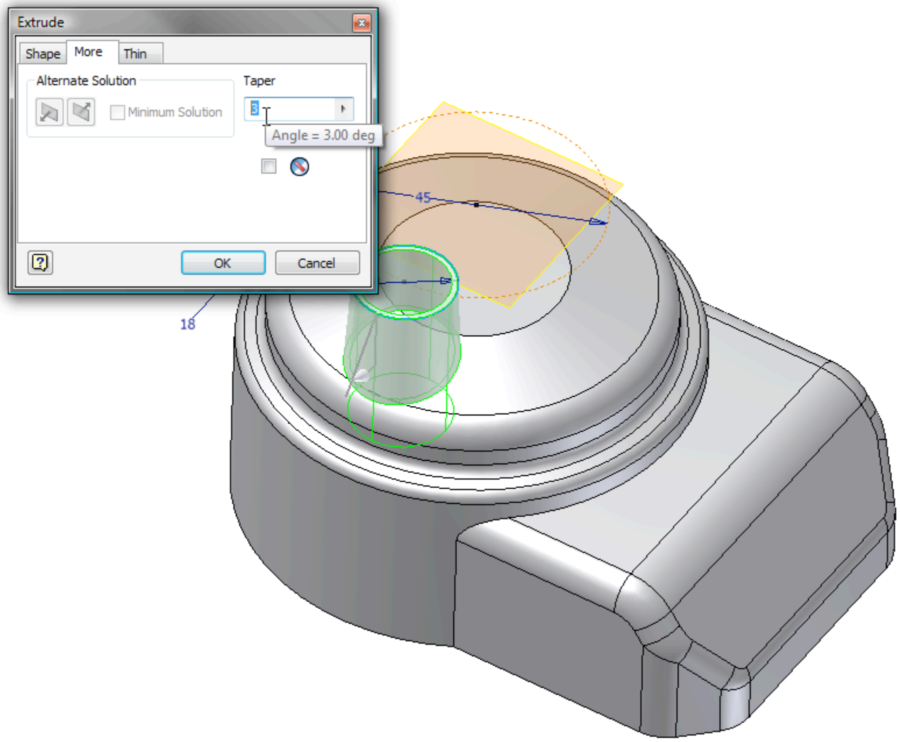
35. Create a construction circle and a normal circle as shown. The construction circle's diameter is **45mm** and the normal circle is **18mm**.



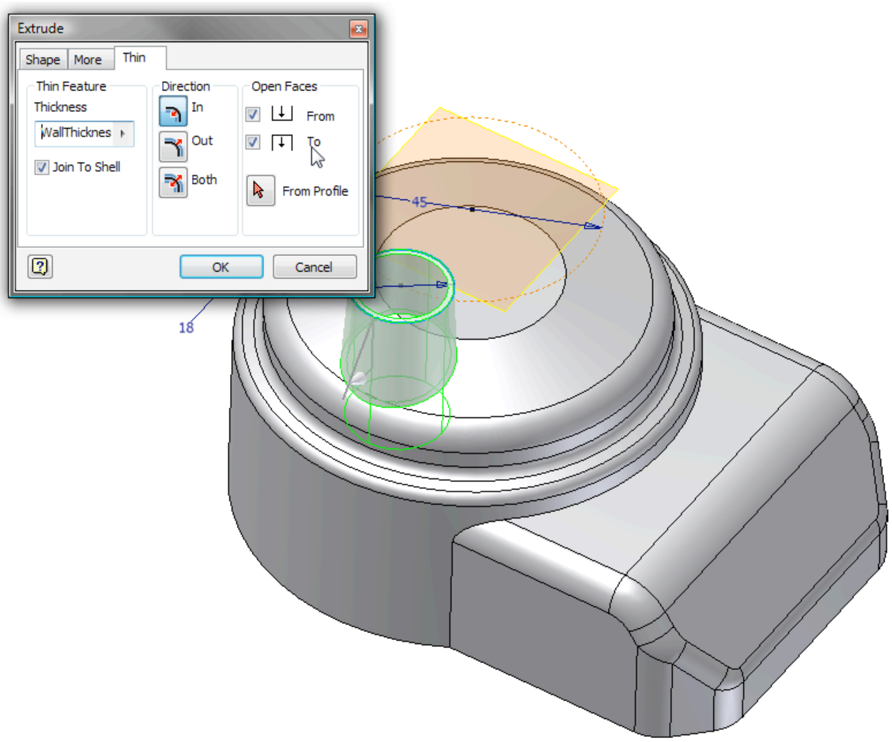
36. Create an Extrude feature and enter a distance of **30mm**.



37. Switch to the **More** tab and enter a draft angle of **3deg**.



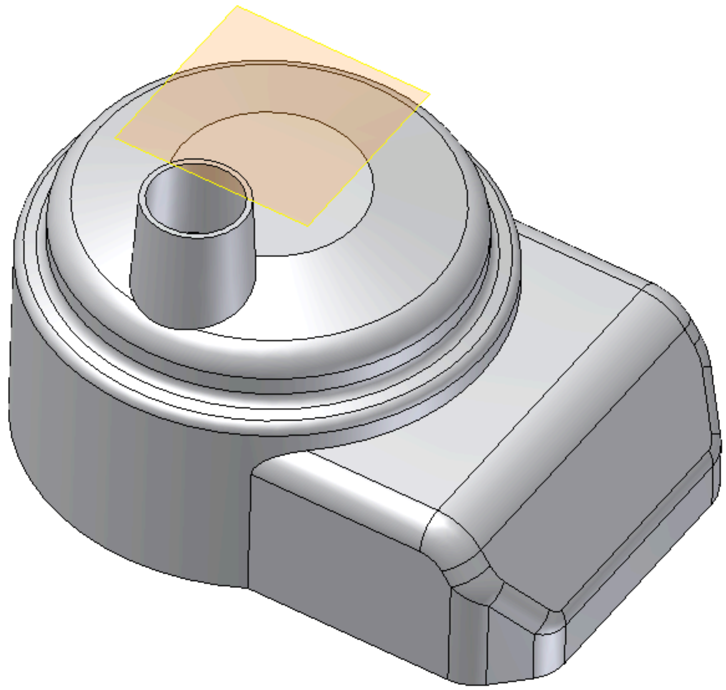
38. Switch to the **Thin** tab, but do not change the default values as shown. Hit **OK**.



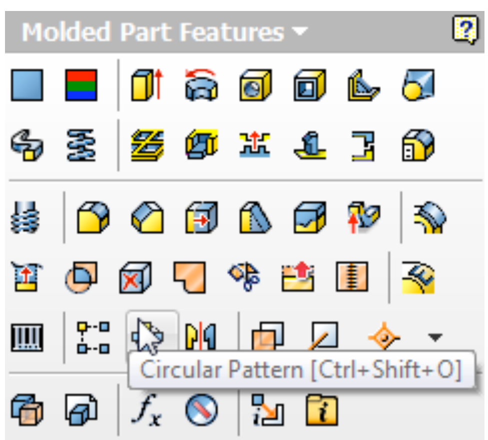
8.

9.

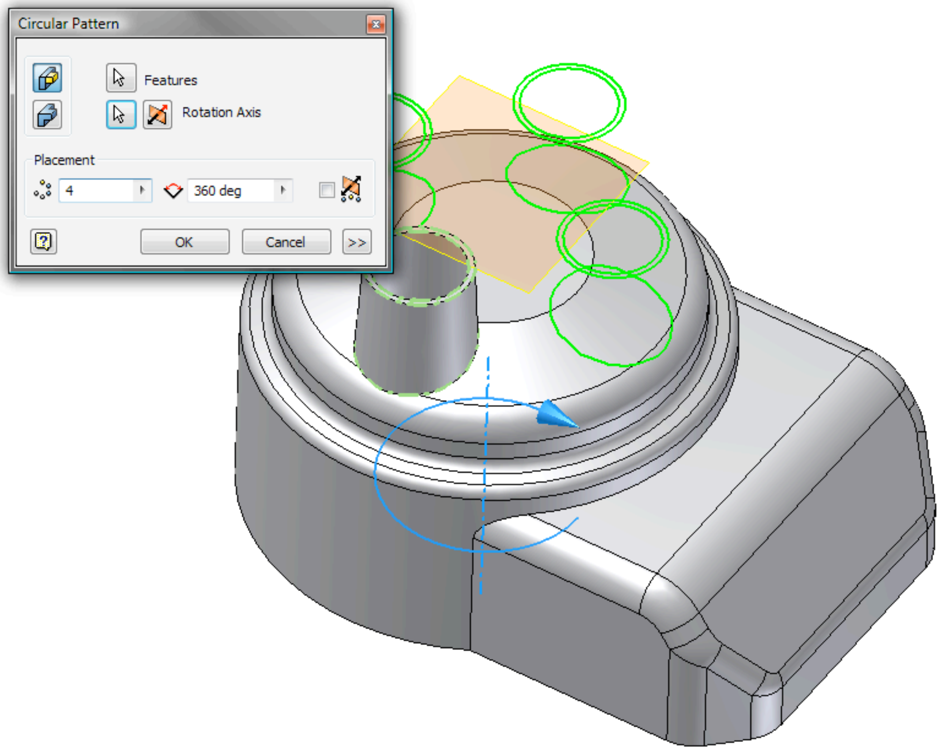
39. The feature is automatically joined to the shell.



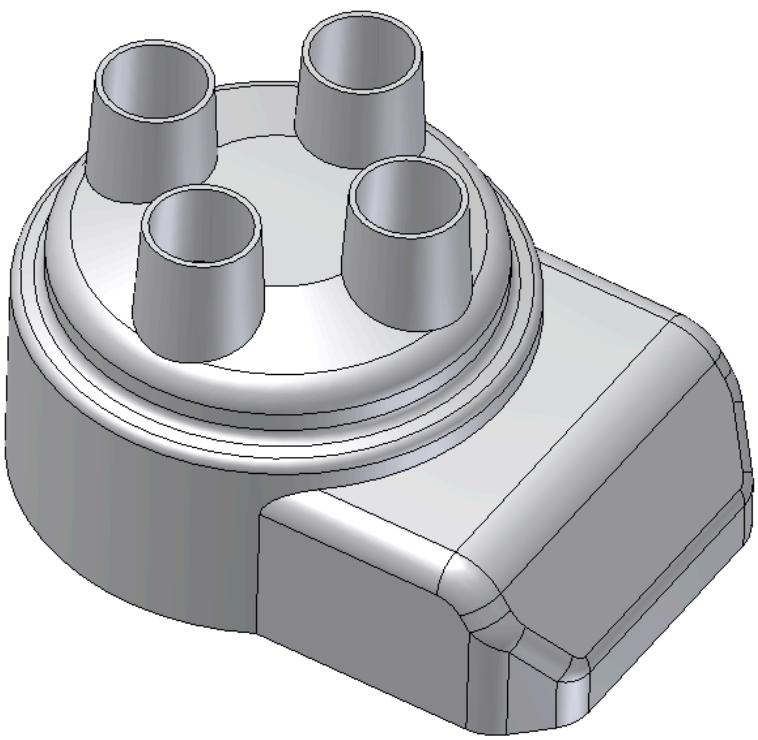
40. We need to pattern our new feature. Select the **Circular Pattern** feature.



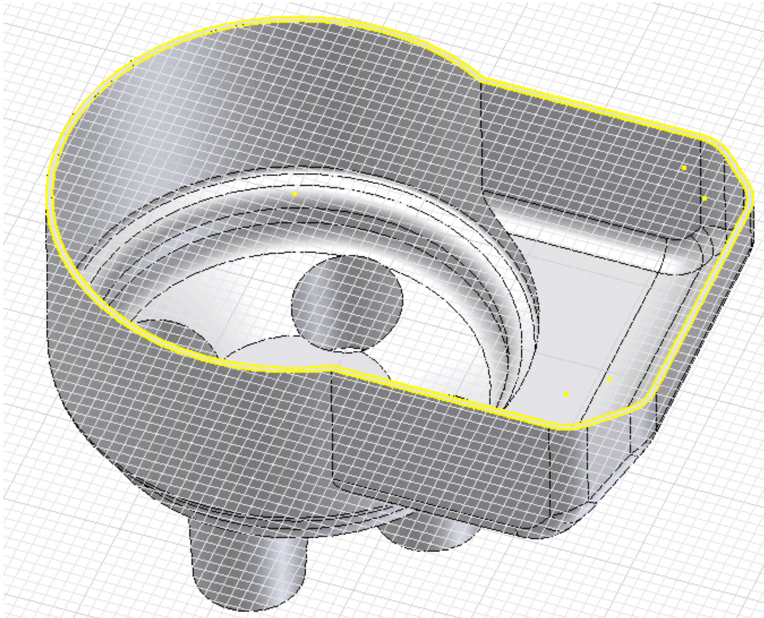
41. Select the last feature to pattern and use any circular face from the first revolve feature for the rotation axis. Enter **4** for the pattern count. Hit **Ok**.



