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Introduction

CATIA Version 5 Functional Molded Part

Upon completion of this course, the student should have a full understanding of the following topics:

- Creating basic part features
- Creating functional part features
- Modifying part features
- Using functional modeling in context of an assembly

Functional Molded Part

The Functional Molded Part workbench in CATIA V5 uses a functional approach to modeling parts. This methodology hinges off of the idea that the part has an inherent inside and outside. The inside of the part, called the core, and the outside of the part, called the cavity, interact with part features in different ways. We can define a part feature that adds material to only the core of the part, for example. By using this modeling technique, we can define a certain class of parts in a more robust fashion. The workbench was initially designed for use in the plastics industry, or any part that would be made from an injected mold. As far as the aerospace industry is concerned, machined parts are typically good candidates for functional modeling. The specific scenarios that benefit from functional modeling will be investigated later.

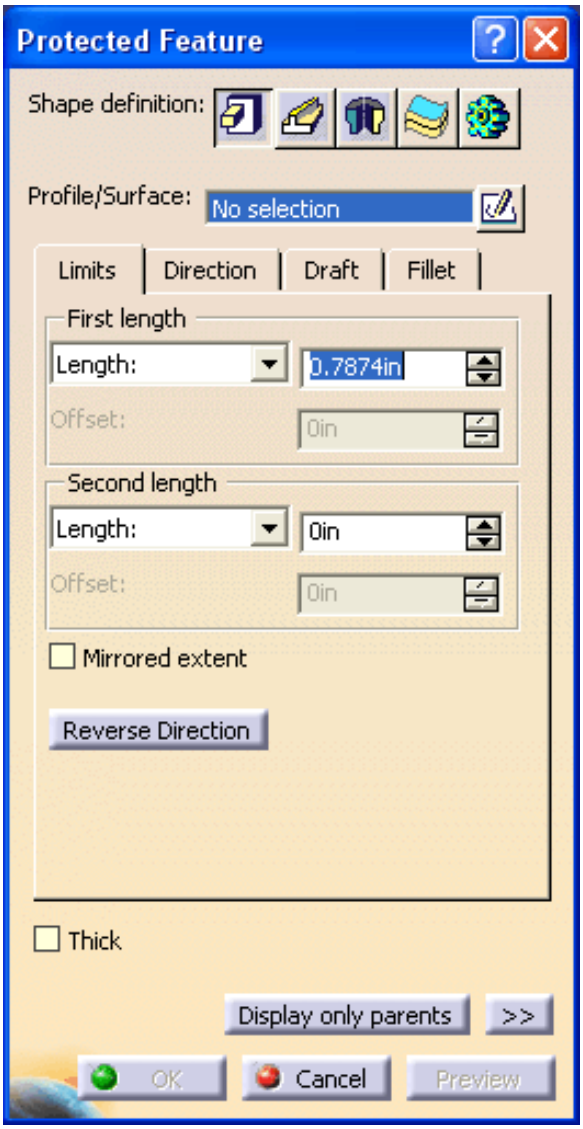
The first step in functional modeling is to define the core of the part. The core of the part is what defines the boundary between the inside and outside of the part. The core will be defined using shape features. Shape features are the basic building blocks of the Functional Molded Part workbench and will be the first focus of this material.

The second step in functional modeling is to add more features to the part using functional features. Functional features have functions built into them (e.g. add material, remove material, or reserve space), and will be applied to specific areas of the part (e.g. core, cavity, or both). For example, the rib functional feature adds stiffeners to the core of the part. Both functional features and shape features can be modified using feature modifiers. Feature modifiers will modify the definition of a feature by using cuts, intersections, and fillets.

After all the necessary shape and functional features are covered, the role of Functional Molded Part within the context of the assembly will be investigated. Functional modeling lends itself well to relational modeling, and useful functions and scenarios will be further discussed. Working in the context of an assembly will be the fourth and final focus of this material.

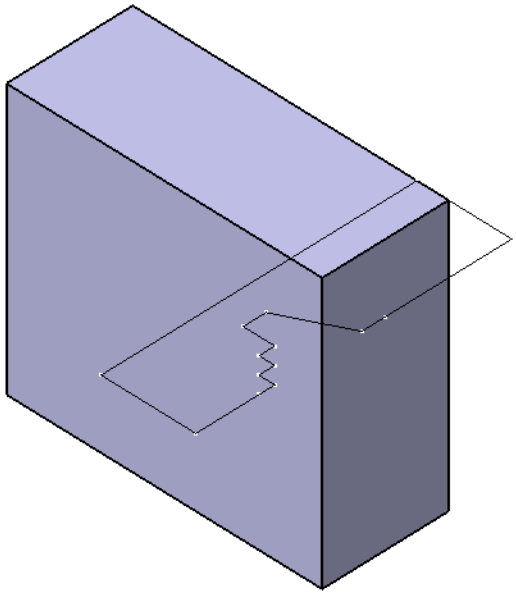
Protected Feature

The protected feature option allows you to create a protected volume without adding any walls or material to the part. The volume will delete any material within it and prevent any material from being added to the inside of the volume in the future. The benefit of this is creating protected spaces within parts where you know another part will fit or where you know you shouldn't have any material.



Notice that the *Core* tab is no longer available because *Protected Feature* does not contribute to the core of the part. The available *Shape definitions* remain the same and will not be covered again in this section. Please refer to the Shellable Feature section for review if needed.

Open the Protected Feature document and save it with your initials. The part should appear similar to the following part.

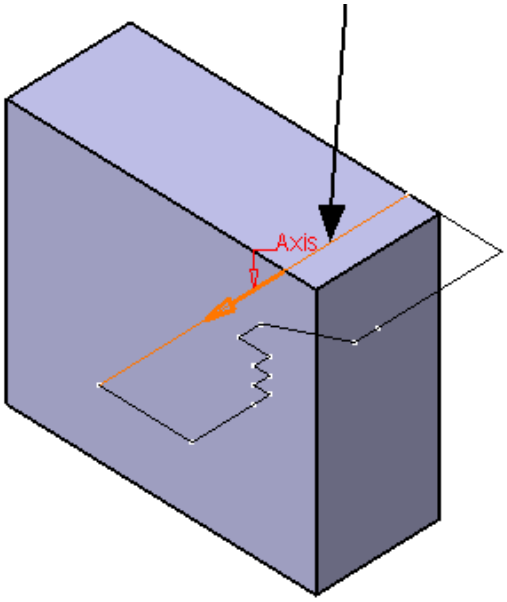


Select on the Protected Feature icon.  The *Protected Feature* window appears.

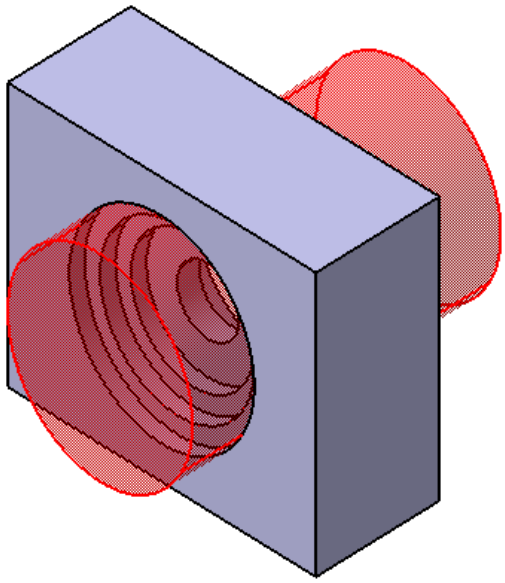
Select on the Revolve icon for the shape definition.  This will bring up the revolve window.

Select *Sketch.2* for the *Profile/Surface*. You will create a revolution of this sketch around the main axis.

