MA322-2L: A View from the Top: Top-Down Design Methods in Autodesk® Inventor®

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MA322-2L Autodesk Inventor has the ability to perform multiple top-down design techniques. Top-down approaches emphasize planning and a complete understanding of the system. It is inherent that no modeling can begin until a sufficient level of detail has been reached in the design of at least some part of the system. The modification of any top-down design technique is inherently productive since it is focused on the modification of the few to update and change the many. This method can drastically reduce time in canvas and repetitive updates to numerous modeling files. This lab is for any users who want to understand how top-down design can aid them in their design intent and efficiency.

About the Speaker:

Mark has been using Autodesk® products since 1999 in many different manufacturing environments. He has implemented Autodesk products for many diverse industries. Autodesk® Inventor® has profoundly augmented Mark's abilities, allowing him to bring 3D digital prototyping to the forefront of the industries with which he has interacted. Mark has extensive experience and a comprehensive understanding of the technical and practical business and human dimensions of implementation. His expertise has helped his clients maximize their project's effectiveness and return on investment. He is an effective and skillful communicator, consulting with his clients to help achieve their business objectives. Mark provides training, support, and implementation on all Autodesk manufacturing solutions.

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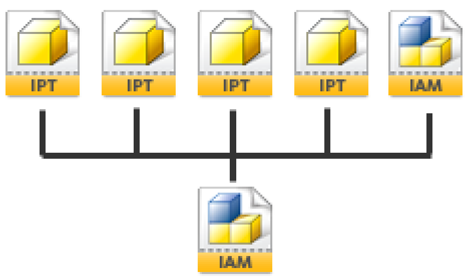
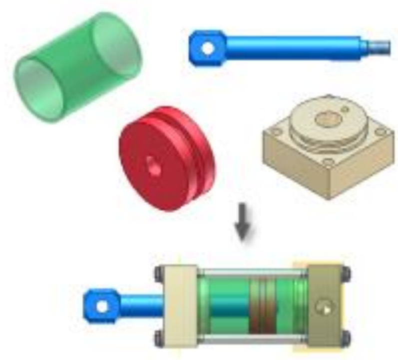
Introduction

Let's take a look at a couple different modeling practices in use in today's Inventor community (Bottom-Up, Middle-Out, and Top-Down), and where Top-Down design can help these scenarios.

Bottom-Up Design (builds IPTs first and constrain one at a time)

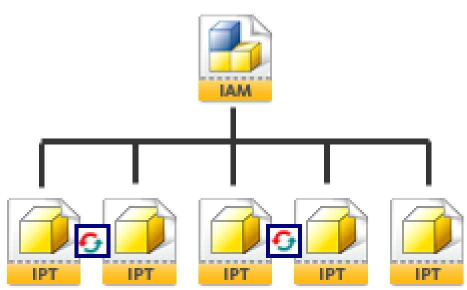
This is the traditional way most users learn Inventor. You start with traditional part modeling. Then after you have your parts you start assembling them together using assembly constraints. The trick comes when you have to start creating mating parts and want to change multiple design criteria across more than one part. With this method you spend a lot of time going back and forth between files and changing the same thing over and over again and hoping the feature tree supports your changes. Furthermore, changes to the modeling geometry can make assembly constraints fall apart if good practices are not followed (such as applying a fillet to an edge where there is a constraint using that edge as a reference). The amount of constraints that need to be added to lock intent can also be tedious.

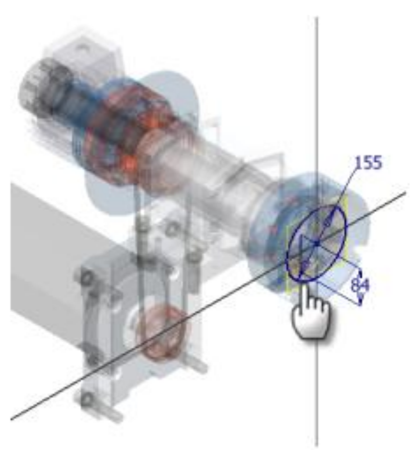




Middle-Out Design (create parts in context and reference to other parts)

This technique is usually picked up by users, but not usually perfected or used correctly. Essentially once components are starting to come together into an assembly, new parts can be created and referenced to existing geometry to quickly reuse it from other parts (*Project Geometry* while holding down **CTRL**) or have it update based on the referenced part (Adaptive Geometry). The problem with this technique is the misuse of Adaptivity, inappropriate updates, and the need to still update many parts for design criteria changes. While powerful for design, this is still not an answer to fast updates or assembly wide changes.

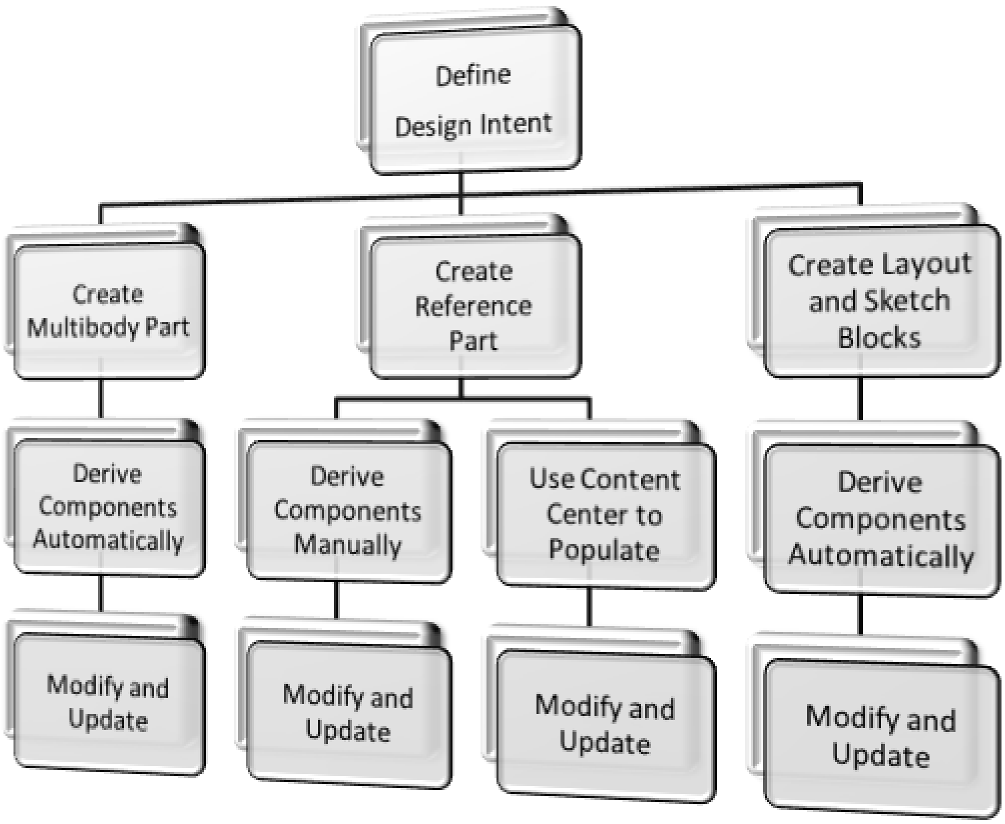




Top-Down Design (build design intent and create assembly and parts from one IPT)

Skeletal Modeling, Multi-Body Modeling, and Layout Design all fall into this design method. Essentially one part controls the design intent and criteria of the entire design and new parts are propagated from this "Master" file. Changes to the Master file are reflected automatically through an Inventor Update or can be set to manual on an as needed basis.

The most important step in Top-Down design is to define the intent before modeling has begun and during the Master part creation. This will influence which Top-Down choice is appropriate (Create Multi-Body Part, Create a Reference Part, or Create Layout and Sketch Blocks).



The topics and exercises in this lab will help users…

- Reduce tedious updates across multiple files based on design intent
- Control assembly relationships with minimal user constraint creation
- Create components faster without having to create and constrain in place
- Check assembly kinematics without having to create a Bottom-Up design only to find out it doesn't work with the intent of the design.
- Aid in complex cross part mating geometry references
- Quickly itterate designs for same-as-but-different scenarios

Note: For time considerations, using Reference Part and Content Center to populate the design will not be covered in this class for time. While it is still a valid method of Top-Down design it pertains mailly to Design Accelerators such as Frame Generator and there are more focused AU classes on those topics.

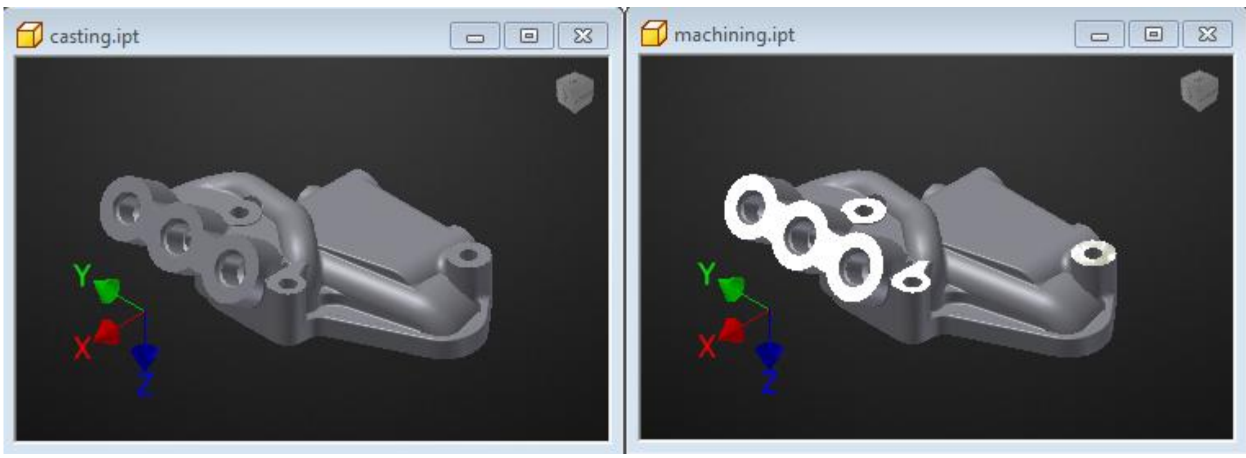
The Backbone Technology

What do all the Top Down design techniques we are about to cover all have in common?

- They will all save me time on design iterations and revisions?
- They will all reduce costly errors in modeling?
- They will all reduce tedious reproduction across multiple files?
- They will all make me look like a super cool CAD man?

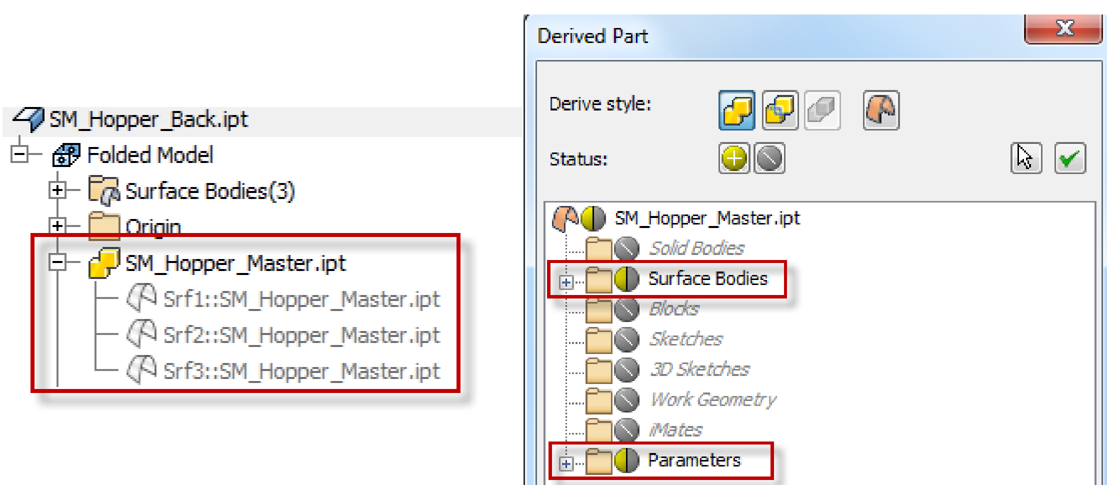
The answer to all of these is yes, but these are not the answers I was looking for. What all of these techniques have in common is the modeling practice of Derived Parts.

A Derived Part creates a link to another part so that when changes are made in that original file, the derived part updates based on those changes. Traditionally this is used in casting and machining to ensure separate file tracking while maintaining geometric updates from the master casting file.



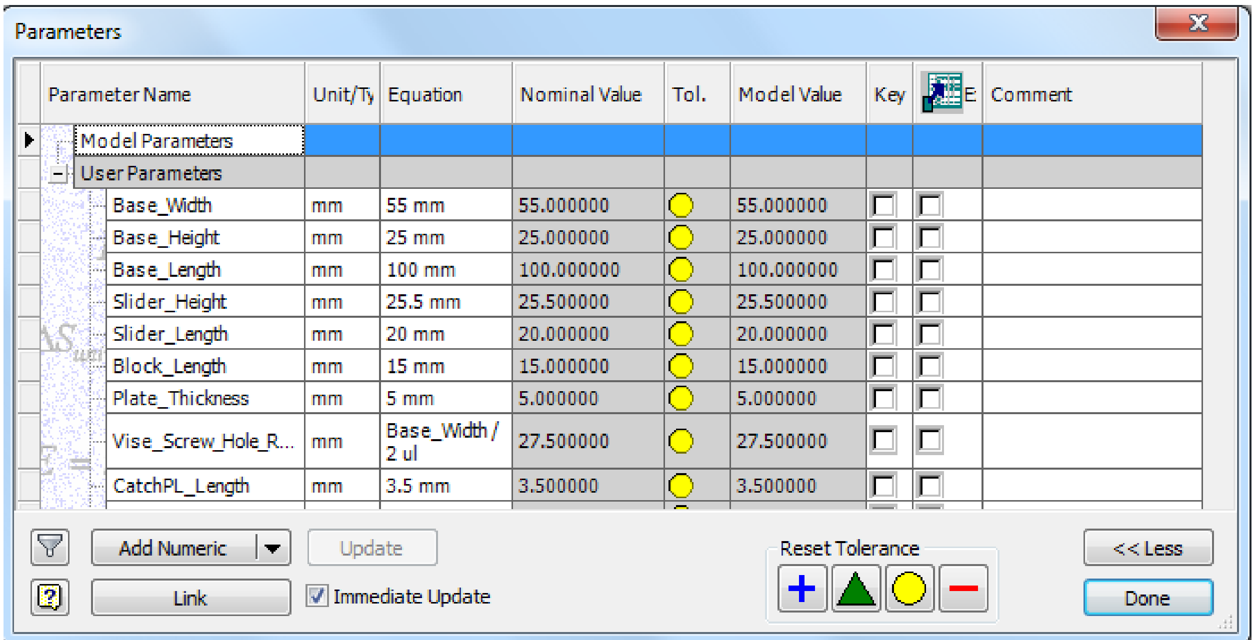
Two of the three methods we will do in the Lab use an automatic approach to this linking and the other one will have to be done manually due to the nature of the method.

Below is an example of the model tree where a part was derived into it for referencing. The part **SM_Hopper_Back.ipt** has a derived reference to **SM_Hopper_Master.ipt** to three Surfaces and some User Parameters that help control the geometry in the file.

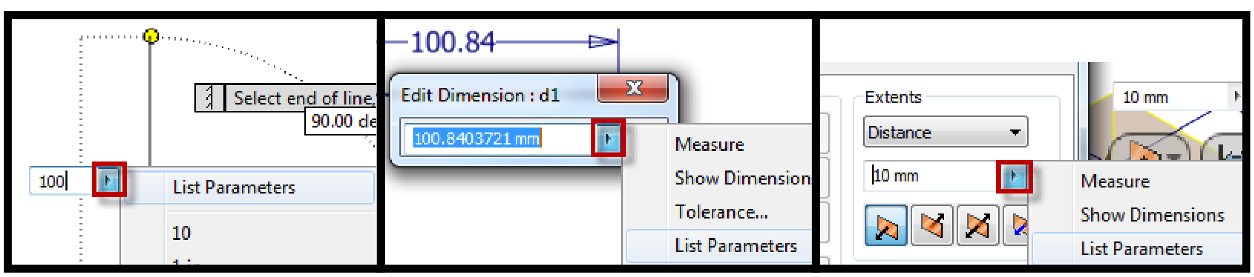


Parameters 2.0

Why 2.0? Because if you learned how to use parameters in the past but never fully appreciated them, you will after this AU session. One of the most primitive concepts of parametric design is the proper use of parameters. To utilize Top-Down design to its fullest it will involve a fair amount of use of the parameters table to not only control design criteria in a logical fashion, but also to allow easy modification of the design criteria.



Parameters can be renamed and accessed without having to constantly open this dialog box. For instance when you see this icon  in an Edit Dimension box or Feature Dialog box, you can list a named parameter to link to the modeling/sketch dimension.