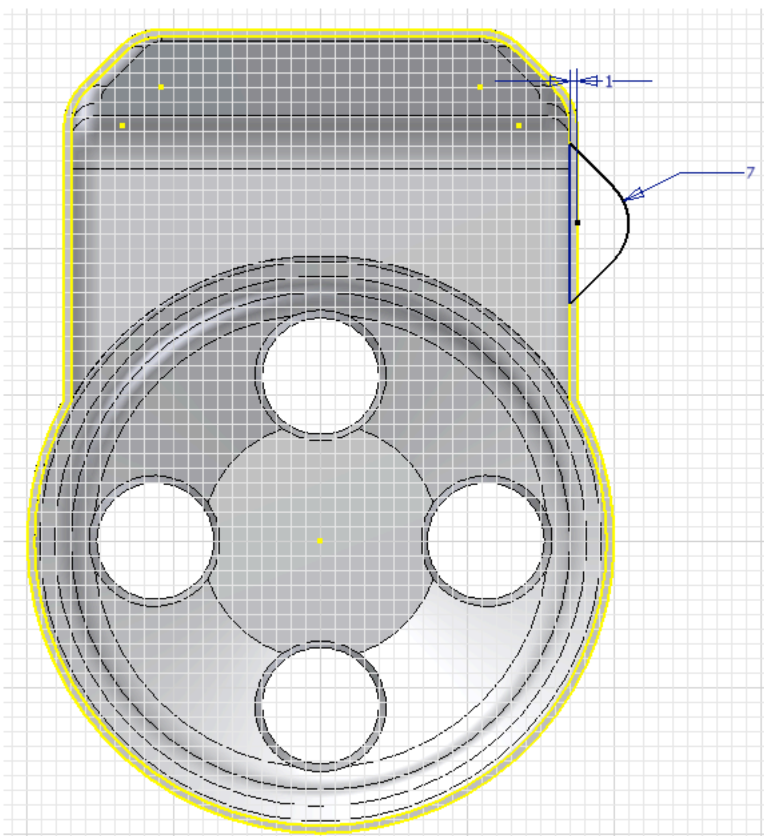
42. Even the new patterned features automatically join to the shell and part still has constant wall thickness.



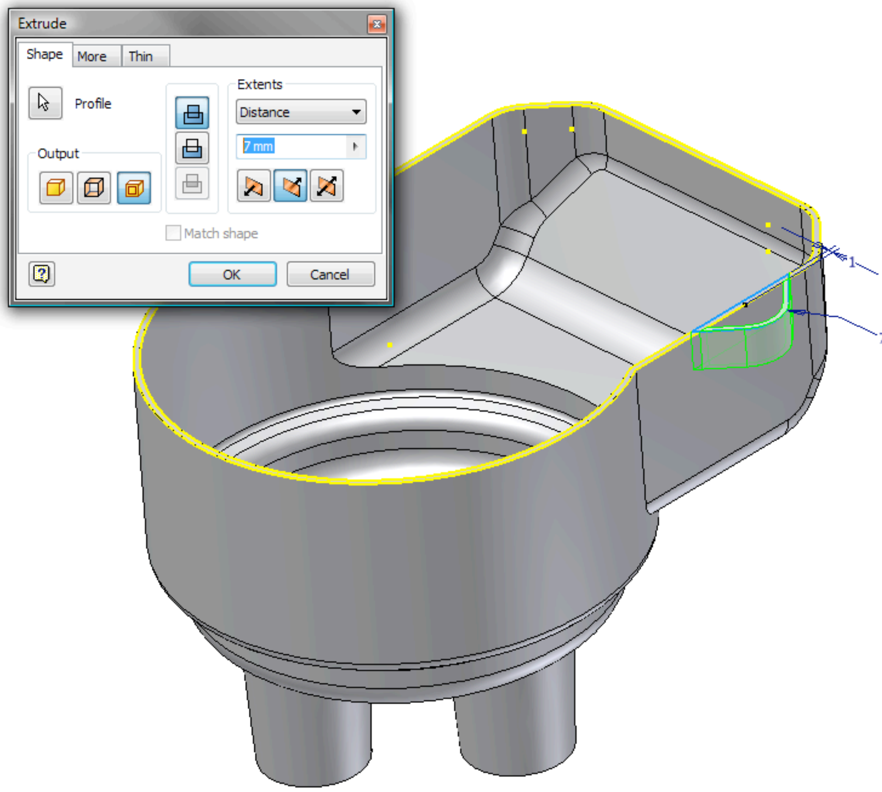
43. Create another sketch on the bottom face.



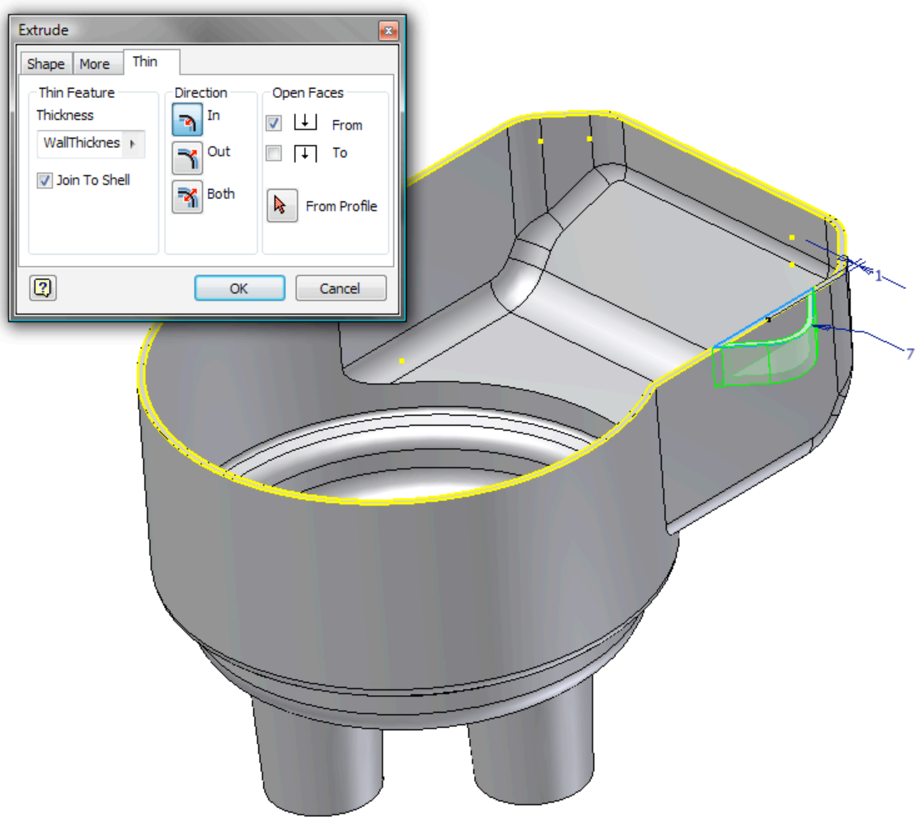
44. Draw a sketch and dimension it as shown. Note that the new geometry is drawn to the inside face.



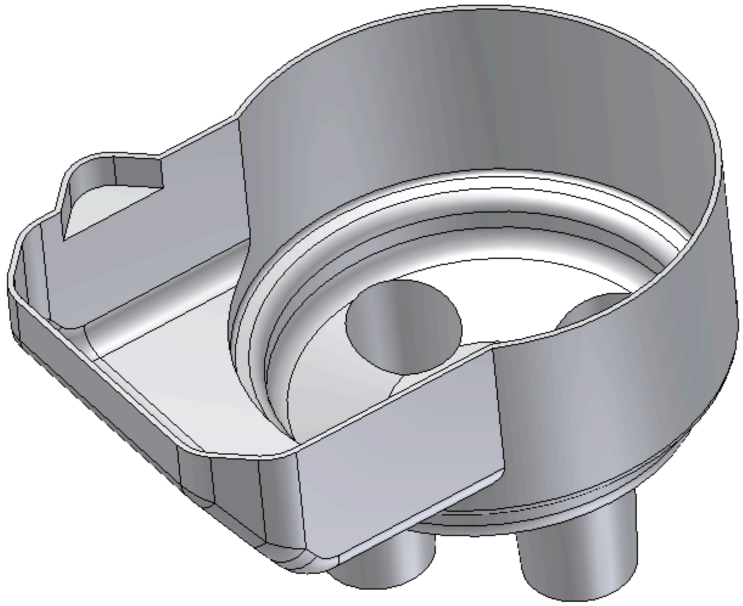
45. Create a **7mm** thin extrude.



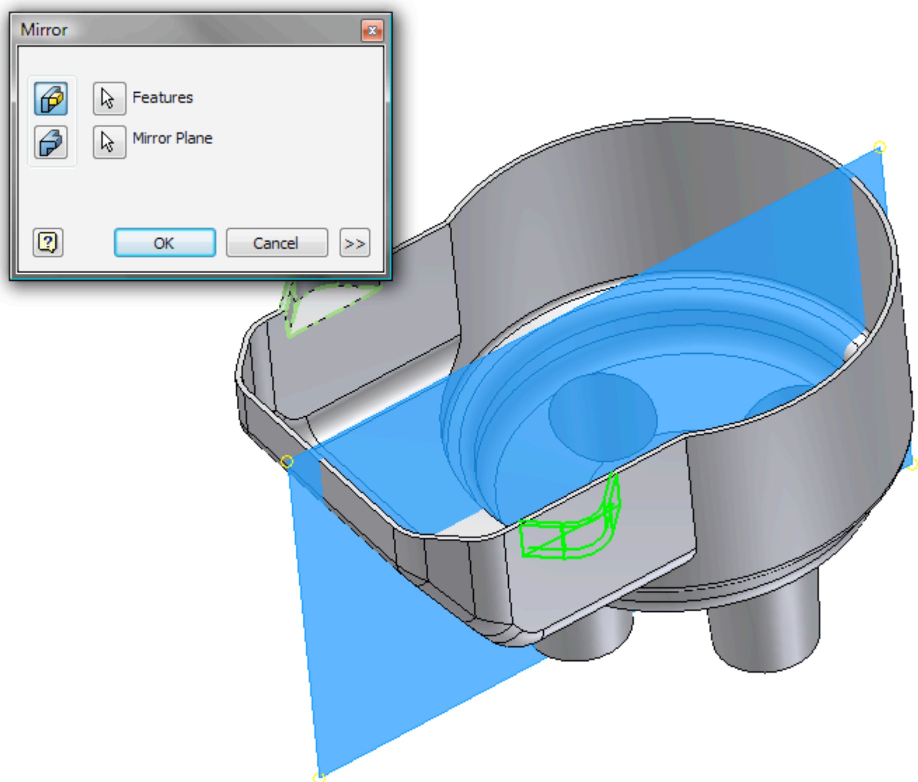
46. On the Thin tab, ensure that only the “From” face is open and hit **OK**.



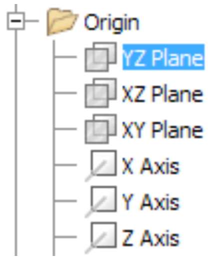
47. The new feature is added.



