CP9884

From Art to Part—Plastic Part Design with Product Design Suite

Jim Swain
Synergis Technologies, LLC

Learning Objectives

- Discover how to use multibody tools and derived parts in product design
- Learn how to set up Moldflow Design to monitor plastic part design
- Discover how to use Showcase to present designs and alternatives to decision makers
- Learn how to use Inventor software's new 3D print environment to set up part creation

Description

Do you need to take a concept and make it a reality? Does that reality include injection-molded parts? Do you need to get your design reviewed and approved quickly? If the answer to even 2 out of 3 of these questions is "yes," you should attend this class. We will also look at answers to these questions, and at an even more important question: How can we make changes on an overall assembly's shape while still developing individual piece details? To do this, we will explore workflows within Product Design Suite software and add a dash of Moldflow Design software for good measure.

Your AU Expert

Jim Swain *has over 25 years of engineering and CAD experience, including working in the consumer electronics and automotive industries as a design engineer, a test engineer, and a CAD administrator. For the last 18 years he has been a project manager, solutions engineer, and trainer with Synergis Technologies, LLC, an Autodesk Reseller in Pennsylvania. He has also taught college-level design classes in plastics design. Jim is an AutoCAD and Inventor Certified Professional.*

Overview

The class is an introduction to several different tools that can be used to create, analyze and present designs. Specifically, it focuses on creating plastic part designs that will be manufactured by injection molding, but some of these tools are very useful in other types of designs.

As an introduction this class won't go into all the functions of these tools. The goal is to get you started using them. Further information can then be found in the various Help files, tutorials and other Autodesk University classes.

In short – this class is to help get you past the "you don't know what you don't know" stage.

Two Keys to a Successful Plastic Part: A Very Brief Discussion

Injection molded plastic parts are manufactured by pushing hot resin into a mold where it begins cooling. Once the resin has cooled enough, the part is removed from the mold and the process is repeated. Sounds easy, doesn't it? Unfortunately as most resins cool they also shrink. This shrinkage can cause parts to stick in the mold when the mold is opened. Shrinkage can also cause aesthetic and performance defects such as: warping, excessive molded-in stresses, and sink marks. Trying to counter these in a poorly designed part can lead to long molding cycle times.

Entire college courses have been built around the techniques available to counter these challenges. There are also numerous design texts and guides available in print and online covering these topics. For this class we are going to focus on two fundamental keys that many of these techniques are built on: sufficient face draft and uniform wall thickness.

Draft

As the part shrinks it squeezes onto any protrusions into the mold cavity. The part then needs to be pushed off those protrusions to remove it from the mold. Doing this reliably and quickly is key to a low cost part. Doing this gently enough so the part isn't damaged can be critical to a product's success.

To help with this all sides of a part that are parallel to the opening direction of the mold are typically tapered away from vertical. In common terminology, they have draft.

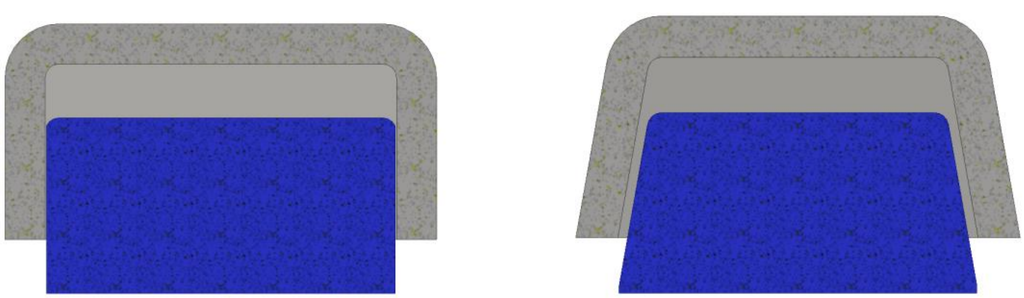

FIGURE 1: VERTICAL WALLS NOT DRAFTED VS. VERTICAL WALLS DRAFTED

Uniform Wall Thickness

A part with even shrinkage is enough of a challenge to manufacture. A part with uneven shrinkage makes all the challenges even harder.

One of the most basic methods to avoid uneven shrinkage is to keep the cross-sectional thickness of the volume that the resin has to fill as uniform as possible. This helps keep the flow of the resin into the mold smooth and avoids regions that will take longer to cool. Slow cooling leads to more shrinkage and slower manufacturing speeds.

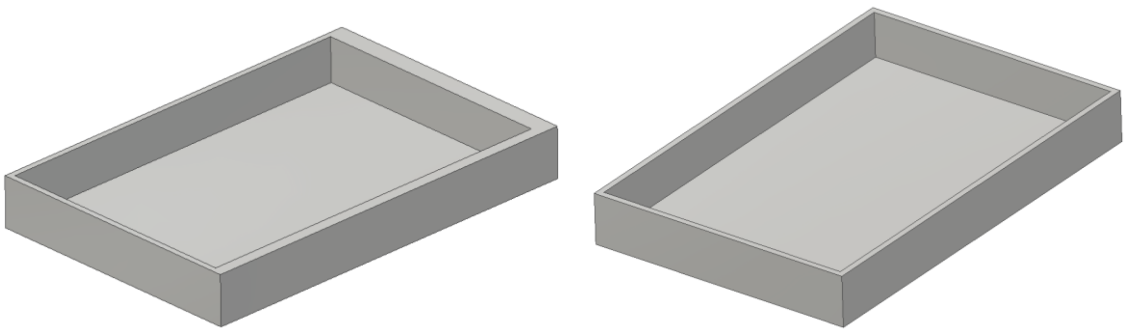

FIGURE 2: NON-UNIFORM VS. UNIFORM THICKNESS EXAMPLES

Other challenges will come up, but if you keep these two concepts firmly in mind when designing a plastic part you will have a better start.

Now let's start looking at the tools that can help us design these parts.

First comes using derived parts and multibody parts. These techniques don't address drafting and wall thickness directly. Instead, they allow parts to be related to each other. This helps changes in overall shape, which can be frequent during a design's early stages, to be incorporated throughout all the design components more easily and faster.

Using Derived Parts and Multibody Tools in Product Design

Derived Parts

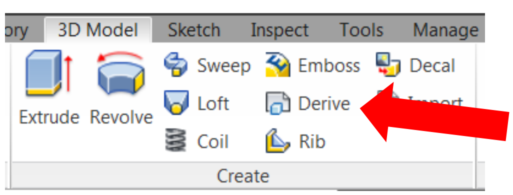

FIGURE 3: DERIVE TOOL, 3D MODEL TAB, PART FILE

A derived part is a part that is linked to another part or assembly. Its shape and parameters are based on those of the original model. The shape can be scaled and mirrored from the original, but it is still linked to the original. That link is active unless it is suppressed or broken.

Deriving from another part file allows access to the solids, surfaces, sketches, work features, parameters, etc. of the original file. This can be used to make a right hand/left hand pair of parts where the new part's shape is controlled by the original. Additional features, such as bosses and holes, can then be added to the new part. If the original part changes an update brings those changes into the new one.

If a part is derived from an assembly file all the same data is available from every part within the assembly, plus additional options like creating a simplified representation of the assembly.

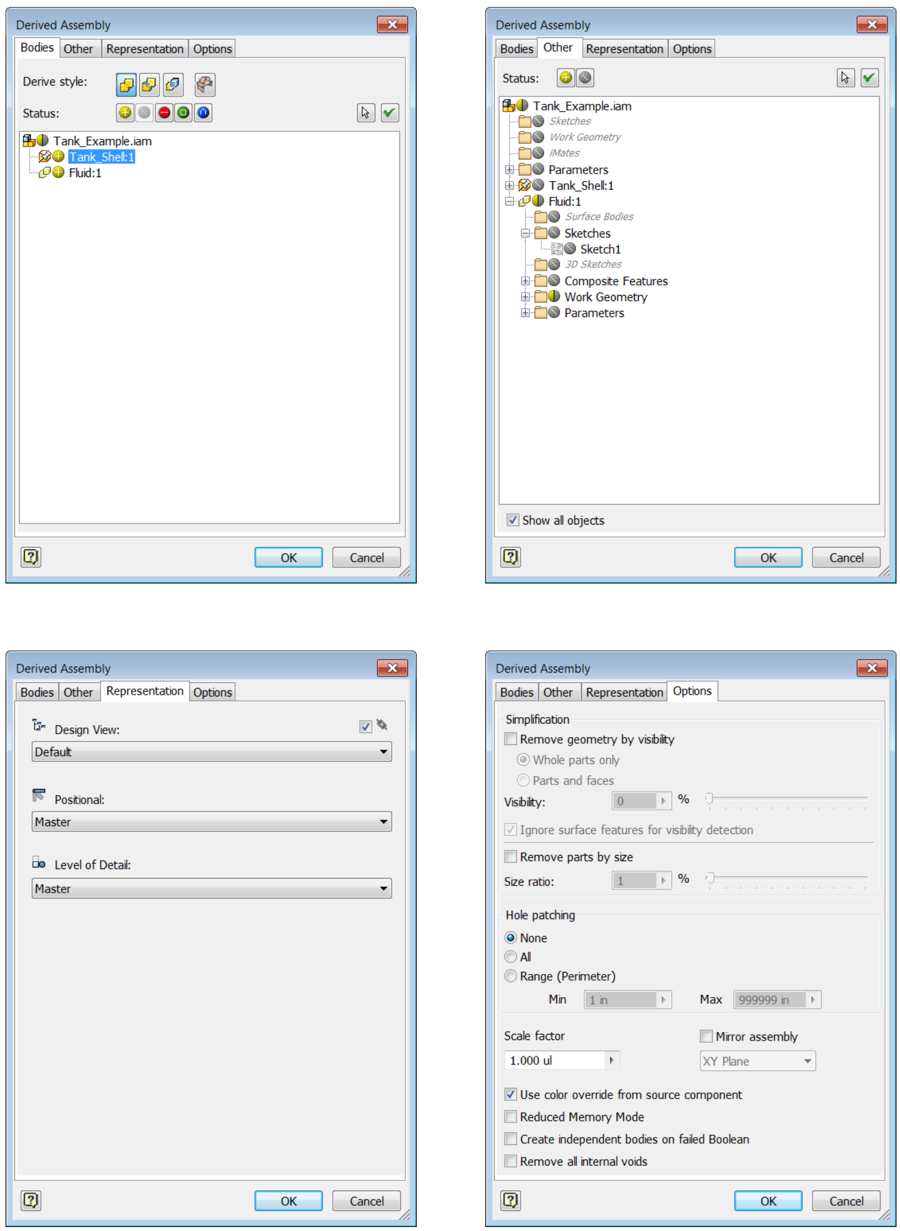

FIGURE 4: DERIVED (FROM) ASSEMBLY DIALOG BOX

Multibody Parts

A part can be modeled such that the various volumes that form the overall part's volume are divided into individual solid bodies. In many ways the part file acts like its own, self-contained assembly. Features can be created or edited and they affect all, or some of the bodies in the part. Because of this, if the overall shape of a part changes, all the bodies in the part can adjust to the change right away.

Creating multiple solid bodies can be done in two ways:

- By making new features create new solids.
- By splitting existing solids into additional solids.

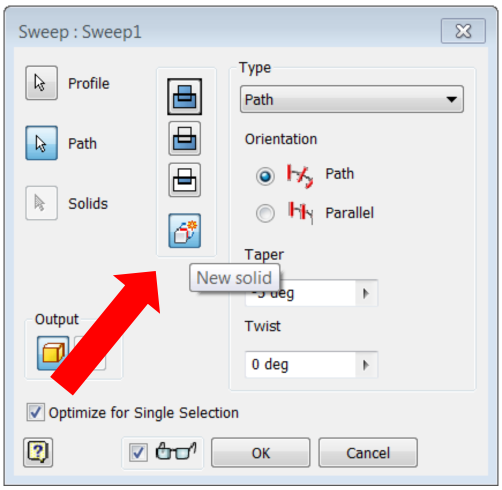

FIGURE 5: NEW SOLID OPTION (SWEEP DIALOG BOX)

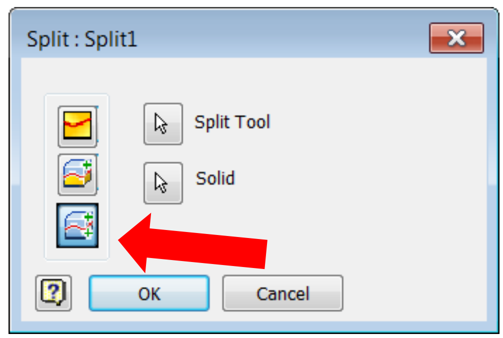

FIGURE 6: SPLIT SOLID OPTION (SPLIT DIALOG BOX)

The Browser will then show the solid bodies, and the features that were used to create them. As new features are added to the part you decide which of the various solid bodies will be affected by the feature.