Select *Sketch.2\Edge.1* for the *Axis*. The edge is shown in the following picture.



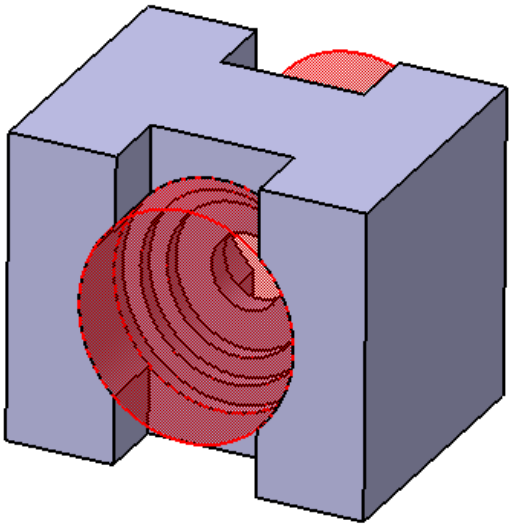
Select *OK*. The part will now have a protected volume through it and will look similar to the following picture.



Show *Sketch.3*. You will add this to the shell to demonstrate how material is added to the protected volume.

Select the Shellable Feature icon and select *Sketch.3* for the *Profile/Surface*. 

Select *To plane/surface* for the *First length* and select the opposite side of the shell body, select *OK*. The part should now look similar to the following picture.

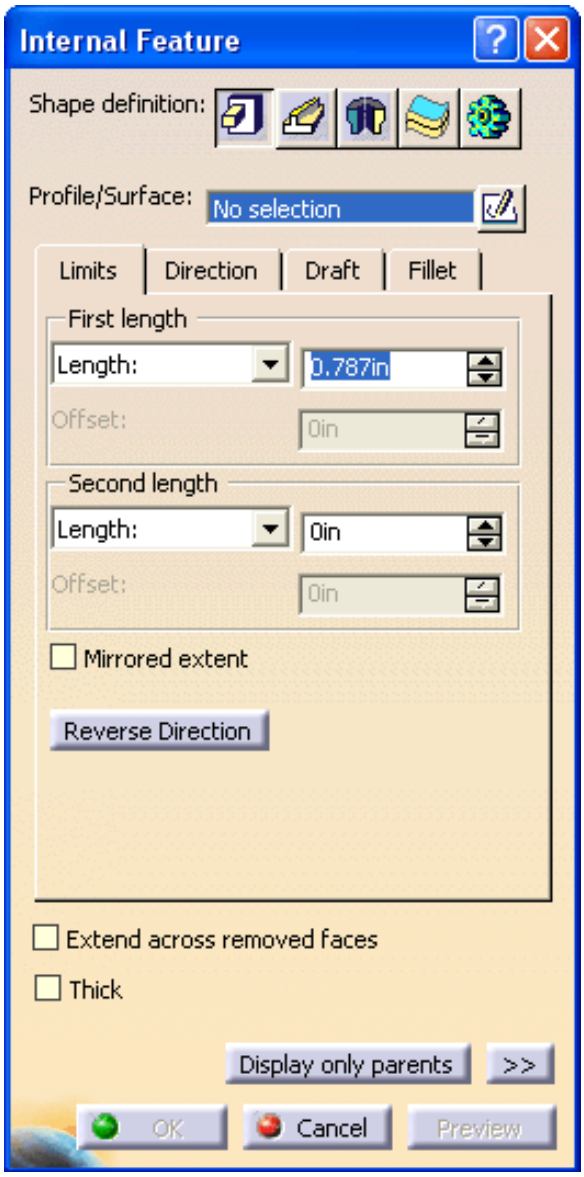


Notice that no material is added within the protected volume.

Save and close your document.

Internal Feature

The internal feature will allow you to add material internally to a shelled or a core feature. An internal feature can be very helpful on odd shaped parts where you want the new feature to be trimmed off at the outside surface and only be shown internally.

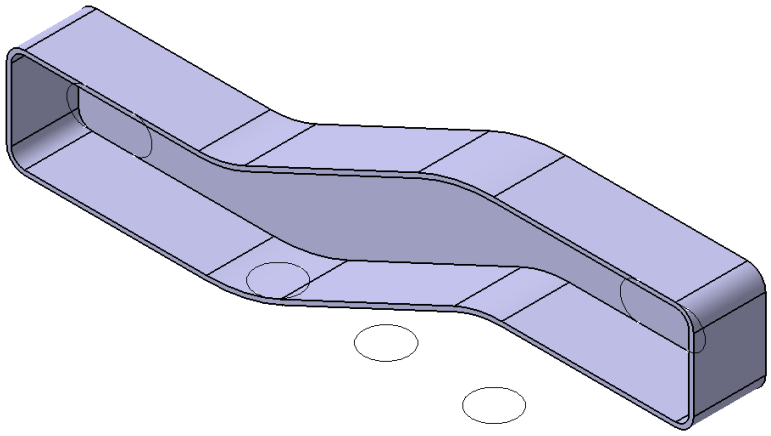


To Shell — Prism extends until it intersects the core of the part

Extend across removed faces — Allows the profile to extend past outside face, if face was removed by Shellable Properties

Notice the rest of the window is the same as the *Shellable Feature* window, with the exception of the *Core* tab. The available *Shape definitions* remain the same and will not be covered again in this section. Please refer to the Shellable Feature section for review if needed.

Open the Internal Feature document and save it with your initials. You should see a part similar to the part shown below.

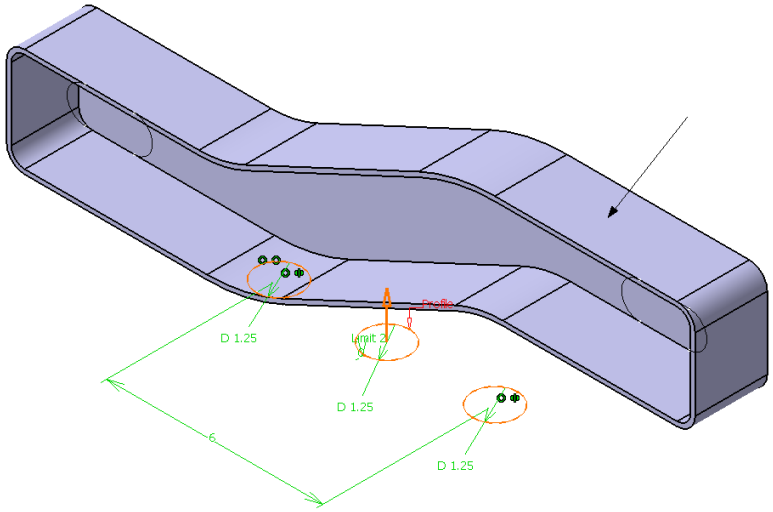


Select the Internal Feature icon.  The *Internal Feature* window appears. This will allow you to use a sketch to create an internal prism.

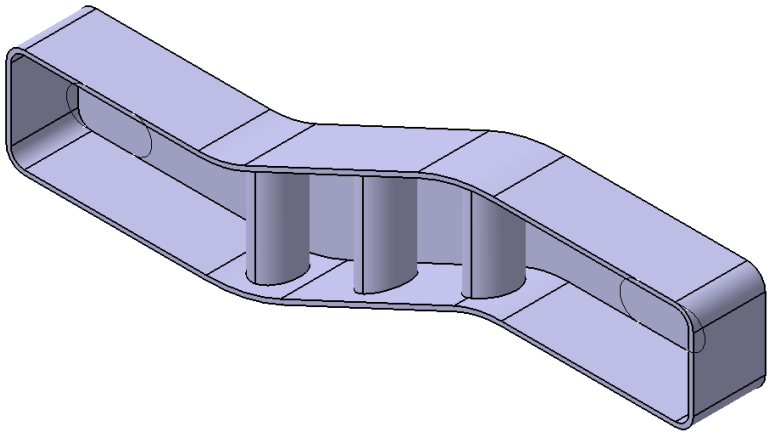
Select *Sketch.3* from your specification tree for the *Profile/Surface* selection box. This defines which sketch you will be using to create the internal feature.

