

AIM Quick Start Manual

For New Design Engineers

Date March 27, 2020

Writers Jordan Lindstrom (jlindstrom@briskheat.com)
 William Lien Chin Wei (wwei@briskheat.com)

Version 2019-v2

Contents

1 Preparation for Physical Models	3
1.1 In SOLIDWORKS	3
1.2 In SpaceClaim	19
2 Start of Simulation	31
3 Workflow	36
3.1 Geometry	36
3.2 Mesh	36
3.3 Flow	43
3.3.1 Solver Options	44
3.3.2 Physics Regions	44
3.3.3 Material Assignments	45
3.3.4 Interface Conditions	46
3.3.5 Physics Options	48
3.3.6 Fluid Flow Conditions	49
3.3.7 Solid Thermal Conditions	51
3.4 Results	55
4 Appendix	57
4.1 Pressure Settings	57
4.2 Filter Settings	58
4.3 Expressions	60
4.4 Design of experiments	63

The resources and examples for ANSYS AIM are located at [P:Simulation Resources](#), as shown below.

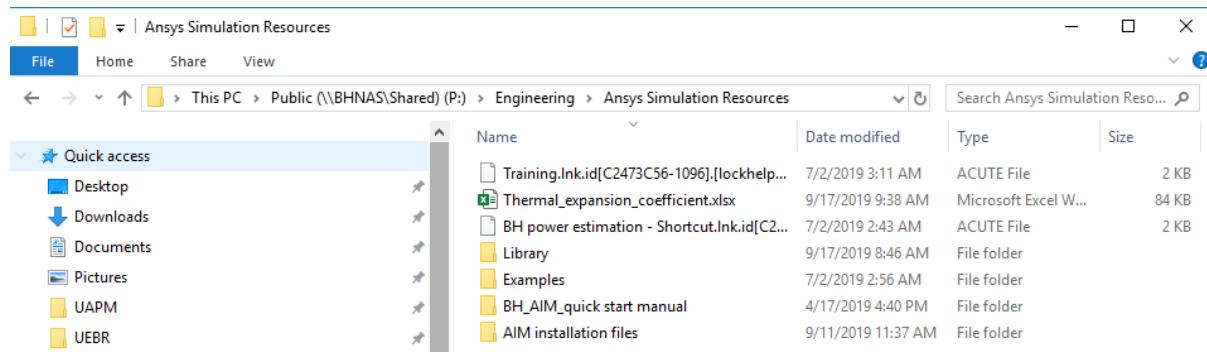


Figure 0.1:

1 Preparation for Physical Models

1.1 In SOLIDWORKS

To create the simulation model, we will be following the jacket construction method depicted on our drawing files, as shown below in Fig. 1.1. (The weldment has been added to the layer construction for this example: P/N – UAPM23065TSN-263.). This example can be found with a file name of “263 Flow Test Assembly.SLDASM” or “263 Flow Test Assembly.STP.”

The simulation model will be an assembly consisting of 4 components: The Weldment, Liner, Heating Element, and Insulation. (We do not model our facing.)

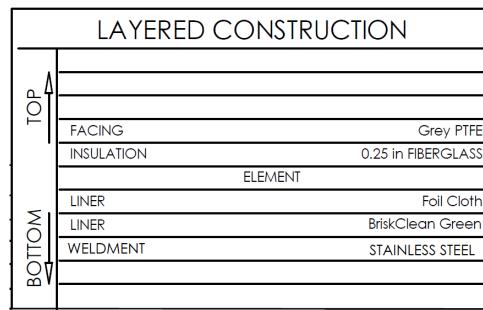


Figure 1.1:

1. Starting from the bottom and working our way up – the first thing we need to create is the Weldment, as shown in Fig. 1.2. This can be done like our patterning process where we go to the customer provided step file and replicate the part in SolidWorks. However, unlike the patterning process, we need to include the geometrical details within the part as well as the outside (Specifically, we are trying to capture the flow path/fluid region that is within the part we are simulating). If a pattern has already been created, you can copy/rename it and modify that file to fit your needs.

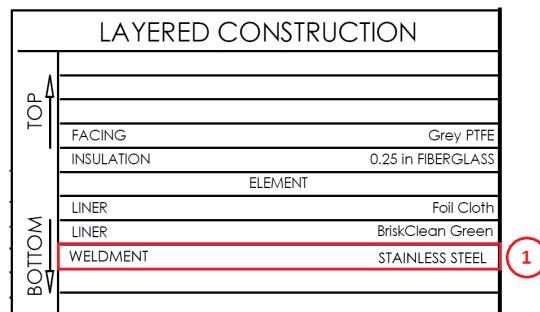


Figure 1.2:

Circled in red is the Weldment section for -263 we will be creating for our simulation model. (Fig. 1.3, in Figures 1.3(b)-(d) the fluid volume we are trying to capture is highlighted in blue.)

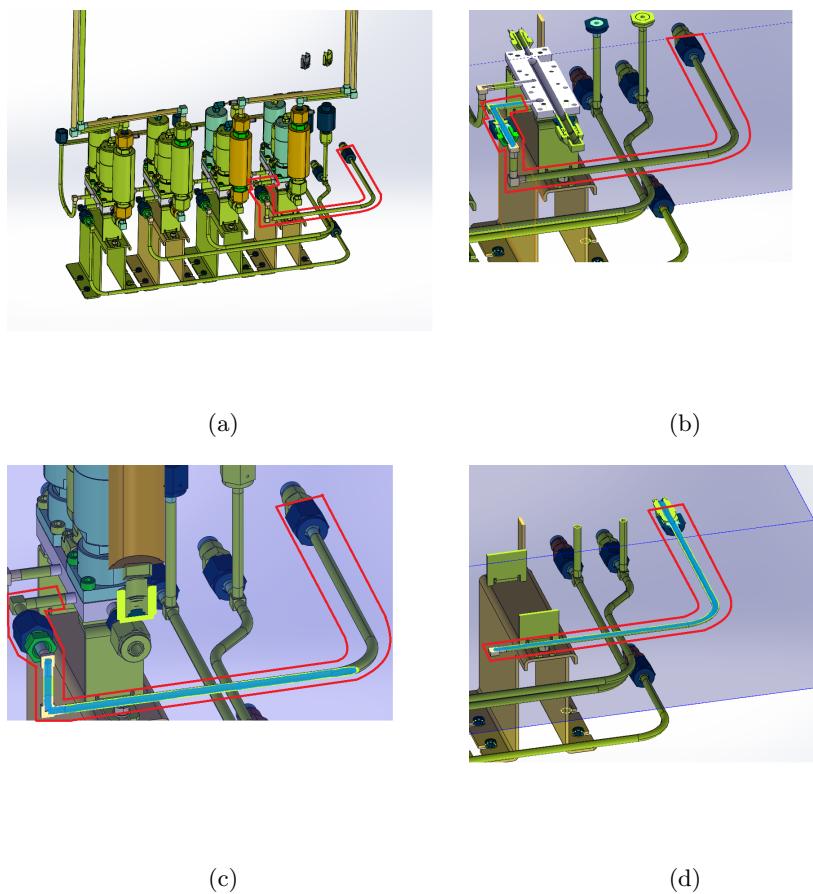


Figure 1.3:

After replicating the geometry and cutting out the appropriate flow path we have the below result in figures 1.4(a) & (b). Once again in figure 1.4(a) the flow/fluid region is highlighted in blue (this is just hollow space in the model and is only highlighted in blue for visual representation). Simplification of the Weldment is still acceptable where necessary – such as in the VCR regions for this case.