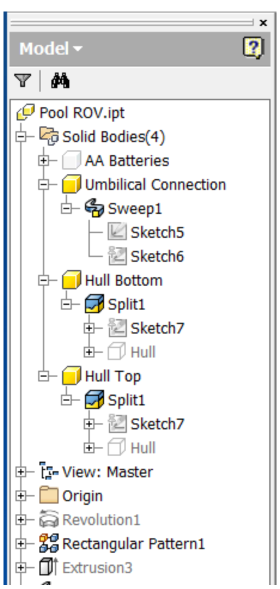

FIGURE 7: BROWSER WITH MULTIPLE SOLID BODIES

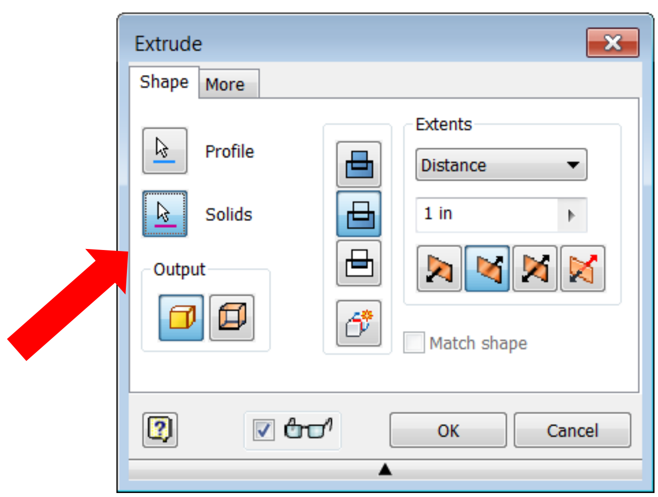

FIGURE 8: SELECT SOLIDS OPTION (EXTRUDE DIALOG BOX)

While these multiple solids make it seem like this is an assembly file it isn't, yet. Here is a comparison of the two:

Assemblies vs. Multibody Parts	Assemblies	Multibody Parts
Materials	Each part has its own material.	The entire part has the same material.
Colors	Parts can have individual colors.	Bodies can have individual colors.
Constraints	Assembly constraints, joints and other techniques develop relative positions by referencing geometric features.	Built in place. *Move Bodies* to relocate and reorient bodies. Relies on dragging and/or typed coordinates.
Tool bodies	N/A. Use Derived Part workflow for similar result.	*Combine* to join, cut and intersect existing bodies.

Advantages with Multibody Solids

When working on solo design projects it can be easier to make broad changes across an entire design when using a multibody solid. This is especially true early in a design, when the aesthetics are still under development. Sculpts and other changes to the part can affect all the solid bodies in that part at the same time.

Advantages with Assemblies

Assemblies allow multiple designers to work on individual portions of the overall design. Parts can be assigned different materials and a working Bill of Materials (BOM) can be developed early on.

Constraints can be easier to use, since they create relationships between geometry on different parts. Moving bodies in a single part file is done by discrete vectors. Constraints can also be animated to check motions and clearances.

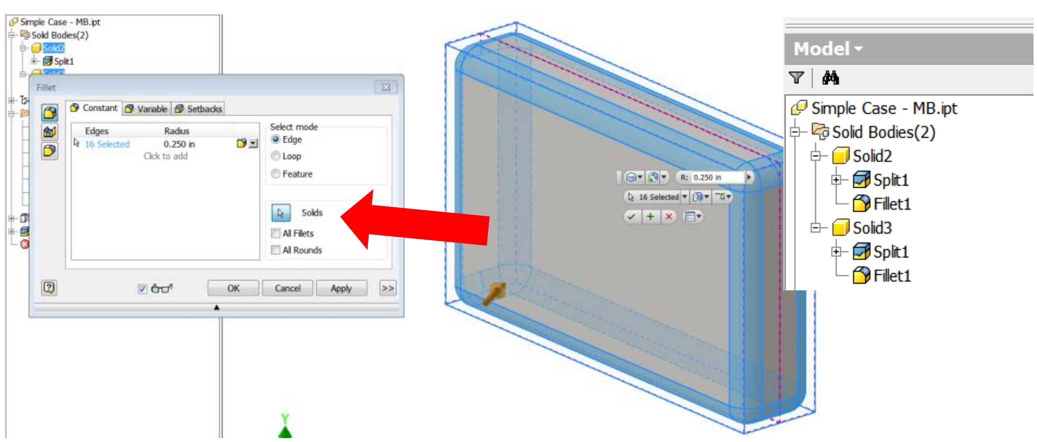

FIGURE 9: FILLET ON BOTH BODIES
INSET: BROWSER RESULT SHOWN IN INSET

Right-clicking on a solid body in the Browser gives access to several commands. **Visibility** can be toggled on or off. The **Hide Others** choice is similar to the Isolate command in an assembly. The **Properties** command allows the body to be renamed and to have its color changed from the overall part.

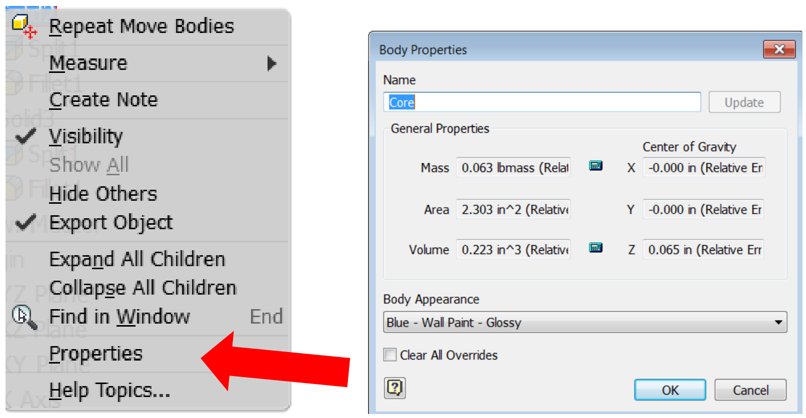

FIGURE 10: RIGHT-CLICK MENU FOR SOLID BODIES AND THE PROPERTIES OPTION DIALOG BOX

Converting Multibody Parts into Assemblies

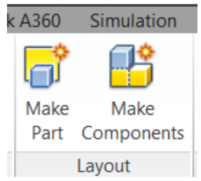

FIGURE 11: MAKE COMPONENTS TOOL, MANAGE TAB, PART FILE

Once the overall design is developed the individual solid bodies can be used to create separate part files and automatically be placed in an assembly. By default the new parts take the name of the solid bodies, so it's recommended that those bodies be renamed something descriptive before creating the assembly.

The **Make Components** tool is on the Manage tab.

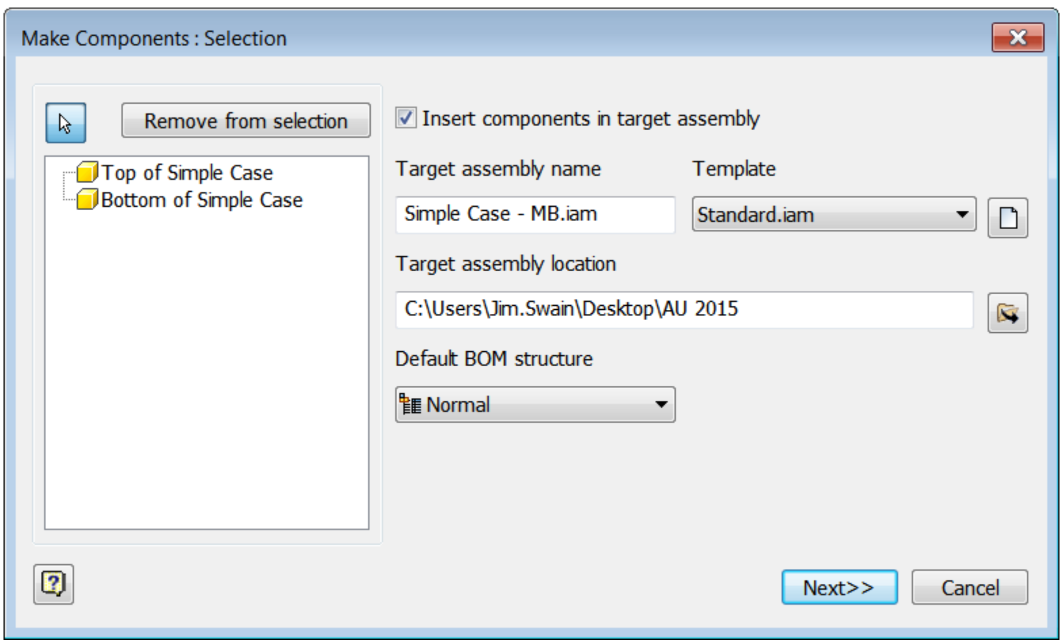

FIGURE 12: MAKE COMPONENTS DIALOG BOX, SELECTION

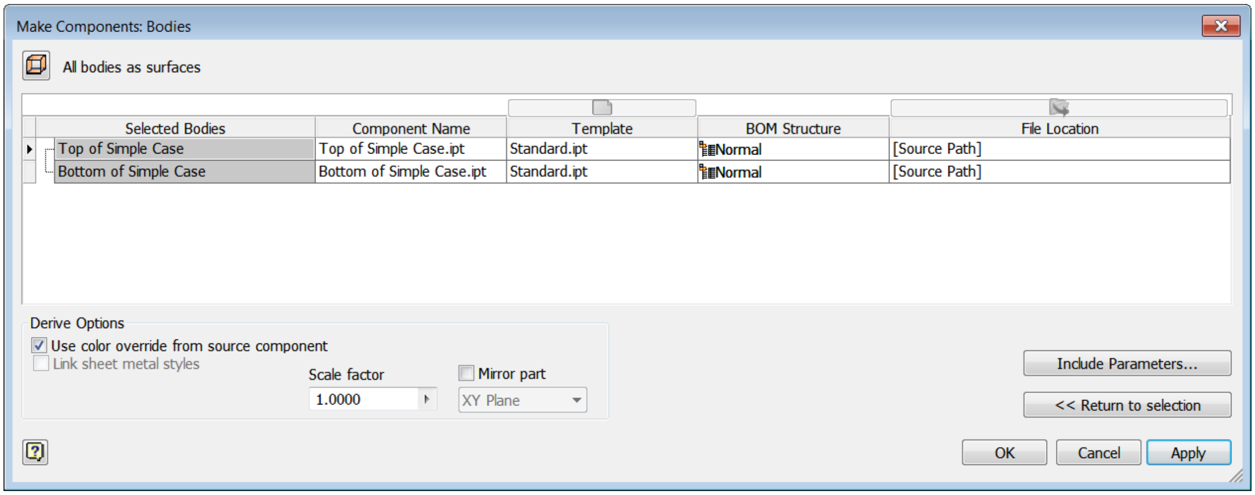

FIGURE 13: MAKE COMPONENTS DIALOG BOX, FILE NAMES AND LOCATIONS

The resulting assembly has the various parts inserted at their original locations, and grounded.