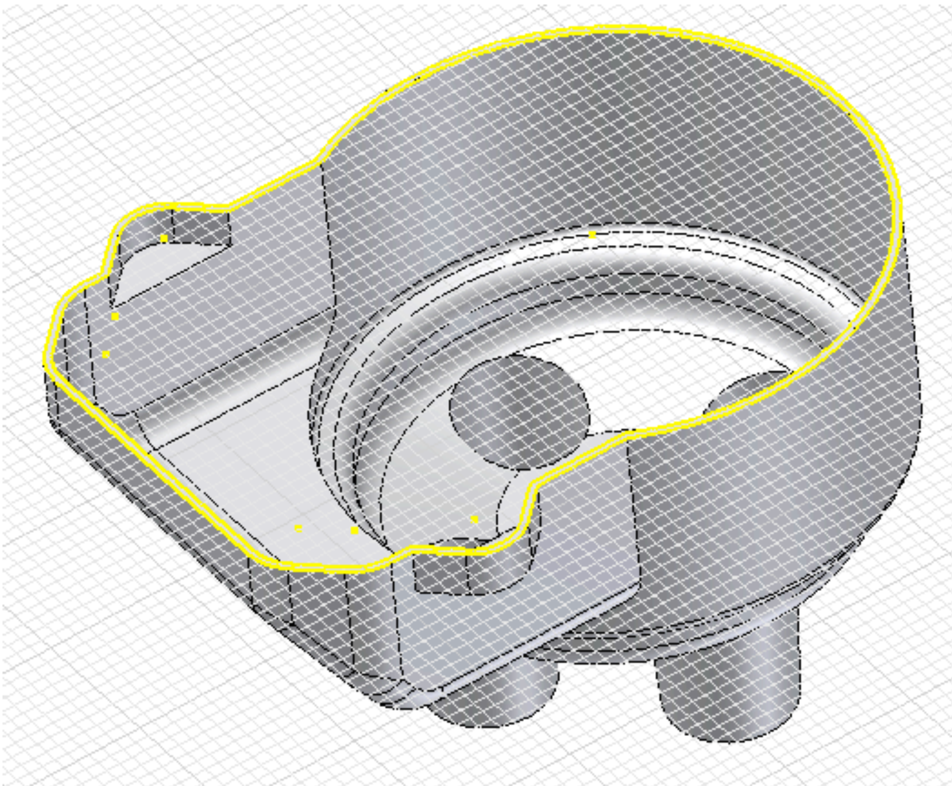
48. Select the **Mirror** feature command and select the last feature as the feature to mirror.



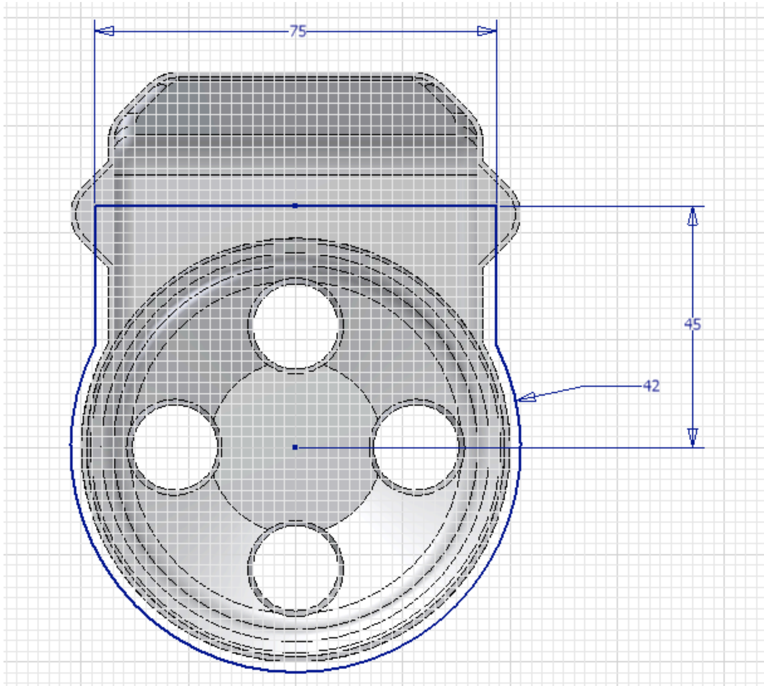
49. In the browser select the YZ plane as the mirror plane. Click **OK** to complete the Mirror.



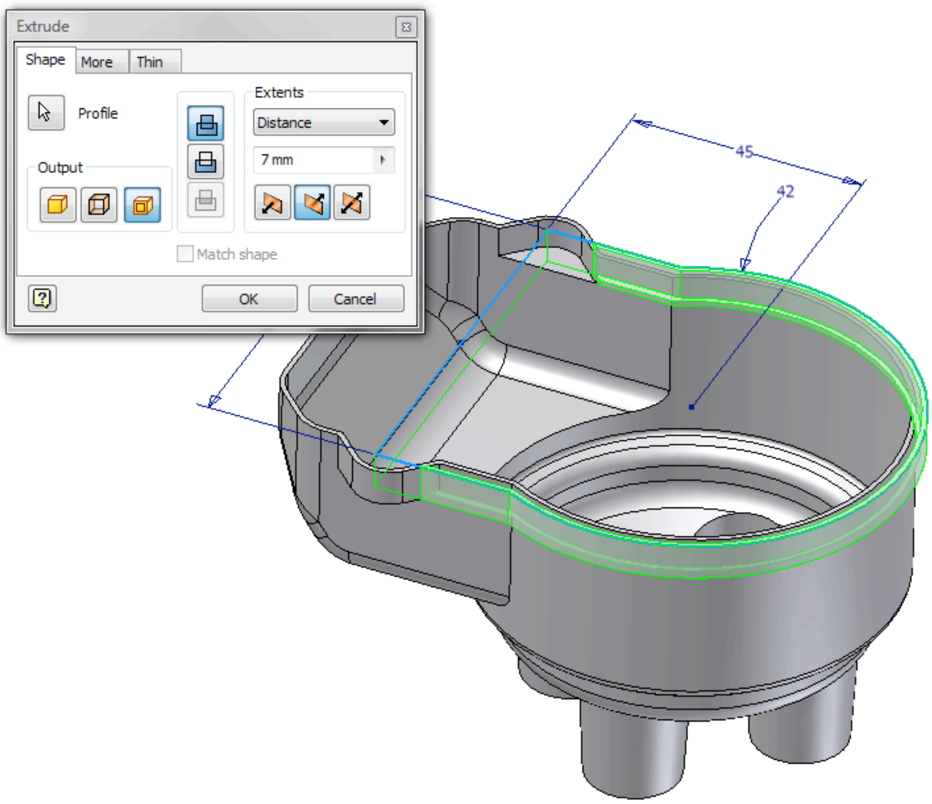
50. Finally, our last feature. Create one more sketch on the bottom face.



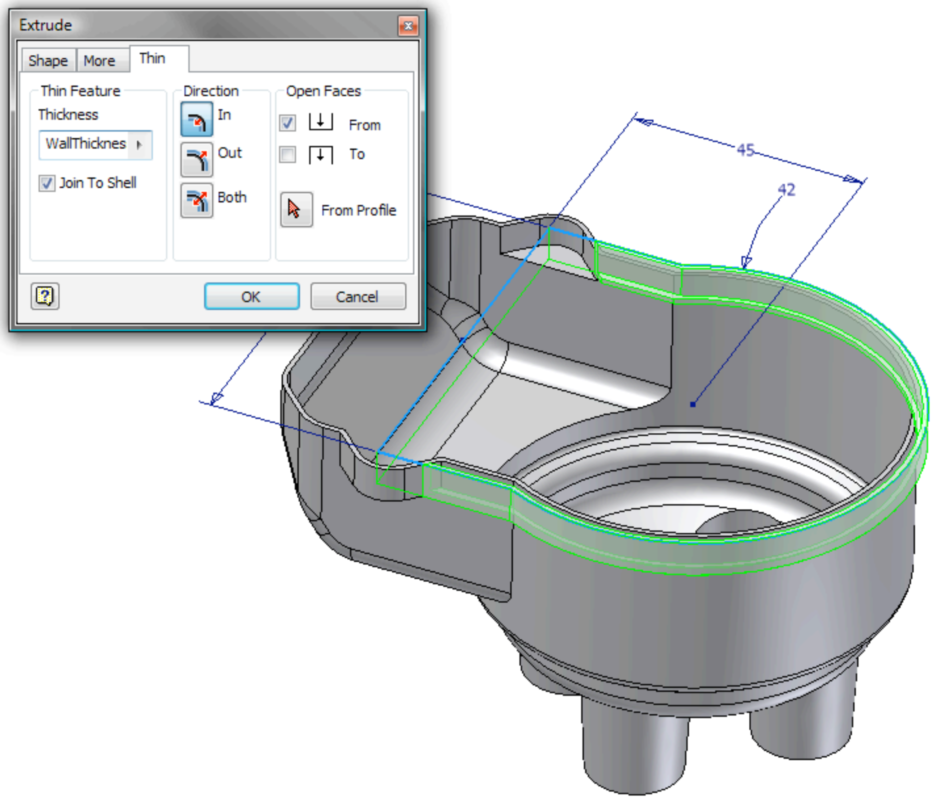
51. Draw and dimension a sketch as shown.



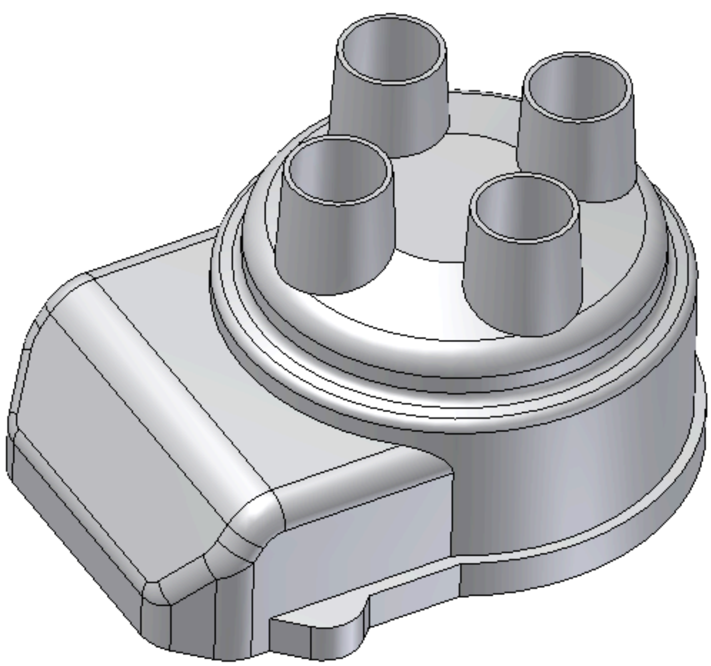
52. Create a **7mm** thin extrude.

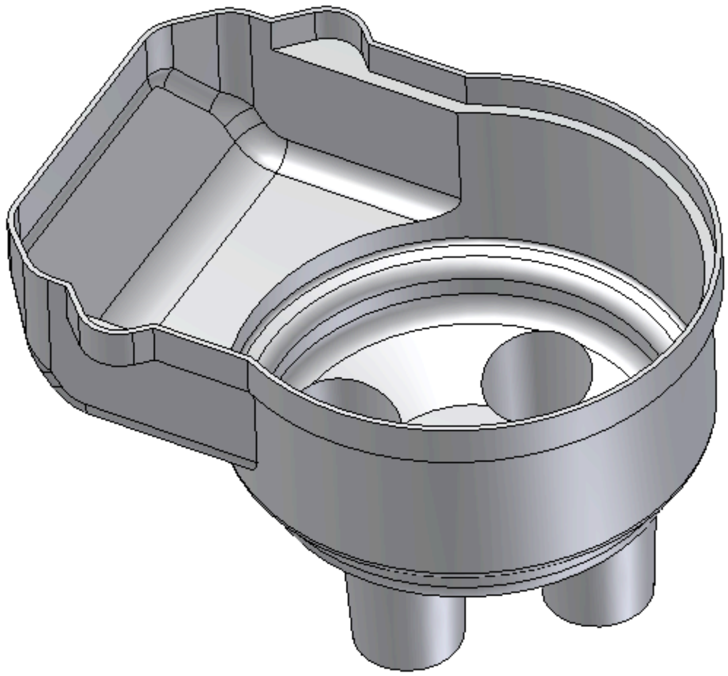


53. On the Thin tab, ensure that only the “**From**” face is open and hit **OK**.



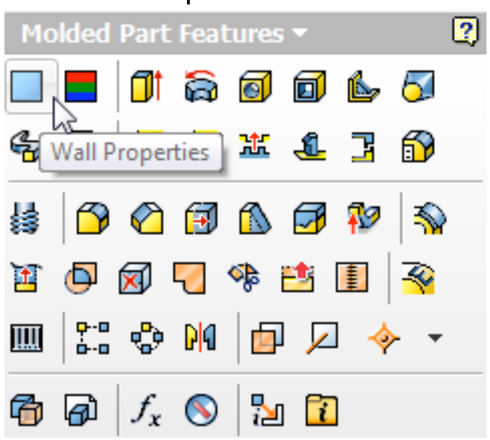
54. The new feature is added.