Open the drop-down box for *First length,* and select *To Plane/Surface.* This will allow you to specify a plane or surface that you want the *First length* to be extruded up to.

Select on the upper face of the part like shown below.



Select *OK.* Your part should now look like the one shown below. Notice that even though the sketch is being extruded before and past the surfaces of the part, the only place that it is creating material is in the internal area of the shelled prism.



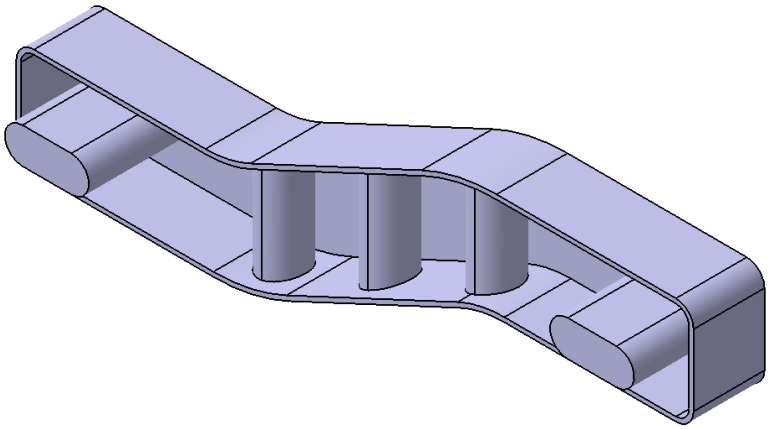
Select on the Internal Feature icon.  You will now be creating another internal feature with the second sketch given.

Select *Sketch.4* from your specification tree for the *Profile/Surface* selection box. This defines the sketch you will be using to create the internal feature.

Change the *First length* to 2.0 and select *Mirrored extent.*

Select *Preview*. Notice that even though the dimensions given will extend past the outside surface of the part, it will not create the material.

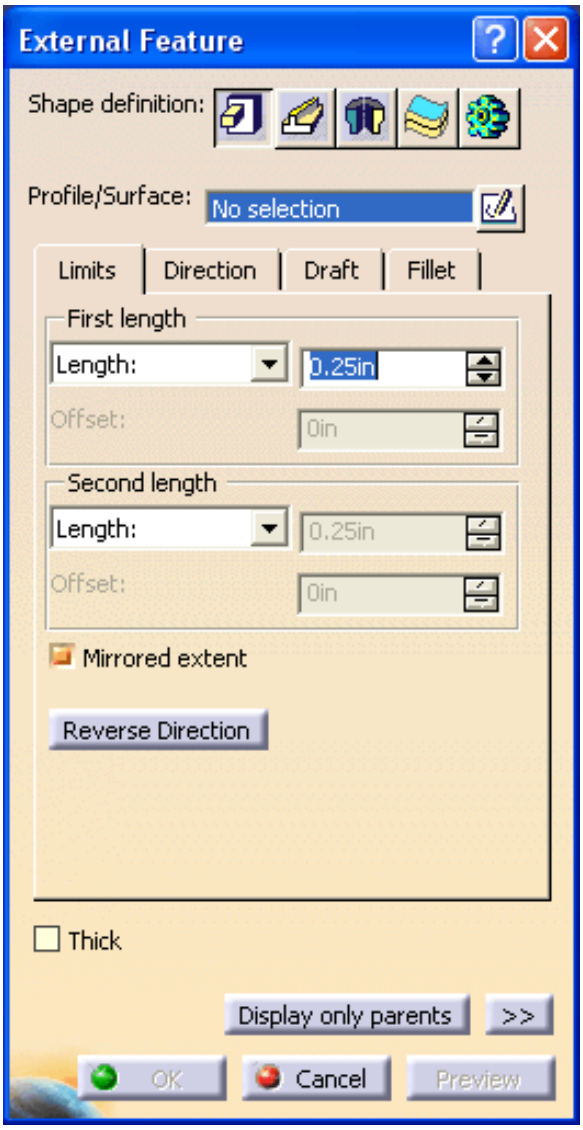
Select *Extend across removed surfaces.* Select *OK*. Notice it will extend out the full length specified only on the side where the face of the part was removed using the shell properties. Your part now should look like the one shown below.



Save and close your document.

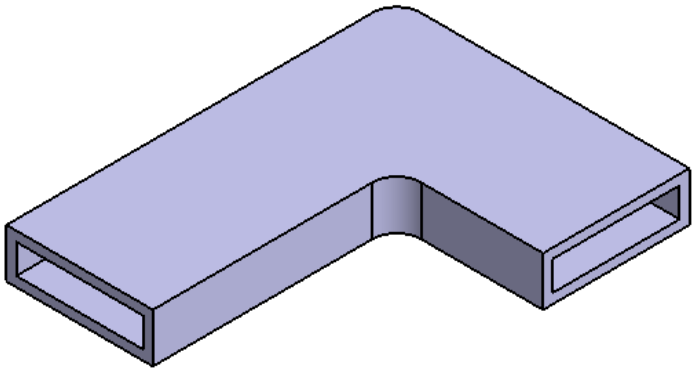
External Feature

The external feature is used to add a solid to the cavity of the part. The cavity of the part is defined as the volume outside the core of the part, which is defined by shellable features that were discussed in the previous section. External features are good for adding stiffeners to the exterior surface of the part, among other things.



Notice that options are all similar to those of the internal feature. Please refer to the Internal Feature section for review if needed.

Open the External Feature document. You should see a shellable prism with shell properties already applied to it, as well as three hidden sketches. You are going to add some external features to the part.

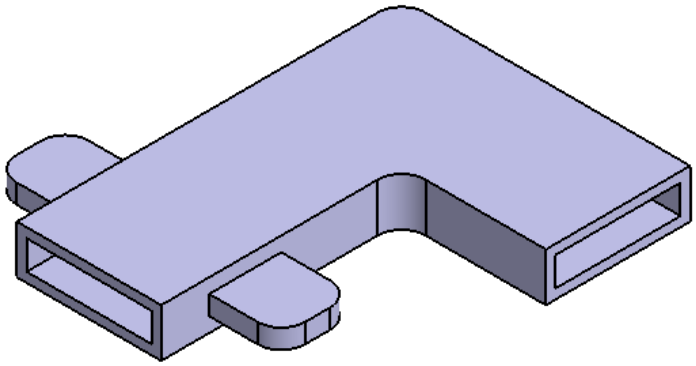


Select the External Feature icon.  The *External Feature* window appears.

Show *Sketch.2* from the specification tree. Make sure that the *Shape definition* is set to *Prism* and select *Sketch.2* to define the *Profile* for the *External Prism*.

Set the *First length* to 0.1 and make sure that *Mirrored extent* is checked.

Switch to the *Fillet* tab and set the *Lateral radius* to 0.25 and select *OK*. Notice that although the profile of the sketch extends through the core of the part, material is only added to the volume outside the core. This is the function of the external feature.



Select the External Feature icon again.  You are going to add a stiffener to the part using *Sketch.3*.

Show *Sketch.3* from the specification tree. Make sure that the *Shape* definition is set to *Prism* and select *Sketch.3* from the specification tree. This will define the *Profile* for the external prism. Notice that the *Thick* option was automatically enabled. Functional Molded Part will detect an open profile and enable the *Thick* option.