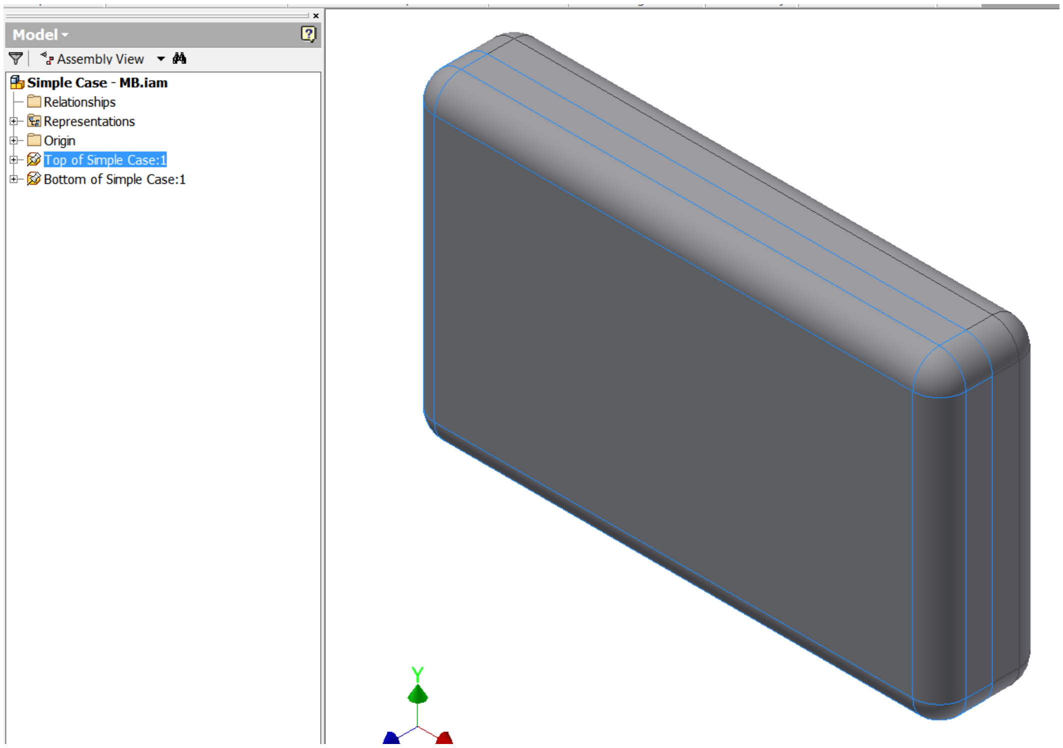

FIGURE 14: NEW ASSEMBLY FROM MAKE COMPONENTS TOOL

Using Inventor's Plastic Design Tools

Common Inventor Tools

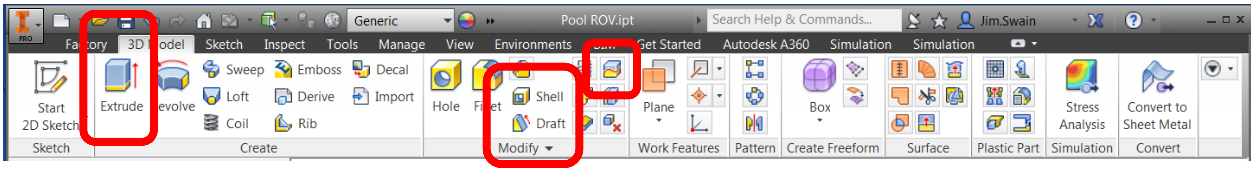

FIGURE 15: USEFUL TOOLS, 3D MODEL TAB, PART FILE

Here are some brief comments about using a few typical Inventor tools when designing plastic parts:

Extrude

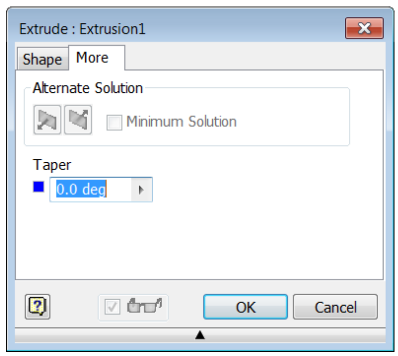

FIGURE 16: EXTRUDE DIALOG BOX: MORE TAB

The More tab of the Extrude dialog box allows a taper angle to be set for the entire extrusion. Consider leaving this at the default of 0° except for very simple shapes, such as for a basic rectangular block. Otherwise the model's parting line could be forced to an unintended location. Instead add draft using the Face Draft tool, as this gives more precise control over the final parting line location.

Shell

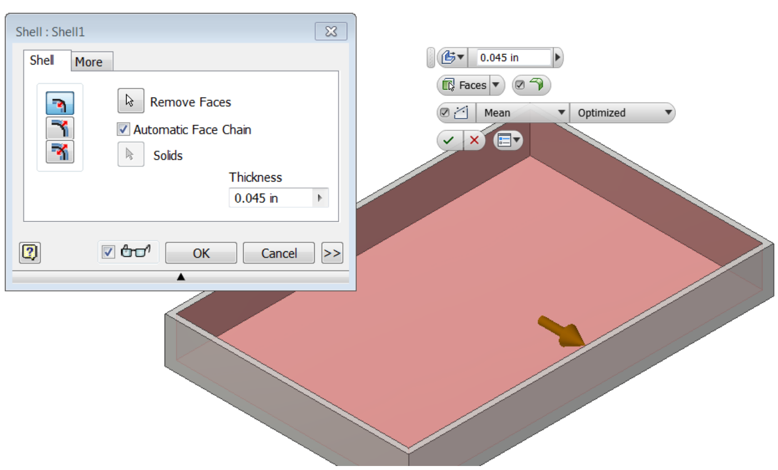

FIGURE 22: SHELL COMMAND: DIALOG BOX AND IN CANVAS MENU

Shell sets up the initial uniform wall thickness for a part. The thickness will be by the resin selection, desired part strength and other design requirements.

After the basic shape is shelled, internal details can be added. If the overall shape of the design needs to be altered it may be necessary to move the End of Part (EOP) marker to before the shell, make the changes, then move the EOP marker back to the bottom of the Browser. Even with this technique it is still possible that existing features will need to be redefined or recreated.

Draft/Face Draft

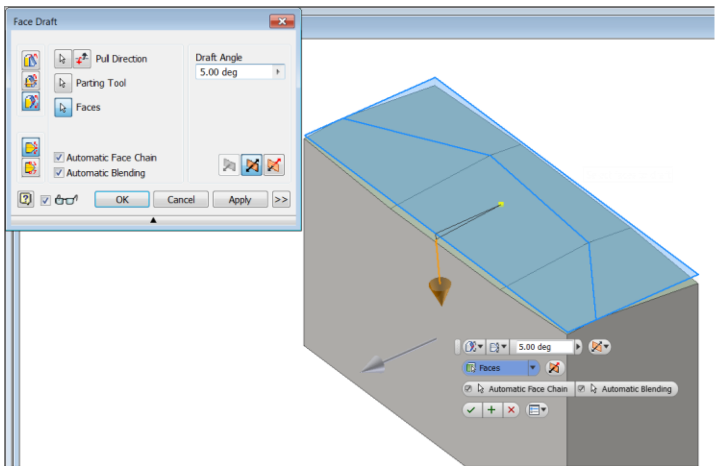

FIGURE 17: FACE DRAFT

The Face Draft tool allows individual faces to be rotated. This allows taper to be added to those faces that are parallel to a defined pull direction. Faces can be done individually, or in a group.

A challenge with the Face Draft tool is that having a fillet on faces being drafted will often prevent the draft from completing. Add any face drafts first, then add fillets and chamfers.

In this example the inner faces are being drafted to a different angle than the outer faces. This might be done for aesthetic reasons or for better ejection. The face drafts are added first, then the fillets along inner edges. Finally any outer fillets are added.

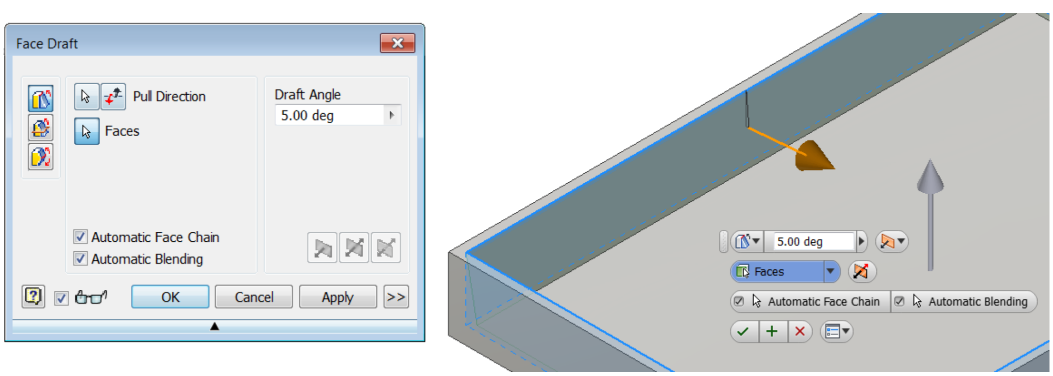

FIGURE 18: FACE DRAFT BEFORE FILLETS

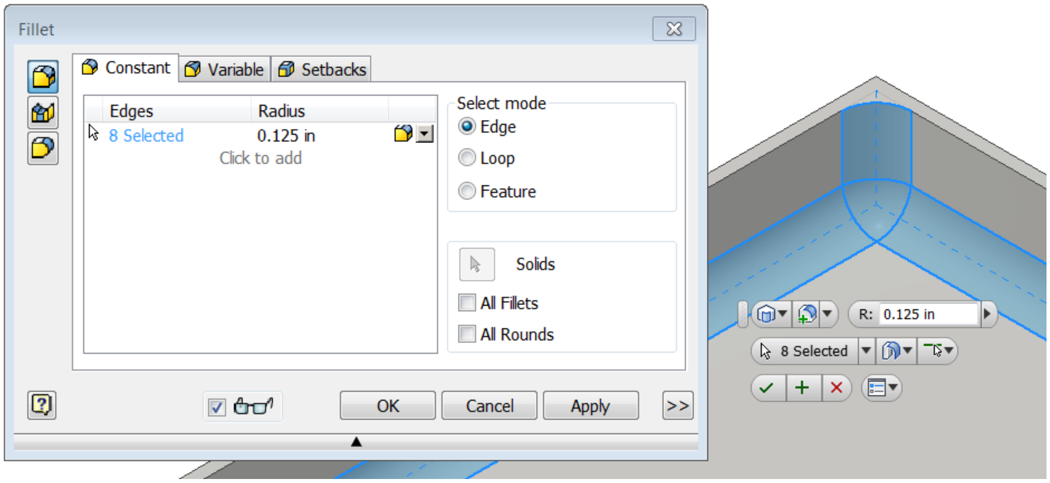

FIGURE 19: INTERIOR FILLETS AFTER FACE DRAFTS

NOTE EXTRA WALL THICKNESS IN CORNER

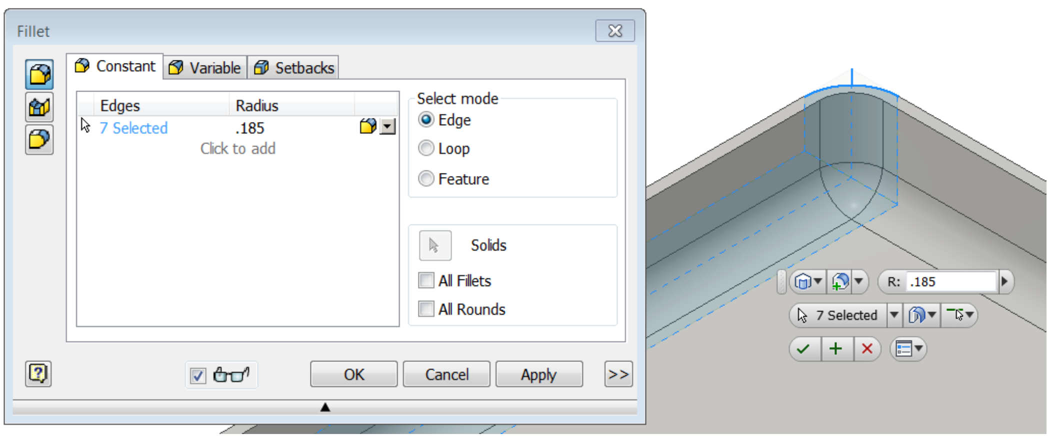

FIGURE 20: EXTERIOR FILLETS AFTER INTERIOR FILLETS

Split

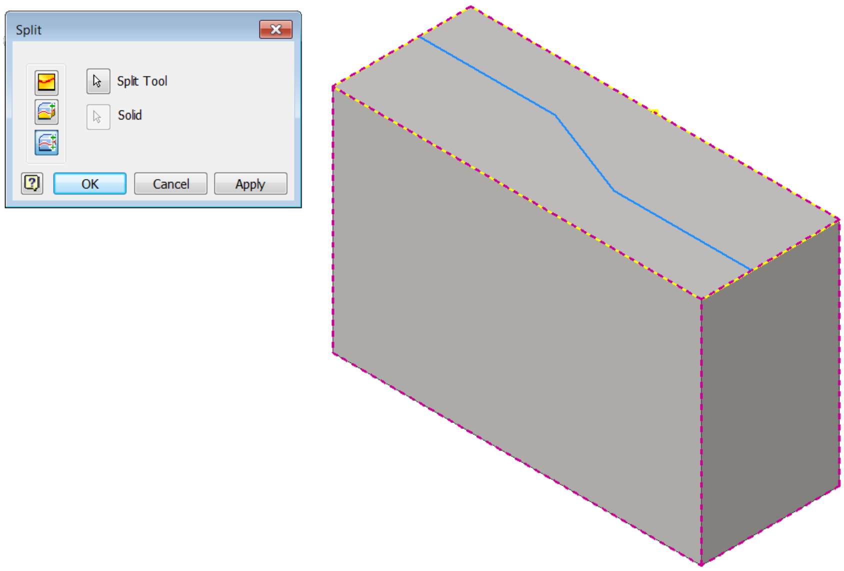

FIGURE 21: SPLIT COMMAND: SPLIT SOLID OPTION HIGHLIGHTED

As shown earlier, the Split tool may be used to break existing solids in two. For injection molded individual solids it is also useful for breaking faces along sketched curves or surfaces to define the mold's parting line.