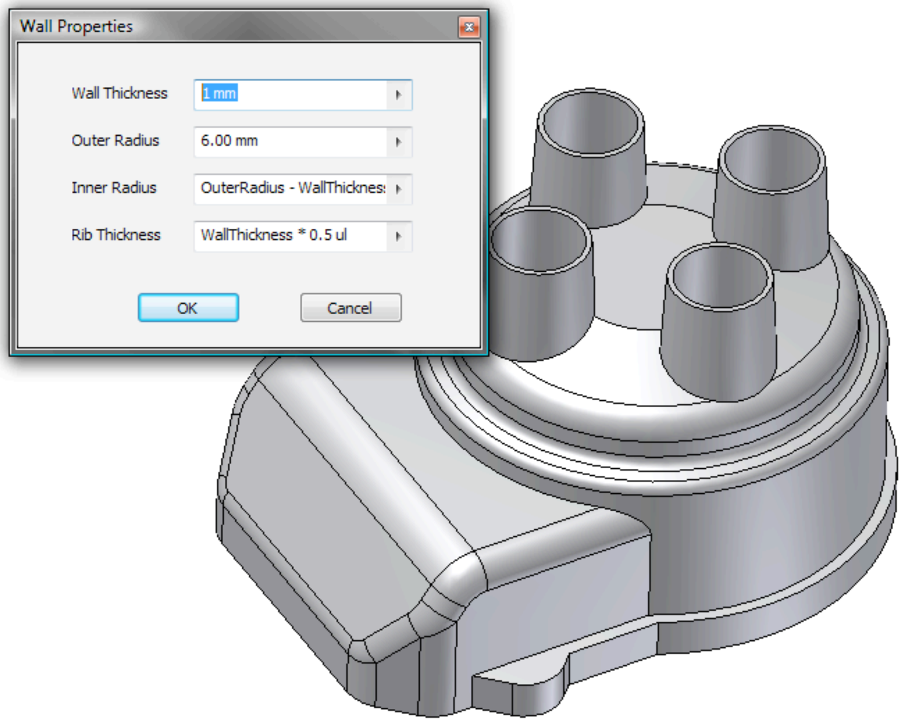


Changing Wall Properties

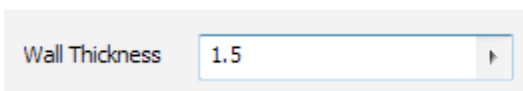
1. On the Molded Part Features panel bar select the **Wall Properties**.



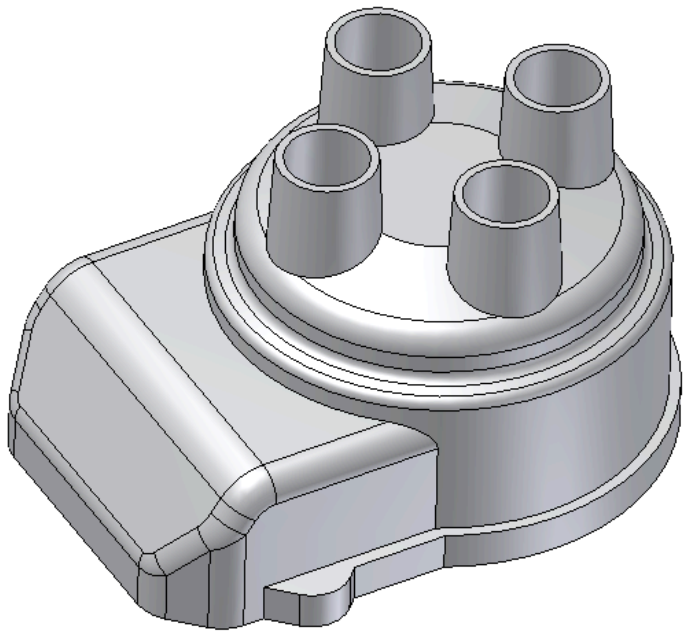
2. The current wall thickness is 1mm.



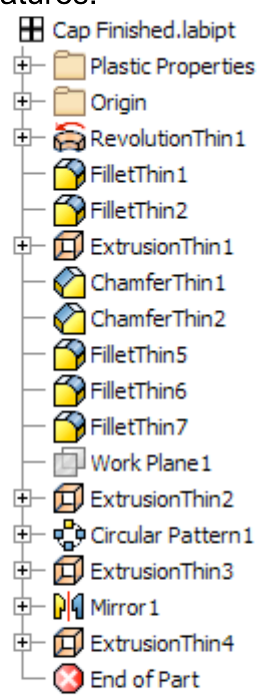
3. Change the wall thickness to **1.5mm** (Note: lager values will not work as we have created small internal fillet values that will begin to be consume and cause errors. This is a limitation of this technology preview)



4. The model automatically updates the entire design to the new wall thickness.



5. We have completed a simple example to illustrate the simplicity and power of these thin features.



Plastic Features Tutorial

Objective

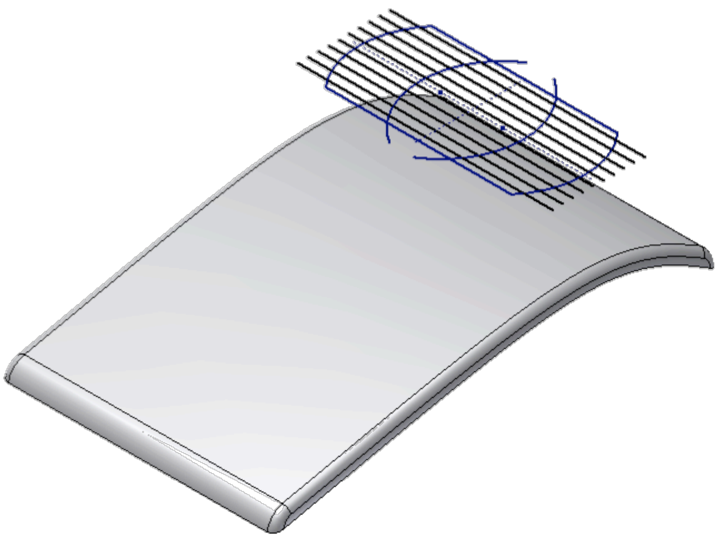
The goal of the tutorial is to introduce the new Plastic Features of Inventor.

We will model a simple plastic case for an answering machine-like product.

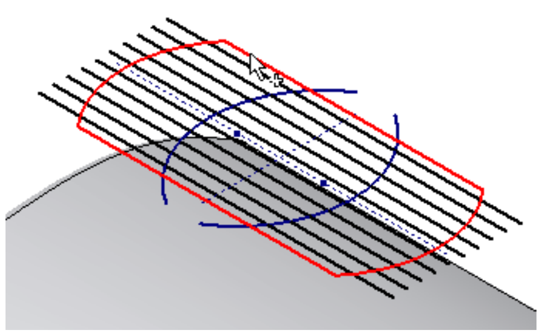
Building a Grill

1. Open the PFTutorialTop.labipt part file.

2. Turn on visibility of *GrillSketch*



3. Click on the **Grill Feature** button



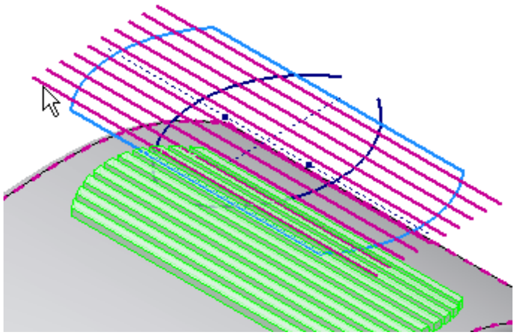
4. In the Boundary tab:

   Select the closed profile

   Thickness = 0 mm

   Height = 2 mm

   Outside Height = 0 mm

5. 

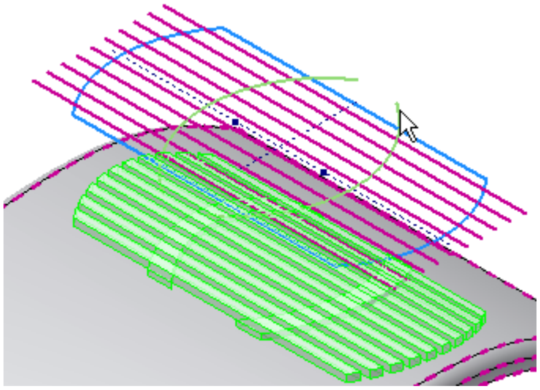
6. Skip the Island tab, in the **Rib tab**:

   Select the lines

   Thickness = 2.5 mm

   Height = 1.3 mm

   Top Offset = 0.2 mm



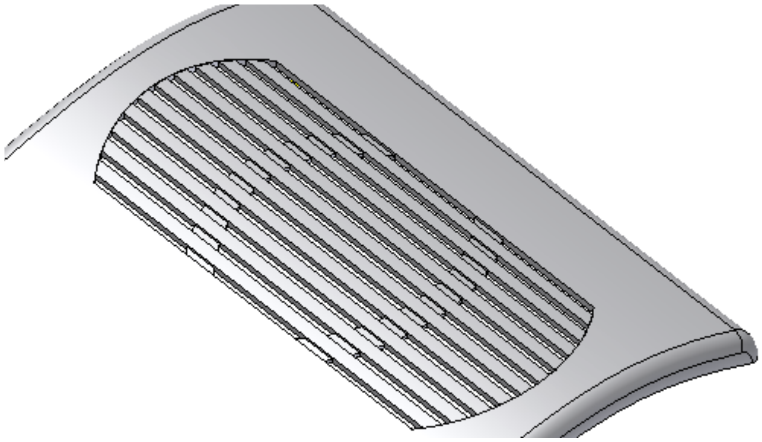
7. In the Spar tab:

   Select the arcs

   Top Offset = 0.5 mm

Thickness = 5 mm

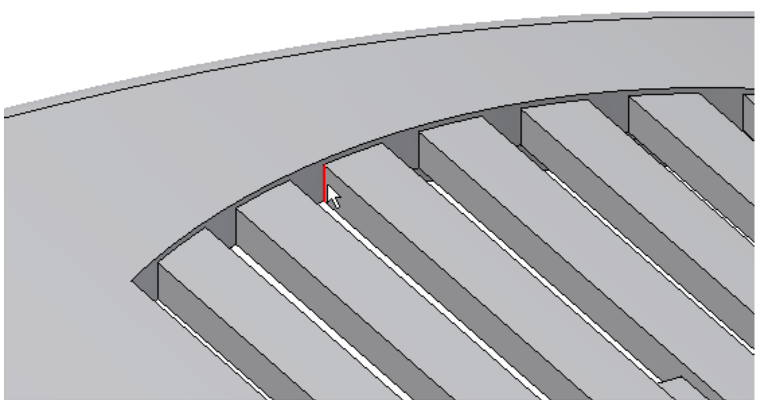
Bottom Offset = 0 mm



8. Hit **OK**

Rule Fillets

9. We now want to fillet all the vertical edges of the Grill like the one indicated in the picture. Sounds like a task for the **Rule Fillet feature**. The strategy will be to select a face where all those edges have a vertex and then we use a vertical direction to identify all of them.