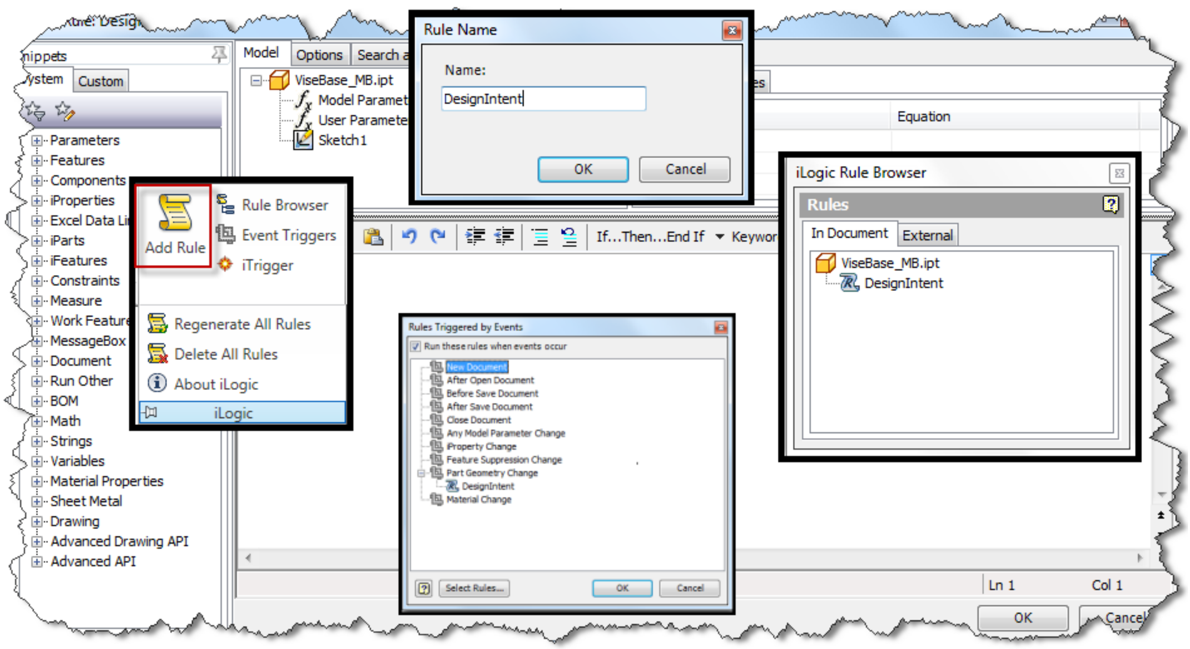
A named parameter is any parameter that does not carry the default d* value where the * indicates a unique identifier number. These are some rules for naming parameters…

- Parameters cannot start with a number
- Parameters are case sensitive (Length and length can both be used)
- Parameters cannot have spaces (you can use underscore instead)
- Parameters have reserved names that are used by Inventor and cannot be used (H, h, V, T, etc)

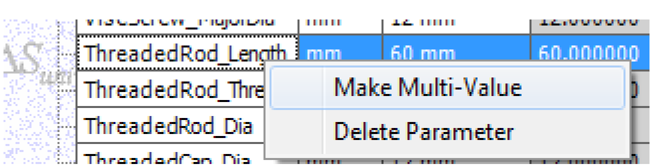
The full incorporation of iLogic into Inventor 2011 has also broadened what can be controlled by parameters. iLogic rules can be triggered and controlled by parameters as well as iLogic rules can perform certain actions to parameters such as IF…THEN and CASE statements.

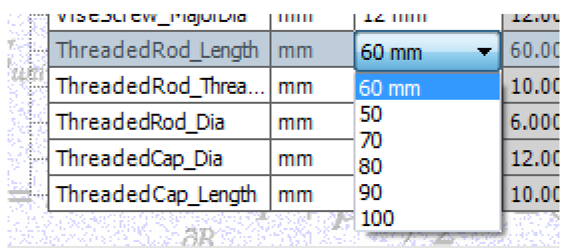
Simplified workflow for iLogic Rules:

- Use **Add Rule** to create a new iLogic rule
- Define criteria for the rule in the iLogic dialog box using snippets and model data
- Setup iTriggers to control how and when a rule is run
- Test the rule for the design intent you created it for



Another added bonus of iLogic was the incorporation of Multi-Value lists in the parameters table. While this has great usage for iLogic code, it still benefits design intent even if you do not choose to use iLogic as it will allow the designer to choose a standard set of values for what a parameter can be.



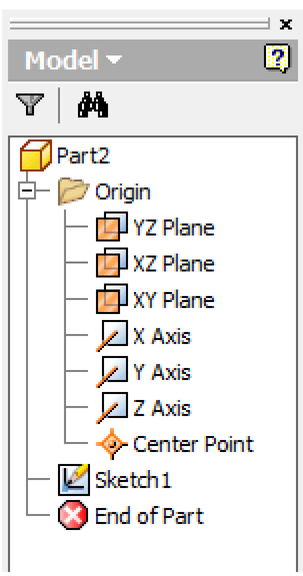


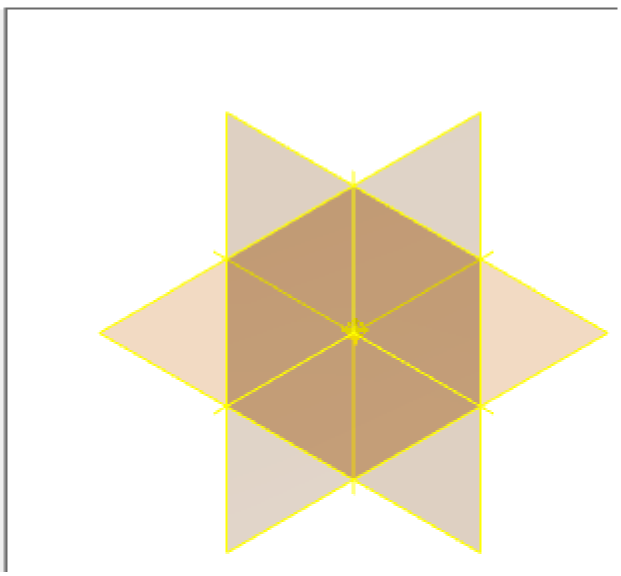
Ultimately, the use or misuse of parameters is where most Top-Down designs succeed or fail. Take time before your first Line command or Work Plane and set up some defining characteristics for your Top-Down design. Create some user parameters up front to help during the design process. If you are going to want something easy to find and use make sure you give it a name and maybe some common values.

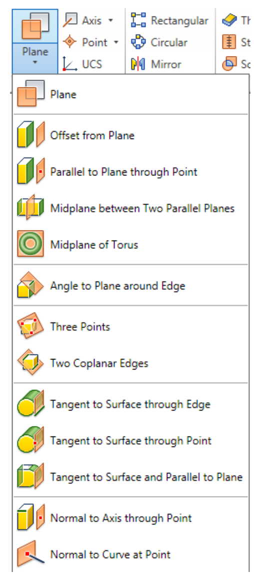
TIP: There are Parameter filters in the Parameters dialog box to help sort Key, Renamed, and Equation based rows in the dialog. I prefer to keep everything in the User Parameter area as I am just accustomed to this. It makes it easier to export or derive the User Parameter area instead of hunting for the right ones.

Origin and Work Planes

It is a well-known fact that Origin Planes are the most stable reference in any modeling file due to the fact they cannot move around the environment. In fact, entire modeling techniques are named after using the Origin Planes and Work Planes created from them as the basis for Design (commonly referred to as B.O.R.N. and Horizontal Modeling Techniques). This stability is the reason is why they are important to Top-Down design. When changes occur to a model, you want it to change in an expected manner and not cause geometry to fall apart after the change occurs. Features, sketches, and work planes based on feature geometry and edges are prone to these parent child errors more so than origin based references.

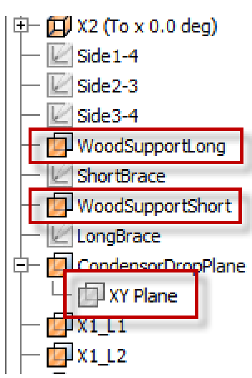


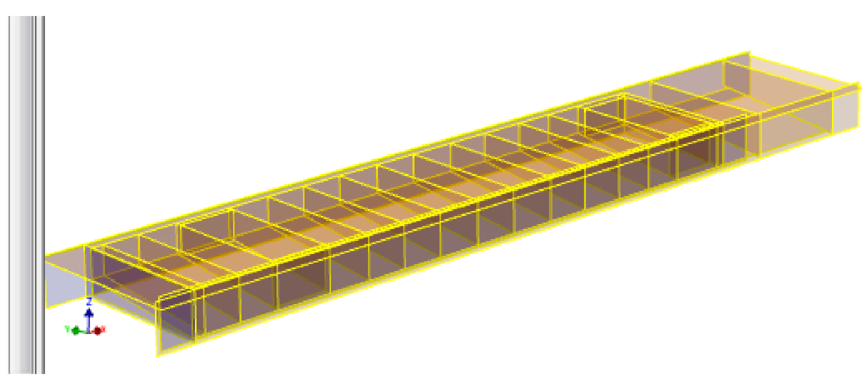




Good Practices using Origin based references

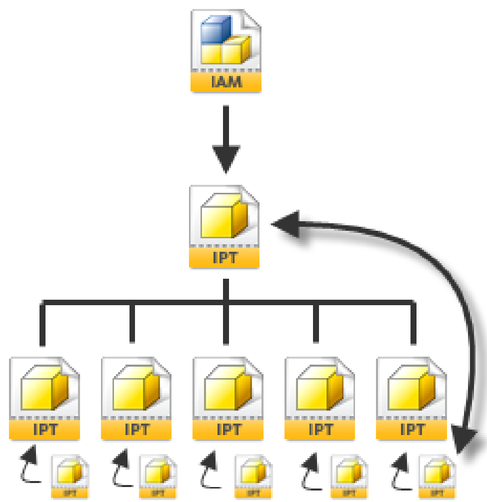
- Use Parameters to control Work Plane dimensional values.
- Use the Origin references as much as possible; Work features based on other user Work features are okay too, just don't get carried away.
- Even if a Work Plane is coplanar with a modeling face, consider using the Work Plane instead for a sketching reference.
- Rename Work features to something easy to reference in Top-Down design and Deriving.
- Use your Work features as datum references in your drawings if logical.



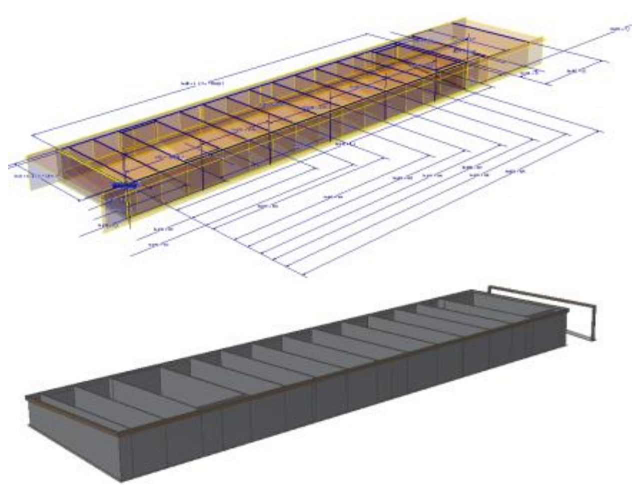


Skeletal Modeling

Modeling with geometry, parameters, and planes has been a long standing advanced technique in the Inventor community for many years. However time consuming it may be to setup initially, the end result cannot be argued with. This technique uses a Reference or “Master” Part to house all the locational and dimensional values for use in an assembly. When new parts are created, the Master file is derived into the file and only the references and parameters needed for that particular model are loaded. That part is then modeled with those references as starting points for geometry that follows the intent of the design.



When it comes time to assemble the parts created in this manner, the Master part is placed first and by Inventor default is grounded to the origin of the assembly based on the origin of the Master part. All the parts can then be added to the assembly and by using simple commands can be automatically constrained and/or grounded to the Master in the correct location. If you ground all the components based off this method it can create a faster opening assembly as it will not have to reevaluate constraints that are not needed and the geometry will still update as expected. Any parts not able to be created in the Master, can still be added in the traditional Bottom-Up or Middle-Out way.

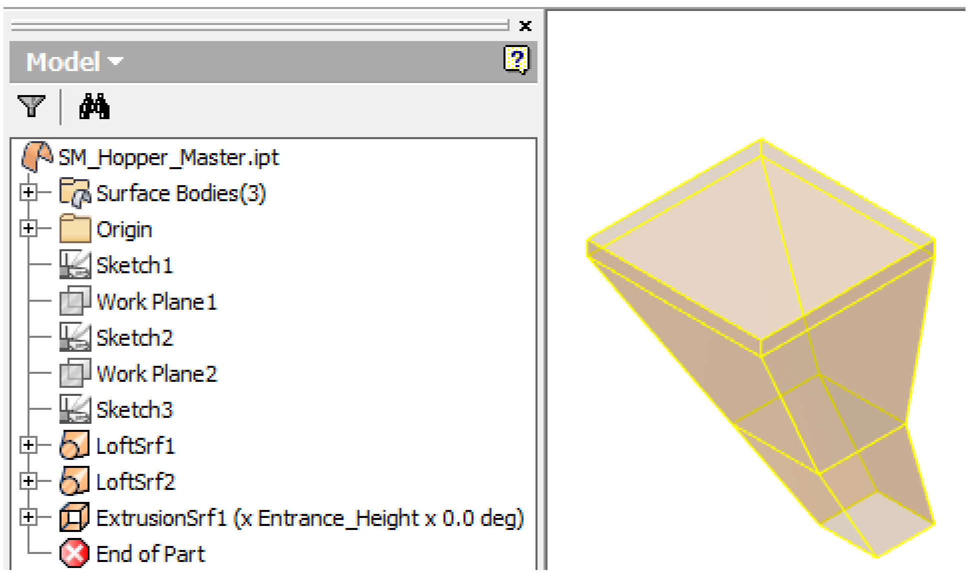


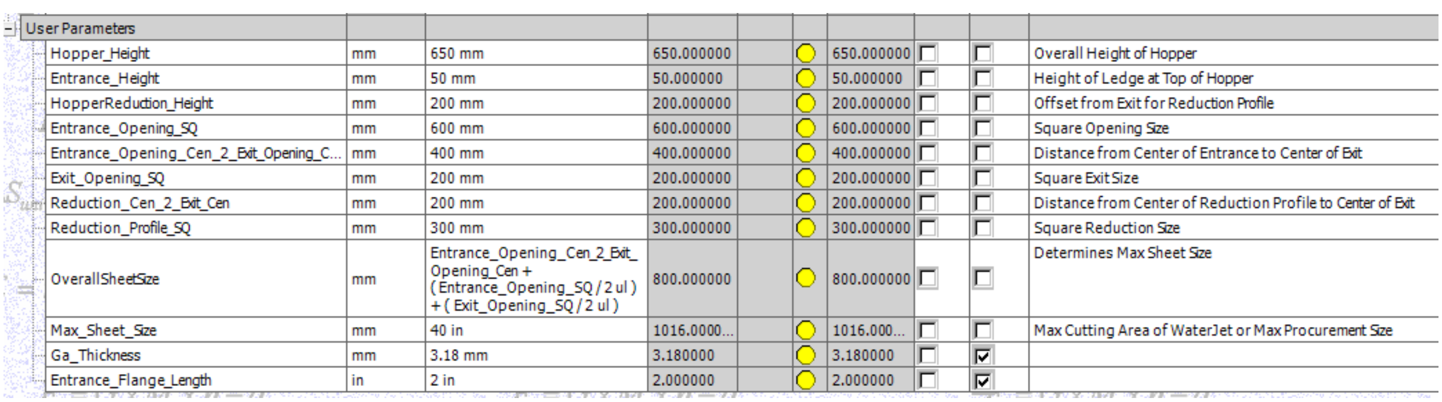
Skeletal Modeling Exercise

In this Exercise we are going to modify a Skeleton model that will control sizing of various components based on the parameters and geometry of one IPT. This methodology will allow the majority of design intent for sizing, spacing, and parametric values to be controlled by a Master file and then derived to individual parts. These individual parts are linked to the original by the Derive command so that any changes to referenced geometry update the new geometry as intended. The goal of this Exercise is to introduce Skeletal Modeling with a simple example to be completed in the allotted time for the course.

Task 1: Examine Skeleton Model

1. Open **SM_Hopper_Master.ipt**. This file contains some lofted surfaces that represent the design intent of the assembled material hopper. Click on the Parameters button in the Quick Access Toolbar to see the list of User Parameters and examine the equations.