Accessing Additional Plastic Part Tools

Additional Inventor plastic part tools are available on the Inventor part's 3D MODEL tab, though they are likely not displayed by default. You can access the Plastic Part panel with the drop down arrow at the far right edge of the panel area or by right clicking on an existing panel.

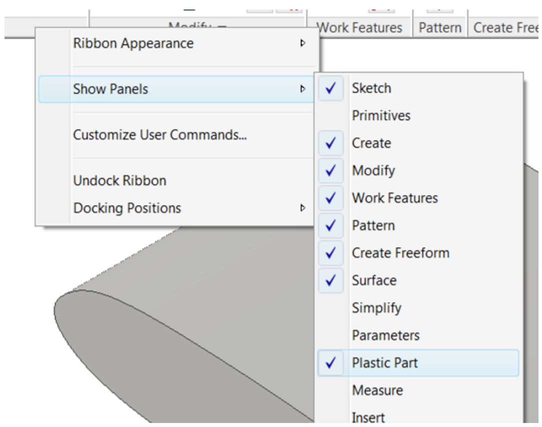

FIGURE 23: DISPLAYING THE PLASTIC PART PANEL

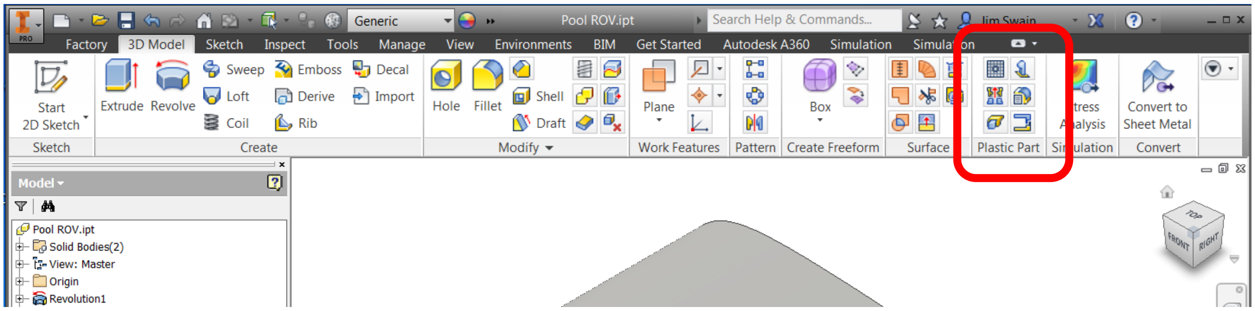

FIGURE 24: PLASTIC PART PANEL, 3D MODEL TAB, PART FILE

FIGURE 25: PLASTIC PART PANEL UNDOCKED

The Plastic Part panel is shown above in both its default docked position and undocked. Let's take a look at the individual tools.

Grill

A grill feature uses one or more sketches to create complex grill work. The grill can be used to protect an air vent or speaker opening. When the dialog box is expanded the flow area through the grill is also displayed.

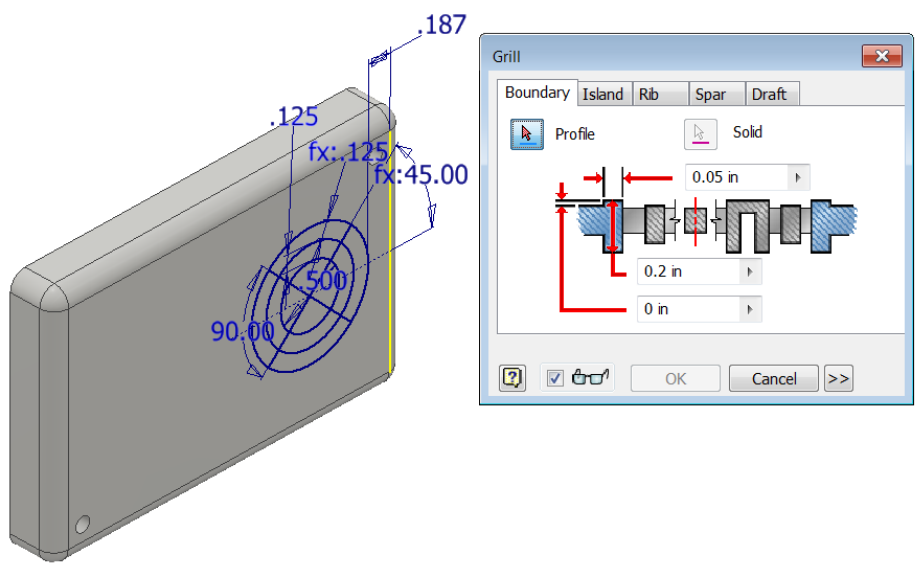

FIGURE 26: GRILL DIALOG BOX (NOT EXPANDED)

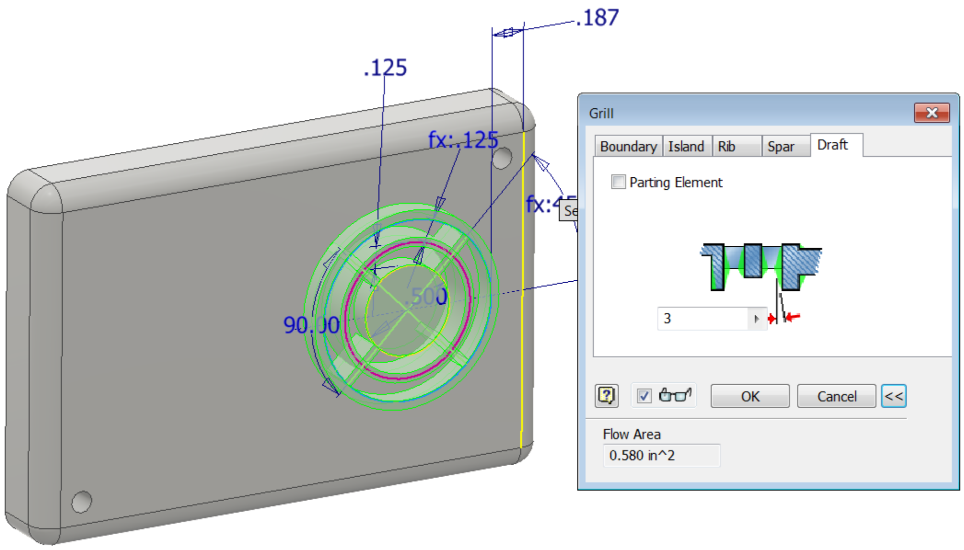

FIGURE 27: GRILL DIALOG BOX (EXPANDED)

Snap Fit

A cantilever beam latch or hook is developed at a point. The various details of the beam's design are controlled by parameters in the dialog box. You will need to calculate the stress levels in the beam to know if the resulting shape will be strong enough for on your design conditions. That isn't included in the tool.

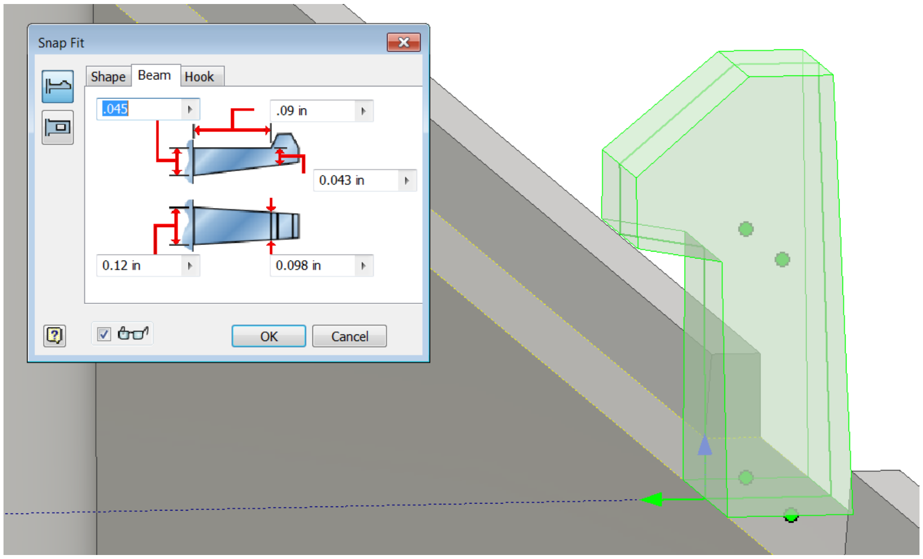

FIGURE 28: SNAP FIT DIALOG BOX (HOOK OPTION)

Boss

A boss feature typically is either a projection for supporting the head of a screw or a projection for the screw to thread into. Both can be created from this tool. Supporting ribs can also be included directly in the feature.

A sketch is used to locate the bosses on the part. The height of the sketch plane determines the height of the boss above the rest of the part.

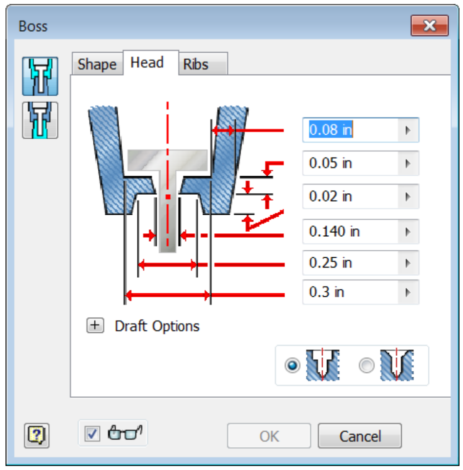

FIGURE 28: BOSS DIALOG BOX

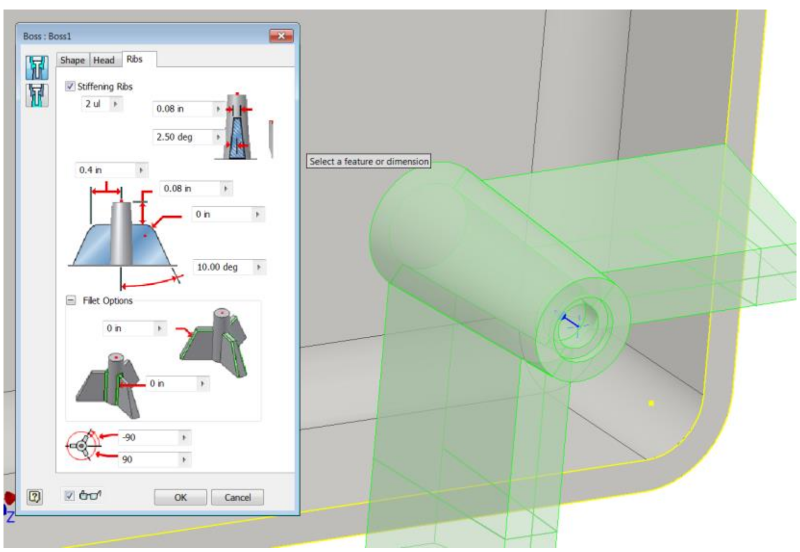

FIGURE 29: BOSS DIALOG BOX, RIB OPTION WITH PREVIEW

Rule Fillet

The Rule Fillet tool allows the creation of large numbers of fillets with a single step. The "rules" refer to the ability to apply different radii and tangency conditions, based on what type of edges the fillet would affect.