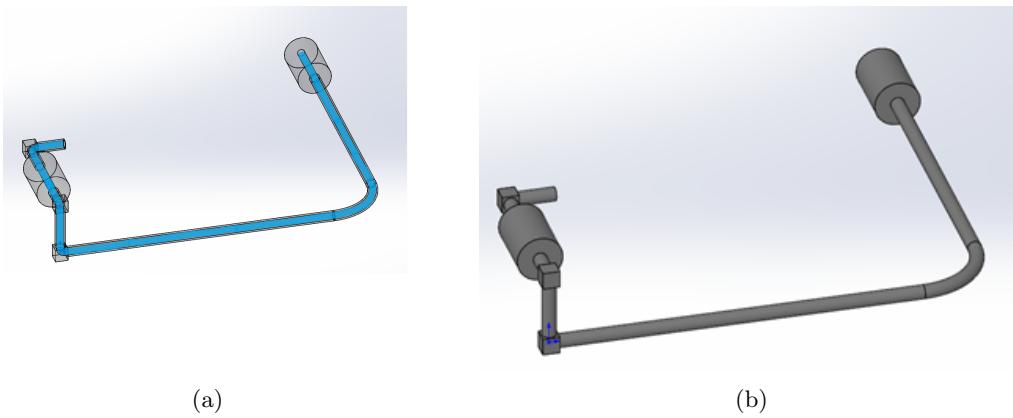


Figure 1.4:

- Once the Weldment is complete, we can move on to creating the next layer of the simulation model, the Liner. Regardless of how many liner layers are listed on the drawing we will only be creating one for them all. (Figure 1.5)

LAYERED CONSTRUCTION	
TOP	
BOTTOM	
FACING	Grey PTFE
INSULATION	0.25 in FIBERGLASS
ELEMENT	
LINER	Foil Cloth
LINER	BriskClean Green
WELDMENT	STAINLESS STEEL

Figure 1.5:

The Liner is to be modeled at 1 mm thick encompassing the weldment. This can be accomplished in two ways:

- 1.) The Shell Function
- 2.) The Move Face function coupled with the Cavity function

The easiest place to start for the Liner is to copy and rename you weldment file and delete/suppress any cutting/hollowing features you performed. Then from here you can –

Shell Method

Select the Shell function under the features tab and select all the open faces of the jacket as shown in Fig. 1.6. Next type in the desired thickness (1 mm) then hit the green check mark.

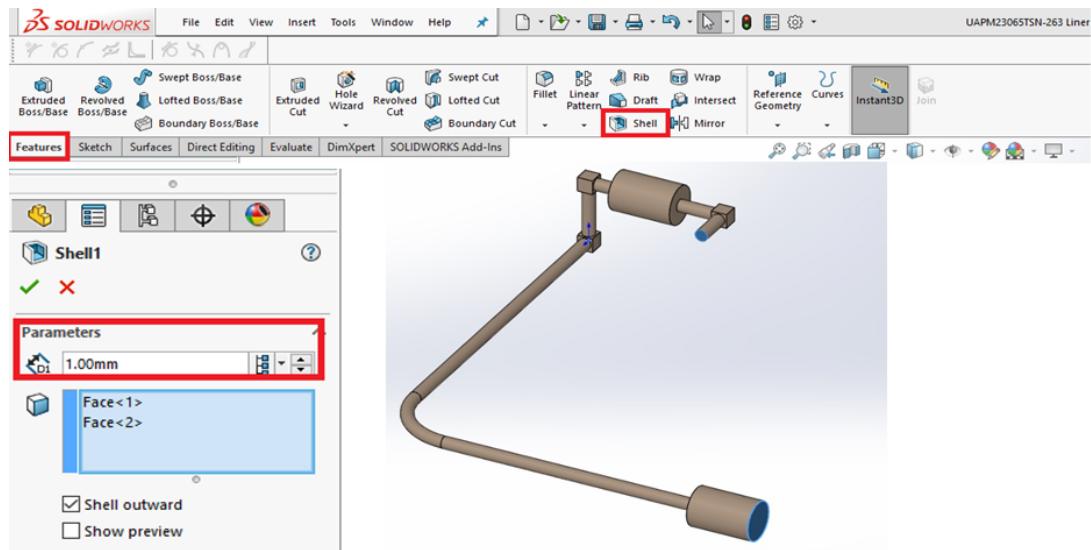


Figure 1.6:

Move Face Method

Under the Direct Editing tab (Direct Editing tab may need to be added) select the Move Face command as shown in Fig. 1.7. From here select all the faces except the open faces of the jacket. Next type in the desired thickness (1 mm) then hit the green check mark.

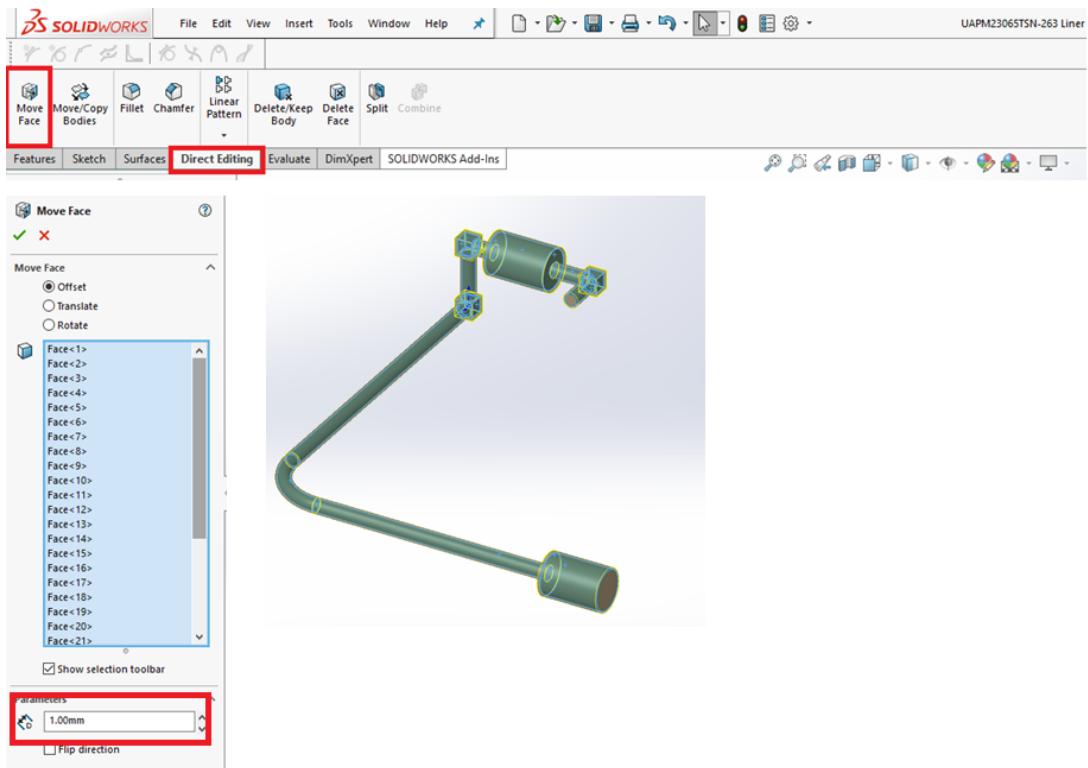


Figure 1.7:

After performing the Move Face command, the next step is utilizing the Cavity function. To use this, we must first make an assembly to start putting our jacket components into.