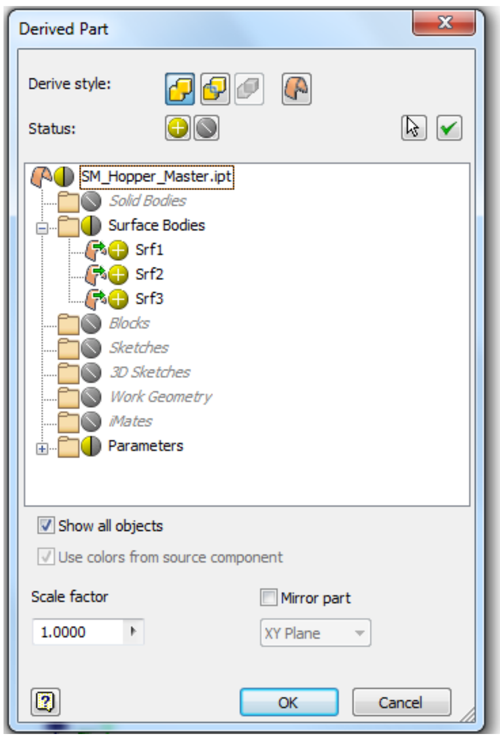


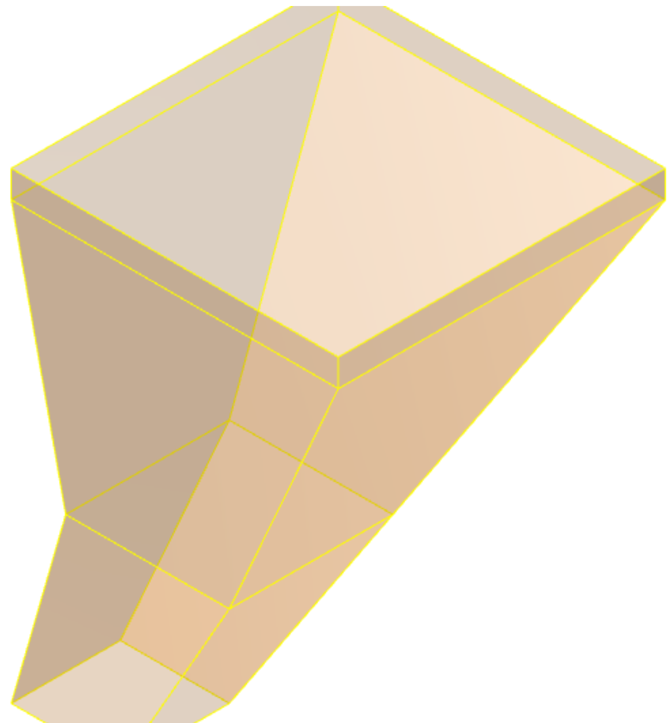
User Parameters									
Hopper_Height	mm	650 mm	650.000000			650.000000	☐	☐	Overall Height of Hopper
Entrance_Height	mm	50 mm	50.000000			50.000000	☐	☐	Height of Ledge at Top of Hopper
HopperReduction_Height	mm	200 mm	200.000000			200.000000	☐	☐	Offset from Exit for Reduction Profile
Entrance_Opening_SQ	mm	600 mm	600.000000			600.000000	☐	☐	Square Opening Size
Entrance_Opening_Cen_2_Exit_Opening_C...	mm	400 mm	400.000000			400.000000	☐	☐	Distance from Center of Entrance to Center of Exit
Exit_Opening_SQ	mm	200 mm	200.000000			200.000000	☐	☐	Square Exit Size
Reduction_Cen_2_Exit_Cen	mm	200 mm	200.000000			200.000000	☐	☐	Distance from Center of Reduction Profile to Center of Exit
Reduction_Profile_SQ	mm	300 mm	300.000000			300.000000	☐	☐	Square Reduction Size
OverallSheetSize	mm	Entrance_Opening_Cen_2_Exit_Opening_Cen + (Entrance_Opening_SQ / 2 ul) + (Exit_Opening_SQ / 2 ul)	800.000000			800.000000	☐	☐	Determines Max Sheet Size
Max_Sheet_Size	mm	40 in	1016.0000...			1016.000...	☐	☐	Max Cutting Area of WaterJet or Max Procurement Size
Ga_Thickness	mm	3.18 mm	3.180000			3.180000	☐	☑	
Entrance_Flange_Length	in	2 in	2.000000			2.000000	☐	☑	

2. Close the Parameters dialog box and select each sketch in the Model tree examining the equations in the sketches. This geometry was modeled with surface lofts, but could just as easily be created with solid geometry. Close the file, do not save changes.

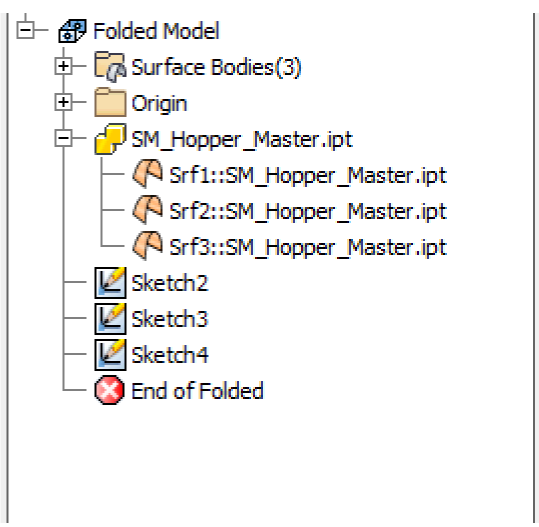
Task 2: Create New Parts and Derive

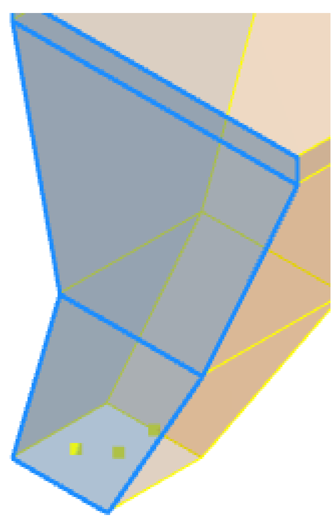
1. Start a new file using the metric **Sheet Metal (mm).ipt** template. Immediately *Finish the Sketch*. You may also delete **Sketch1** if you wish, as it will not be needed. Save the file as **SM_Hopper_Front.ipt**.
2. Start the *Derive* command (*Model Tab > Create Panel* in the Ribbon) and select **SM_Hopper_Master.ipt** from the course directory. You will see the lofted surfaces are already selected to derive as well as two user parameters (*Ga_Thickness* which will control the sheet metal thickness and *Entrance_Flange_Length* which will be used for a **Flange** length. These were pre-selected in the Master file by using the **Export Objects** command.



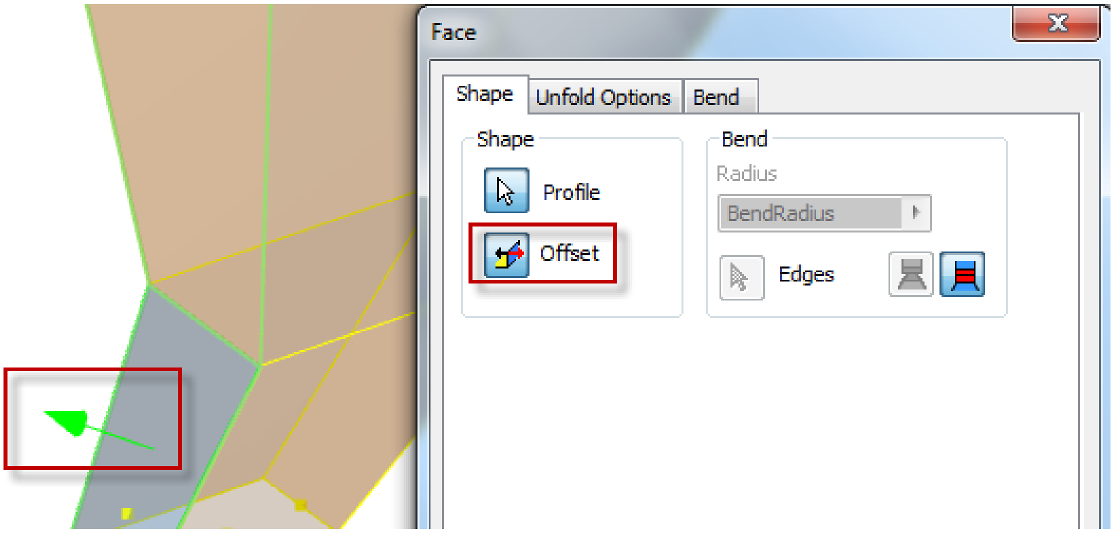


3. Create 3 new sketches on each of the highlighted faces. You may need to project the edges manually if *Auto-Project Edges* is turned off in the Application Options.

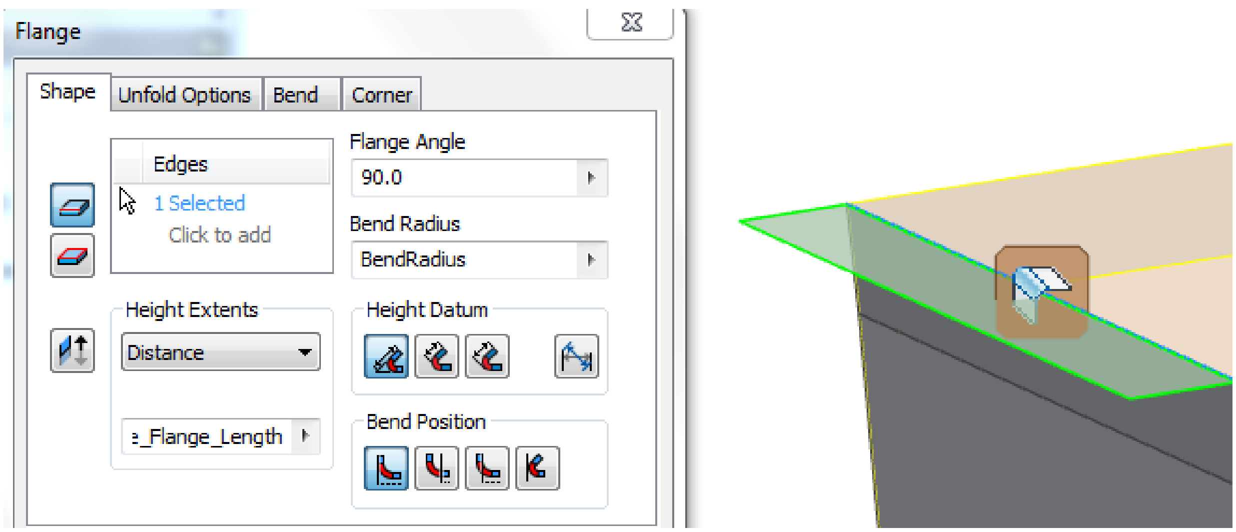




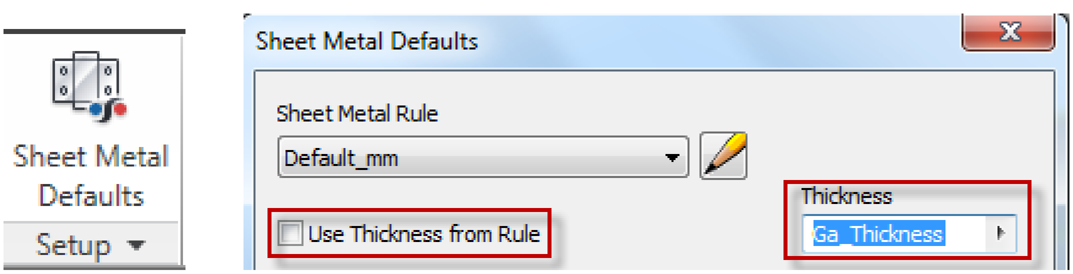
4. On the *Sheet Metal Tab*, start the **Face** command. Select one of the profiles created by the sketches. Make sure the geometry for the Face command creates the geometry to the outside of the surface loft since our design intent modeled the inside extents of the Hopper. Repeat for the remaining **2** sketches to create the **2** remaining faces.



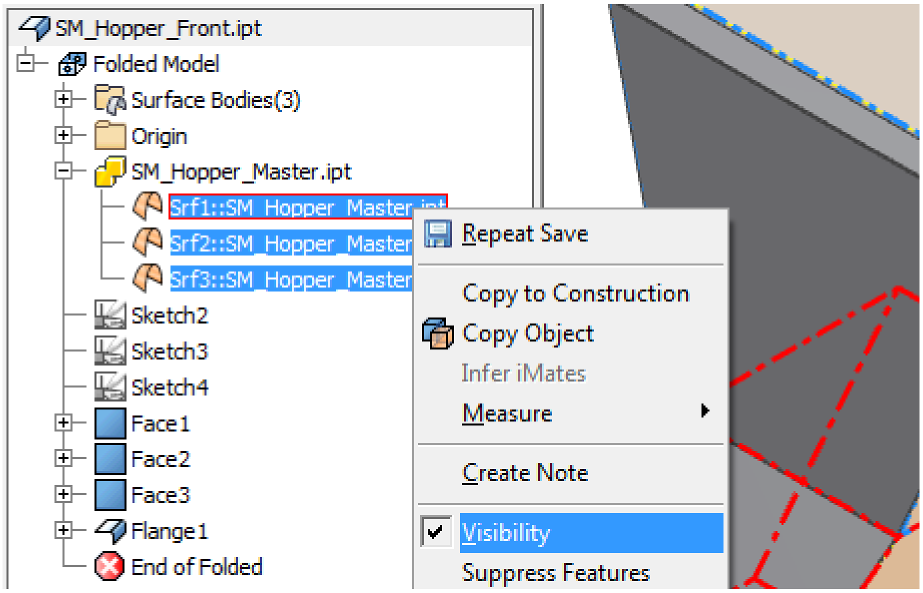
5. Add a **Flange** (*Sheet Metal Tab > Create Panel*) to the top of the Hopper Piece. Set the *Height Extents* to the Parameter *Entrance_Flange_Height*.



6. Start the **Sheet Metal Defaults** command (*Sheet Metal Tab > Setup Panel*) and uncheck the *Thickness from Rule* box and set the value equal to *Ga_Thickness*.



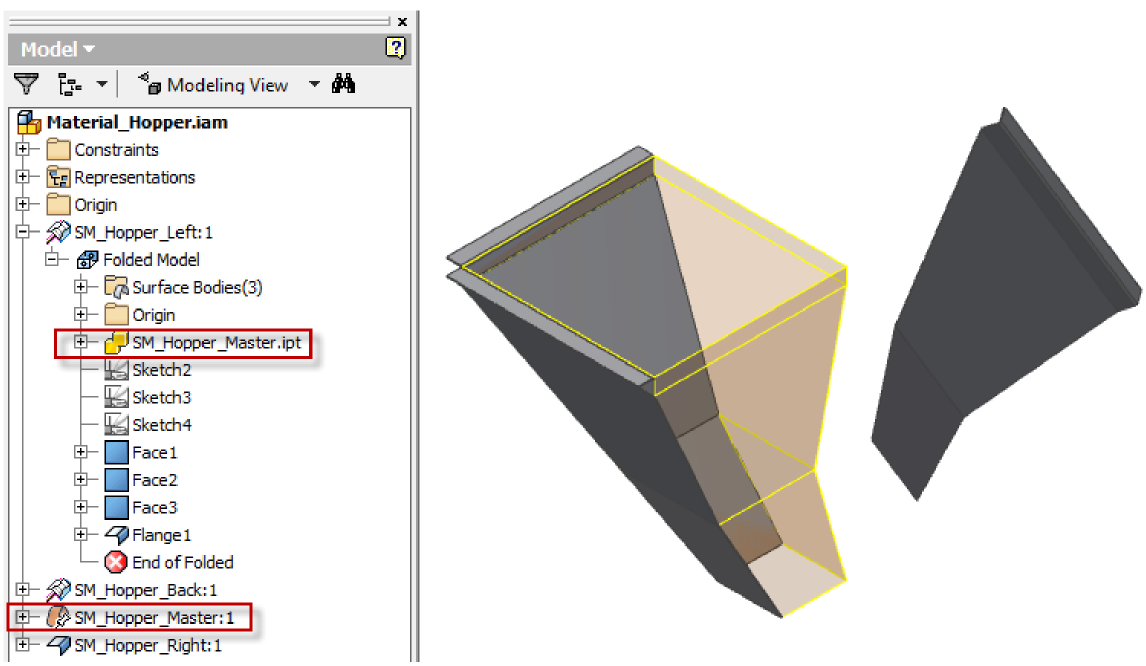
7. Expand the derived reference node for **SM_Hopper_Master.ipt** in the model tree and turn off the Visibility of the three surfaces.



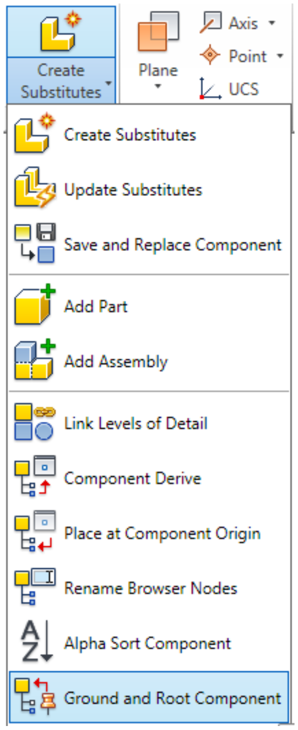
8. Save the file selecting to save to all the dependents and close the file.