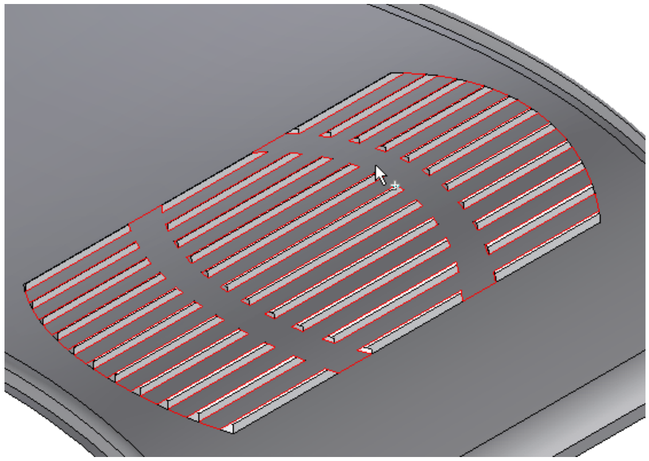


10. Click on the Rule Fillet button:

Source = *face*

Select the face of the Grill on the internal side of the solid

Radius = 0.5 mm

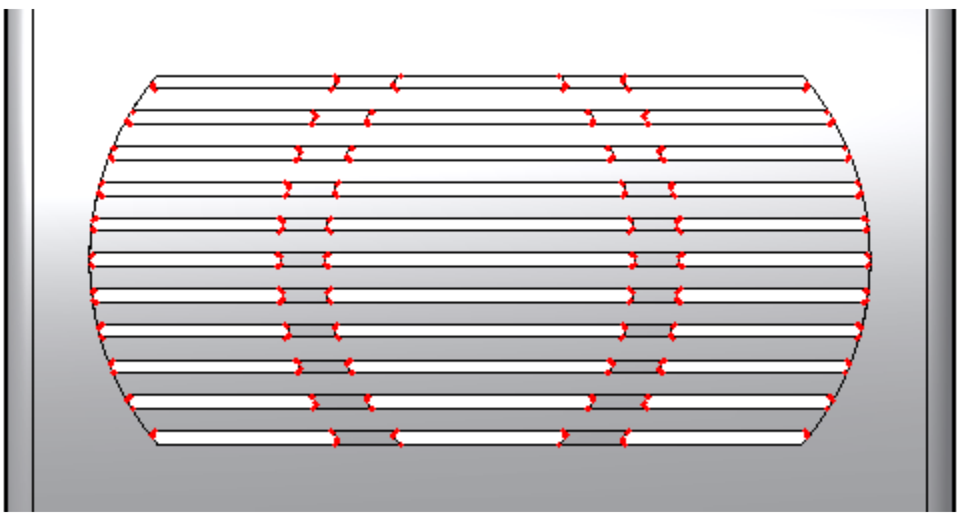


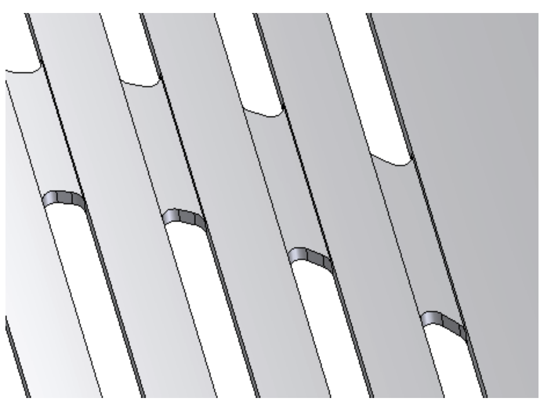
Rule = Incident edges

Select the Y axis for the direction

Tolerance = 1 deg

11. Hit OK. All the 128 fillets are done in one simple feature.





Create a rest

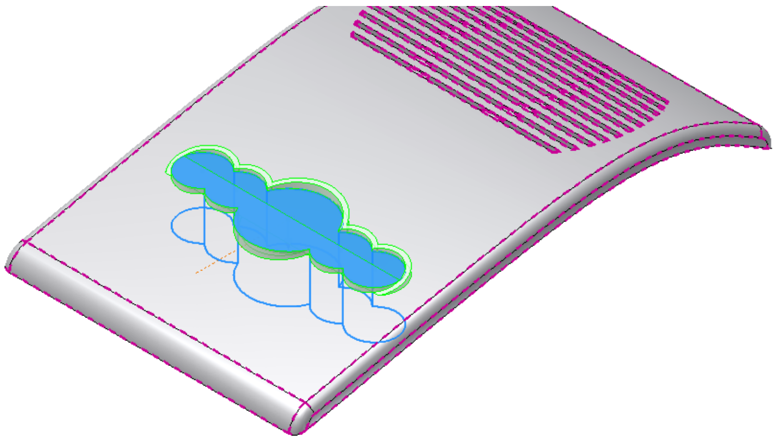
We now want to build a flat area for the answering machine buttons.

1. Turn on the visibility of *RestSketch*.
2. Click on the **Rest feature** 
3. In the **Shape tab**:

   Select the *RestSketch* as the Profile

   Thickness = 1.5 mm inside

   Through All

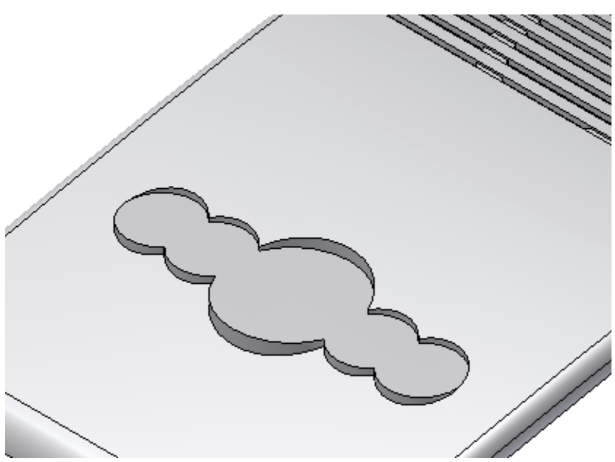


4. More tab:

   Landing Distance = 0 mm

   Landing Taper = 0 deg

   Clearance Taper = 0 deg
5. Hit **OK**

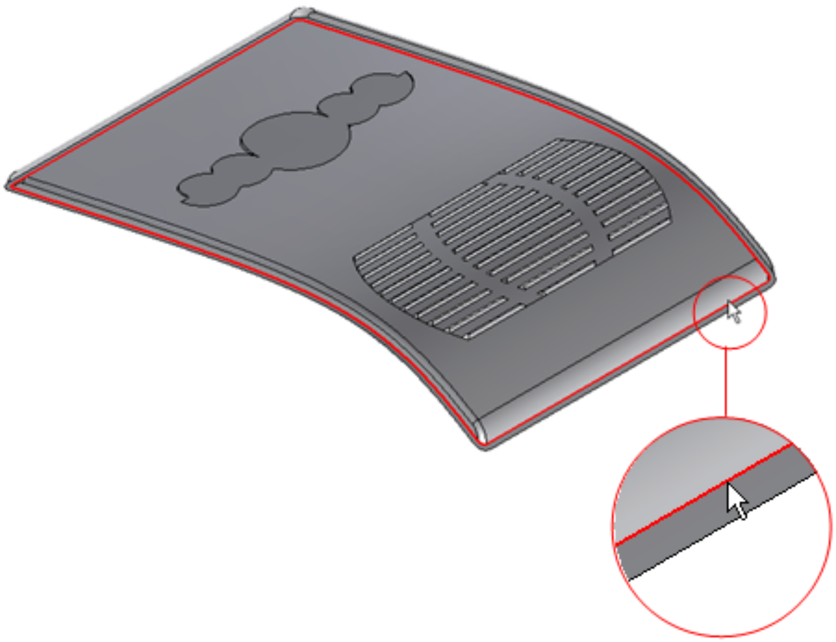


Create a Lip

1. We now build a lip-groove combination to facilitate the assembly.
2. Click on the **Lip feature**.

3. Make sure the **Lip** button is selected.

4. In the **Shape tab** select the inside edge as the *Path Edges*.



5. Leave the *Path Extents* unchecked
6. Click on the *Pull Direction* check box.
7. Select the *Y axis* (in the Origin folder) as the Pulling Direction.
8. In the **Lip tab**:

   Outside Angle = 0 deg

   Inside Angle = 0 deg

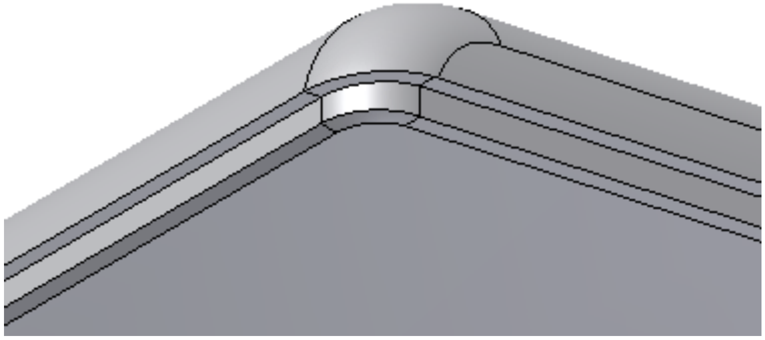
   Height = 1 mm

   Shoulder Width = 0 mm

   Width = 0.75 mm

   Clearance = 0.5 mm

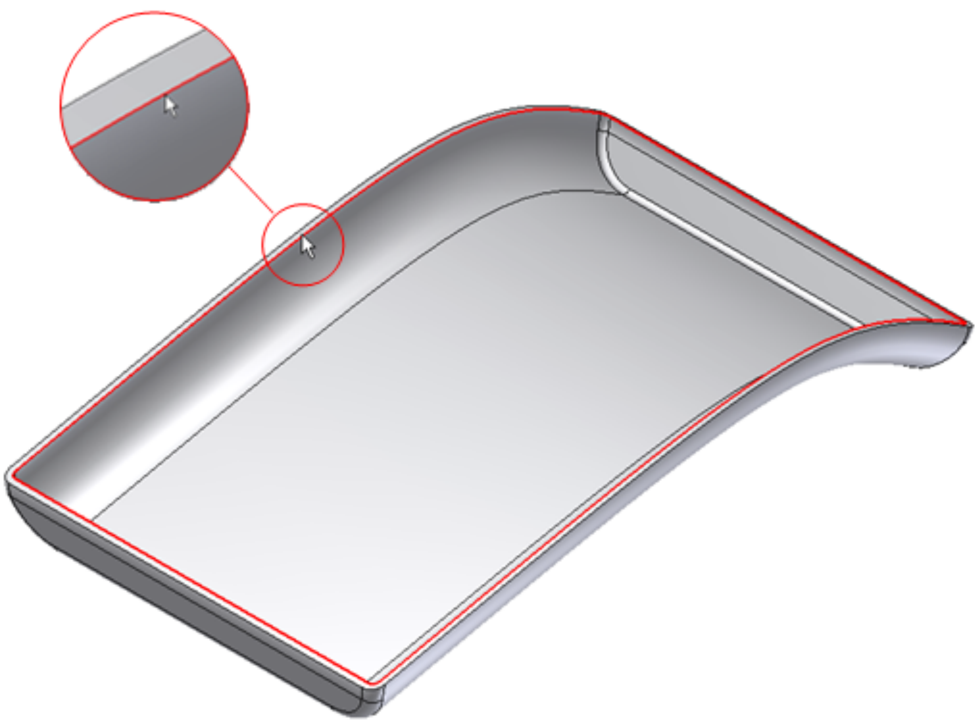
9. Hit **OK**.



10. Open the PFTutorialBottom.labipt part file.
11. Click on the **Lip feature**

12. Make sure that the **Groove** button is selected.

13. In the **Shape tab** select the inside edge as the *Path Edges*.



14. Leave the *Path Extents* unchecked.
15. Click on the *Pull Direction* check box.
16. Select the *Y axis* (in the Origin folder).
17. In the Groove tab:

    Outside Angle = 0 deg

    Inside Angle = 0 deg

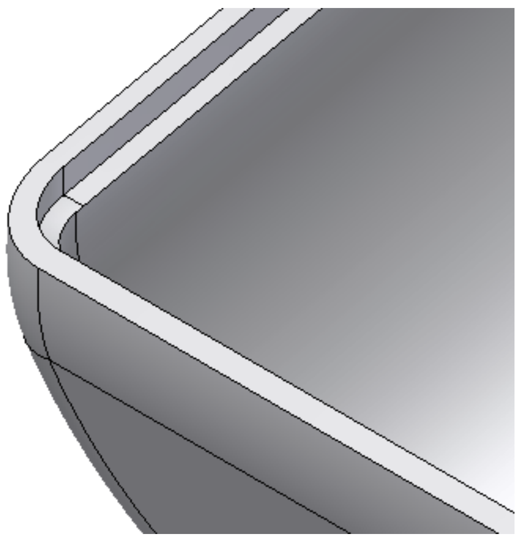
    Height = 1 mm

    Shoulder Width = 0 mm

    Width = 0.75 mm

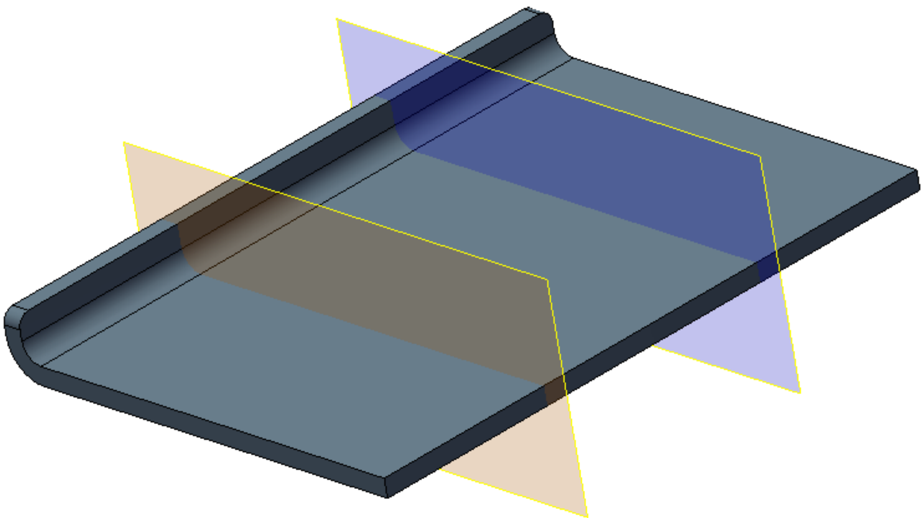
    Clearance = 0 mm

18. Hit **OK**.



Build a cover

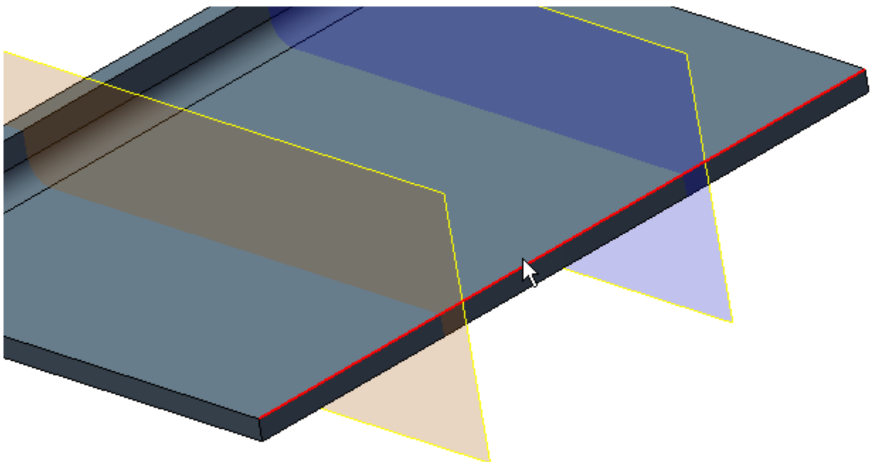
1. We now build a sliding cover for the batteries compartment. We need a snap in mechanism for the battery cover. First we build a lip-groove combination on one side of the battery cover limited by two planes.
2. Open the part PFTutorialBatteryCover.labipt



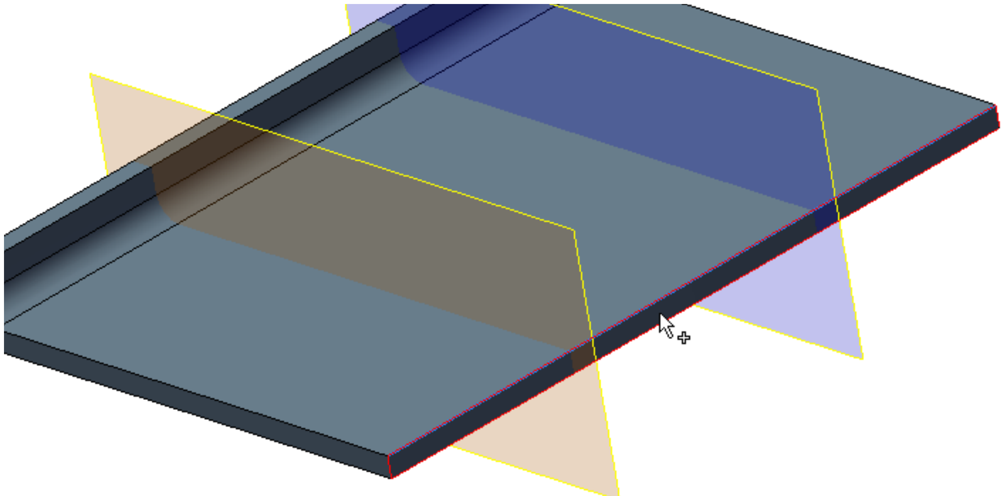
3. Build two work planes parallel to the side faces of the battery cover at 15 mm offsets
4. In the Plastic Features panel click on the **Lip feature.**

5. Make sure the **Lip** button is selected.

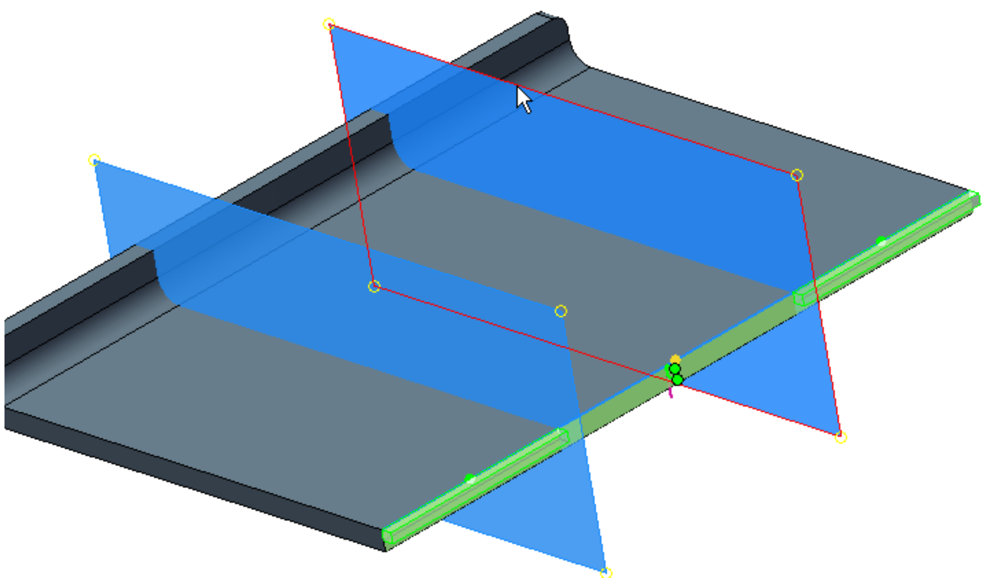
6. In the **Shape tab** select the internal edge as the *Path Edges*.



7. Select the *Guide Face*.



8. Leave the *Pull Direction* unchecked.



9. Check the *Path Extents* box and select the two planes.
10. The preview already shows the portions of the Lip that have been chosen, the first and the last, clicking on the green and yellow dots may change that selection.
11. In the Lip tab:

    Outside Angle = 0 deg

    Inside Angle = 0 deg

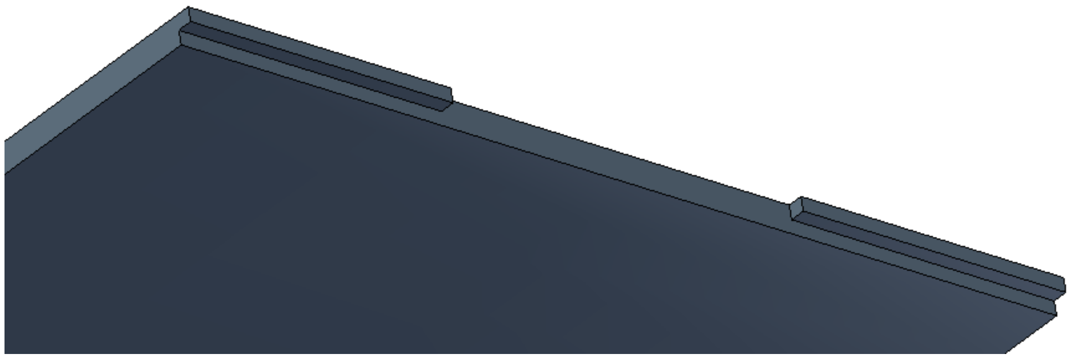
    Height = 0.8 mm

Shoulder Width = 0 mm

Width = 0.75 mm

Clearance = 0 mm

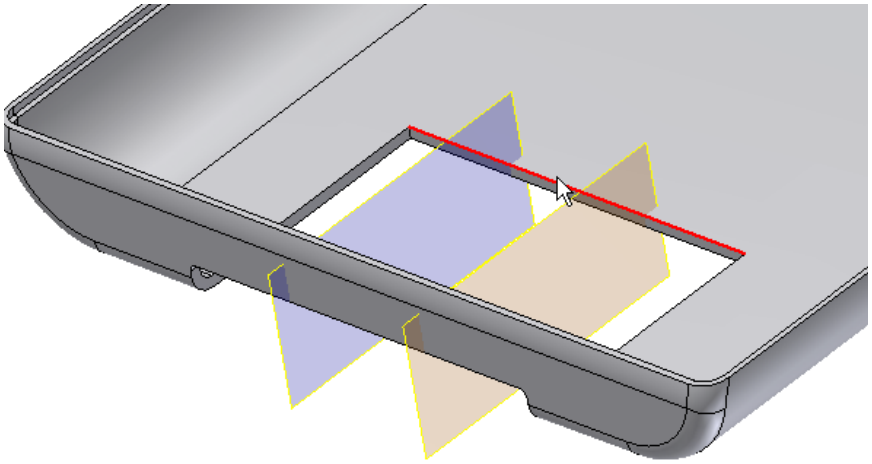
12. Hit **OK**.



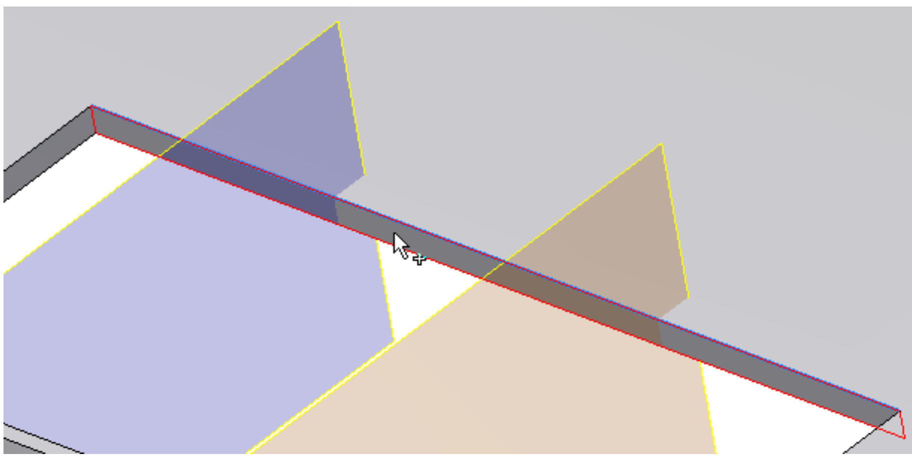
13. Activate the window the part PFTutorialBottom.labipt is displayed in.
14. Make Battery Split Surface visible
15. Use the **Split** feature and Battery Split Surface to remove Battery Cover material.
16. Make Battery Split Surface invisible
17. Build two work planes parallel to the side faces where the battery cover will be placed at 15 mm offsets.
18. Click on the **Lip feature.**

19. Make sure that the **Groove** button is selected.

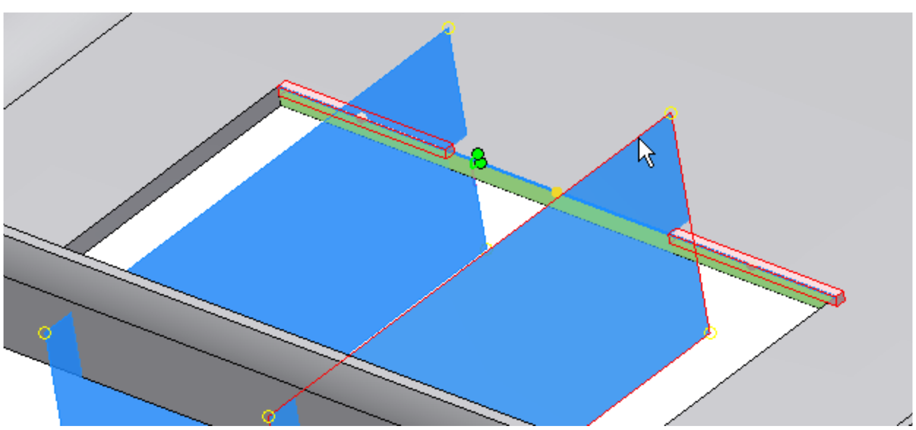
20. In the **Shape tab** select the inside edge as the *Path Edges*



21. Select the *Guide Face*



22. Leave the *Pull Direction* box unchecked.
23. Click on the *Path Extents* check box and select the two limiting planes, the default selection of the portions of the groove is the right one.