- Create an assembly for our simulation model and insert the Weldment file
- Drag in your Liner file and mate it to your weldment
- Right click your Liner component and select edit part
- Go to Insert → Molds → Cavity and select Cavity (Figure 1.8)

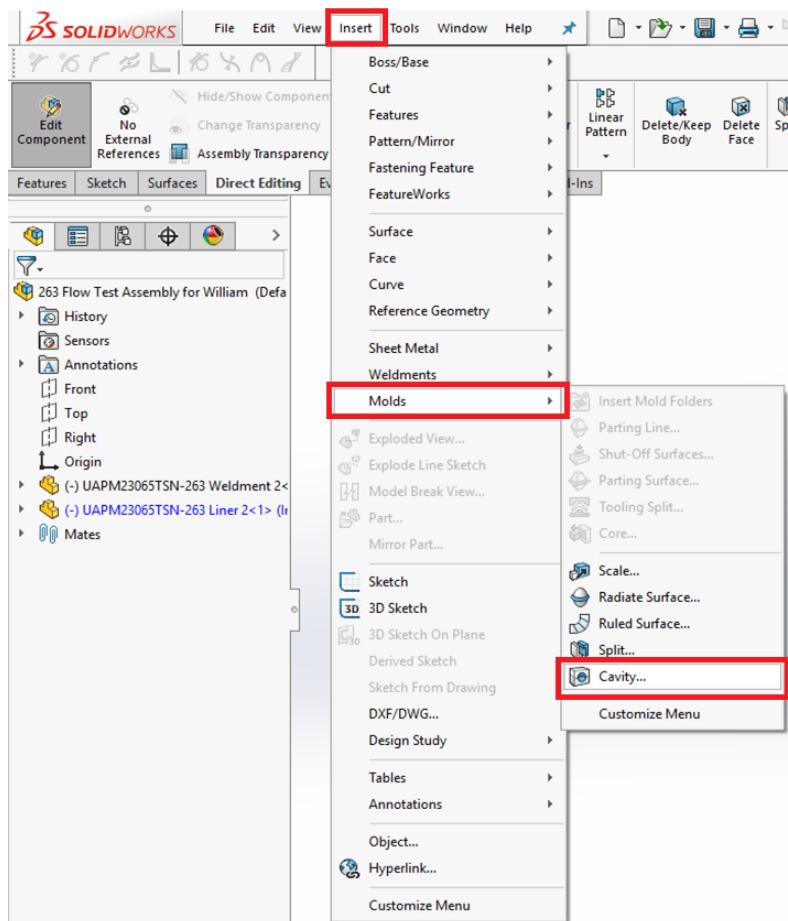


Figure 1.8:

We will be using the Weldment to make a Cavity in the Liner so that there are no interfering sections between them. A dialog box will pop-up asking what bodies we would like to keep. We only want the Liner and not any volumes inside it. So be sure to only select that body. (Figure 1.9)

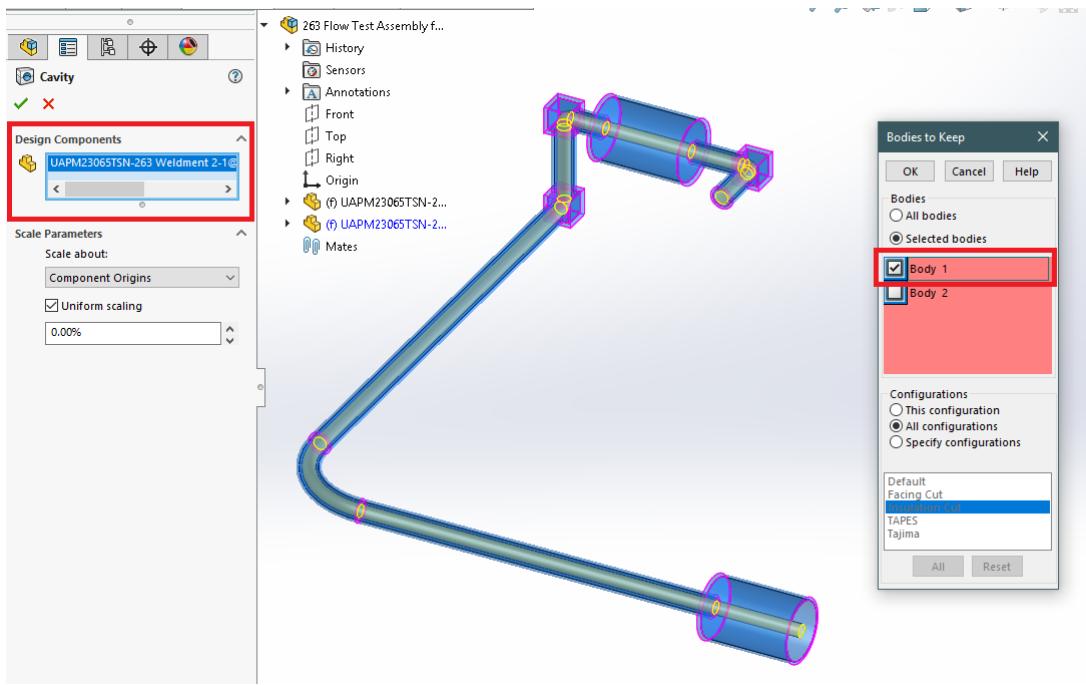


Figure 1.9:

The result for both methods should be what is displayed below in figure 1.10(a) (Liner only) and figure 1.10(b) (Liner and Weldment).

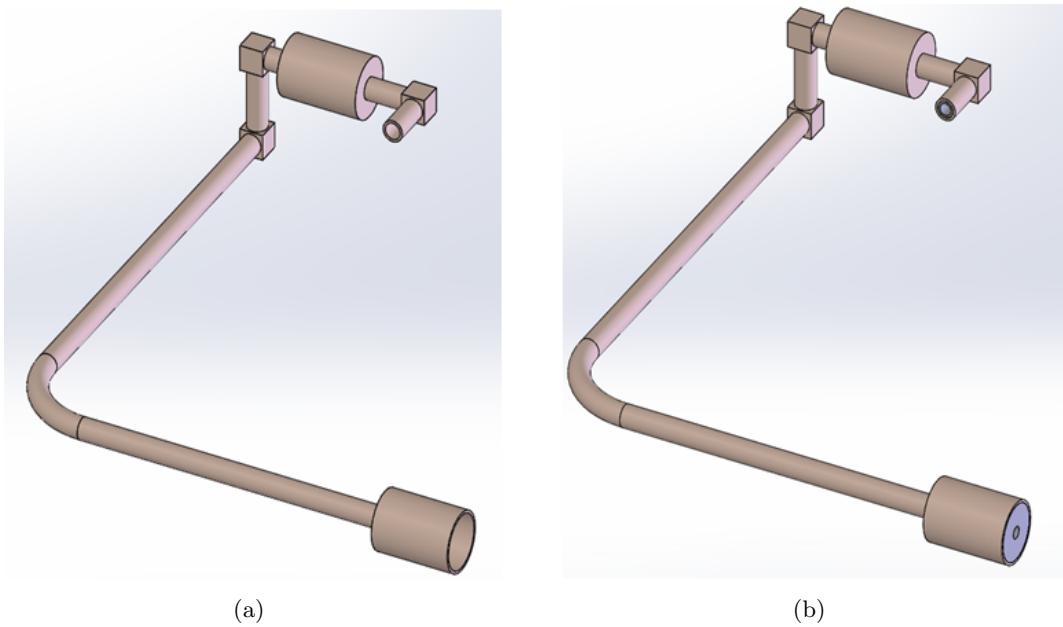


Figure 1.10:

3. Now that we have finished creating the Liner, we can move on to creating the Insulation layer (We will be skipping the Element layer for now). This layer can be created using the same functions shown when making the Liner. The quickest way to create the Insulation would be to copy your Liner file, rename it as your Insulation file, and edit it appropriately.

LAYERED CONSTRUCTION		
TOP		
FACING	Grey PTFE	
INSULATION	0.25 in FIBERGLASS	3
LINER	Foil Cloth	
LINER	BriskClean Green	
WELDMENT	STAINLESS STEEL	

Figure 1.11:

Shell Method

After making your Insulation file, go into the shell feature and edit your thickness to be 1mm + the thickness of your insulation as stated on your drawing – for this case it is 0.25" = 6.35 mm. Making our shell function thickness $1\text{ mm}+6.35\text{ mm} = 7.35\text{ mm}$ as shown in Fig. 1.12.