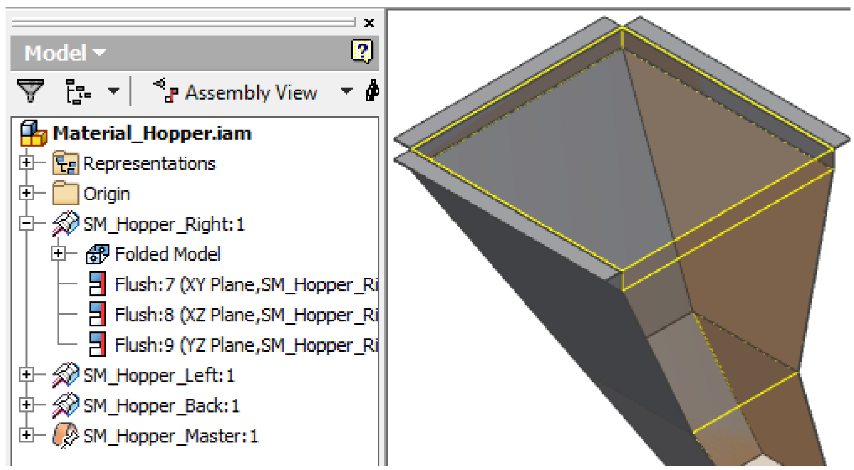
Task 3: Place Parts into Assembly

1. Open the **Material_Hopper.iam** assembly. This assembly is already populated with other parts created in the same way as in Task 2. This has been done for time restrictions of the lab and the fact that the steps are similar. There are two seemingly put together components (**SM_Hopper_Left** and **SM_Hopper_Back** and one that is rotated askew and not in place (**SM_Hopper_Right**). The **SM_Hopper_Master.ipt** is also here as a reference for easy access modification. Its Bill of Material Structure is set to *Reference* as to not partake in Part Lists.

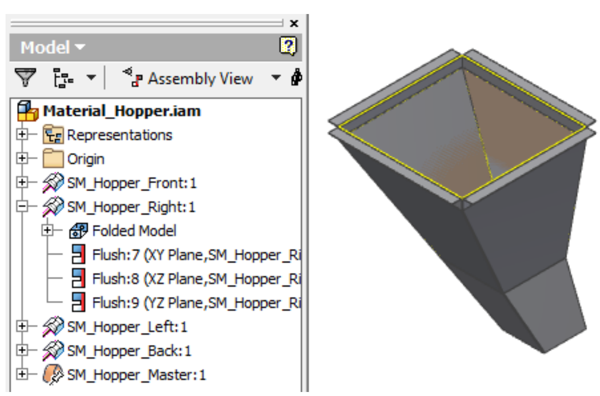


2. Go to the fly out on the *Assemble Tab > Productivity Panel* and choose **Ground and Root Component** from the list. Select the **SM_Hopper_Right** component. The component will automatically be flush to the 3 Origin planes of the assembly and will also be in a *Grounded* state. This is because when the Assembly was created the first part placed was the **SM_Hopper_Master.ipt** and by default grounded to the Origin Planes of the Assembly based its own Origin Planes. Since **SM_Hopper_Right** shares the same reference to the **SM_Hopper_Master** origin references, the components go together perfectly.



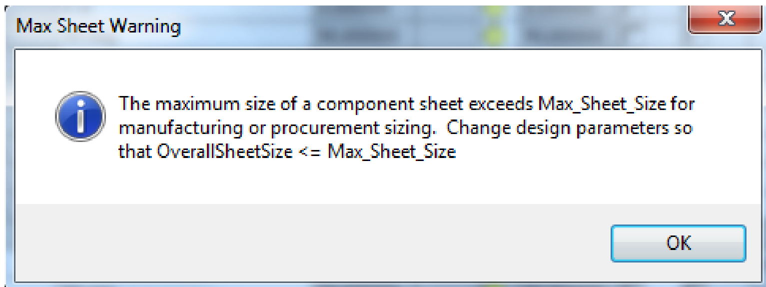


3. Another command in this list is the **Place at Component Origin** which will place a new component not yet in the assembly to the origin of a selected component. If you prefer to have your components grounded rather than simply constrained, place it with a normal **Place Component** and then use the **Ground and Root Component** command. Once a component is grounded with this type of modeling approach you do not need assembly constraints on your derived components and is one of the best advantages of skeletal modeling. Place the **SM_Hopper_Front.ipt** using one of these commands to finish up the assembly.



Task 4: Update Assembly from Skeleton

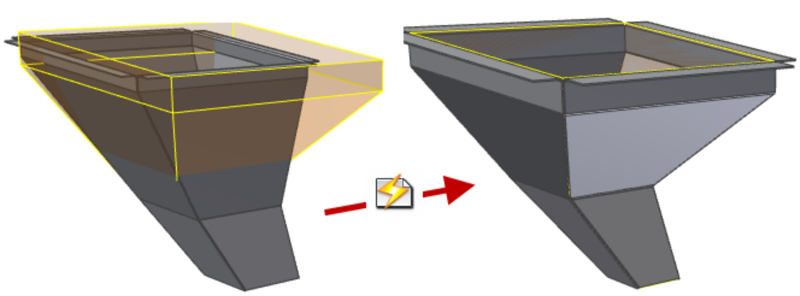
1. Right click on **SM_Hopper_Master:1** in the assembly and select Open. Launch the parameters box from the Quick Access Toolbar. f_x
2. Adjust *Entrance_Opening_Cen_2_Exit_Opening_Cen* to a value of 800. You will get a Max Sheet Warning dialog box letting you know you have stepped outside the design parameters of the Hopper. Select OK. This is an iLogic rule that fires when *OverallSheetSize* > *Max_Sheet_Size* and will continue to pop up so long as this is true. This is a great way to build added design intent into Skeletal Modeling or any Top-Down modeling approach.



3. Change the following user parameters:

User Parameters		
Hopper_Height	mm	700 mm
Entrance_Height	mm	100 mm
HopperReduction_Height	mm	250 mm
Entrance_Opening_SQ	mm	750 mm
Entrance_Opening_Cen_2_Exit_Opening_C...	mm	200 mm
Exit_Opening_SQ	mm	150 mm
Reduction_Cen_2_Exit_Cen	mm	300 mm
Reduction_Profile_SQ	mm	300 mm
OverallSheetSize	mm	Entrance_Opening_Cen_2_Exit_Opening_Cen + (Entrance_Opening_SQ / 2 ul) + (Exit_Opening_SQ / 2 ul)
Max_Sheet_Size	mm	40 in
Ga_Thickness	mm	4.76 mm
Entrance_Flange_Length	mm	3.5 in

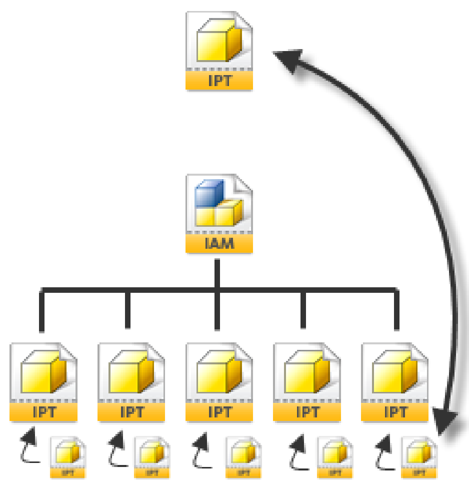
4. Toggle back to the **Material_Hopper.iam** and click the Update button in the Quick Access Toolbar. Keep in mind changes in parameters must also allow for valid geometry.



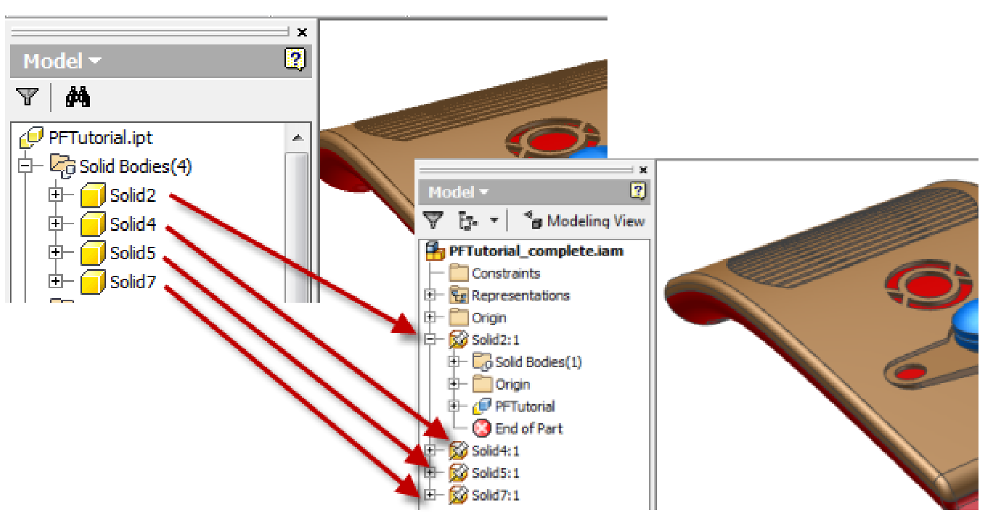
Multi-Body Modeling

Compared to Skeleton Modeling which uses a Master part as a reference, Multi-Body modeling streamlines the process by taking out the manual deriving and assembly process. This type of modeling is geared more for plastic parts or small static designs rather than large structures or sheet metal designs (you cannot directly make a Sheet Metal part a Multi-Body part). Multi-Body uses standard modeling to create normal geometry and to simply assign new Solids () throughout the part modeling process.

Each new Solid will be a corresponding new part when assembled. The new parts are automatically created by Inventor deriving each solid body into a new part file and then creating and adding them to a new assembly in an already grounded state. Updates to the Master, prompts updates to the components to update their referenced geometry.



Unlike Skeletal Modeling, where the Master part is placed first into the assembly, Multi-Body designs do not automatically add the Master file to the assembly when the automatic routine is run. I add it anyway and place it at the origin of the design, ground it, and make it inivisible just so it is easy to access from the assembly browser and lets anyone know that is working in that assembly that it is a Multi-Body design that created the assembly. Remember to make it a Reference component so it does not show up in the BOM.



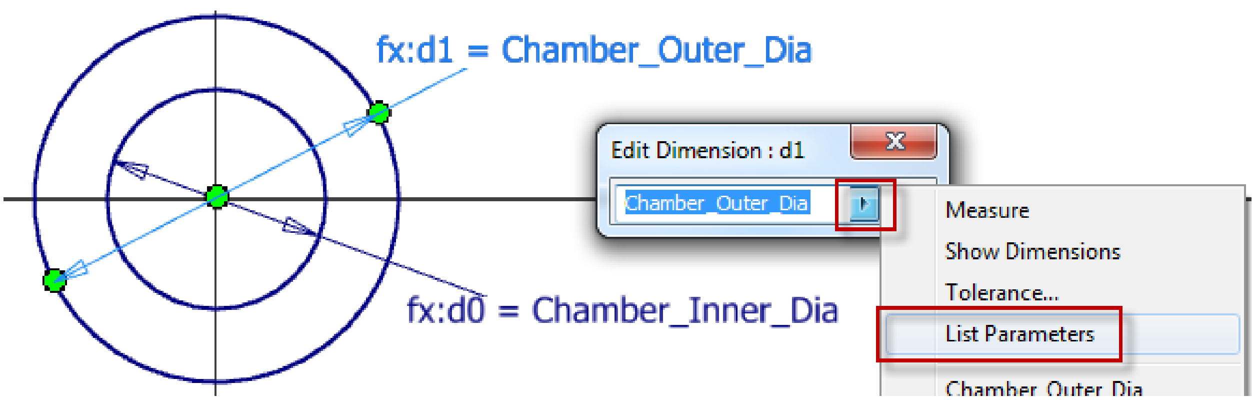
Multi-Body Exercise

In this Exercise we are going to create a Multi-Body Inventor model that will control sizing of various components based on the parameters of one as well as allow for use of geometry from separate solid bodies. This methodology will allow an assembly to be modeled as one part so that geometry lines up correctly as to avoid errors in mating parts and avoid advanced assembly level cross part relationships. The goal of this Exercise is to introduce Multi-Body modeling with a simple example to be completed in the allotted time for the course.

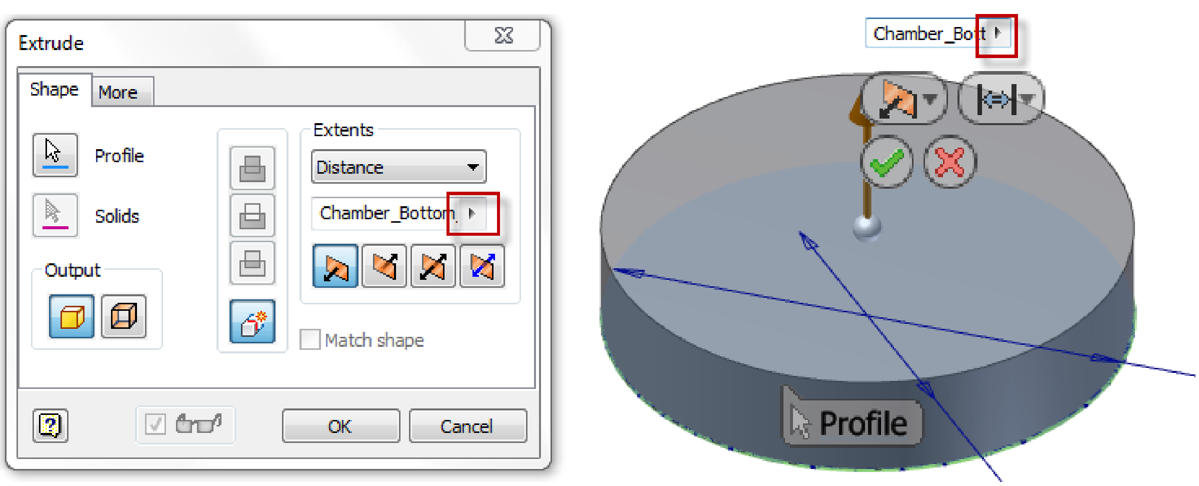
Task 1: Create Initial and Ancillary Bodies

1. Open **Pressure_Cylinder.ipt** from the Course Directory. This will be a seemingly blank file with only Sketch 1 in it, but the file actually contains user parameters. Click on the *Parameters* button in the Quick Access Toolbar to see the list and examine the equations.

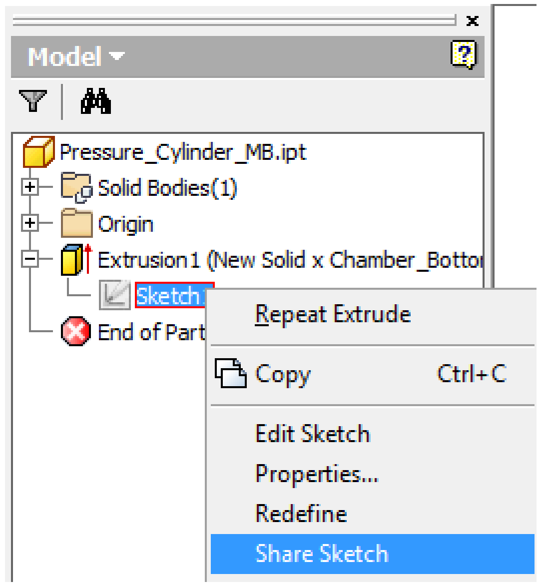
2. Activate Sketch 1 and create the following geometry. Use the *List Parameters* option when placing dimensions to link the user parameters to the sketching dimensions.



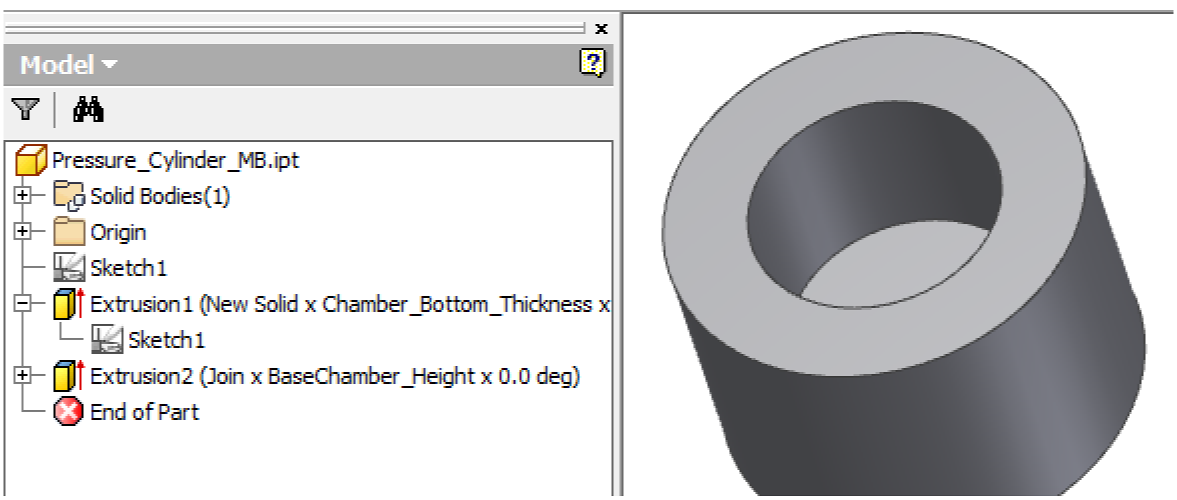
3. Finish the Sketch and **Extrude** by the Parameter *Chamber_Bottom_Thickness*. Remember to use the right arrow on the input boxes to get the *List Parameters* option to make sure the parameter name matches exactly.