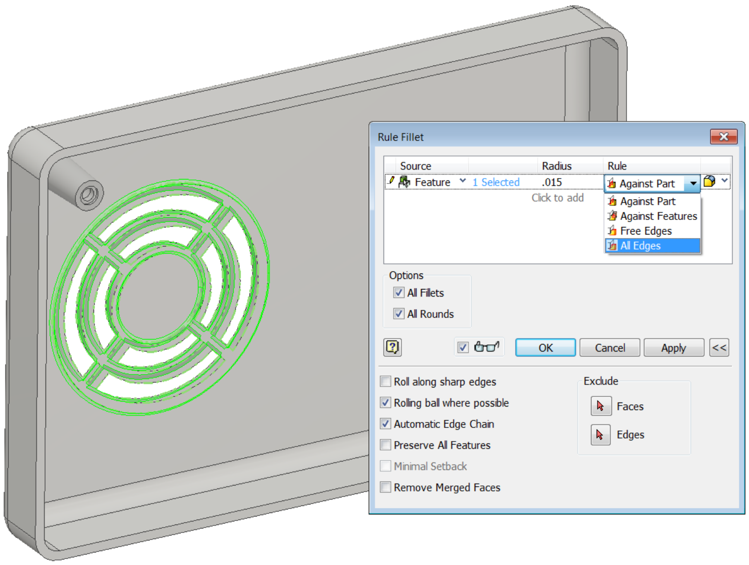

FIGURE 30: RULE FILLET DIALOG BOX (EXPANDED)

Rest

A rest is a platform molded into the part. The shape of the platform is sketched, then Inventor attaches it to the remainder of the part by creating new walls. The platform could be used for mounting other parts, or as a positioning feature.

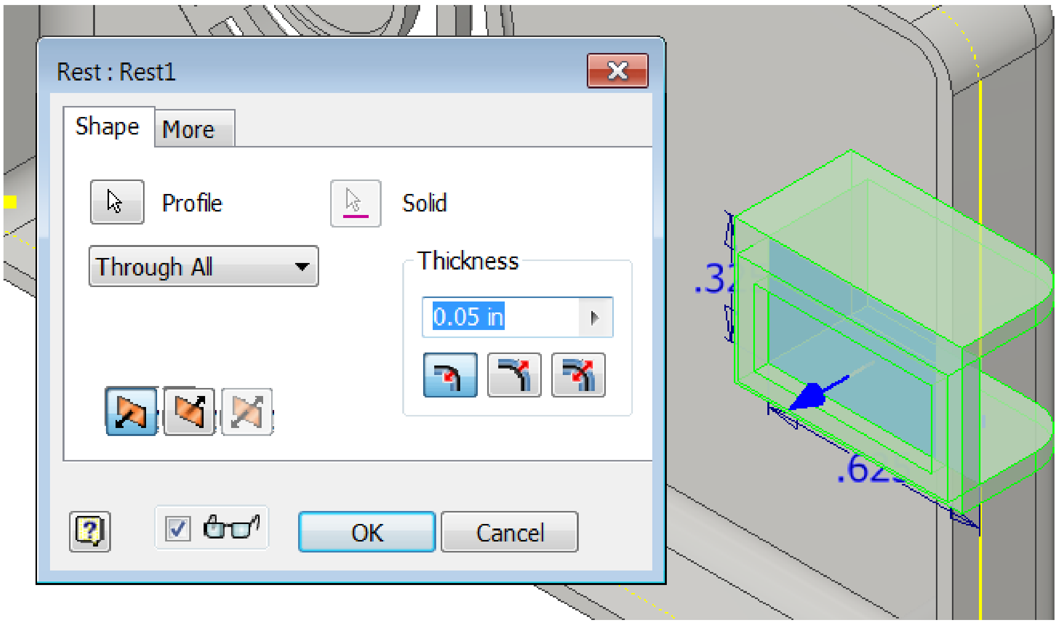

FIGURE 31: REST DIALOG BOX

Lip

Overlapping lips are often applied to edges where plastic parts meet. These lips help hide any deformation of the molded parts. They could be created with sweeps, but the Lip command simplifies the process.

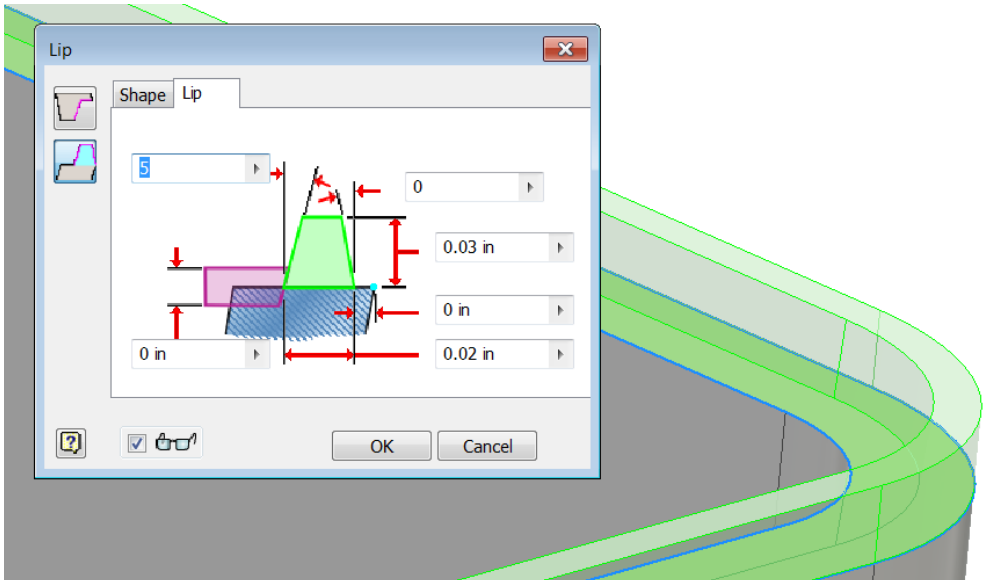

FIGURE 32: LIP DIALOG BOX

Helpful Analysis Tools in Inventor

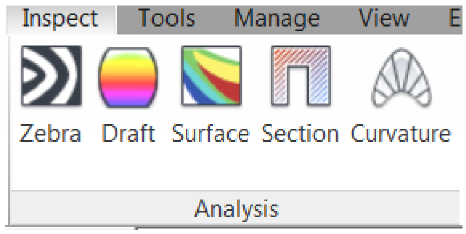

FIGURE 33: ANALYSIS PANEL, INSPECT TAB, PART FILE

Inventor has some analysis tools that are very useful when working with plastic parts.

Zebra

This analysis projects a striped image onto the part, almost like looking at the reflection of a picket fence onto a car's fender. This is useful for finding discontinuities along a part's surface.

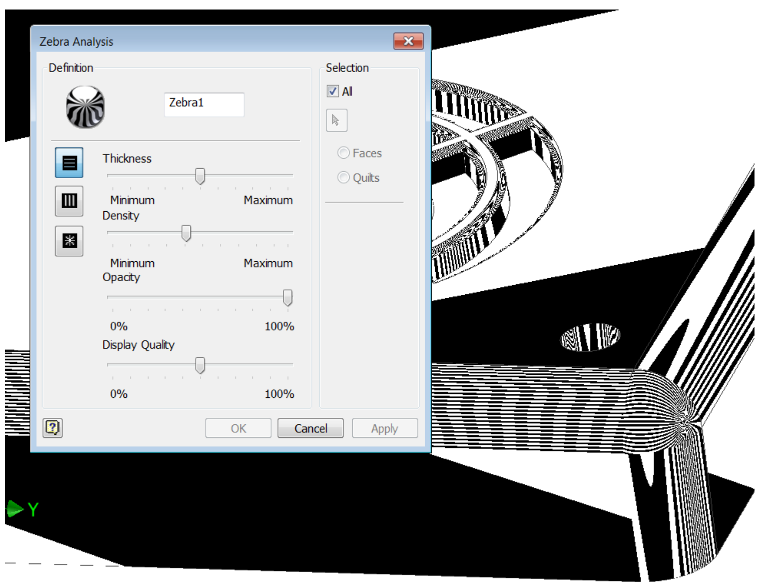

FIGURE 34: ZEBRA ANALYSIS BOX AND RESULT

Draft

The draft analysis will give a color coded display of the relative angle of a part's surfaces. This angle is measured with respect to either a face or edge defining the expected mold opening direction. This is especially useful for parts that will have molded in texture.

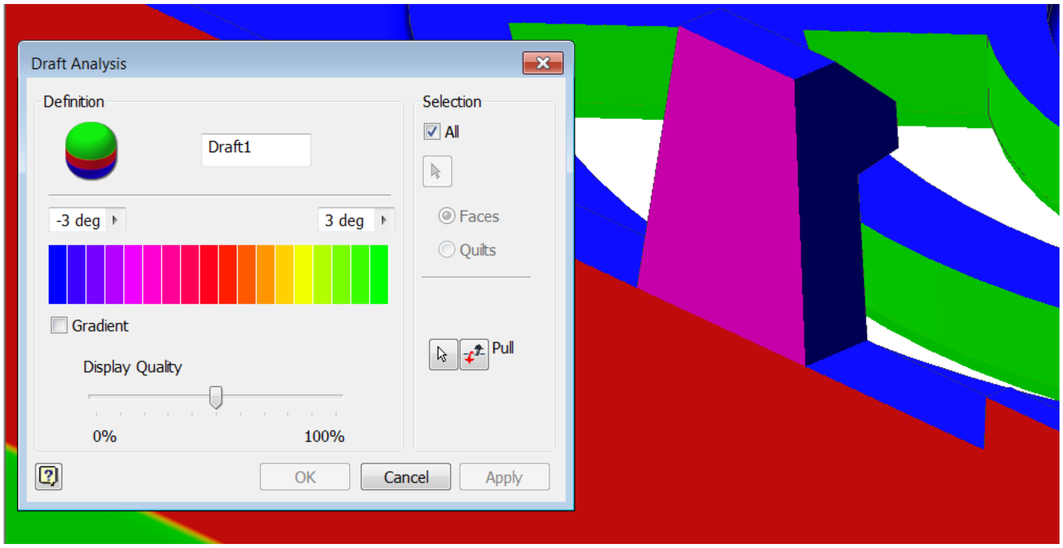

FIGURE 35: DRAFT ANALYSIS DIALOG BOX AND RESULT

Surface

This analysis gives a color coded display of the curvature of a part's surfaces. It gives a better overall impression of a part's shape than the Curvature analysis.

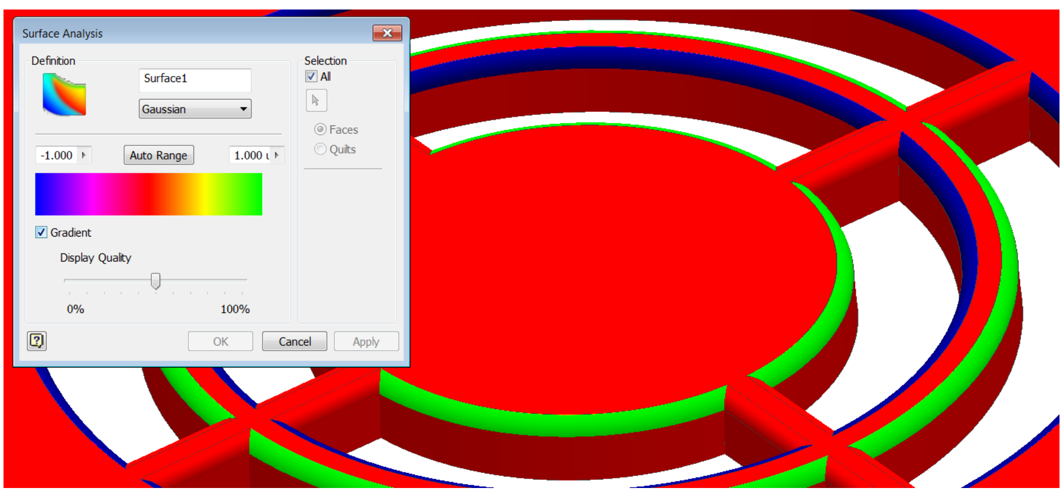

FIGURE 36: SURFACE ANALYSIS DIALOG BOX AND RESULT

Section (Cross Section Analysis)

Single or multiple cross sections can be calculated through a part. Results can include wall thickness and cross sectional areas. This is helpful for checking for wall thickness variations that can cause shrinkage problems and choke points for mold filling.