24. In the Groove tab:

    Outside Angle = 0 deg

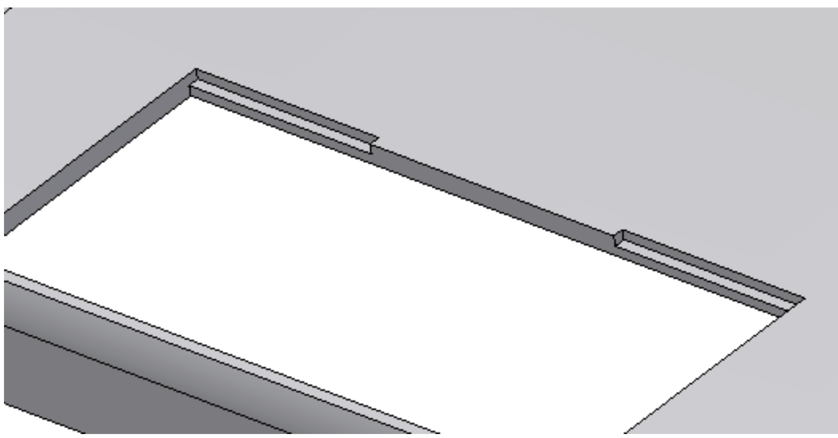
    Inside Angle = 0 deg

    Height = 0.8 mm

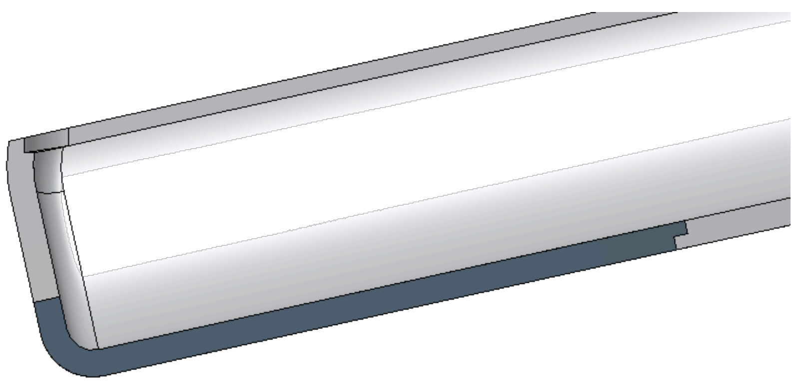
    Shoulder Width = 0 mm

    Width = 0.75 mm

    Clearance = 0 mm
25. Hit **OK**.



26. When placed in an assembly the Lip and Groove will mate as pictured.

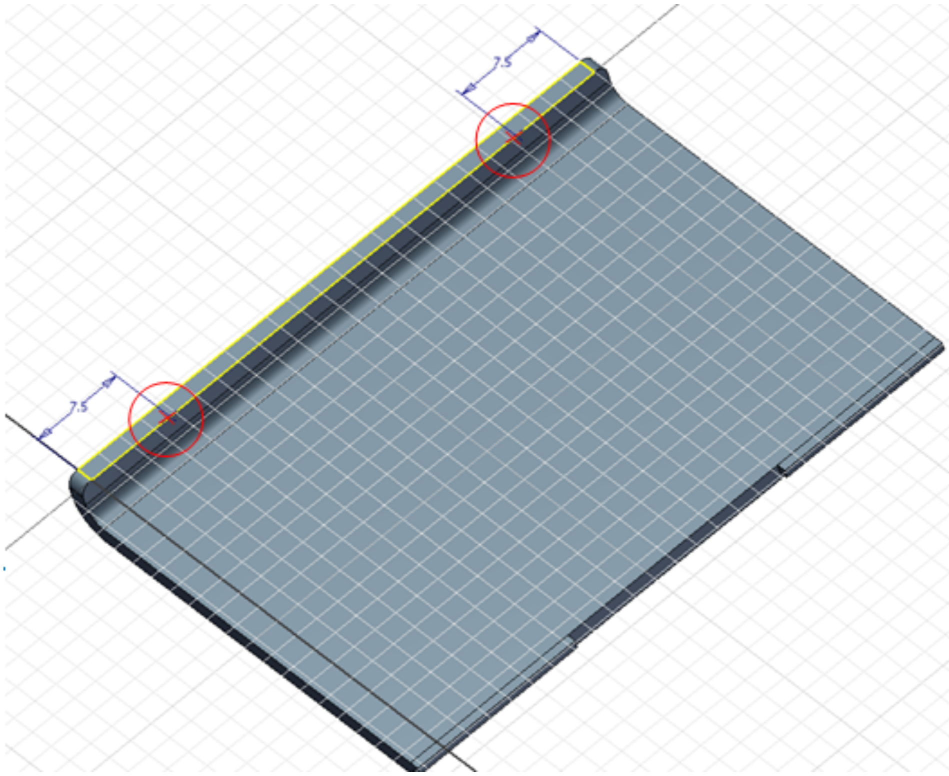


Snap Fit feature

We build now a retention mechanism.

1. Activate the window the part PFTutorialBatteryCover.labipt is displayed in.

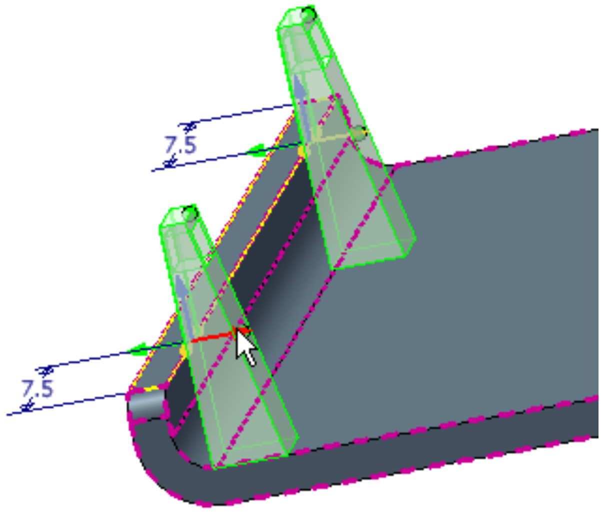
We need two positioning points.



2. Build a sketch on the wall narrow face as indicated in the picture. Make sure you have the internal edge projected since the positioning points should stay on this edge to make the Snap Fit face flush with the internal face of the bottom solid.
3. Build two points on the internal edge projection at 7.5 mm from the sides.
4. Click on the **Snap Fit** feature button.

5. Select the Cantilever Snap Fit Loop style.

6. In the **Shape tab**:
7. *From Sketch* Placement type
8. Select the two sketch points as *Centers*
9. Click on the Flip button and the Catch direction manipulator arrows on the graphics area until the orientation of the clips is like in the picture.



10. In the **Clip tab**:

    Clip Length = 4 mm

    Clip Width = 5 mm

    Clip Thickness at Wall = 0.5 mm

    Clip Thickness At Top = 0.3 mm

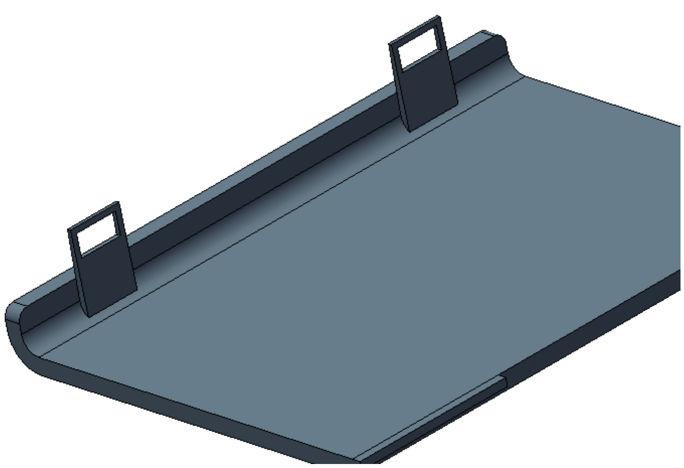
11. Catch tab:

    Catch Width at Side 1 & 2 = 0.5 mm

    Catch Opening Length = 2 mm

    Catch Width At Top = 0.5 mm

12. Hit **OK**.



We now want to fillet the edges at the intersection between the Snap Fit and the battery cover solid. A **Rule Fillet** can do the job.

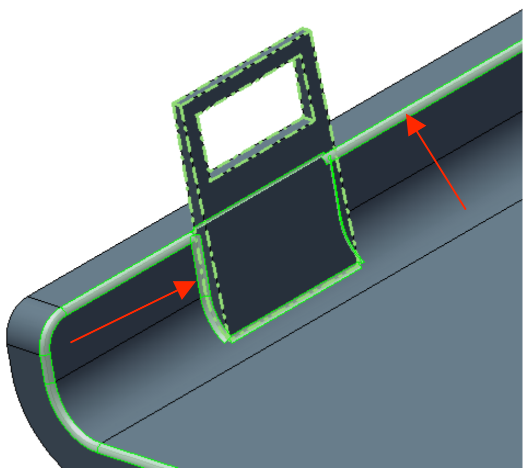
13. Click on the Rule Fillet feature.

Source = *Feature*

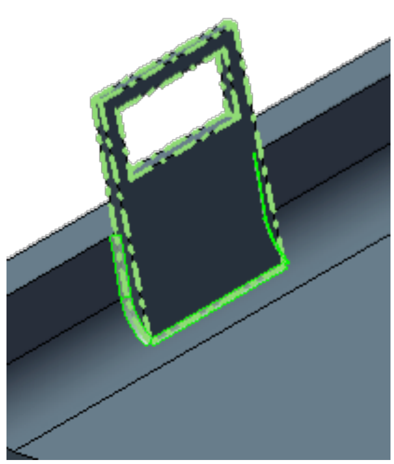
Select the Snap Fit feature

Radius = 0.2 mm

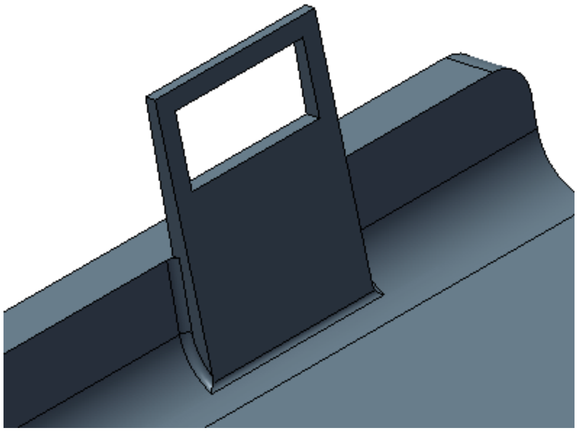
Rule = Against Part



14. We do not want the fillet around the top edges (red arrows) The edges where included because they share the curved face of the Snap Fit that coincides with the curved face of the battery body (they are called “merged faces”), we can simply skip such merged faces (and all the edges they share)



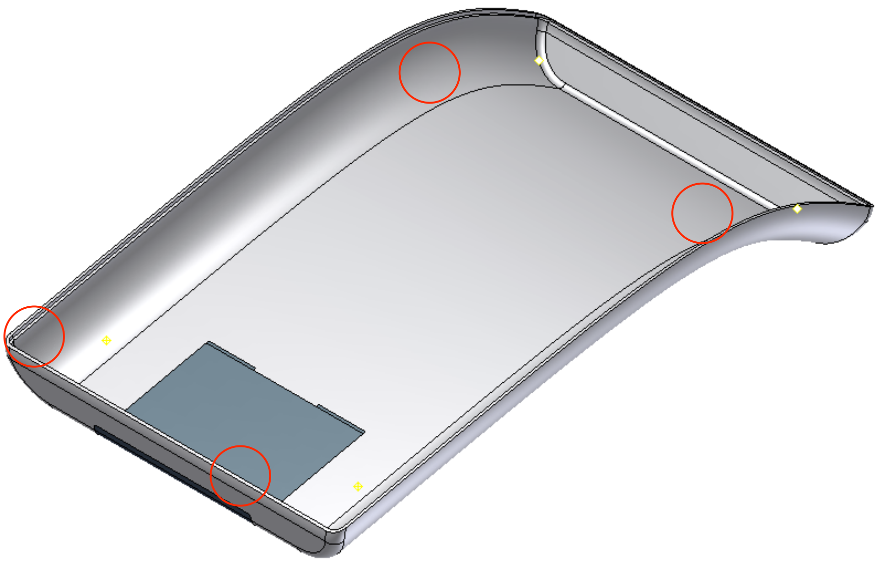
15. Click on the More button that expands the dialog
16. Check the *Removed Merged Faces* box on. The preview now shows that the fillet on the long top edge is gone.
17. Hit OK, both clips get the fillet since they both belong to the same feature.
18.



Constructing bosses

We model now the mounting bosses to join the assembly together.