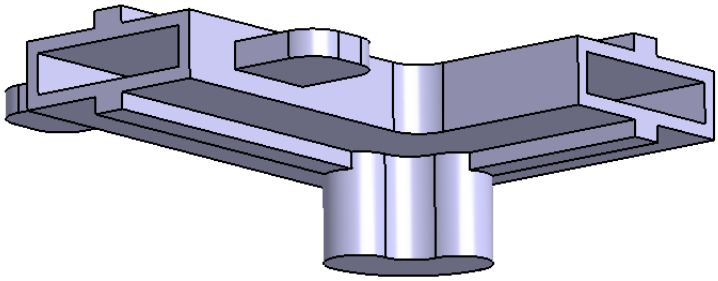
Set the *First length* value to 0.375 and make sure that *Mirrored extent* is checked. Under the *Thin solid* section, set the *Thickness1* value to 0.25 and make sure that the *Neutral fiber* option is checked. Select *OK* when done. This will fully define a stiffener that appears on both sides of the part. Notice again that no solid is created within the core of the part. Next you will explore an additional limit option that is available.

Select the External Feature icon again.  You are going to create another external prism to demonstrate a different limit option.

Show *Sketch.4* from the specification tree. The sketch is on a plane offset from the bottom of the part.

Select *Sketch.4* as the *Profile* for the *External Prism*. Make sure that the *Mirrored extent* option is deactivated and under *First length*, set the limit type to *To Shell* and select *Preview*. Notice that the prism extends until it reaches the shell, or core of the part.

Select *OK*. The final part should appear as below.



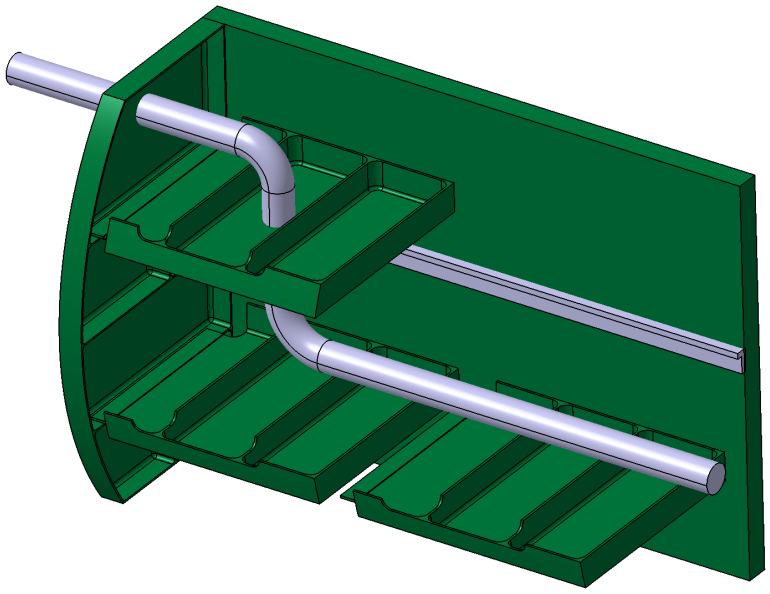
Save and close your document.

Assembly Context

Nearly every part you make will be a part of some assembly. The Assembly workbench is the same regardless of whether the specific parts are created in part design or functional molded part. However, due to the intrinsic properties of a functional modeled part you can create a more dynamic assembly. In this section you will see how to use these properties to your advantage.

All functions that you will use have already been covered so the basic functionality will not be discussed. You will now learn how to use them in the context of an assembly. These functions are the *Push*, *Pull*, *Fitting*, *Cut* and *Fillets*. You also have the ability to reuse features using the copy paste special options.

Open the Assembly_Context document and save it with your initials. Remember that since this is an assembly that you should use save management to propagate all the files to one location and save your documents. The assembly should look similar to the following picture.

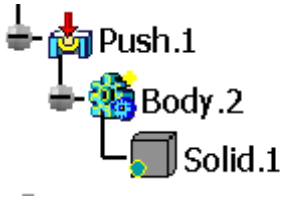


Note: When working in assembly you will want to turn on the Keep link with selected object option located in the pull down menu Tools, Options, Part Infrastructure, General tab. This will keep any external references linked with the part that created those references. This allows the assembly to update if the position of those references change. Without this option turned on all external references will be created as datums and will not update with respect to the original geometry.

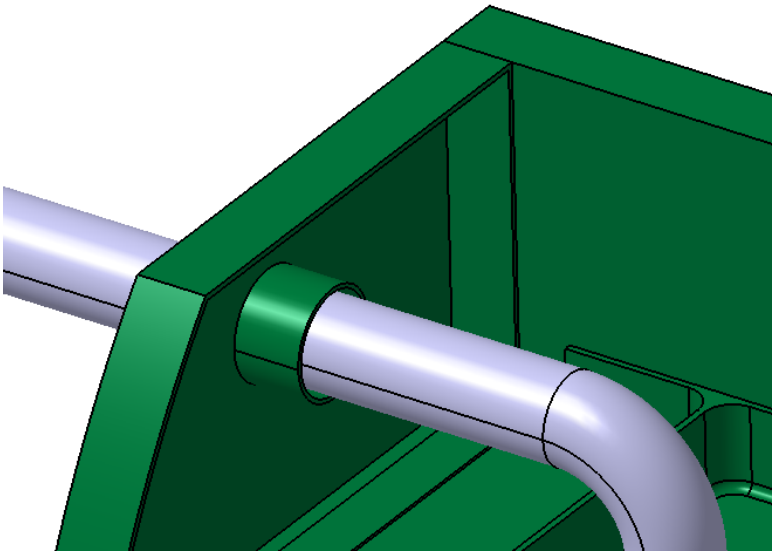
Double select on the *Part Body* of *Frame-001* then set the *Solid Functional Set.1* to be the *Define In Work Object*. This should put you within the partbody and in the functional molded part workbench.

Select the Push icon.  You are going to push the tube through *Frame-001* creating a hole and reinforcement wall.

Select the tube for the *Tool body*. You should be able to select it by just selecting on the part body. In the *Push* definition window you should see *Body.2* in the *Tool body* selection box. This is because FMP will automatically insert a body with the solid used to create the push. This is shown in the following picture of the tree.



In the *Clearance* selection box enter 0.08 and select *OK*. The tube should now be going through a hole in the frame with a reinforcement wall as shown in the following picture.

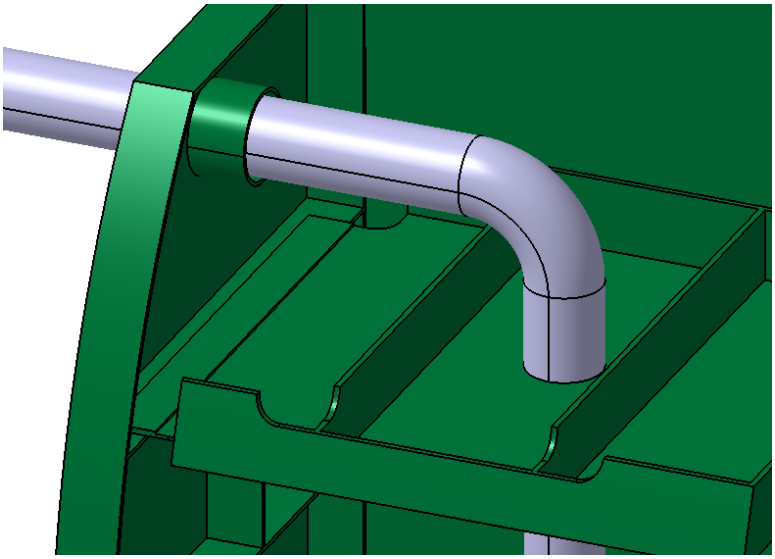


Double select on the *Part Body* for *Deck FWD-002* then set the *Solid Functional Set.1* to be the *Define In Work Object*. You are going to put the tube through the deck as in the frame but this time you will use *Fitting*.