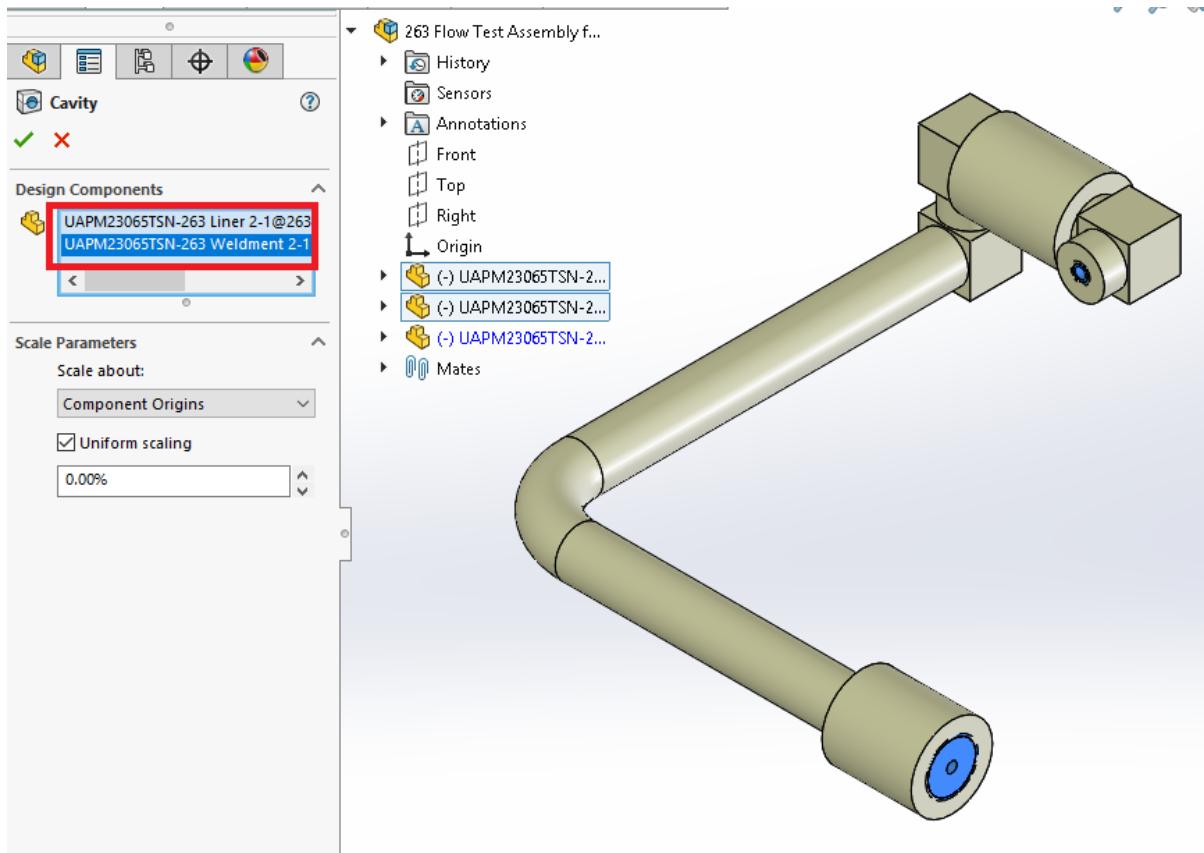


Figure 1.12:

Once you have done this, we will need to use the cavity function to cut out the regions that the Weldment and Liner intersect the Insulation.

Move Face Method

The process for the Move Face method is very similar. We will go into the Move Face command and change the distance from 1 mm to 7.35 mm as shown in Fig. 1.13. Then

once again using the Cavity function and selecting the Weldment and Liner we will cutout any intersecting areas. Remember when the keep bodies dialog box comes up, we only want to keep the Insulation not any internal bodies.

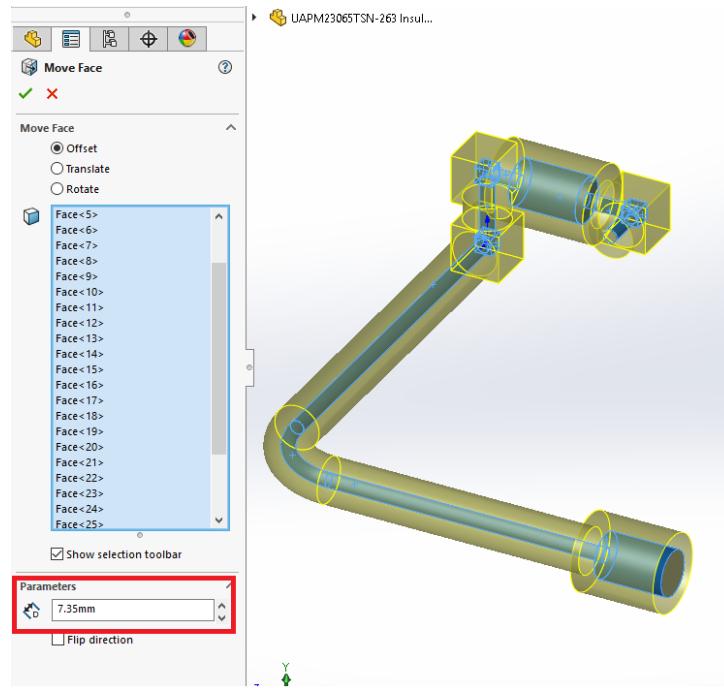


Figure 1.13:

For both methods the resulting part should look like the model in Figure 1.14 below.

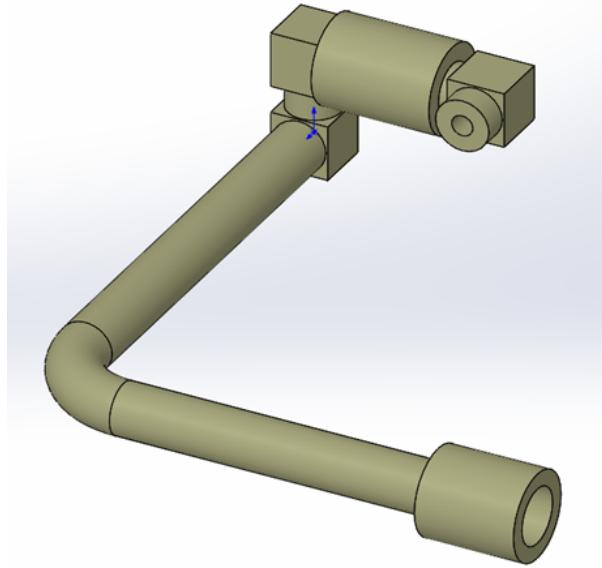


Figure 1.14:

The entire assembly now should resemble the model displayed in Figure 1.15.

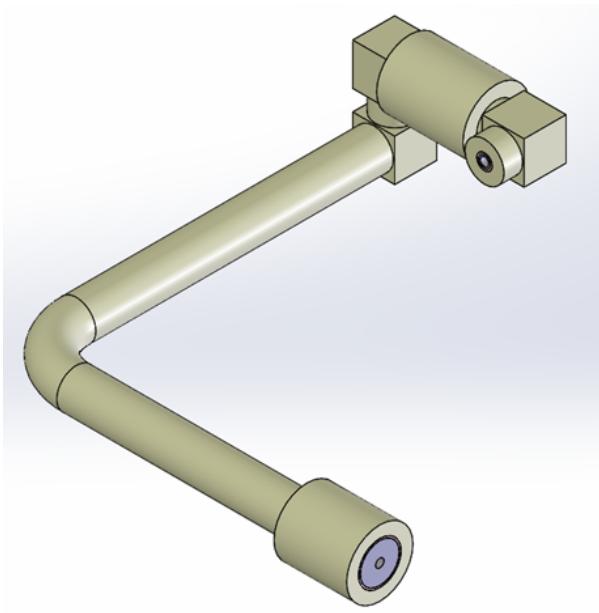


Figure 1.15:

- Now that we have the Weldment, Liner, and Insulation complete we can move on to the last component we will need for this simulation model, the Heating Element, as shown in Fig. 1.16. In this case that would be a wire layout not a tape layout. Since that is the case the first thing to do would be to fill out our High-Performance Excel Spreadsheet for this part so that we have a good idea of how much wire should be on the part (specifically trying to find a starting point for the VCRs).