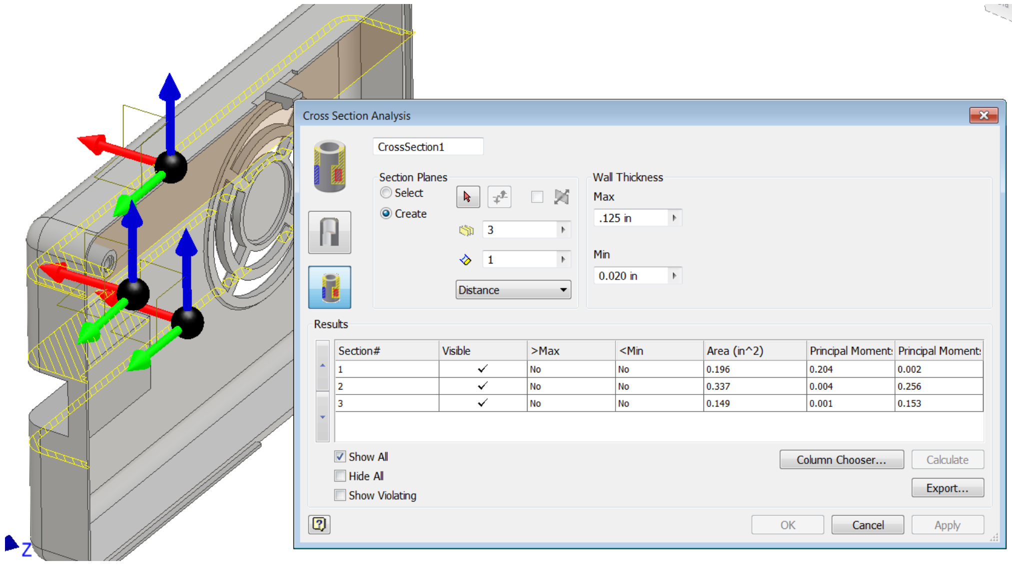

FIGURE 37: CROSS SECTION ANALYSIS DIALOG BOX AND RESULT

Curvature

This analysis is essentially the same as the Surface analysis, but the results are displayed as splines instead of color bands. The distance the spline is offset from the part's surface is proportional to the curvature of the part at that location. Large offsets indicate rapid changes in curvature.

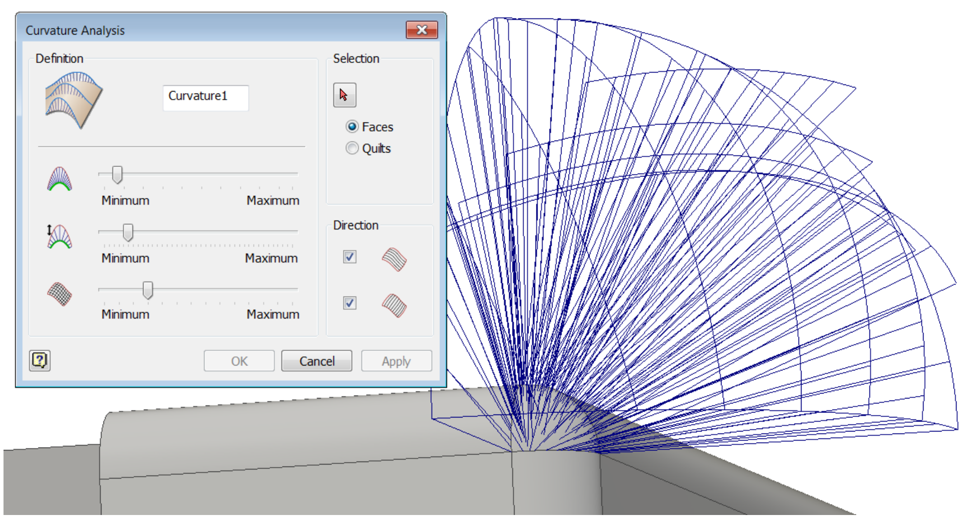

FIGURE 38: CURVATURE ANALYSIS DIALOG BOX AND RESULT

The analyses are listed in the Browser, above the Solid Bodies node. They can be turned on and off or deleted from the Browser.

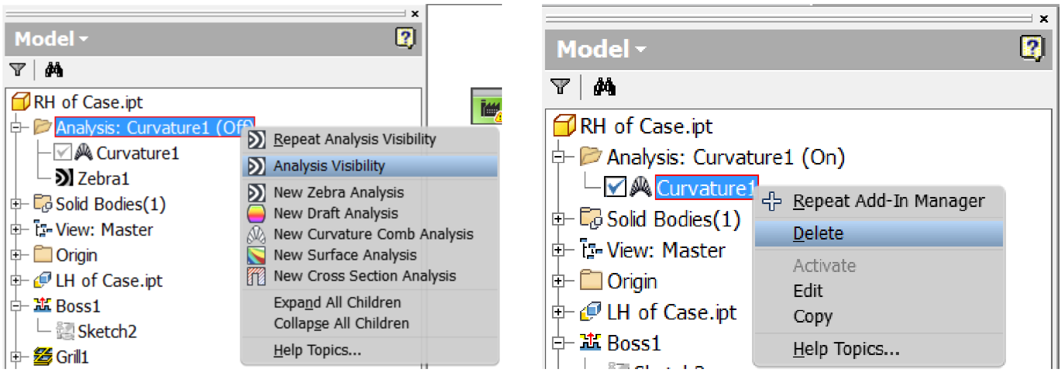

FIGURE 39: ANALYSIS NODE IN BROWSER

Using Moldflow Design for Injection Molded Parts *(formerly Simulation DFM)*

As stated at the start, two of the major design goals of an injection molded part are to keep a uniform wall thickness and have sufficient draft across the design. Moldflow Design is a tool that keeps real-time track of how close to these ideals a given design is. It was developed from the Moldflow family of injection analysis software, and though it isn't a part of the Product Design Suite it does run within Inventor.

Moldflow Design monitors part wall thickness, draft angles, undercuts and other design features. It can also predict filling patterns, resulting weld lines and potential sink marks. And it does this continuously as your design evolves.

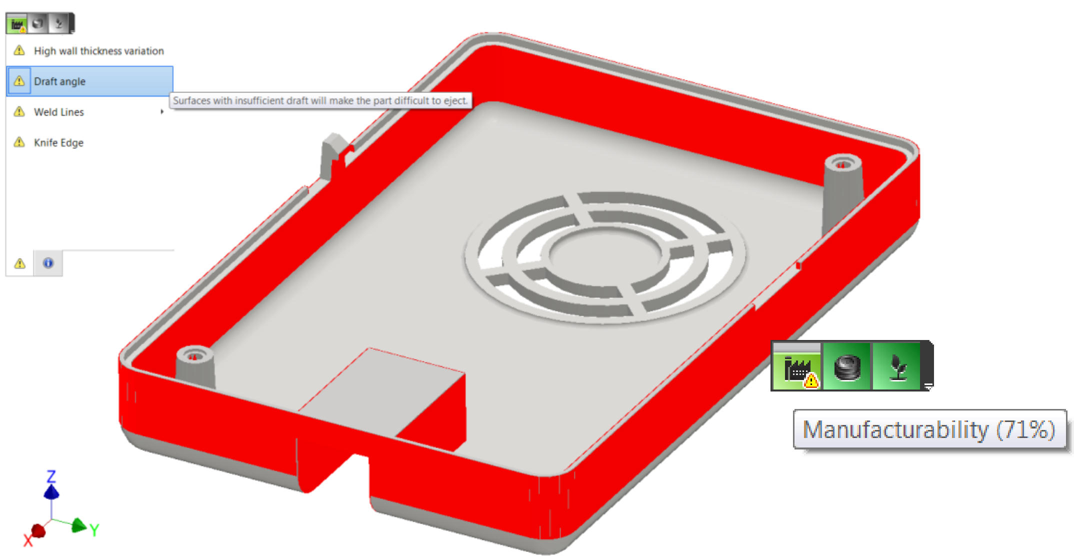

FIGURE 40: MOLDFLOW DESIGN, SHOWING DRAFT RESULTS
INSET: CURRENT OVERALL MANUFACTURABILITY RATING FOR DESIGN

Clicking on the Moldflow Design widget will allow you to select which of the analysis output you want to see. Clicking on it again will hide the output display.

Hovering over the widget will give the current design performance score.

Since the tool is continuously monitoring the design it can affect system performance. You can toggle the automatic Click on the Refresh arrow on the lower right corner of the widget to recheck the design. This workflow means you have to remember to check your design, but it does improve system performance. You can also turn off the tool by right clicking on it, or by using the Add-in Manager on the Environments ribbon tab.

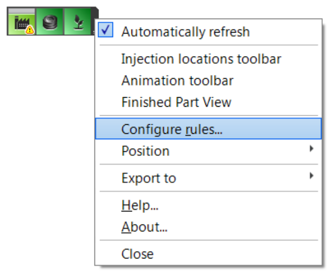

FIGURE 41: MOLDFLOW DESIGN RIGHT-CLICK MENU

Here are the results you can expect from Moldflow Design with respect to uniform wall thickness and draft angle:

Wall Thickness Variation
This tool will look for areas of the design that significantly change from the average. The alerts badge indicates if there are areas that violate the design rules. The information tab will shade the entire part, based on the wall thickness.

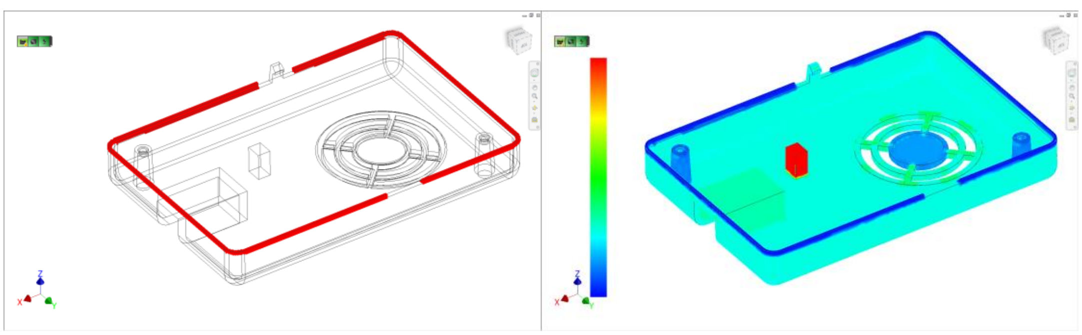

FIGURE 42: MOLDFLOW DESIGN: WALL THICKNESS VARIATION

Draft Angle
This tool will look for surfaces that are too close to vertical, with respect to the direction of pull for the design. Red areas may drag in the mold during opening and ejection.

The direction of pull is automatically determined by the software. The software checks possible undercuts, the bounding box for the part, and finally the part's Z axis. A more complete description of the calculation is included in Moldflow Design's Help system.

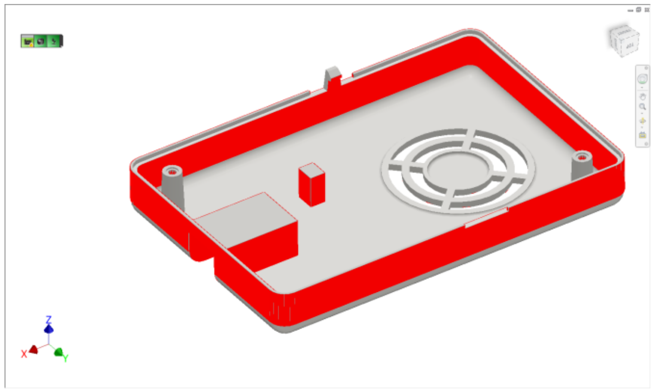

FIGURE 43: MOLDFLOW DESIGN: DRAFT ANGLE

Filling

Moldflow Design will show an animation of the expected flow paths for the resin as it is injected into the mold, based on wall thicknesses and injection locations.