Select the Fitting icon and select the tube for the *Tool body*. 

Select *OK*. Unlike push that added a wall around the tube, fitting simply removes any material that intersects the tool body solid.

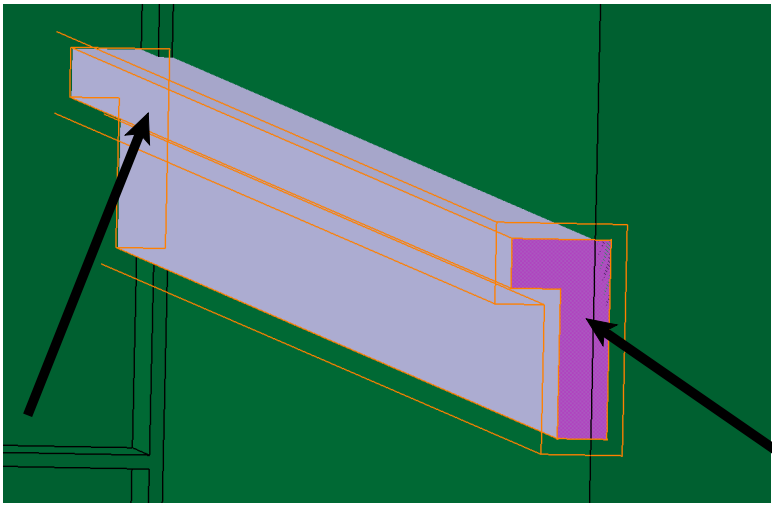
Hide *Fitting.1* in the *Solid Functional Set.1*. The deck should look similar to the following picture.



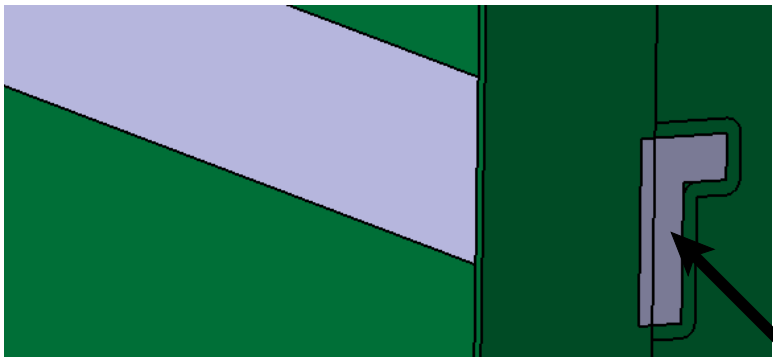
Double select on the *Part Body* for *Keel-001* then set the *Solid Functional Set.1* to be the *Define in Work Object*. Now you will use the stiffener for a pull.

Select the Pull icon and select the stiffener for the *Tool body*. 

For *Faces to remove* under *Openings* select the two outside faces of the stiffener. The faces are shown in the following picture.



Turn on *Constant wall thickness* and select *OK*. The keel will now look similar to the following picture.



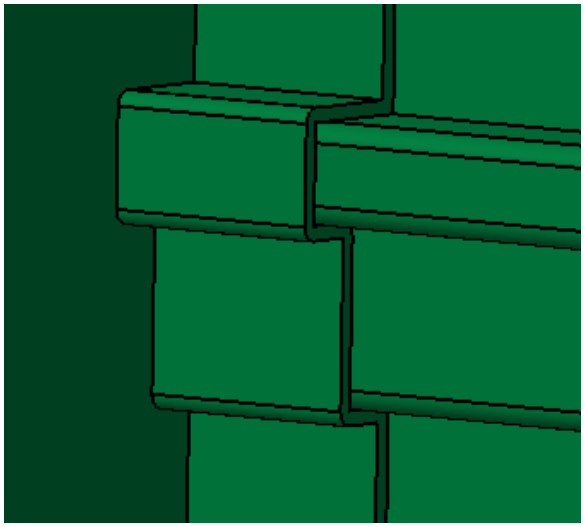
Note: While the keel is pulled around the stiffener, Frame-001 is unmodified. So the frame and the keel now intersect around where the arrow is located though this may be hard to see. You will need to do a Push on the frame to fix this.

Double select on the *Part Body* for *Frame-001* then set the *Solid Functional Set.1* to be the *Define In Work Object.*

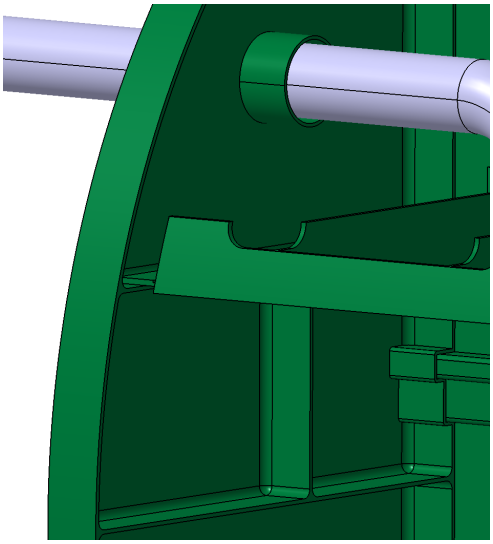
Select the Push icon and select the stiffener for the *Tool body*.  You should see *Body.3* appear in the selection box.

In the *Clearance* selection box key in 0.08. This will offset the *Tool body* from the stiffener, accounting for the thickness of the keel.

Turn on *Constant wall thickness* and select *OK*. The frame will now appear to wrap around the keel. *Frame-001* should now appear similar to the following picture.



Set the *PartBody* to be the *Define In Work Object*. You can now see the part design fillets that have been applied to the frame. Notice that there are no fillets around where you pushed the tube and stiffener through, you will need to add them.



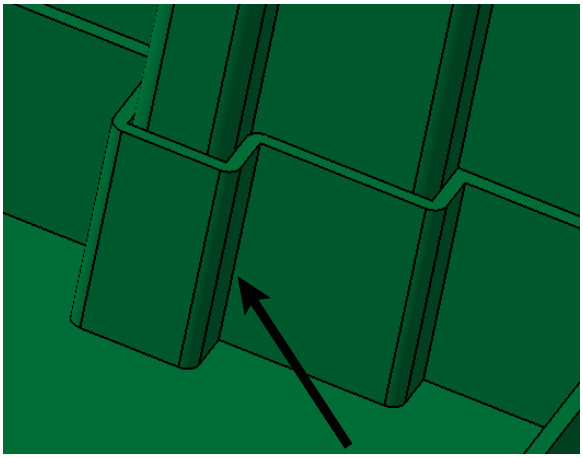
Select the Edge Fillet icon. 

Change the *Selection mode* to *Intersection with selected features*. For the *Objects(s) to fillet* select *Push.1* for the *Selected features* select *Rib.2*, enter 0.12 for the *Radius* and select *OK*. The fillet you created will not be seen until the *Tube* is moved later in the exercise.

If a *Warnings* window appears that states the fillet is unnecessary, just select *Close*. This window might appear several times as you work along. In all cases just close the window.

Select the Edge Fillet, local modifier icon. 

Select the edge shown below, enter 0.06 for the *Radius* and select *OK*. You could use a part design edge fillet but in this case it would have a greater chance of failing.