LAYERED CONSTRUCTION	
TOP	
FACING	Grey PTFE
INSULATION	0.25 in FIBERGLASS
ELEMENT	
LINER	Foil Cloth
LINER	BriskClean Green
WELDMENT	STAINLESS STEEL

4

Figure 1.16:

Once that is done, we can move on to modeling the wire in SolidWorks. The first step will be to create a blank SolidWorks file and saving it as our Heating Element. From here we will drag it into our Assembly and mate the primary planes (Front, Top, Right) of our Heating Element to the primary planes of our Liner.

From here we need to create a series of references sketches based on the liner to draw our wire path along. You will most likely need to create a few planes to accomplish this as shown in Fig. 1.17.

Apply the same rules used when creating a tape layout when drawing these sketches (offsets from openings, spacing between wires, etc.). Each sketch is offset from the Liner a fixed distance of 0.766 mm.

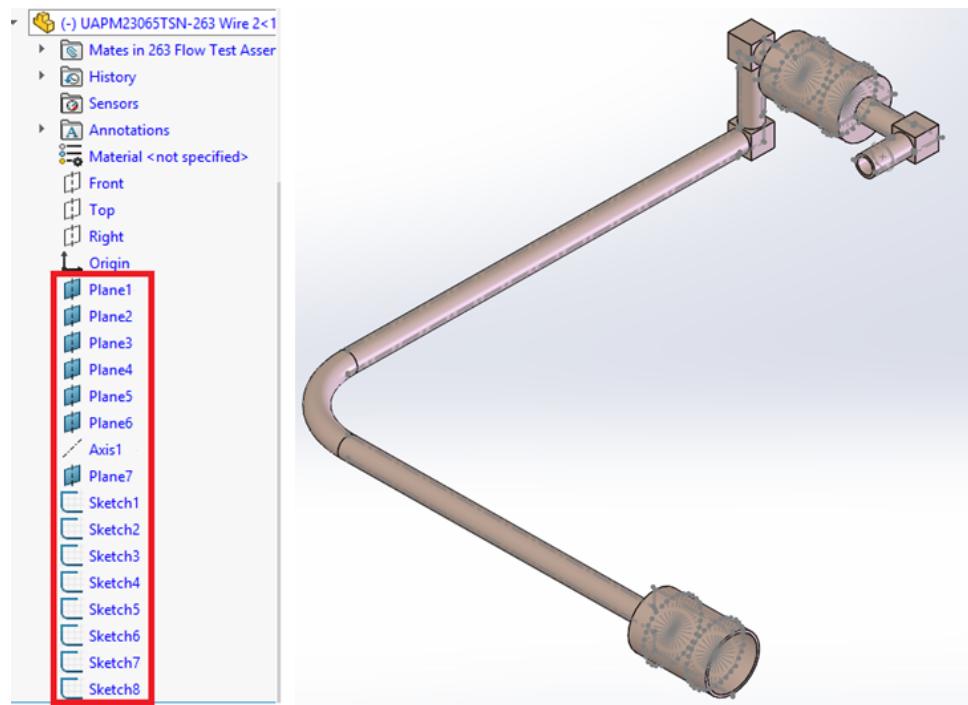
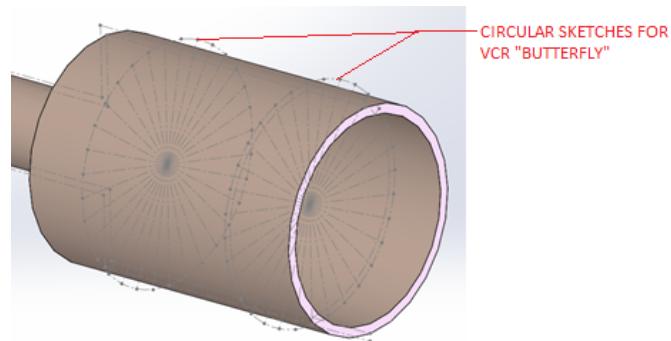
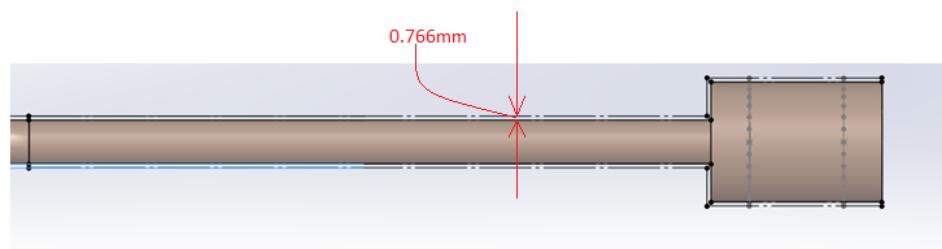


Figure 1.17:

Below in Fig.1.18 are some close ups of what the sketches would look like for various sections of our geometry.



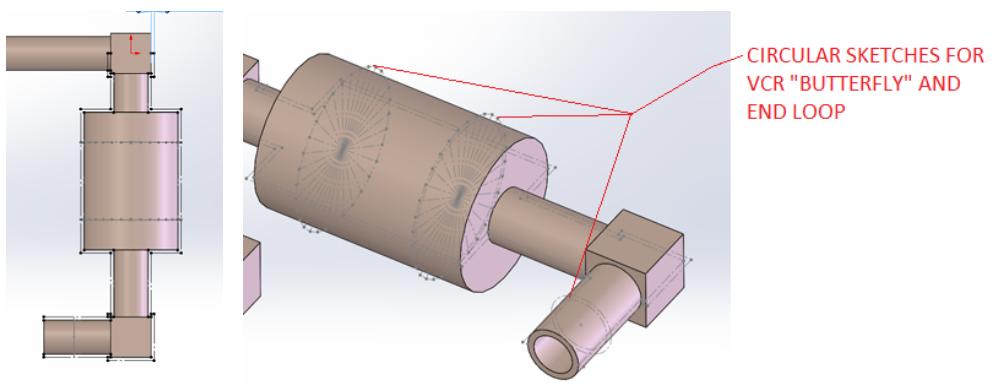
(a)



(b)



(c)



(d)

(e)

Figure 1.18:

After all the initial “guide” sketches have been made we will be creating a 3-D sketch as shown in Fig. 1.19 to connect them all together by tracing over them so that we can complete our wire path.



Figure 1.19:

The result should look something like what is displayed in Figure 1.20. Notice we are leaving a small gap at our start/stop point for our wire this can be modeled between $1/4''$ - $3/8''$.