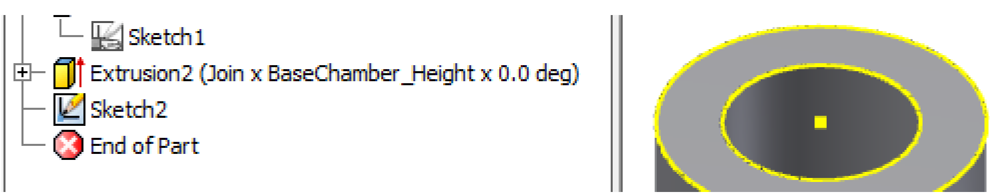
4. Expand Extrusion 1 and right click on Sketch 1 and choose the *Share Sketch* option. This will allow you to use the geometry in the sketch over and over again without having to create a new one for geometry that can be defined on the same plane.



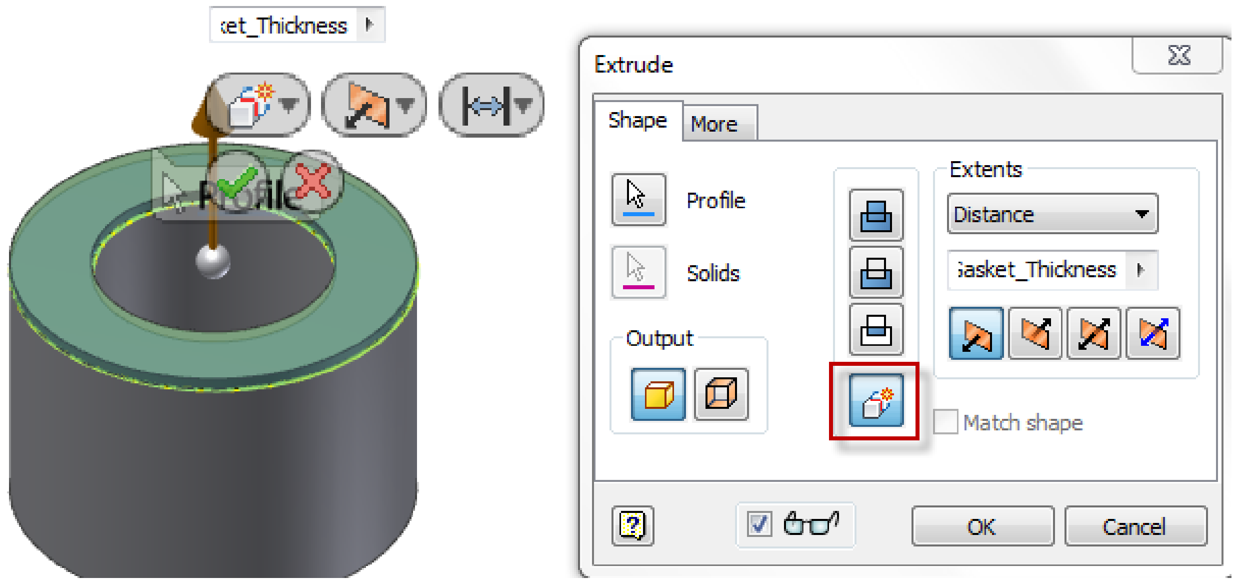
5. Choose **Extrude** again, this time selecting the outer most profile. Set this extents to *BaseChamber_Height*. Make sure you are joining the material instead of cutting. When finished, turn off the Visibility of Sketch 1.



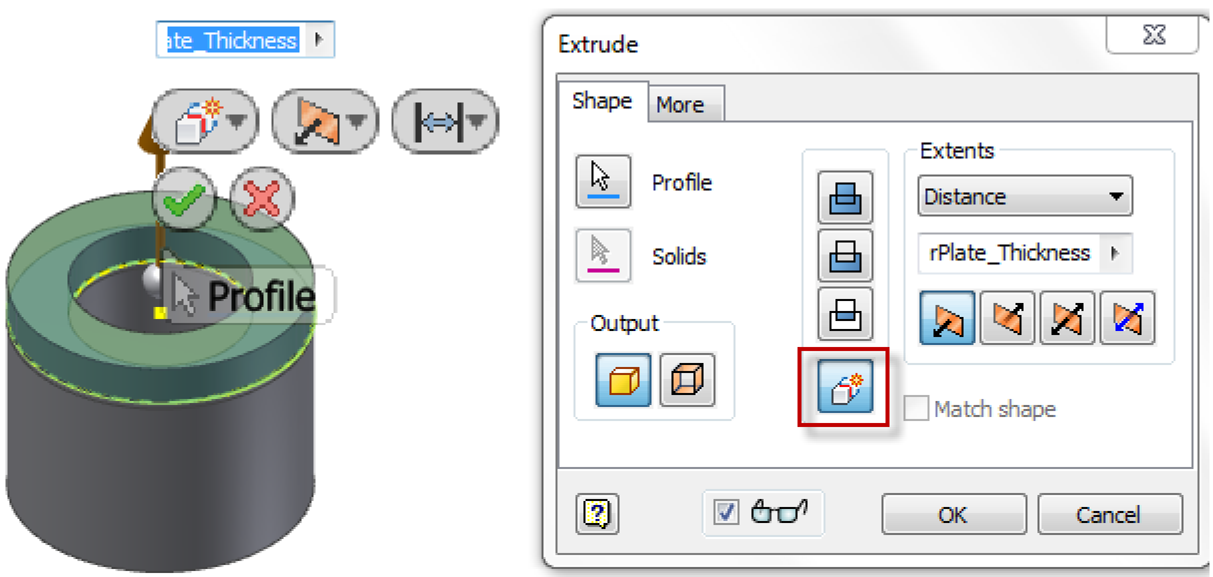
6. Create a new 2D Sketch on the top face of the part. Notice the edges of the cylinder are automatically projected. If they are not, use Project Geometry from the *Draw Panel* and select the two circular edges. Finish the Sketch.



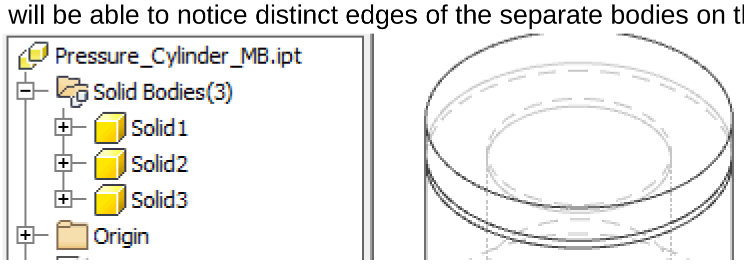
7. Start **Extrude** again and select the outer ring profile. Instead of choosing the Join operation this time, use the New Solid button to create a new Solid Body that will be another part for assembly use. Extrude this new solid with a value of *Gasket_Thickness*. Select OK.



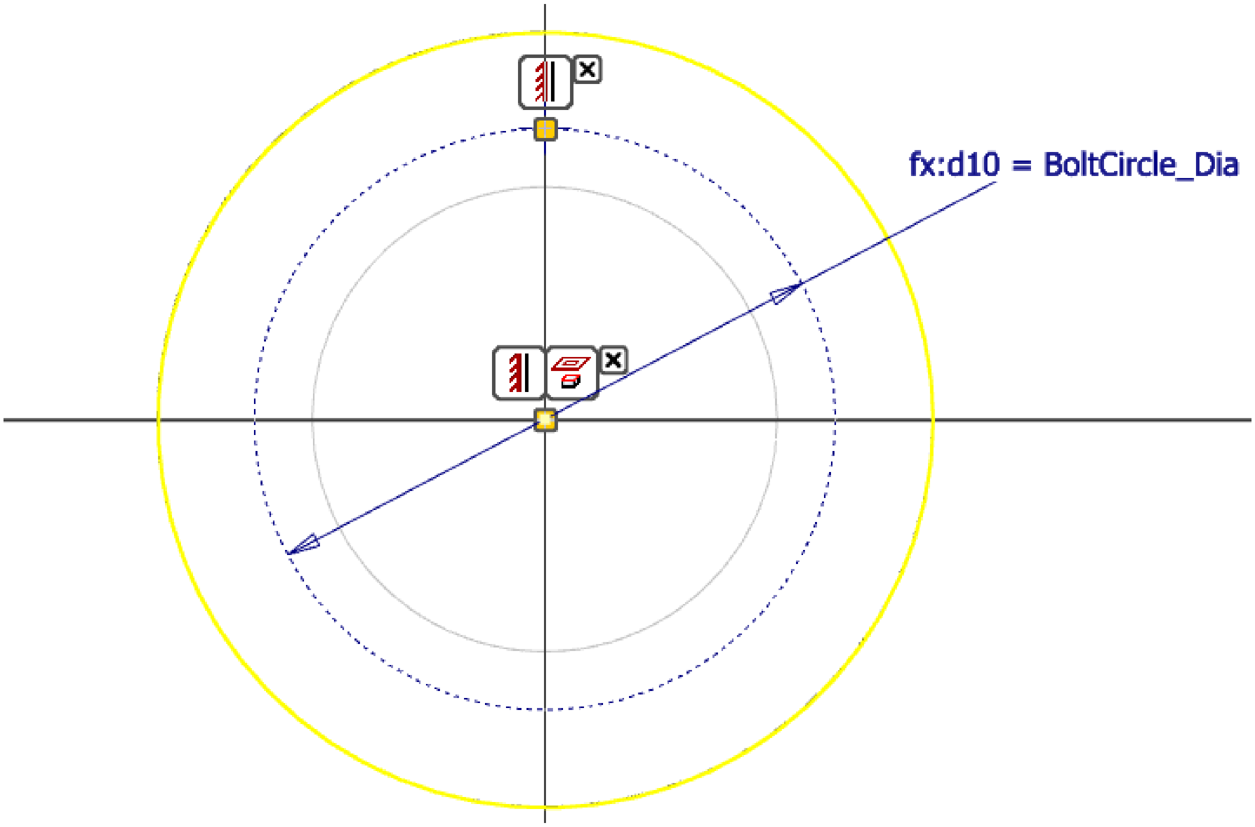
8. Repeat another 2D Sketch on the top of this solid and once again choose the New Solid option. **Extrude** both profiles by *CoverPlate_Thickness*. Select OK.



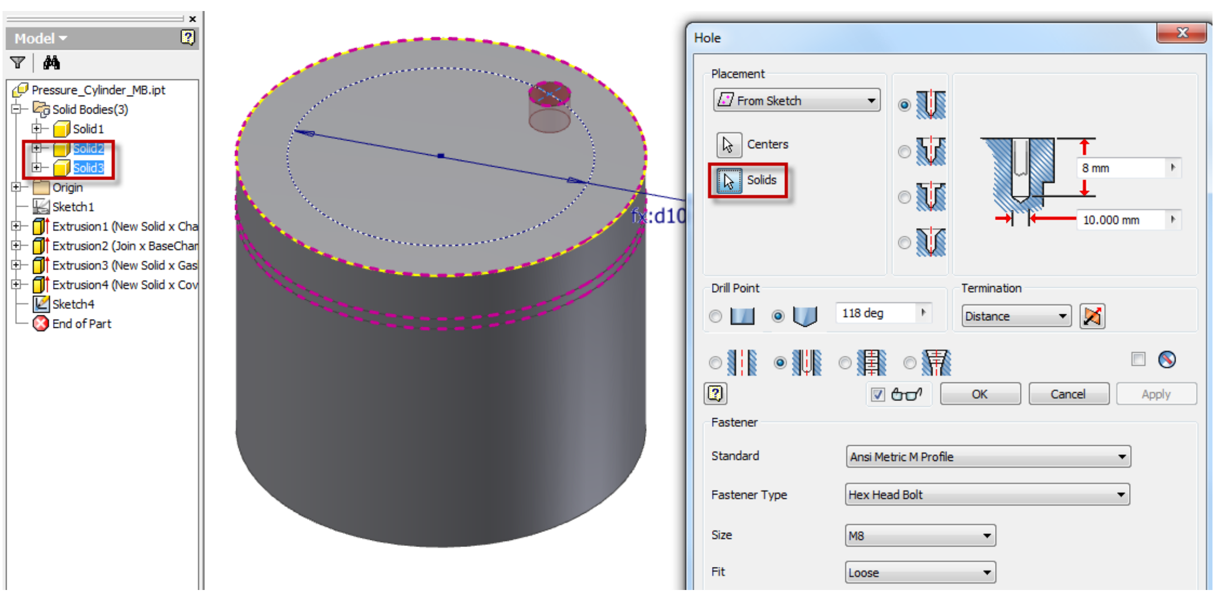
9. Examine the Model Browser. You will now see 3 Solid Bodies under the Solid Bodies folder in the tree and will be able to notice distinct edges of the separate bodies on the model.



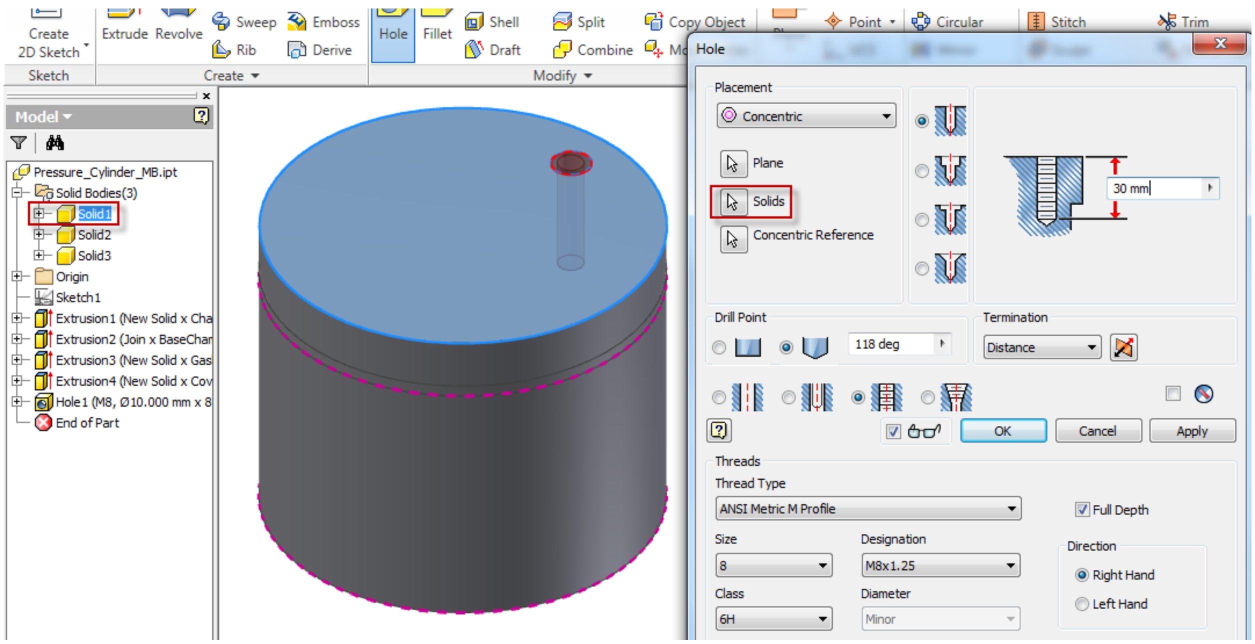
10. Sketch on the Top of the 3rd Solid Body and create the geometry below. Create the Bolt Circle with Parameter *BoltCircle_Dia*. Also create a *Point* (-+-) vertically or horizontally constrained with the Origin of the part. Finish the Sketch.



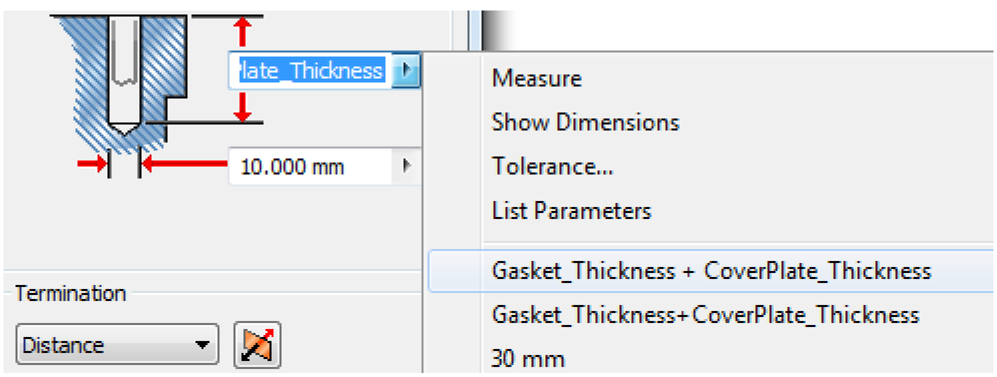
11. Create a **Hole** using the From Sketch placement method. Size a **Clearance Hole** for **ANSI Metric M Profile**, **Hex Head Bolt**, **M8**, **Loose Fit**. Make sure you use your Solids button highlighted below to include both **Solid2** and **Solid3** in the operation. Otherwise the Hole will only permeate through **Solid3**.



12. Create another **Hole** using the **Concentric** placement method. Use the top of the part as the Planar reference and the existing Hole1 as the Concentric reference. Create a **Tapped** hole using **ANSI Metric M Profile**, Size **8**, Designation **M8x1.25**, **Full Depth**, and **Right Hand Thread**. Make the distance of the blind tap **30** mm. Be sure this time to deselect **Solid3** and select only **Solid1**. You can use the Shift key to deselect solids.