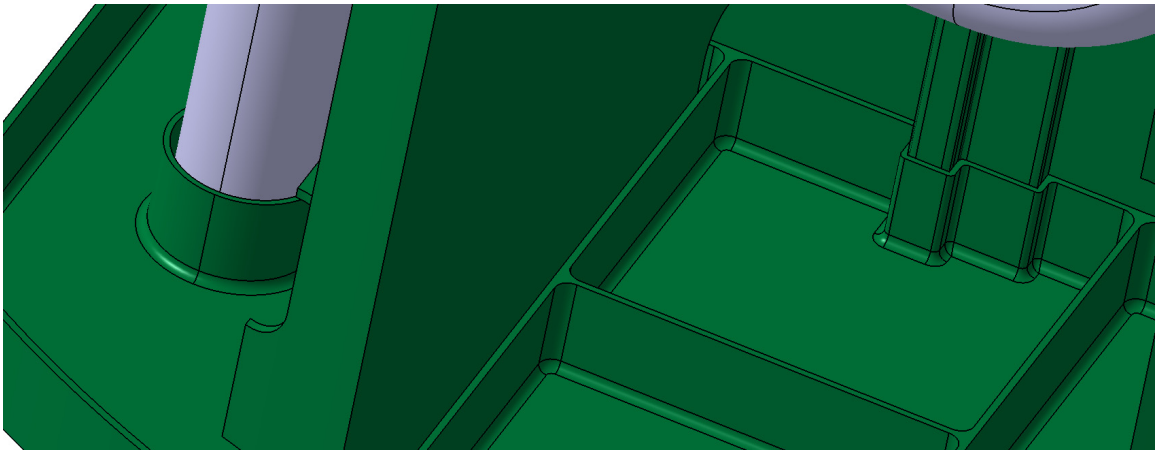
Set the *PartBody* to be the *Define In Work Object.*

Select the Edge Fillet icon. 

Change the *Selection mode* to *Intersection with selected features*. For the *Objects(s) to fillet* select *Push.2*, for the *Selected features* select *Internal Thick Surface.4*, enter 0.06 for the *Radius* and select *OK*. Now you can add the pushes created to an existing edge fillet.

Double select on *EdgeFillet.3*. This fillet is created using the intersection with selected features rather than picking edges. This is why when you pushed the stiffener through the frame and modified that edge the fillets didn't break as they normally would.

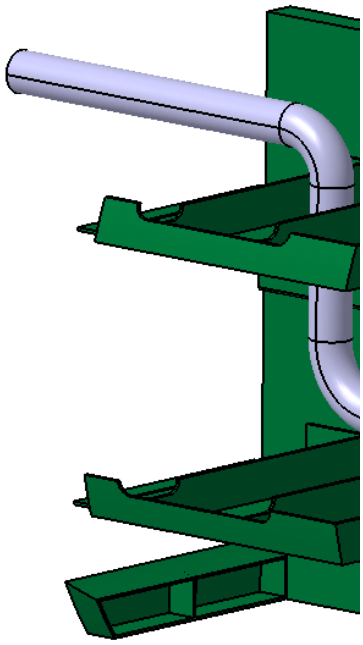
Add *Push.1* and *Push.2* to the *Object(s) to fillet*, change the *Radius* 0.09 and select *OK*. Now the walls of the two pushes will be included in the fillet as you can see in the following picture.



Select the Cut icon.  You are now going to cut the frame as if it intersects a loft surface.

Select the *Shellable Prism.1* for the *Features to cut*. Notice that *Cut.1* appears in the selection box. This is because there is already a cut operation on the shellable prism and FMP automatically recognizes this feature and takes the name of its modifying operations.

Select the bottom surface for the *Cutting element* and select *OK*. Either select the graphical surface, you may have to unhide the *MDF* instance, or select the published *bottom_loft_surface* within the *MDF* instance. You may notice that *Surface.3* is what appears in the selection box. Your assembly should be similar to the following picture.



Notice that even though the wrong side was cut the part doesn't break.

Double select *Cut.2* flip the arrows and select *OK*. Now the correct side of the frame is left.

Set the *PartBody* as the *Define In Work Object*.

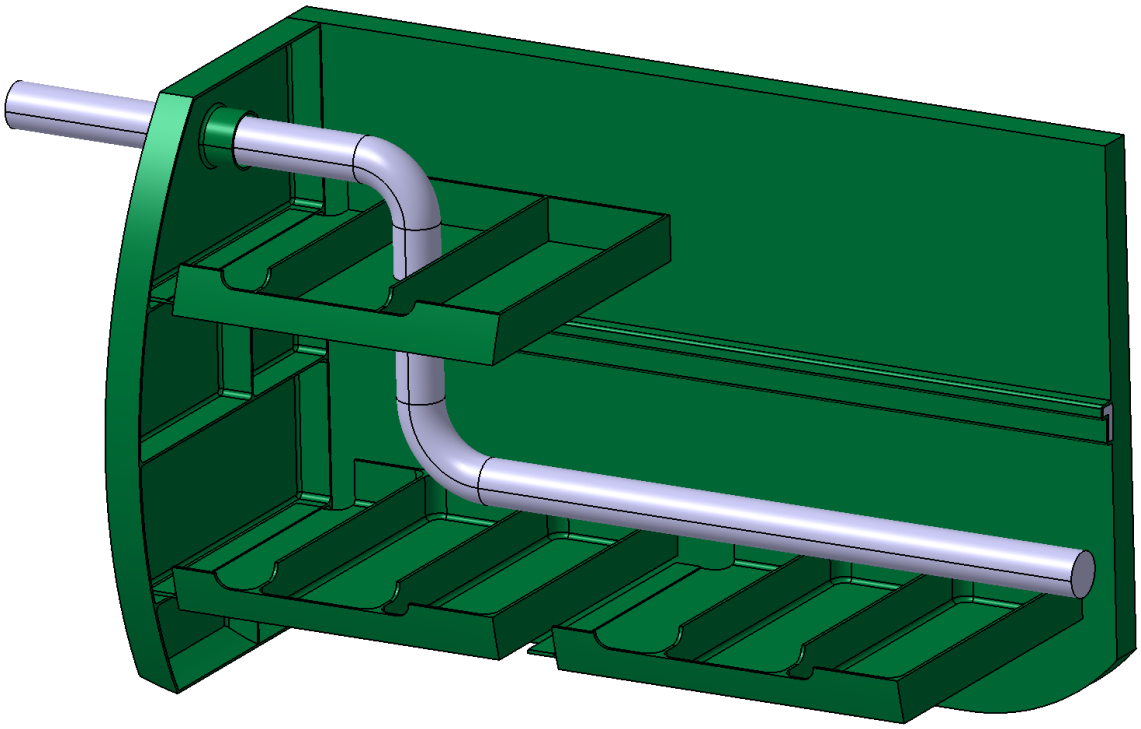
Double select the *Part Body* for *Keel-001* then set the *Solid Functional Set.1* to be the *Define In Work Object*. Now you will cut the keel with the same surface.

Click the Cut icon and select *Shellable Prism.1* for the *Features to cut*. 

Select the *bottom_loft_surface* for the *Cutting* element and make sure the arrows are pointing to the top. Notice that this time *Surface.1* appears in the selection box.

Select *OK*.

Set the *PartBody* as the *Define In Work Object.* The assembly should appear similar to the following picture.

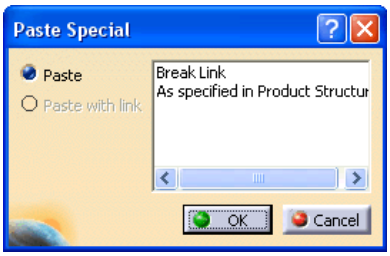


Now you are going to add two more frames to the assembly. Instead of inserting the frames and redoing the various modifications you will use copy, paste special.