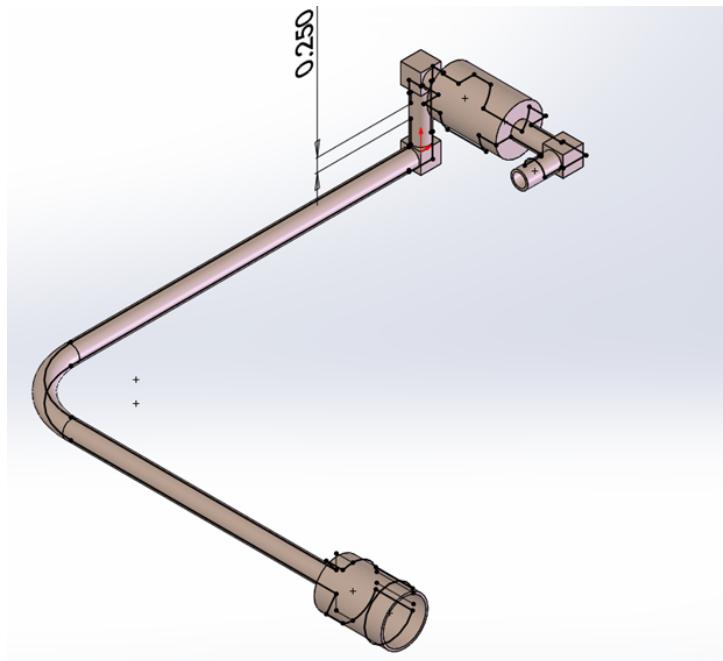


Figure 1.20:

Once our wire path is complete, we can use the Sweep Feature to create the desired 3-D wire model.

For our sweep we will be selecting our 3-D sketch as sketch path and be using a circular profile as shown in Fig. 1.21(a). The diameter we are modeling the wire is a $0.08''$. We also have the boxes for Merge tangent faces and show preview selected. The Merge tangent faces option seems to be more successful when producing the 3-D layouts but is not a necessity. The show preview selection is a quick check to see if the drawn sketch is feasible – if the preview is not rendering then the sketch path may have some issues. Assuming that is not the case we should have a nice preview like the image shown in Fig. 1.21(b).

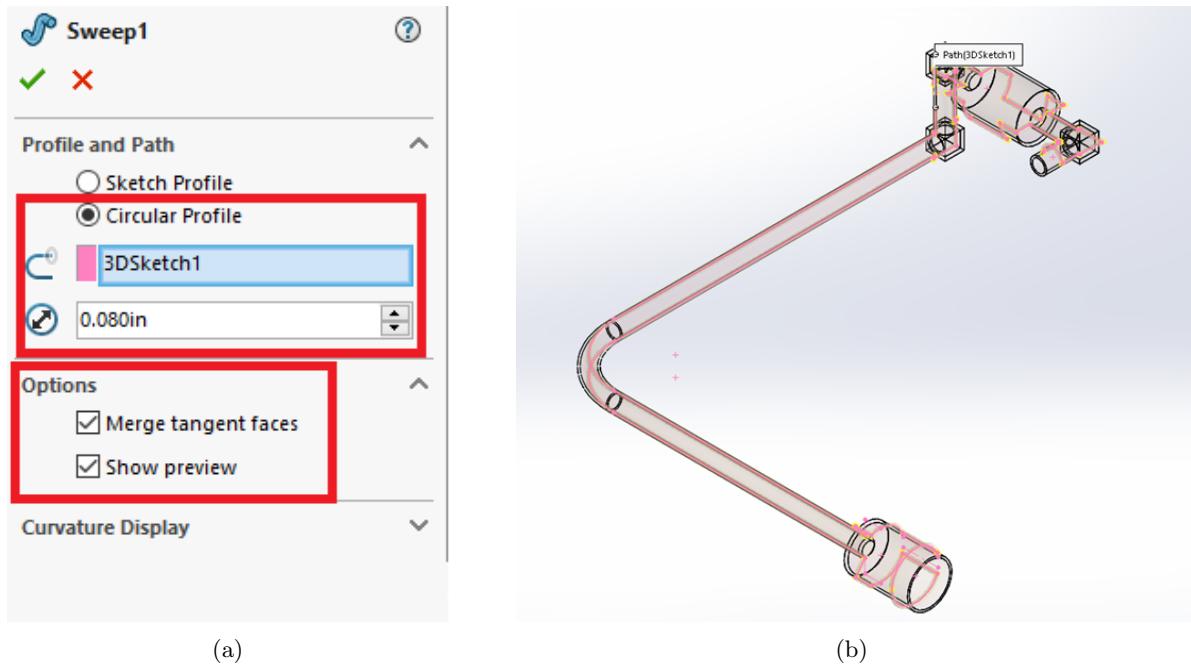
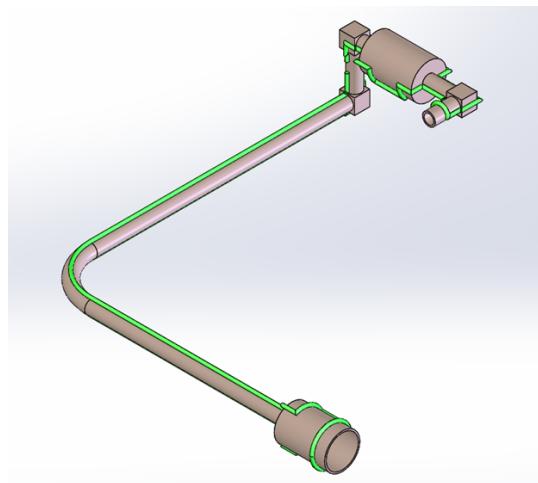


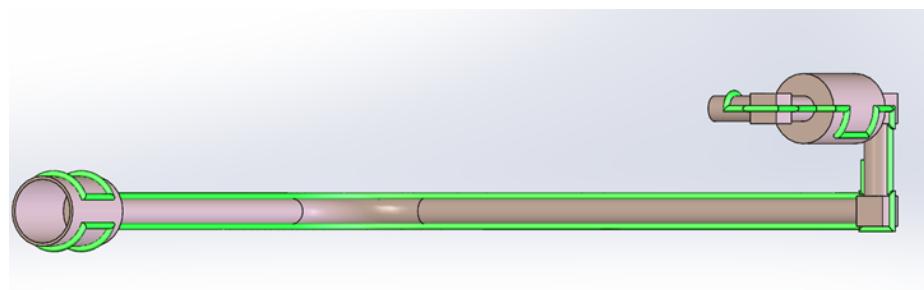
Figure 1.21:

Once we hit the green checkmark our 3-D wire model should be completed, and we should be done with the creation of our jacket simulation model in SolidWorks!

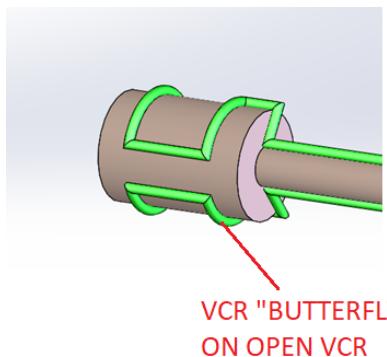
Images of the complete wire are shown below in Figure 1.22



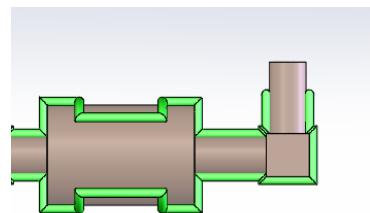
(a)



(b)



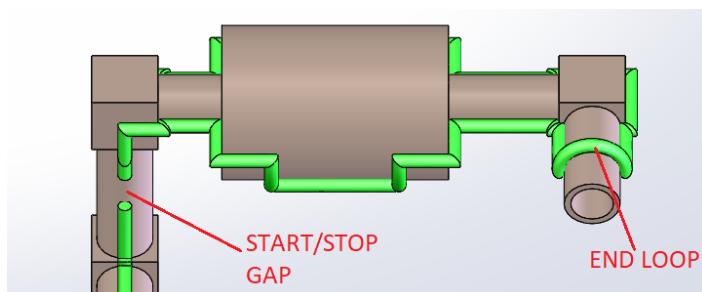
VCR "BUTTERFLY"
ON OPEN VCR



VCR "BUTTERFLY" MID-
JACKET VCR

(c)

(d)



(e)

Figure 1.22:

With the wire now complete your final jacket assembly file should resemble the below

image shown in Fig. 1.23. (The insulation has been made transparent to show the wire layout beneath.)