FIGURE 44: MOLDFLOW DESIGN: MOLD FILLING

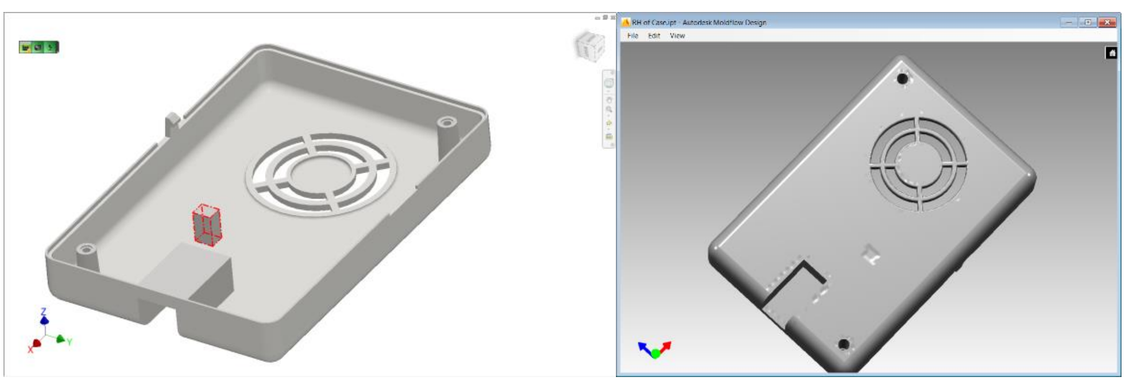

FIGURE 45: MOLDFLOW DESIGN: SINK MARKS AND FINISHED PART VIEW

Setting Moldflow Design Parameters

The calculations in Moldflow Design are based on parameters that can be set in the Configure Rules dialog box. Right-click on the Moldflow Design widget to access this dialog box:

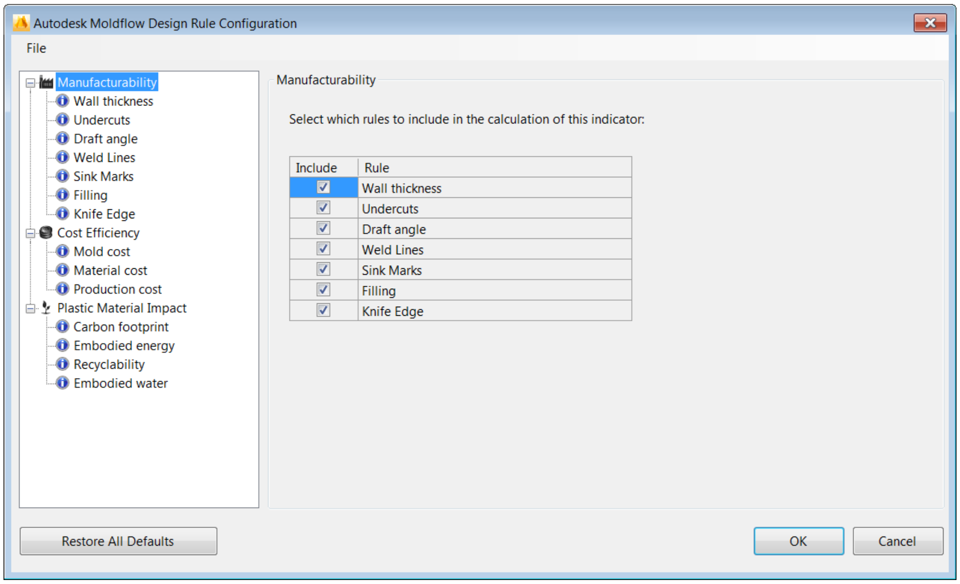

FIGURE 46: MOLDFLOW DESIGN CONFIGURE RULES DIALOG BOX

The default parameter values can be tweaked as needed. The default values for the Wall Thickness tool are shown here:

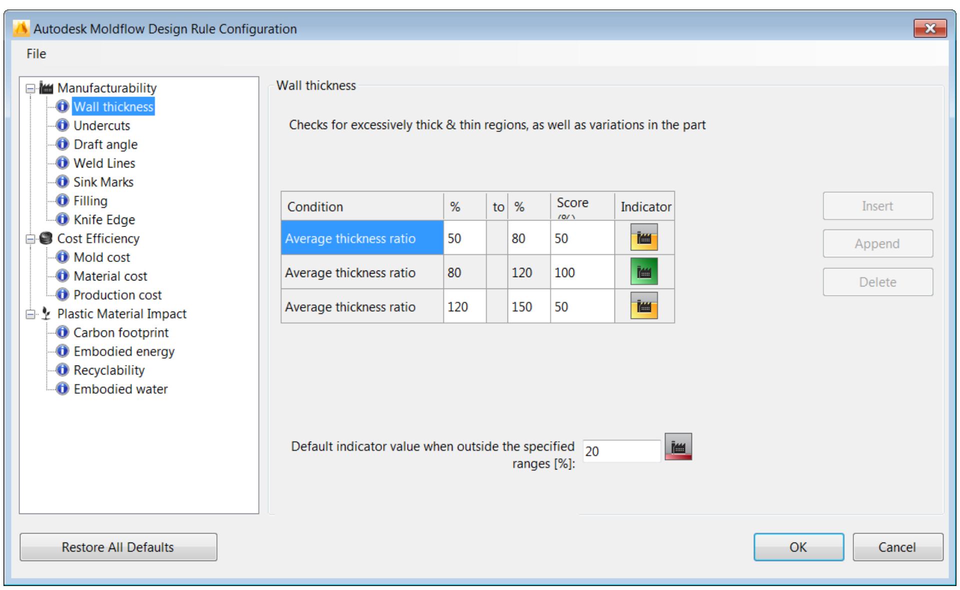

FIGURE 47: MOLDFLOW DESIGN CONFIGURE RULES DIALOG BOX, WALL THICKNESS SETTINGS

Using Showcase for Product Presentations

Autodesk Showcase is a visualization tool included in the Product Design Suite. It can be launched directly from Inventor, or launched separately and files can then be imported. Once files are imported the design can be viewed from any angle. Colors can be changed easily. Lighting and backgrounds can be swapped. Product configurations, including alternate designs and motions can be viewed for discussion.

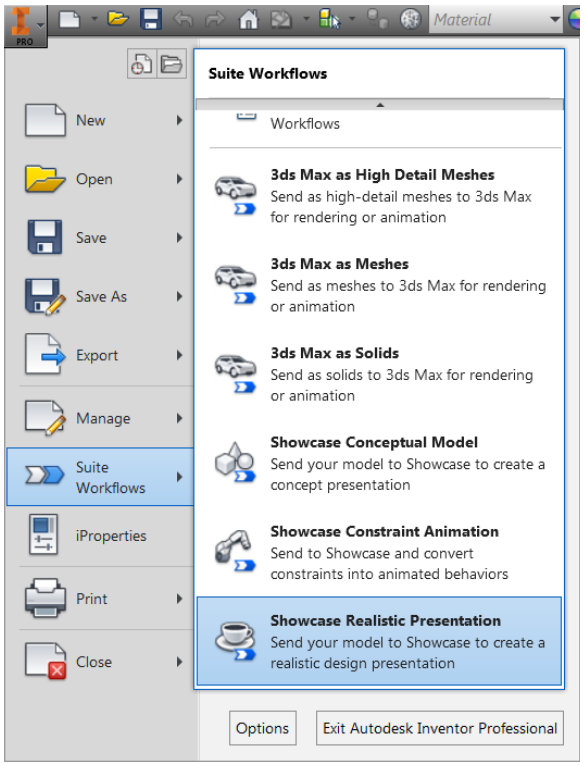

FIGURE 48: STARTING SHOWCASE FROM INVENTOR

Importing Models

Multiple files can be brought into the same scene to help present a design. They can be from many different programs.

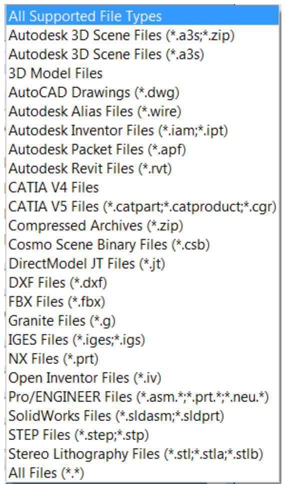

FIGURE 49: SHOWCASE IMPORT FILETYPES

On occasion models will import with the wrong units applied. If this happens, open the **Import Status** dialog box, right click on the model and choose the **3D Model Properties** option. This dialog box will allow you to change the units of the model and toggle its "up-axis".

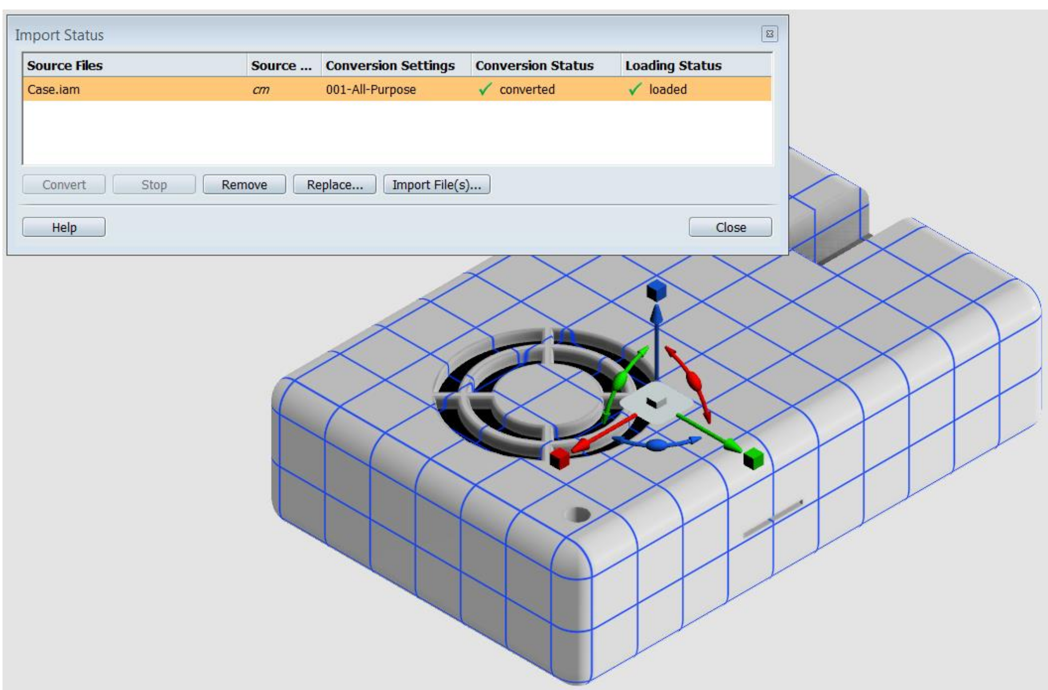

FIGURE 50: SHOWCASE IMPORT STATUS DIALOG BOX

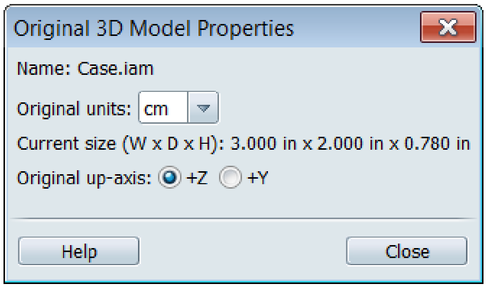

FIGURE 51: SHOWCASE IMPORT 3D MODEL PROPERTIES

Changing Lighting and Backgrounds

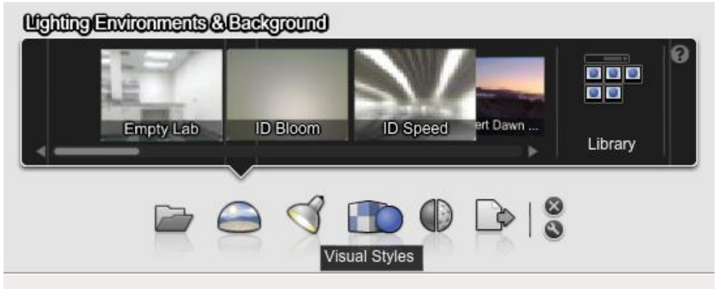

FIGURE 52: SHOWCASE IN-CANVAS MENU: LIGHTING ENVIRONMENTS OPTION

Showcase's Lighting Environments Library contains a wide selection of pre-defined environments. Some are aimed at larger models, such as vehicles. Some, such as the ones with names starting with **ID**, are aimed at smaller models.