Double select on the *Assembly_Context.* This will take you to the assembly level.

Using the third mouse button, *Copy* the *Frame-001* instance. You are going to create another part by using the copy paste special.

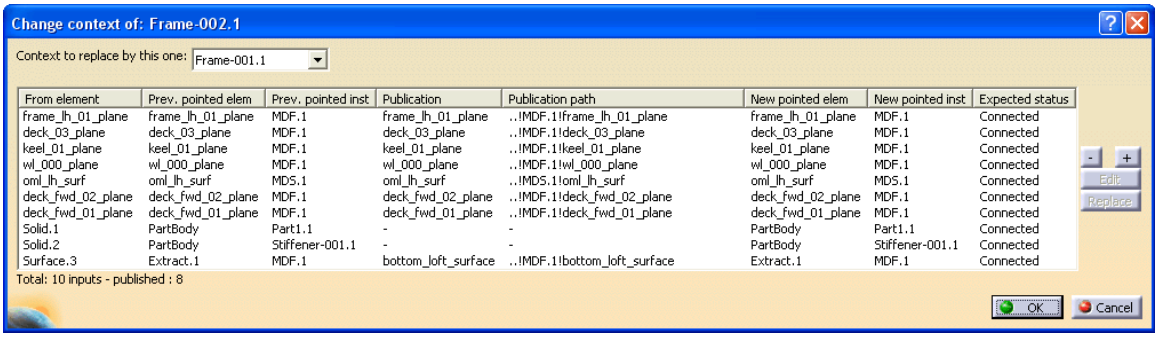
Using the third mouse button, select on *Assembly_Context* and select *Paste Special*. A window will appear similar to the following diagram.



Select *Break Link* and select *OK.* You should see another frame instance appear in the assembly. It will be called *copy (1) of Frame-001*.

Change the *Part Number* to *Frame-002* and the *Instance name* to *Frame-002.1*.

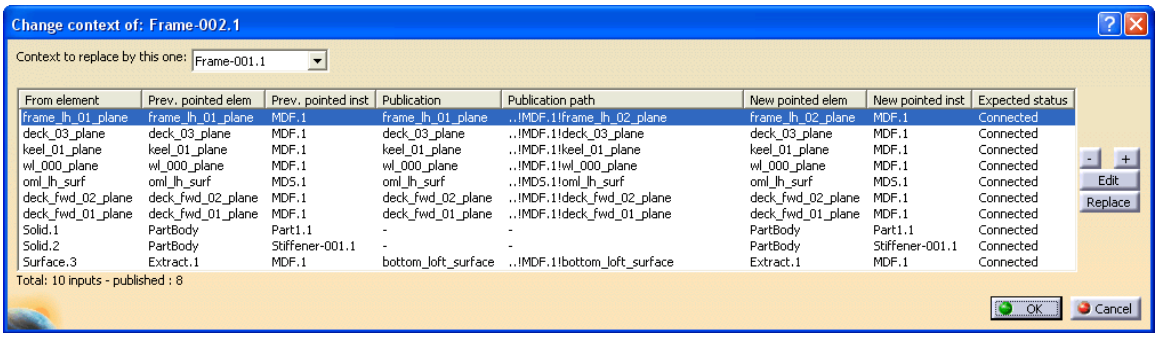
Press the third mouse button while on the instance name of the new part, select *Components* and select *Define Contextual Links.* The following window will appear.



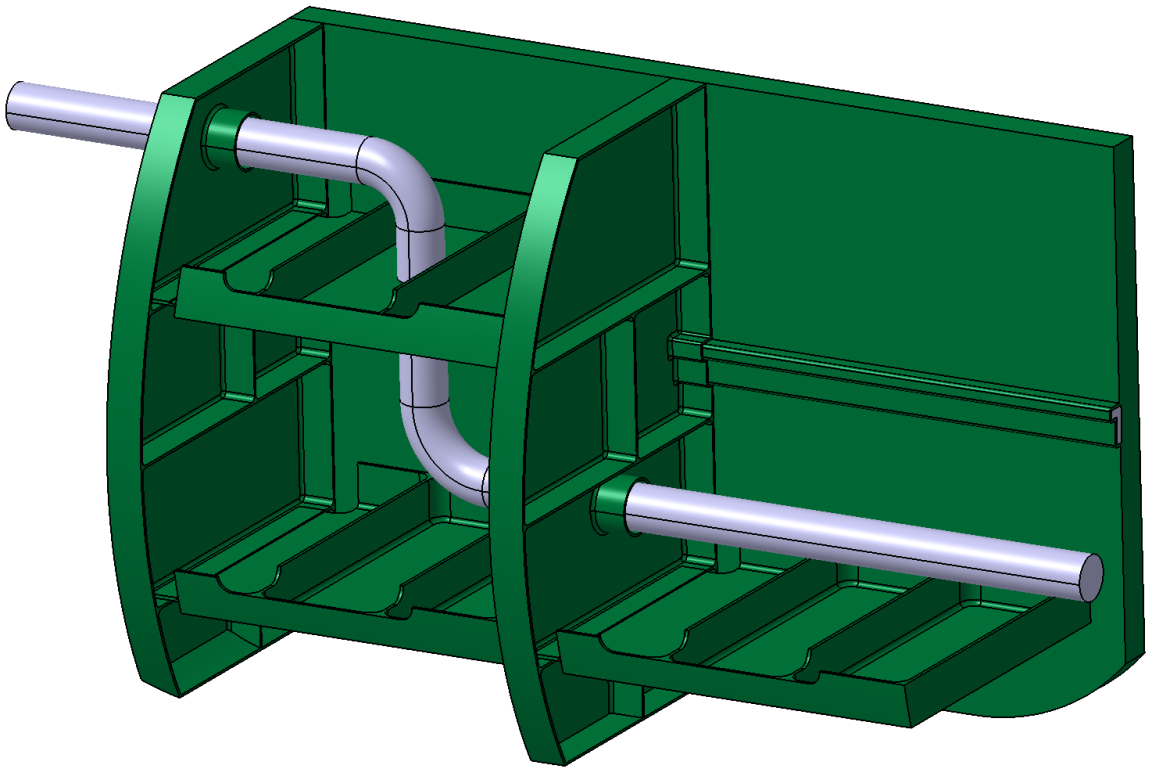
Note: All the external references are listed here with the published elements you started with at top and the solids and surface that you used for the push and cut operations.

Select *frame_lh_01_plane* and then select *Replace*. Notice that the window automatically closes and that you can't graphically select anything so you need to know the name of the element you want to switch to.

From the tree select *frame_lh_02_plane* within the *MDF* instance under *Publications*. The context window appears automatically. Notice that the *frame_lh_01_plane* now points to *frame_lh_02_plane* as show in the following picture.



Select *OK* and then select the Update icon.  The frame is moved to the middle and since the external references are kept it is automatically modified how you want it. The assembly should appear similar to the following picture. Notice that the fillets automatically located to the correct places.



Using the third mouse button, *Copy* the *Frame-001* instance.

Using the third mouse button, select on *Assembly_Context* and select *Paste Special*. You may need to copy the other frame again.