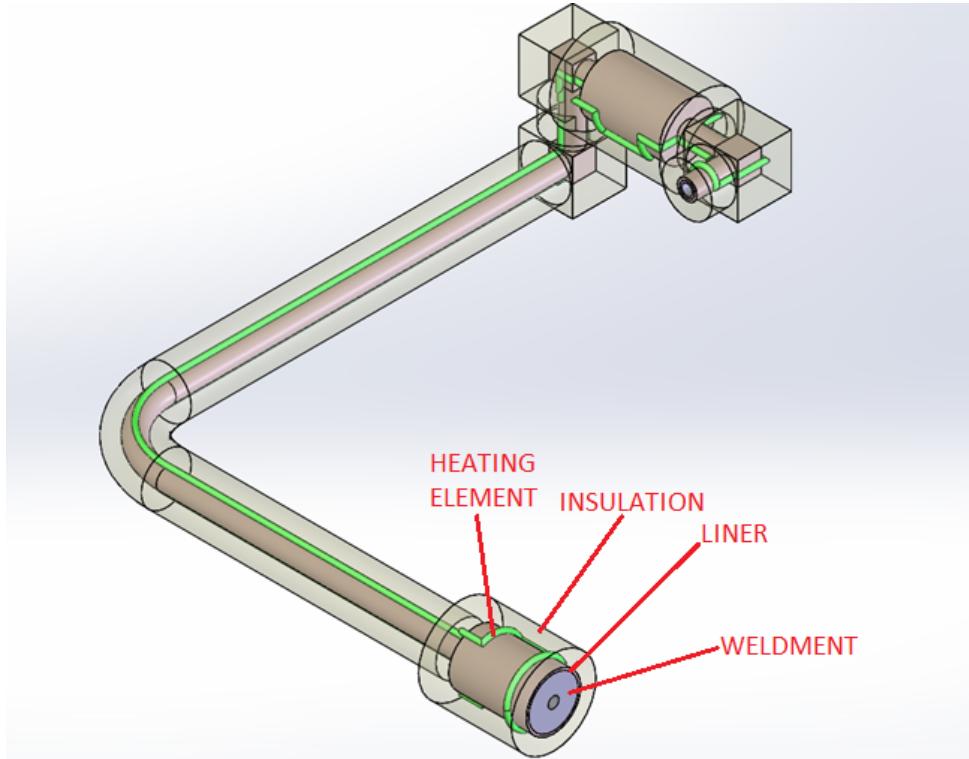


Figure 1.23:

5. Because this model was prepared to help compare our simulation to our lab results, we will need to add an extra component to make sure that our simulation model accurately represents what was tested in the lab. As shown in Fig. 1.24, the additional weldment section circled in red was attached in our lab testing so this needs to be modeled as well to ensure that our results are valid when simulated. To do so simply create the weldment model by either measuring the physical weldment used or using an available step file. After this is completed drop the new component into your assembly and mate it to your jacket weldment section in the proper orientation.

Once this is completed, we can save the assembly as a step file. Saving it as a STEP AP214 step file as shown in Fig. 1.25 is preferred as it captures more details than the STEP AP203. With our step file created we can move on to the model processing in SpaceClaim to make our file simulation ready. (Depending on your version of SpaceClaim and ANSYS you may be able to simply drag and drop your SolidWorks assembly file into the software for processing. However, a step file will always work independent of software version.)

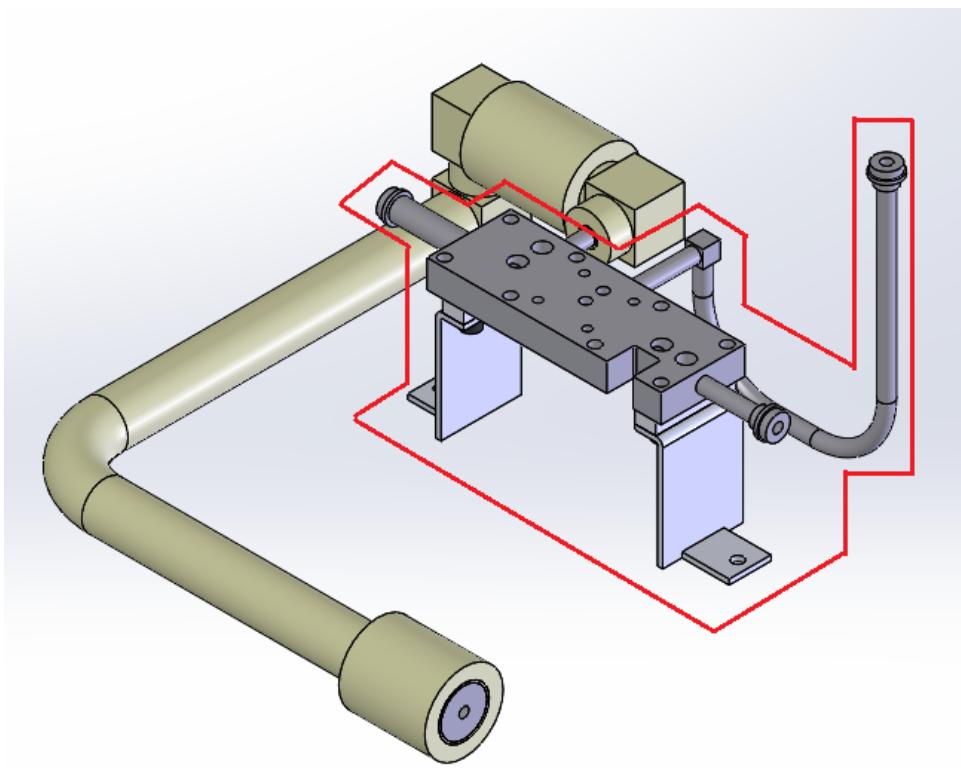


Figure 1.24:

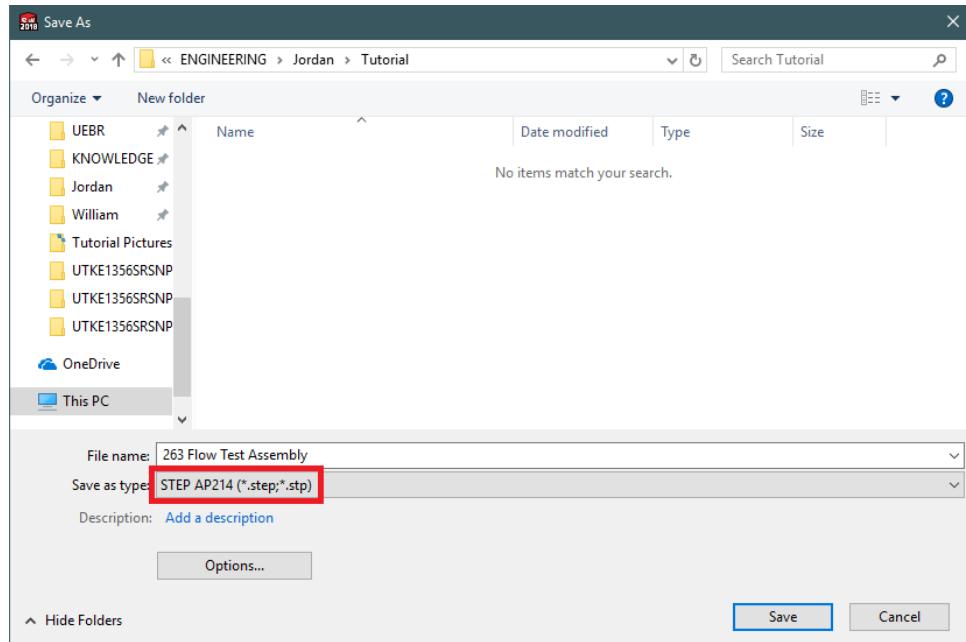


Figure 1.25:

1.2 In SpaceClaim

With all the SolidWorks modeling complete we can move on to processing our model in SpaceClaim. After opening SpaceClaim drag in the step file we created of our simulation model. This example can be found with a file name of “Example_SC.scdoc.”