FIGURE 53: MODEL IN LIGHTING ENVIRONMENT

Moving Models

When changing environments it may be necessary to adjust where the model is with respect to the floor. The simplest tool for this is to use the **Set Environment Floor Position** and adjust the floor as desired.

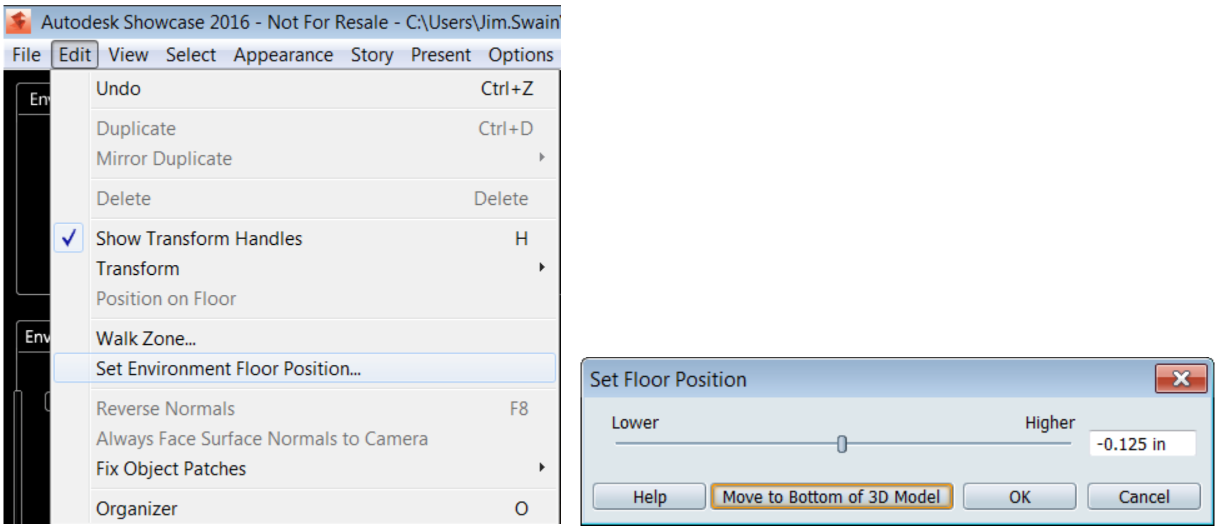

Figure 54: Adjusting the Environment's Floor

If adjusting the floor isn't enough, then the model will have to be moved in the environment. This is done with the transform tool. Select the model(s) as you would in Inventor. Control+A can be used to select all models in the scene. Transform handles will be displayed for the selected items. These can be used to drag and rotate the selected items.

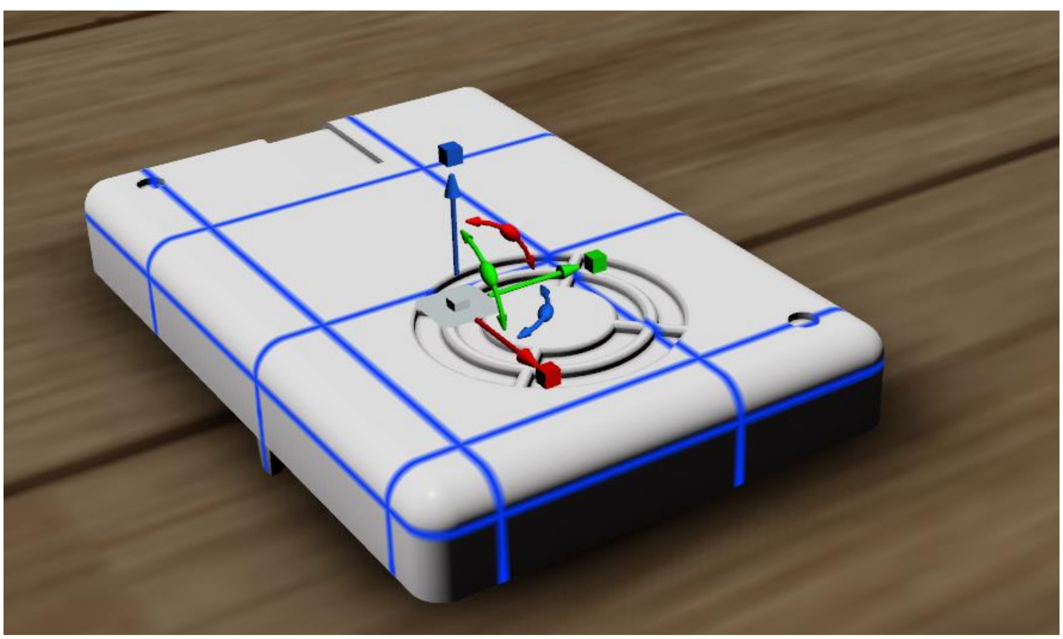

Figure 55: Transform Handles

Changing Appearances

The colors and other factors in a model's appearance is controlled by its material. The Material Library contains a large selection of predefined materials, complete with their colors and textures. Additional materials also can be defined if needed.

FIGURE 56: TWO DIFFERENT MATERIALS ON THE SAME MODEL

Showing Alternative Designs

Once variations are created they can be saved as Alternatives. Alternatives can be toggled to show the various choices.

Alternatives can then be saved as part of a slide show for a more formal presentation.

FIGURE 57: TWO MATERIAL ALTERNATIVES

Using the 3D Print Environment

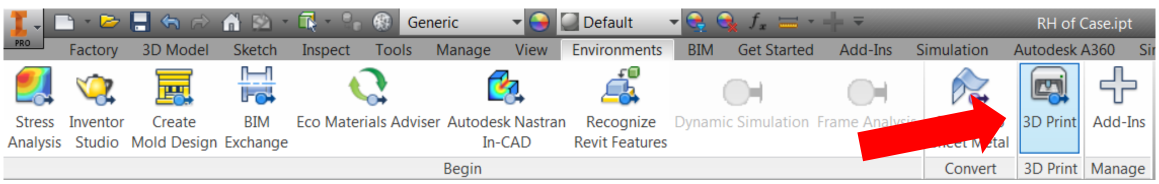

FIGURE 58: 3D PRINT PANEL, ENVIRONMENTS TAB, PART FILE

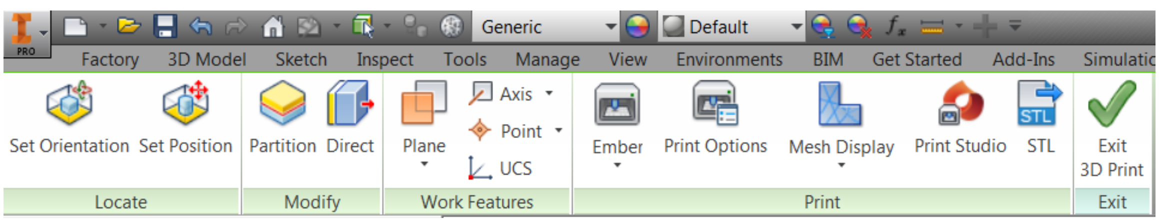

FIGURE 59: 3D PRINT CONTEXTUAL RIBBON

Choosing the Device

The first step is selecting the device to use for the 3D printing. This will set the working envelope for the printer, which will help to determine the orientation and position of the part in the device. Many brands and machines are available from the menu. The choice can either be made from the drop down menu, or take the **Other Printers** option to see the full list. This list is also useful as it shows the working envelope for the different machines.

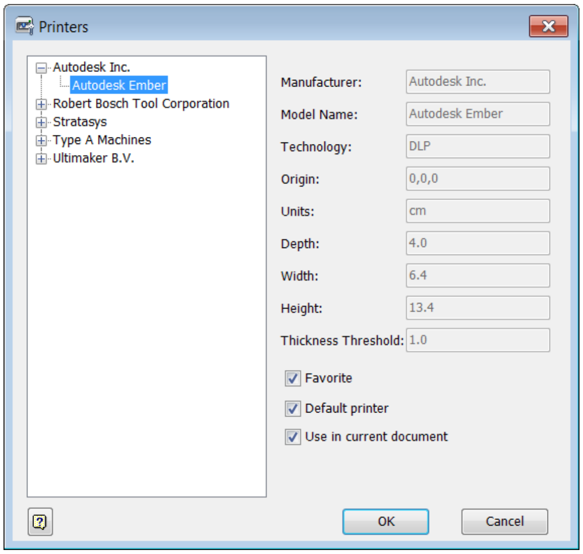

FIGURE 60: OTHER PRINTERS DIALOG BOX

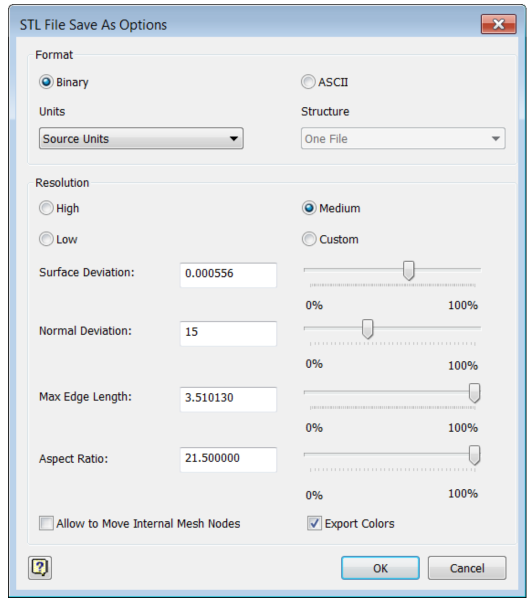

FIGURE 61: PRINT OPTIONS

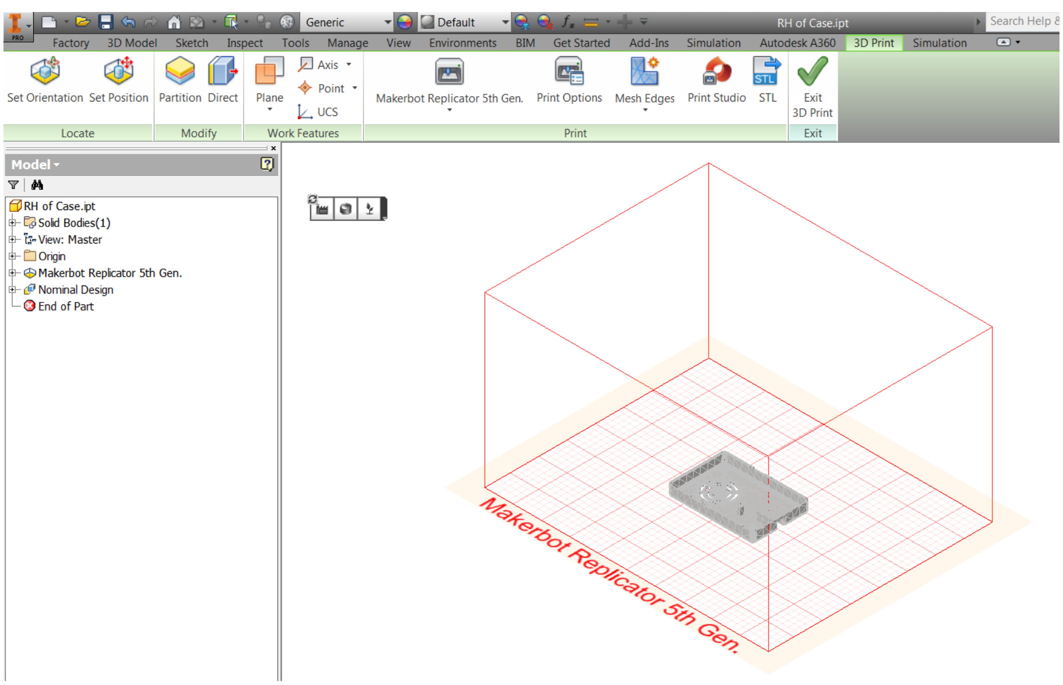

FIGURE 62: SELECTED PRINTER ENVELOPE

Once the basic setup is complete, Print Studio can be used to configure duplicates and see supporting structures for a number of machines, or the STL can be directly exported and a different preprocessor used for the final setup.

Conclusion

The Autodesk Product Design Suite offers selection of tools inside and outside of Inventor to help you create good designs and present them to others for review. Moldflow Design adds real-time capability for monitoring model characteristics for improved performance and reduced costs.