1. Activate the window displayingPFTutorialBottom.labipt.
2. Turn the visibility on for *Work Point1,..., 4*.



3. Click on the **Boss feature** of the Plastic Features panel.

4. Make sure the **Head** button is pressed.

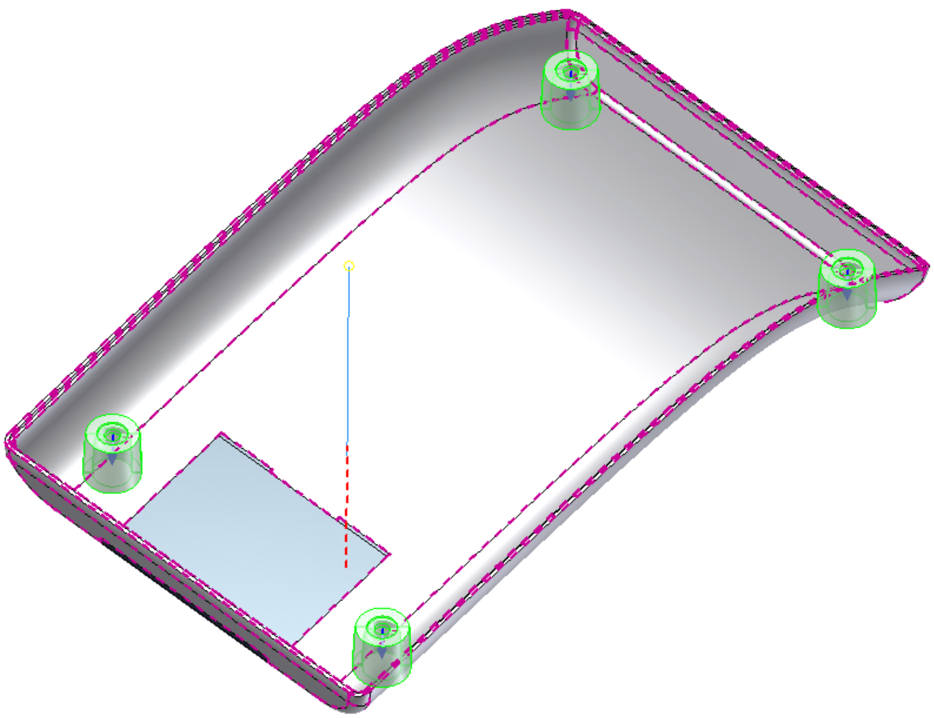
5. In the **Shape tab**:

   On Point placement type

   Select the four points as the *Centers*

   Select the *Y axis* as the *Direction*

   Interaction Fillet = 0.3 mm



6. In the **Head tab**:

   Wall thickness = 1.5 mm

   Shank Height = 1.5 mm

Clamp Height = 0.5 mm

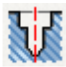
Shank Diameter = 3 mm

Clamp Diameter = 7 mm

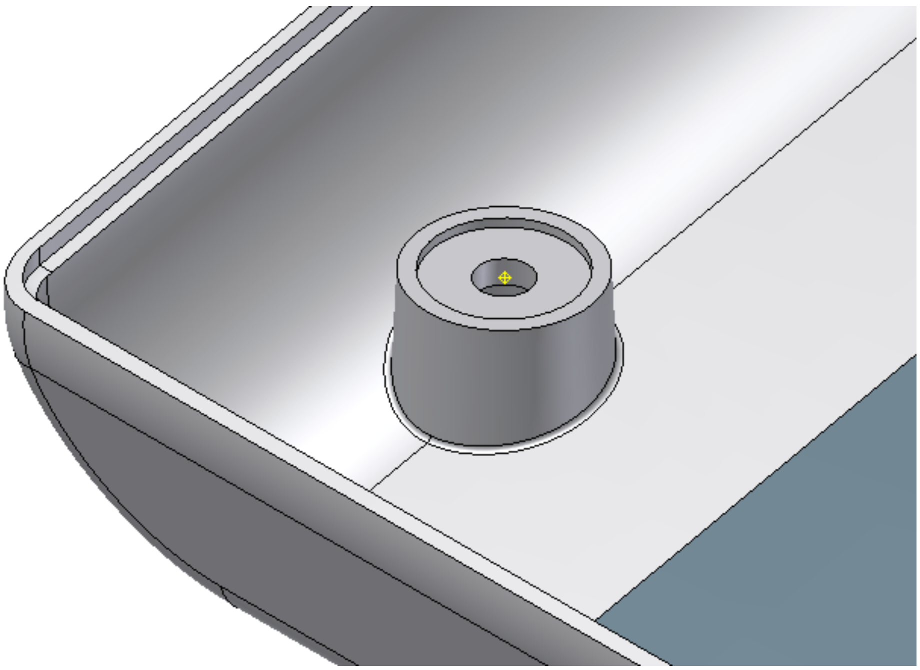
Head Diameter = 8 mm

Draft Options = 2.5 deg everywhere

Counter-bore type



7. Skip the **Ribs tab**.
8. Hit **OK**. The four bosses look like the one in the picture.



We now build the mating bosses for the thread part of the fasteners on the top.

1. Activate the window displaying the PFTutorialTop.labipt

Click on the **Boss feature** of the Plastic Features panel.

Make sure the **Thread** button is pressed.

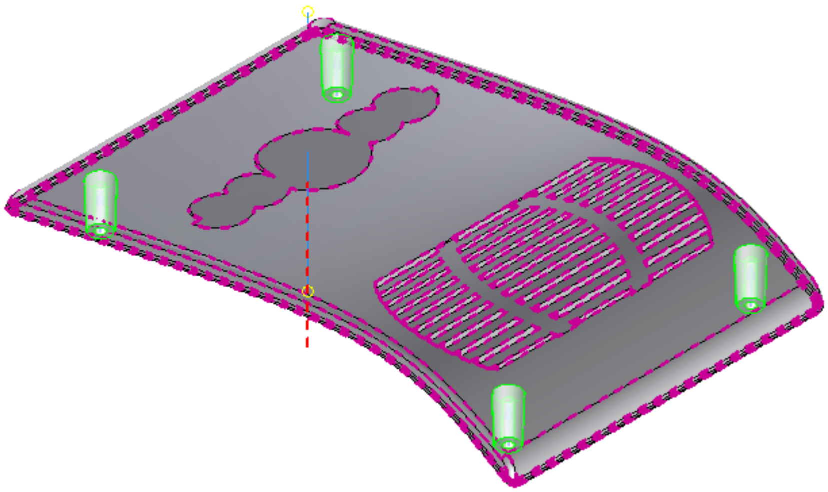
2. In the **Shape tab**:

On Point placement type

Select the four points as the *Centers.*

Select the Y axis as the *Direction*

Interaction Fillet = 0.3 mm



3. **Thread tab**:

   Hole check box on

   Full Depth

   Thread Diameter = 8 mm

   Hole Diameter = 3 mm

   Inner Draft = 1 deg

   Outer Draft = 2 deg

4. **Ribs tab**:

   2 stiffening ribs

   Thickness = 1.5 mm

   Draft = 1.5 deg

   Shoulder Length = 6 mm

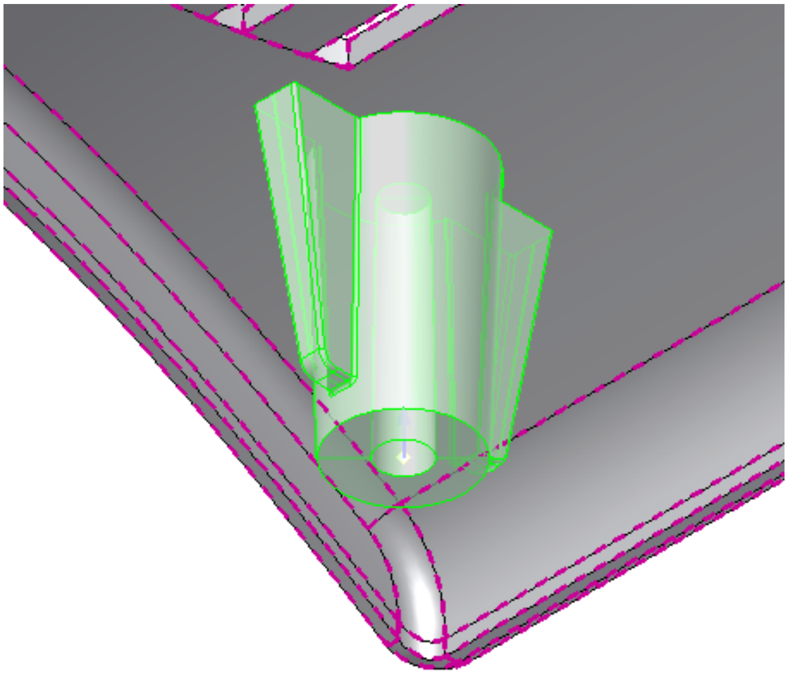
   Shoulder Top Offset = 2 mm

   Shoulder Radius = 1 mm

   Shoulder Flare Angle = 10 deg

   Fillet Options:

5. Blend Radius = 0.2 mm

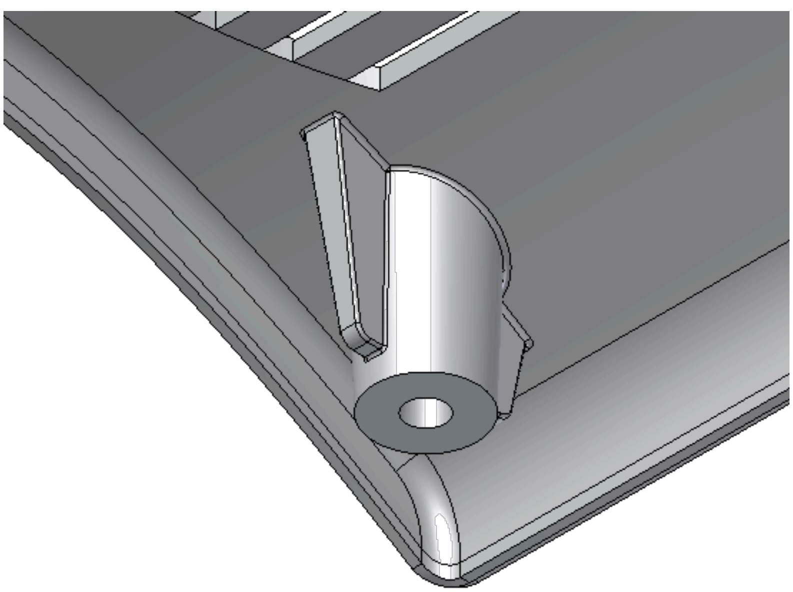


Ribs Direction

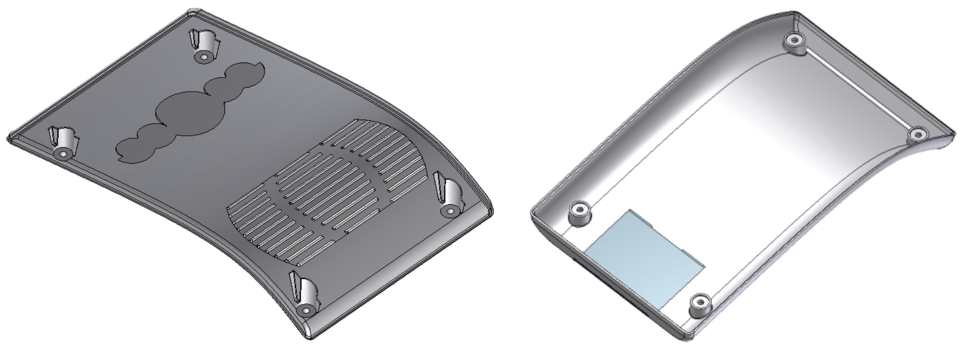
Rib Start Direction Angle = 0

Direction = select the X Axis

Hit **OK**. In the picture one of the four Boss features with strengthening ribs.



6. Now save all 3 parts and place into a single assembly. Use assembly constraints to properly mate the three parts.
7. Congratulations! The models and the Inventor Plastic Features are complete.



Expiration Date

Since it is technology preview evaluation software, Autodesk Inventor Plastic Features will no longer work after December 31st 2008. Consider uninstalling after this date.

Uninstall

From Add or Remove Programs in your Windows Control Panel, simply locate the Autodesk Inventor Plastic Features entry and select **Uninstall**.

Contacting Autodesk

Thank you for your interest in Plastic Parts Design technology for Inventor. Please direct any support issues, questions or feedback to labs.iv.plastic@autodesk.com.

Your feedback helps us make better products and technology, and it plays an important role in determining the future of capabilities to help design plastic parts. All feedback is subject to the Autodesk agreement(s) applicable to your use of Autodesk Labs.