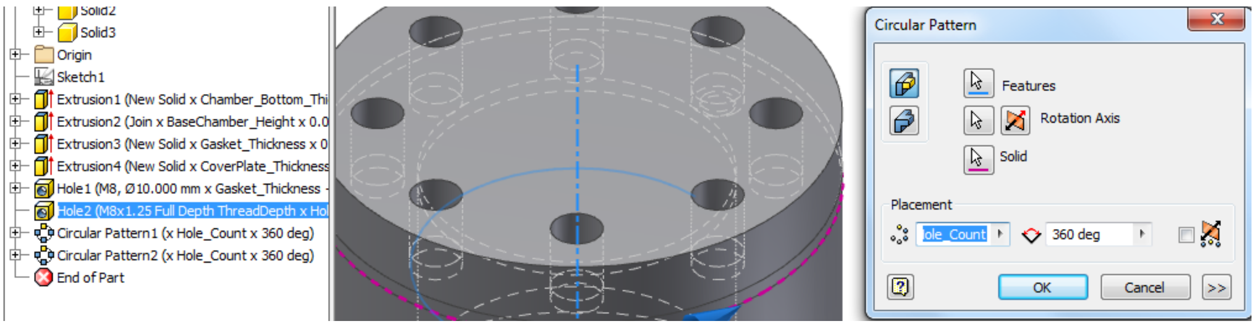
13. Since we did not set the value for the termination distance of the first hole, the connection does not go through the part all the way. Go back to **Hole1** and modify the depth of the hole to be equal to *Gasket_Thickness + CoverPlate_Thickness*.



14. Also adjust **Hole2**'s depth to be *Hole_Depth*. These adjustments will allow the depth of the holes to be consistent with the Gasket and Cover Plate material thicknesses as well as maintain good thread engagement.

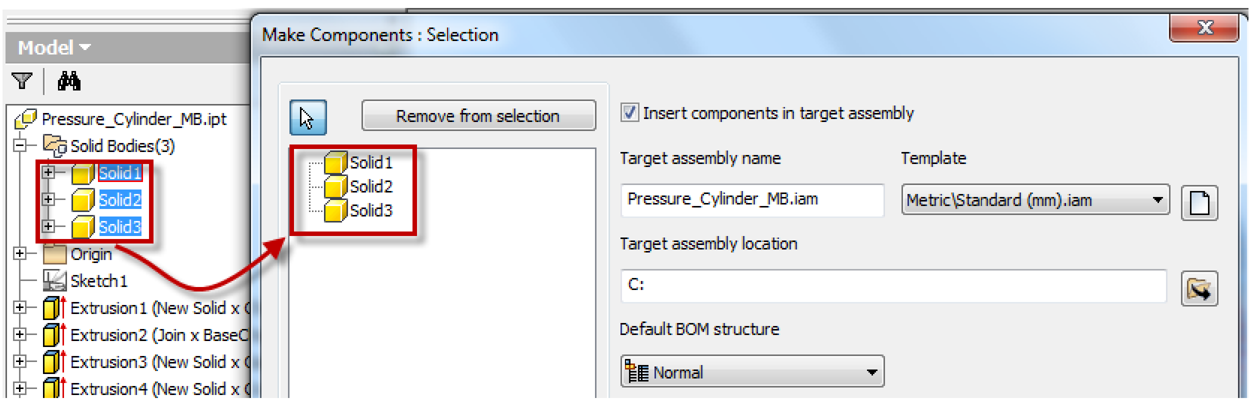
(Conversely, this can be controlled by the Assembly with a **Bolted Connection** Design Accelerator, but not every company uses Content Center and not every design is industrial by nature. This method shows how one type of pick and place command can be used to control design intent and reduce heavy file editing between multiple parts.)

15. To finish off this Task, start the **Circular Pattern** command and select **Hole1** either graphically or from the browser. Select a *Rotation Axis* by selecting the circular face of **Solid3** and then for the *Solid* reference select **Solid3**. Set the placement count equal to the parameter *Hole_Count*. Repeat this process again for **Hole1** and **Solid2** and **Hole2** and **Solid1**. (Patterning of features can only be done 1 solid at a time)

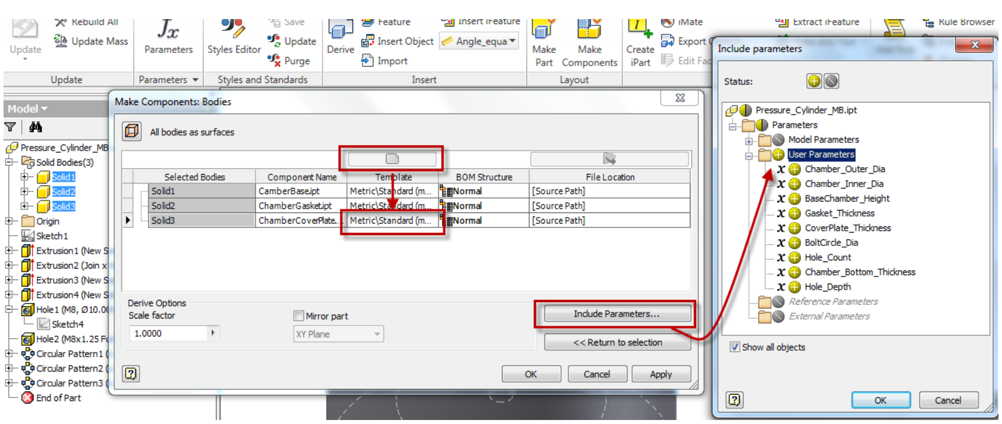


Task 2: Create Assembly from Multi-Body Part

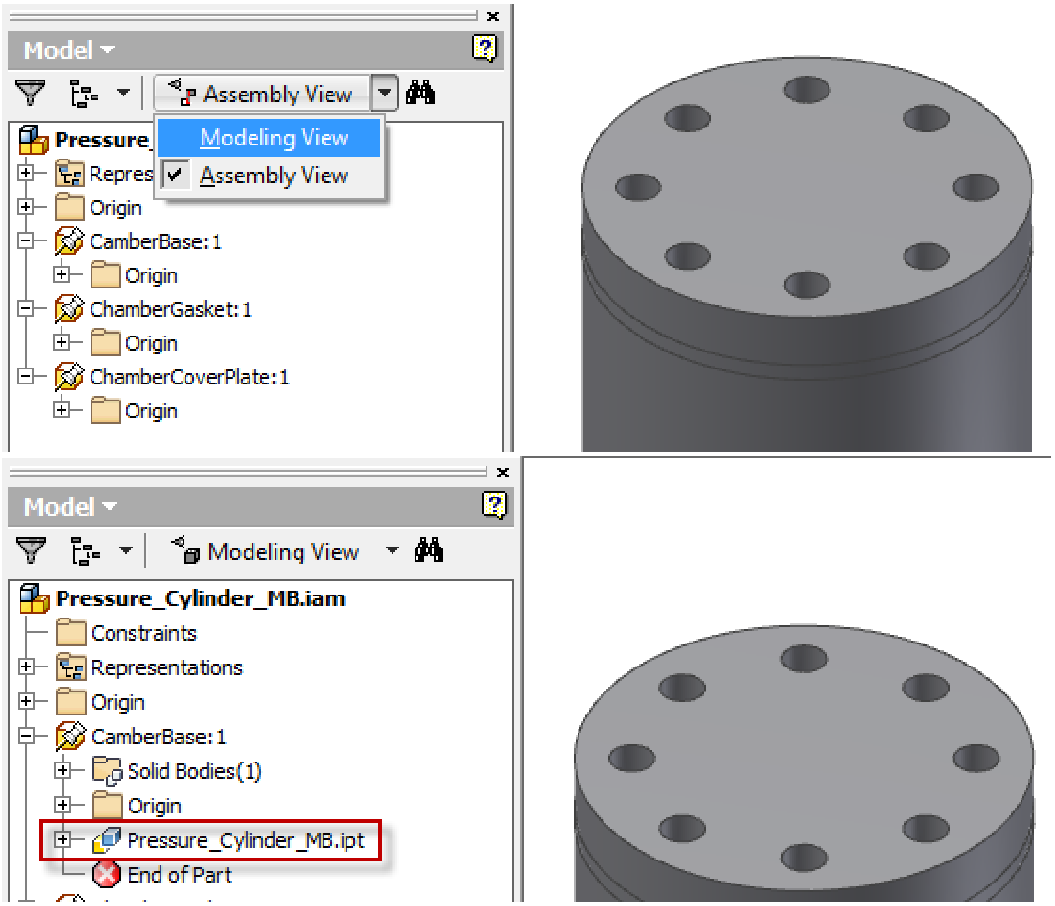
1. From the **Pressure_Cylinder_MB.ipt**, start the *Make Components* command from the *Manage Tab > Layout Panel*. Select the Solids from the Solid Bodies folder in the tree. Click Next.



2. Rename the Solid Bodies into better names. **Solid1**=**ChamberBase**; **Solid2**=**ChamberGasket**; **Solid3**=**ChamberCoverPlate**. Select the Include *Parameters* button and select all the User Parameters. This will be for use at the individual part level if you wish to do modification there and not in the Mutli-Body. Click OK here and the main dialog.

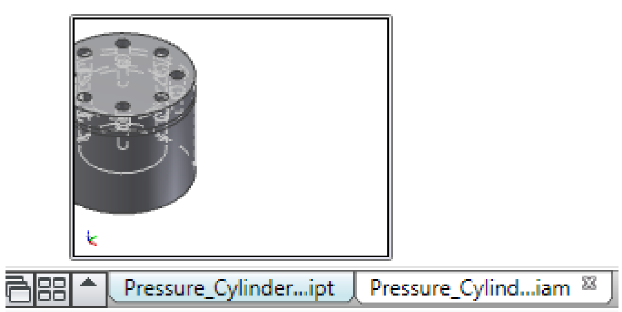


3. A new assembly file is launched with the 3 Solids now individual parts placed and grounded to the root of the assembly. Change to Modeling View, expand the assembly browser nodes and notice that the parts all show Derived links to **the Pressure_Cylinder_MB.ipt** file. This will ensure any changes made to the originating file will propagate to the part files and reflect in the assembly.



Task 3: Update Geometry from Multi-Body Part

1. Toggle back to **Pressure_Cylinder_MB.ipt** as it should still be open in session.

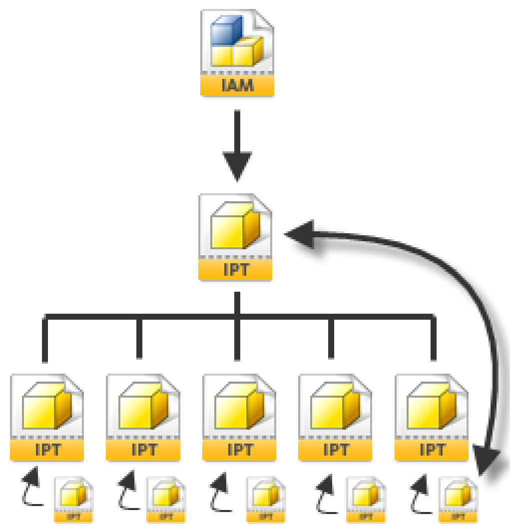


2. Open the Parameters table again and change the *Chamber_OuterDia* to **150** and *CoverPlate_Thickness* to **20** and *Hole_Count* to **10**.
3. Toggle back over the Assembly and hit the Update button in the Quick Access Toolbar to see the updates take place through the derived components. 

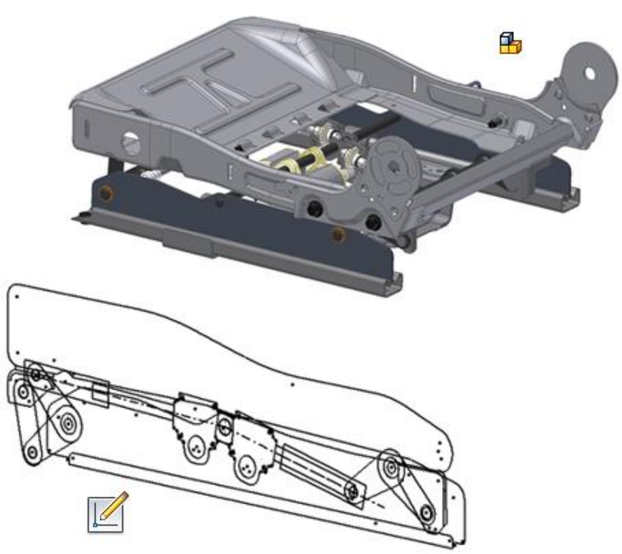
Layout Design

The last method we will cover is Layout Design. This is used primarily for kinematic testing of assembly components before the geometry is created such as linkage systems or hydraulic cylinders.

Here, Sketch Blocks ( ) are created to signify separate components in the assembly. Sketch Blocks can be nested to create flexible ( ) subassemblies to allow movement through the model structure while still maintaining the intent of the kinematic design. The solid geometry is added after the design is tested to meet the intent and is not done in the Master part itself. Parameters can also be passed from the Master for geometry creation.



Unlike Multi-Body design, this method will add the Master part to the assembly and also constrain it and make it invisible. This component will already be set up as a Reference component structure.

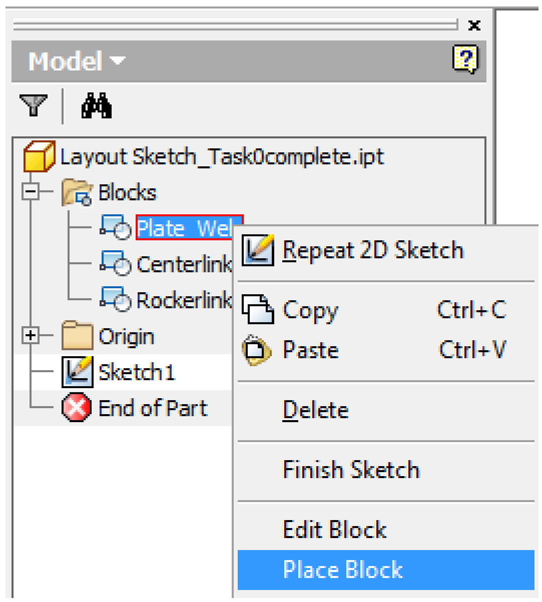


Layout Design Exercise

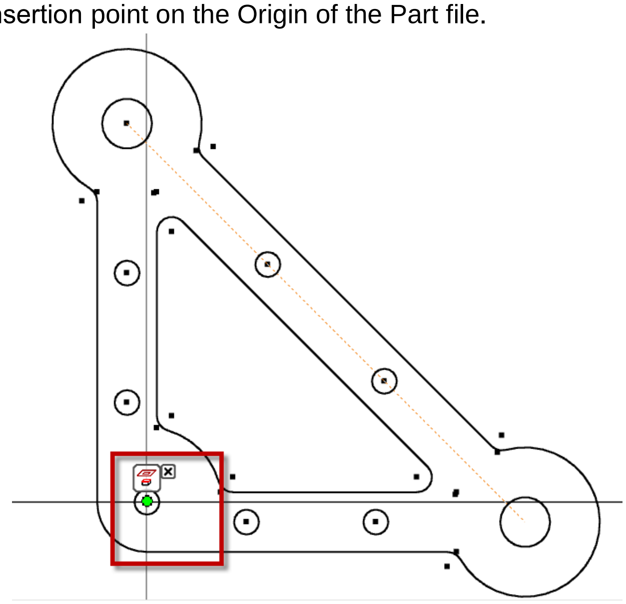
In this Exercise we are going to create a Layout Sketch that will validate sizing and motion. This upfront check will ensure our design intent is met without having to create something that is non-functional and doing a lot of back tracking in the design to make it functional. The goal of this Exercise is to introduce Layout Design with a simple example to be completed in the allotted time for the course.

Task 1: Create and Place Sketch Blocks

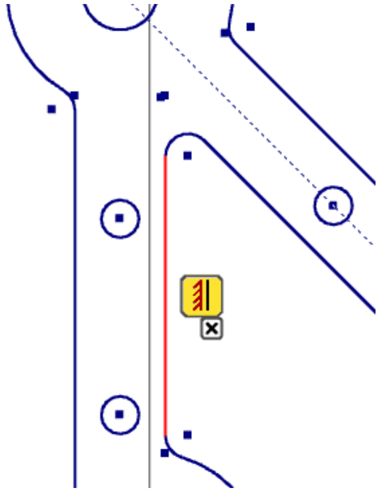
1. Open **Layout_Sketch.ipt** from the Course Directory. This will be a seemingly blank file with only Sketch 1 in it, but the file actually contains Sketch Blocks that are not yet placed.
2. Activate Sketch 1 and place an Instance of the Sketch Block **Plate_Web** into the Sketch by expanding the folder in the Model Browser called Blocks.