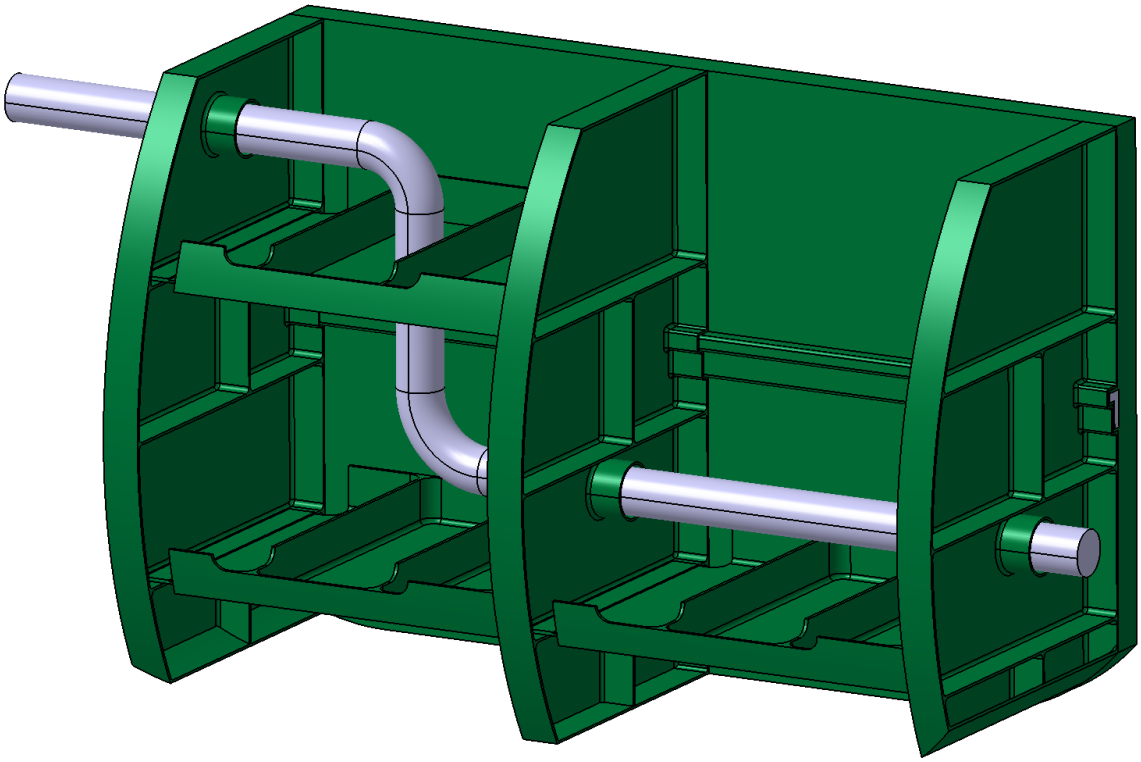
Select *Break Link* and select *OK*.

Change the *Part Number* to *Frame-003* and the *Instance name* to *Frame-003.1*.

Press the third mouse button while on the instance name of the new part, select *Components* and select *Define Contextual Links.*

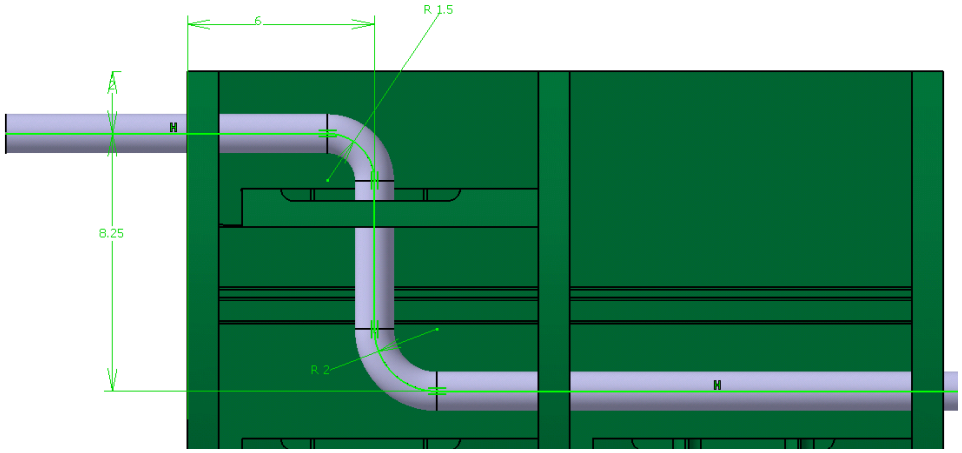
Replace the *frame_lh_01_plane* with *frame_lh_03_plane* and select *OK*. This is done in the same manner as you did it earlier.

Select the Update icon.  Now there should be three frames just as in the following picture.



Save the your document. Make sure you use save management to propagate the files.

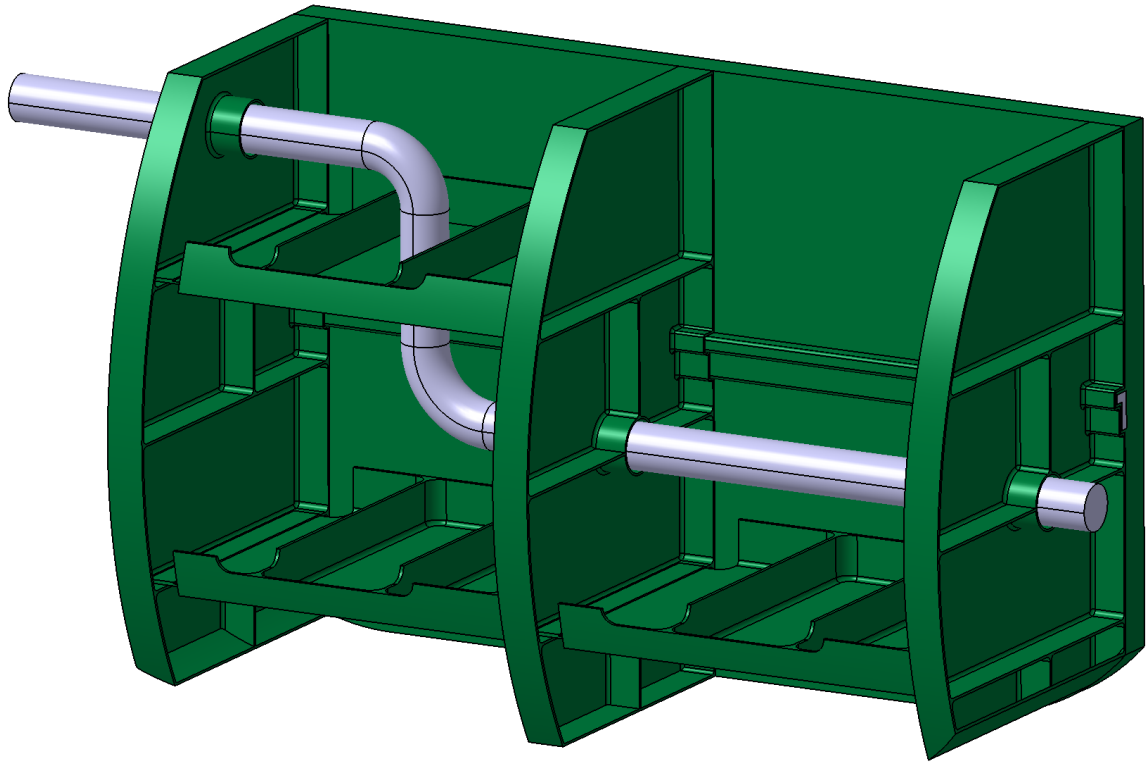
Open *Sketch.1* within the *Tube PartBody*. You are going to modify the position of the tube. You should see a sketch similar to the following picture.



Change the 8.25 measurement to 7.0. This will place the tube through the rib intersections on *Frame-002* and *Frame-003.*

Exit the sketch workbench, make the main assembly active and select the Update icon.

Due to how the fillets were created your frames updated with only the first frame's fillet not being necessary. If you would of used normal filleting procedures without using functional molded part you would of had errors when moving the tube's location.



Save and close your document. Make sure to use save management.