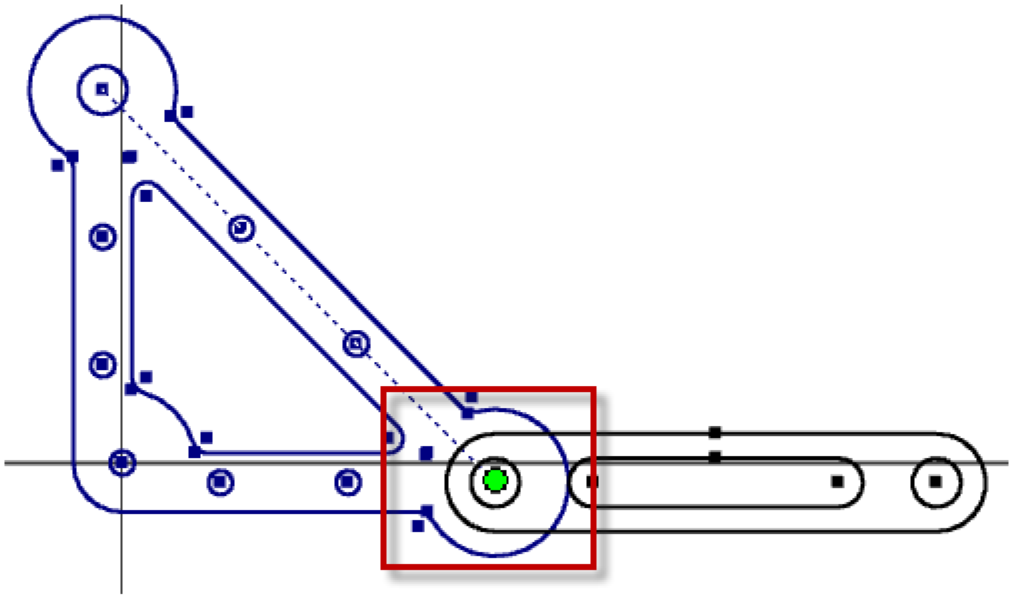
3. Place the Block's insertion point on the Origin of the Part file.



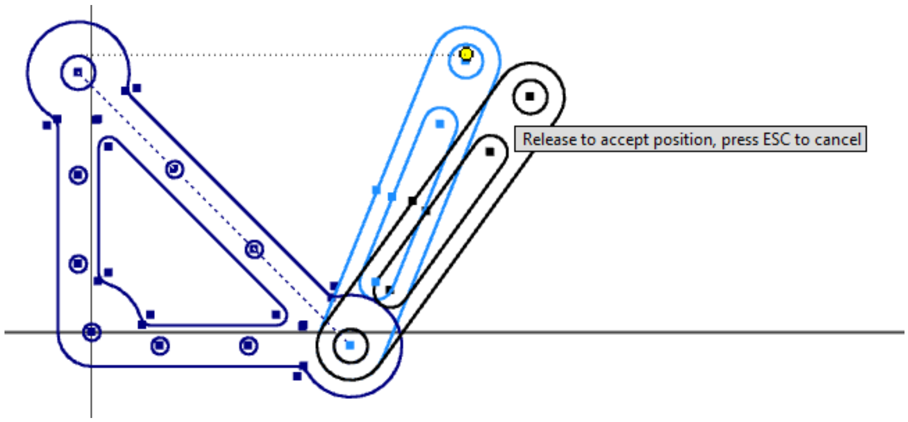
4. Place a Vertical Constraint on one of the vertical edges of the Sketch Block. This will lock the sketch block's position in the sketch.



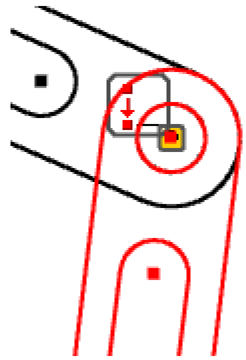
5. Place an Instance of the Sketch Block "**Centerlink"** next. Place this on the lower right hole in the Plate_Web.



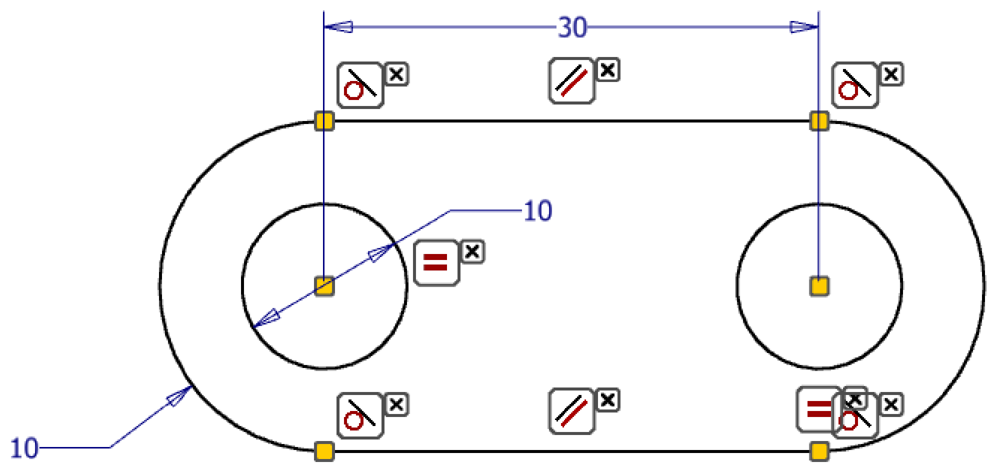
6. Click and hold on the **Centerlink**'s outermost center point and drag it around. Notice the **Centerlink** is constrained to the center point of the **Plate_Web**'s circle.



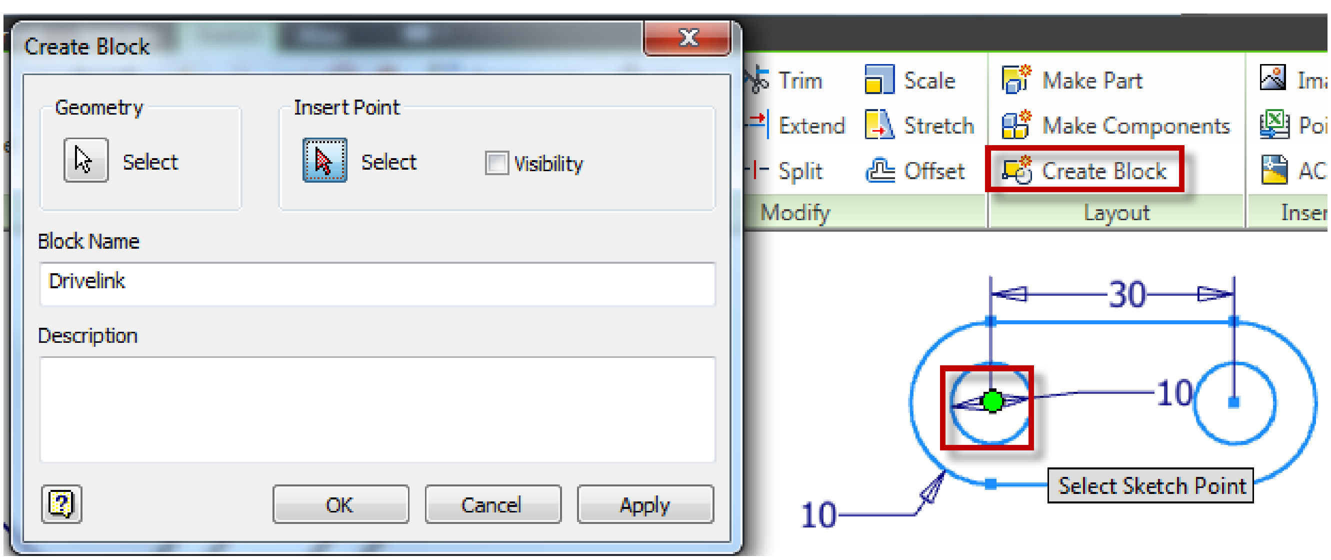
7. Place an instance of the **Rockerlink**, this time place it in space and do not attach it to anything. Place a **Coincident** Constraint between two of the circles on the links.



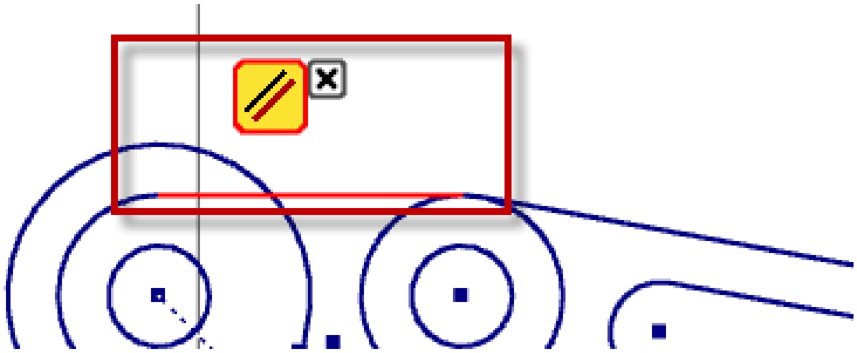
8. Create the following geometry somewhere in the sketch. Make sure no other constraints get created related to the placed blocks or it may cause trouble later.



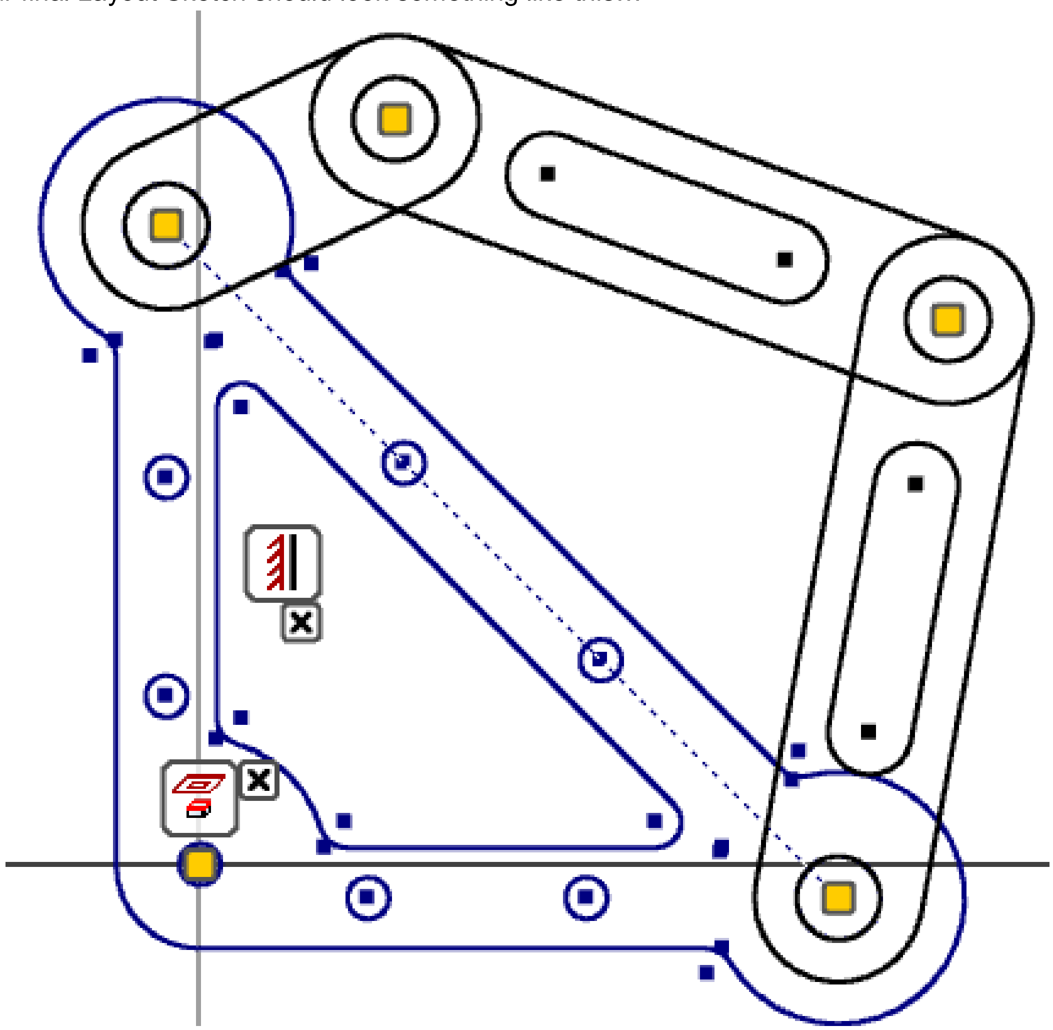
9. Start the **Create Block** command from *the Sketch Tab > Layout Panel*. Select the geometry you just created. Select the left center point as the *Insert Point*. Name the block **Drivelink** and select OK.



10. Apply **Coincident** constraints to the **Drivelink** and **Rockerlink** circle center points as well as the **Drivelink** and **Plate_Web** circle center points. Make sure no other constraints get created that restrict movement. If this does get created, make sure you delete it.



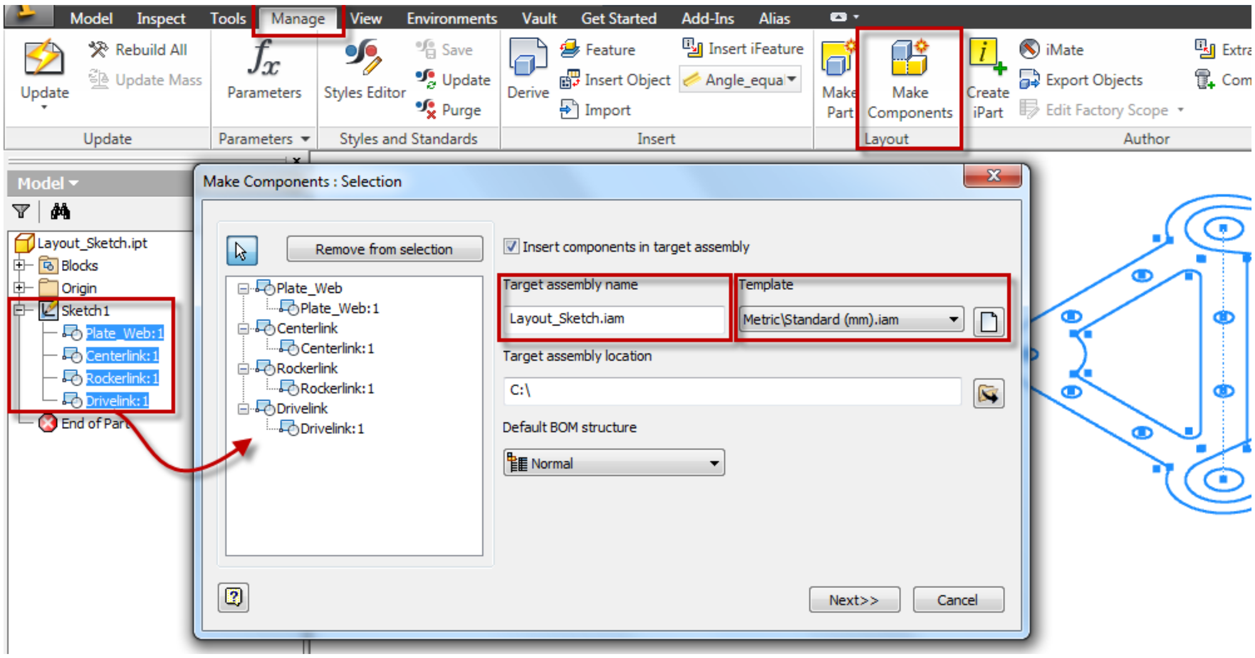
11. Your final Layout Sketch should look something like this…



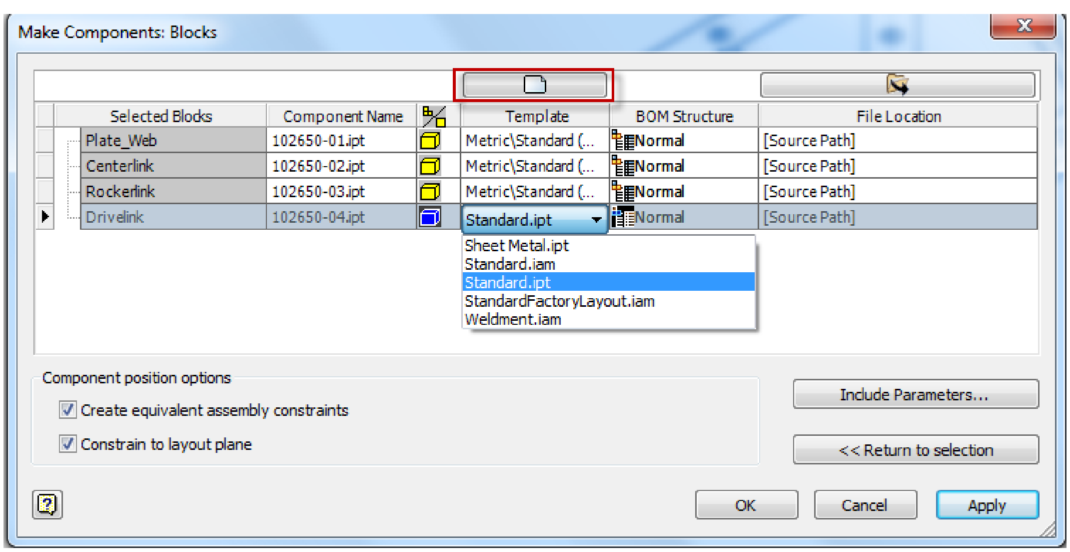
12. Finish the Sketch.

Task 2: Create Layout Assembly

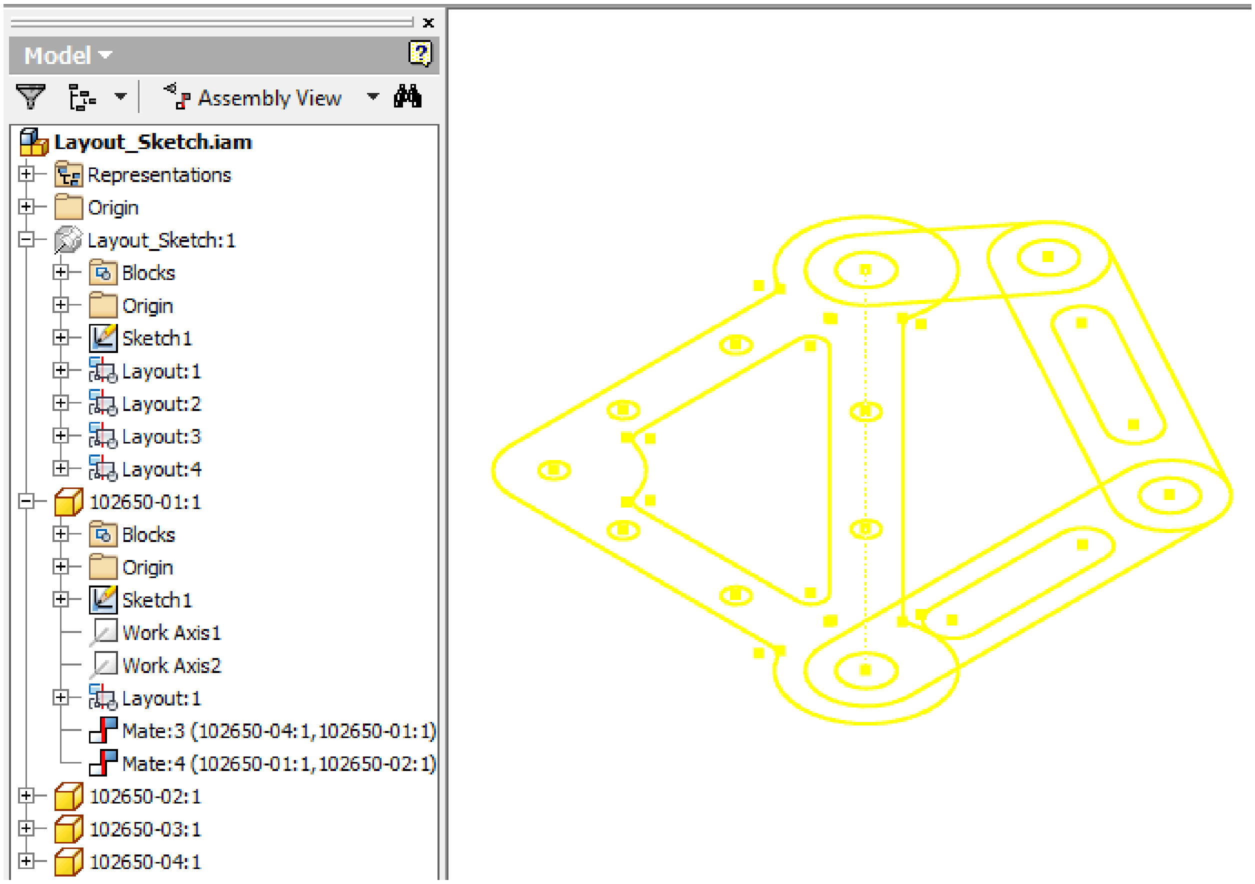
1. After finishing the sketch, start the **Make Components** command located on the *Manage Tab > Layout Panel*. Expand Sketch 1 and select the 4 blocks listed under the sketch. These will appear in the selection area of the dialog box. Select the *Target Assembly Name* as **Layout_Sketch.iam** using the default Standard.iam (mm) template. You might have to click the little icon for template selection. For the *Target Assembly Location*, choose the workspace the **Layout_Sketch.ipt** is located. Click Next.



2. This next box will allow further setup of the files that are to be created from the Sketch Blocks. Template selection, overriding source directories, parameters, and constraint creation could all be done from this box as well as more unique file naming. Change the names of the files as shown below (102650-01, 102650-02, etc.) and also make sure they are using the **Standard (mm).ipt** template. Click Ok.

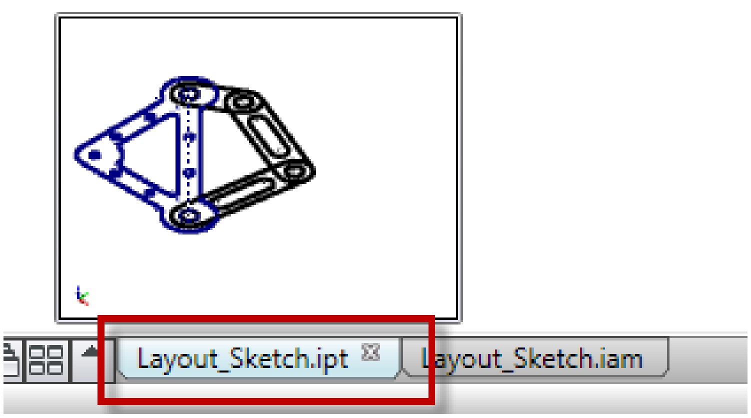


3. A new assembly file will be opened with the **Layout_Sketch.ipt** file placed an Invisible component with a BOM structure set to Reference. Assembly constraints will be setup based on the sketch constraints used during the layout process. There will also be some assembly constraints suppressed based on this same reasoning.

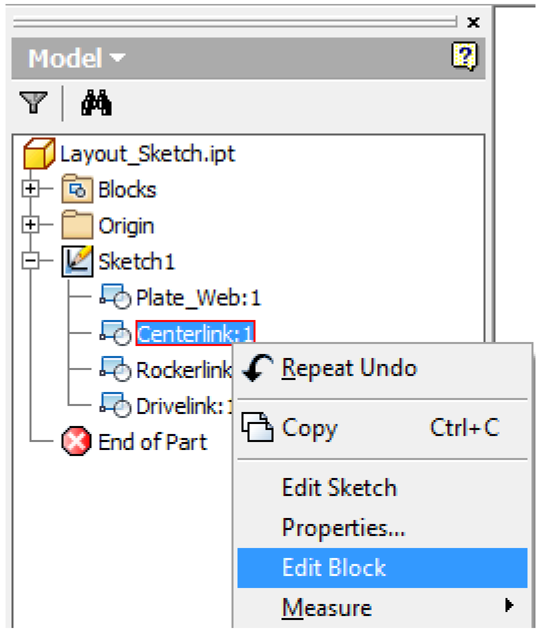


Task 3: Adjust Layout Sketch based on Design Changes

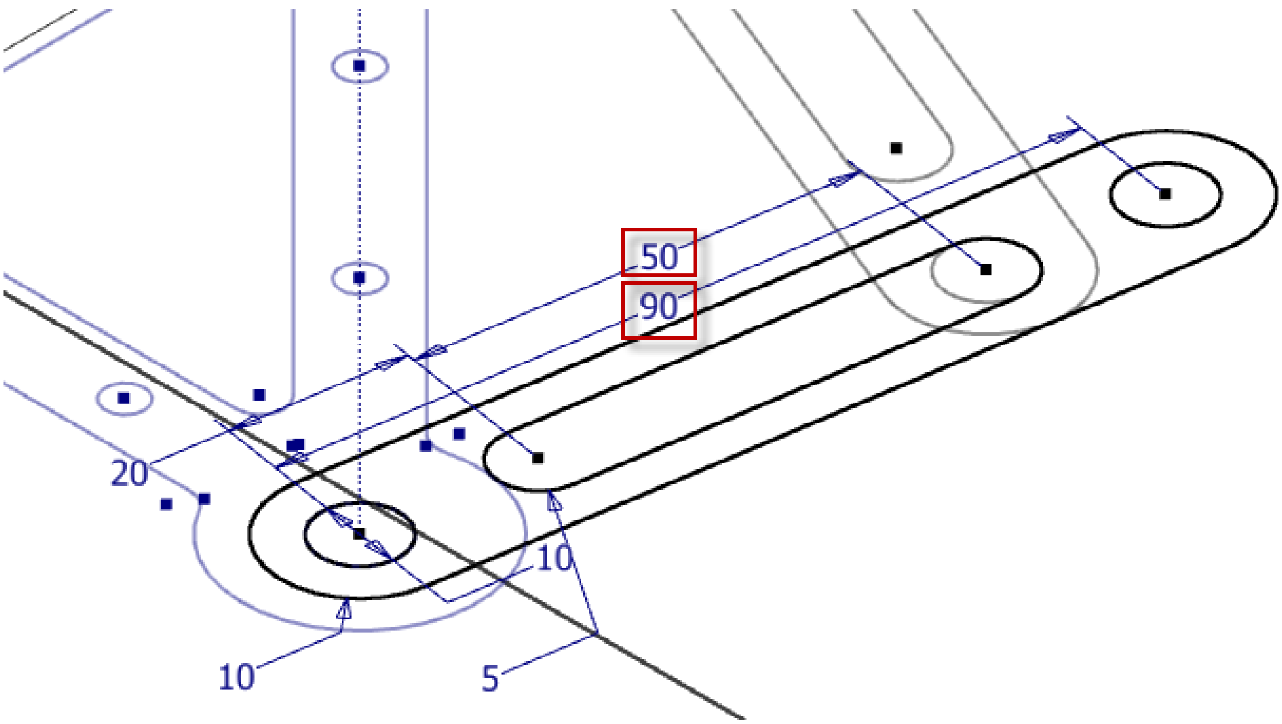
1. Grab the center point where the **Drivelink** connects to the **Rockerlink**. Try to swivel the **Drivelink** <u>around</u> the **Plate_Web**. Notice the Linkage does not permit full revolution to occur because either the **Centerlink** or the **Rockerlink** are too short.
2. Toggle back to the **Layout_Sketch.ipt** which should still be open in session.



3. Right click on **Centerlink:1** under Sketch 1 and select *Edit Block*.



4. Change the values of the **30** dimension to **50** and the **70** dimension to **90**. Right click and chose *Finish Block Edit*.



5. Toggle back to the **Layout_Sketch.iam** and click the **Update** button on the Quick Access Toolbar. Now try to swivel the **Drivelink** around the **Plate_Web** and notice the changes confirm the design intent.

Task 4: Create Geometry for Parts (if time permits in lab)

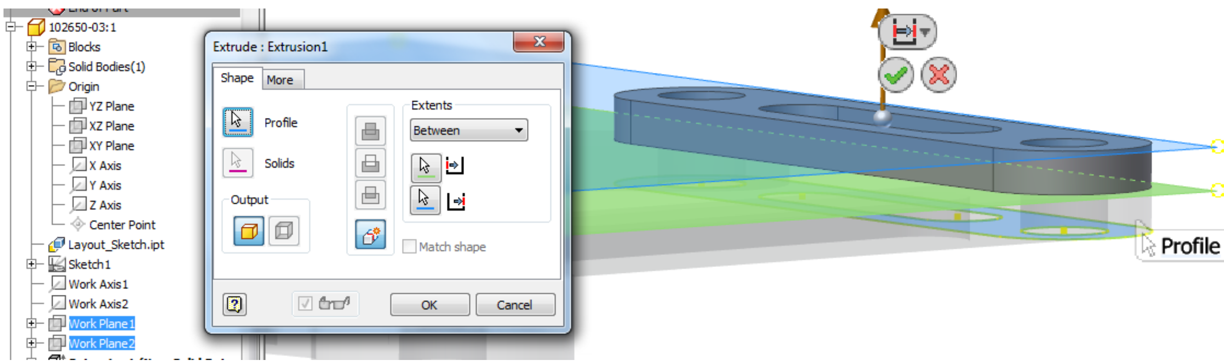
1. Double click on the parts in the assembly to edit them and create extrusions.

   a. Plate_Web → **10** mm in negative Z direction 

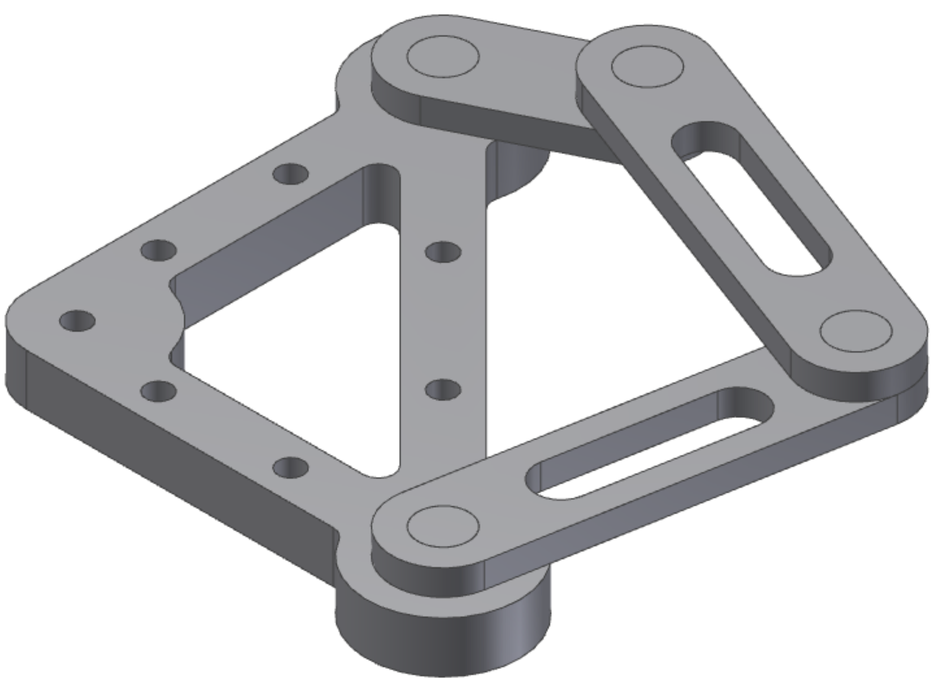
   b. Drivelink → **5** mm in positive Z direction 

   c. Centerlink → **5** mm in the positive Z direction 

   d. Rockerlink → Offset 2 Work Planes from the XY Origin Plane (**5** mm and **10** mm) Extrude using the between option and select the two Work Planes. Turn off the Visibility of the two Work Planes when finished so they do not show up in the Assembly.



   e. You can also create Pins for the links using middle-out design with in place components. (Make sure if you do not want Adaptivity to hold down CRTL when projecting references)



Documentation, Tracking, and Iteration

Now that you have gone through these labs, you are probably getting a lot of great ideas about directions you can take with your Inventor designs. Take a deep breath and remember these last few pointers.

Documentation

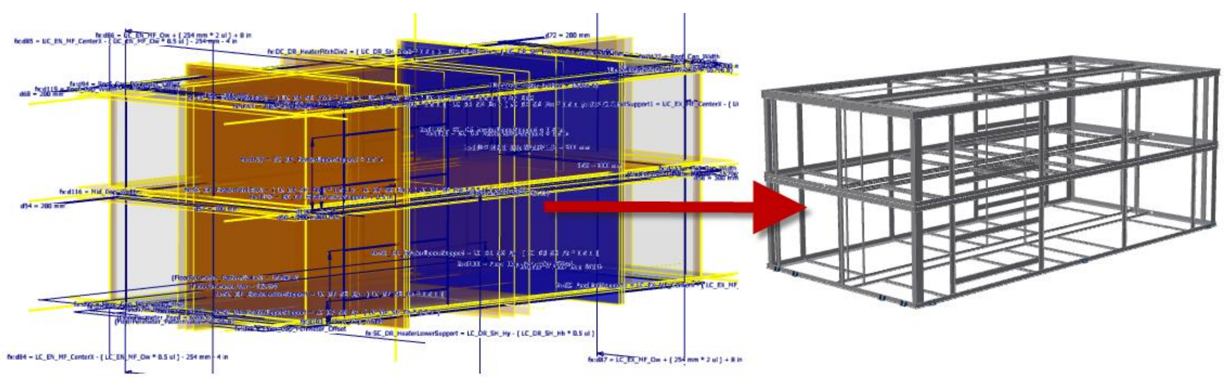
Perhaps the most important thing to remember about any Top-Down design method when working in a collaborative group is that it will only be as good your weakest link. The key to success with these methods and across your user base will be accurate and concise documentation of your design method. This can be done with good parameter naming and commenting, iLogic rules and VB forms, word files with screen grabs or even the Inventor Engineer's Notebook. If you are a lone gun, then you have free reign. But documenting your own design has its own merits as well including troubleshooting and new employee training when your team grows.

Tracking

File management is always an important background task of 3D modeling and Top-Down designs are no different in Inventor. Due to the nature of derived parts and links to other files, good workspace project management is just as important as ever. If you are using Vault, this workspace management is taken care of and files are checked into the Vault while automatically retaining the links and folder hierarchy so the next time you need to work on the design there will be no missing references or name mismatches and the files will have track-able versions to go back on.

Iteration

When it comes time to iterate a Top-Down design to a same-as-but-different build for a new project or client, the use of Vault takes the iteration time down to minutes instead of hours with the use of Vault's Copy Design. Consider if you have a Skeleton Model that references close to 100 or 200 parts. You will have to play a very dangerous game with project files and workspaces when you start renaming the Master part since every file has to be able to find that file for its references, but Vault will manage all of this for you.



Class Summary: Autodesk Inventor has several Top-Down design techniques to aid users in the tedious task of design intent changes and iterations. Top-Design unleashes the power of parametric design with Inventor to speed up the time of design validation and conformity to help companies shorten lead time to market and production of their products through use of well-developed modeling techniques.