There are four main steps when processing our model in SpaceClaim before it is simulation ready –

1. Extract Fluid Volume(s)
2. Imprint Heating Tape on Liner OR Imprint Liner on Heating Wire AND Cut out groove for Heating Wire from Liner
3. Remove Insulation Material that is overlapping/intersecting our Heating Element
4. Imprint with all parts (except heating element) as a check that all edges needed for simulation are captured

**Quick Tip – to exit a function double click in the white space surrounding your model

1. In this stage we will be extracting the fluid volume that flows within the weldment. To do this start by hiding all other components beside that of the weldment(s) as shown in Figure 1.26.

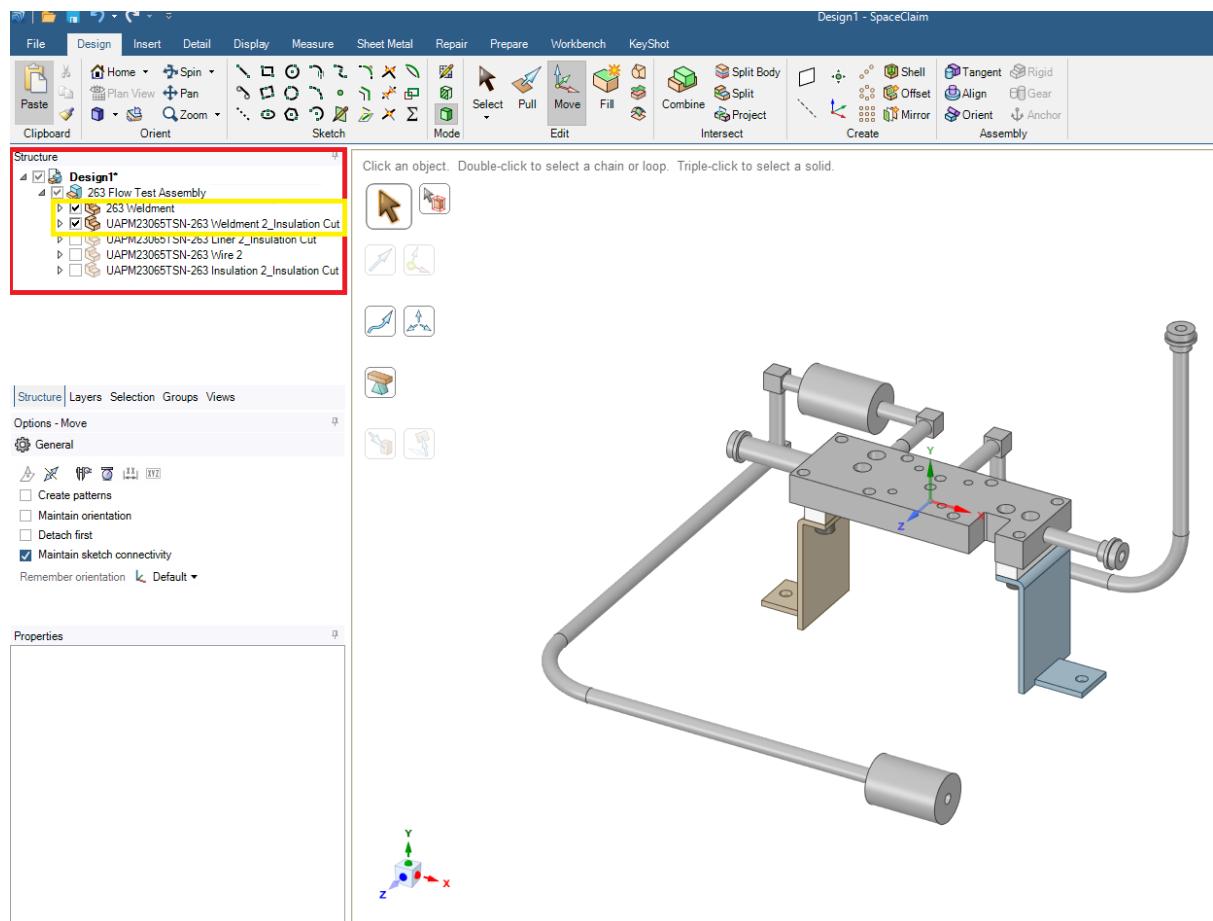


Figure 1.26:

Once this is complete, we want to go to the Prepare Tab and select the Volume Extract Function (Figure 1.27). By selecting the edges of the weldment openings where the fluid(s) begin/end the software will create a fluid volume to fill the space in between. For our case there are 10 edges we need to select which will result in 4 fluid volumes being extracted (Figure 1.27-1.29). Each fluid volume needs to be created separately – meaning its individual beginning/end edge needs to be selected then the check mark pressed to create that volume (so this process will be repeated 4 times.)

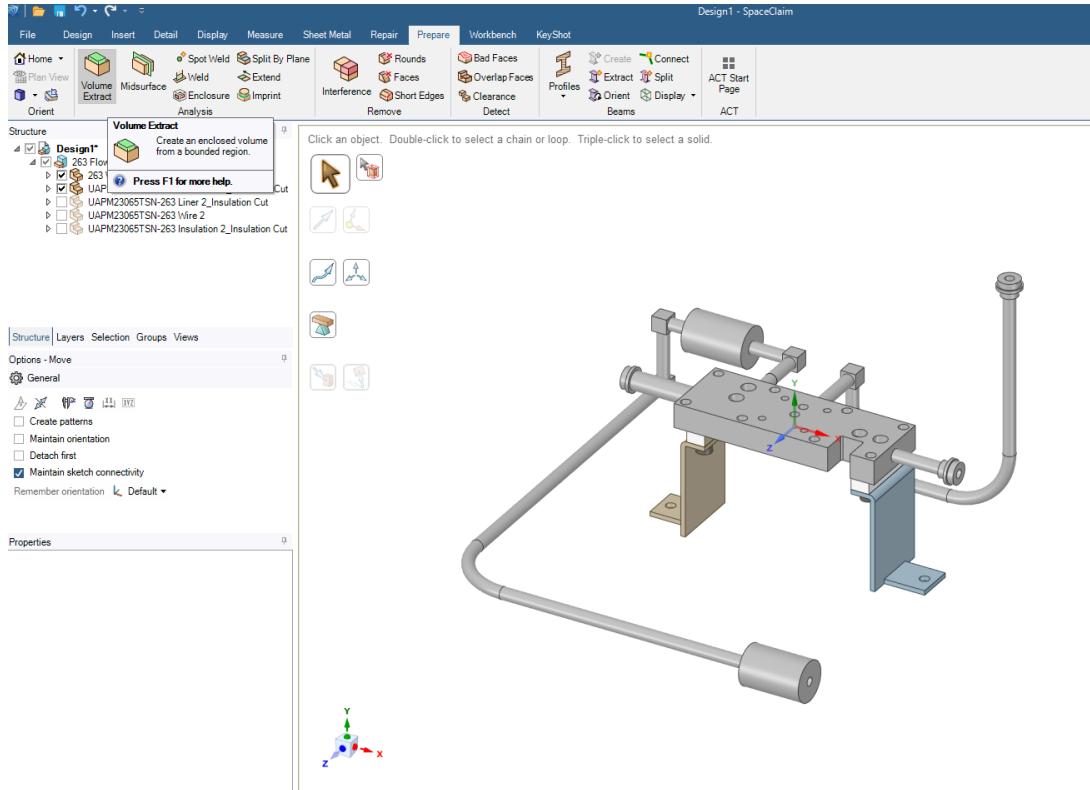


Figure 1.27:

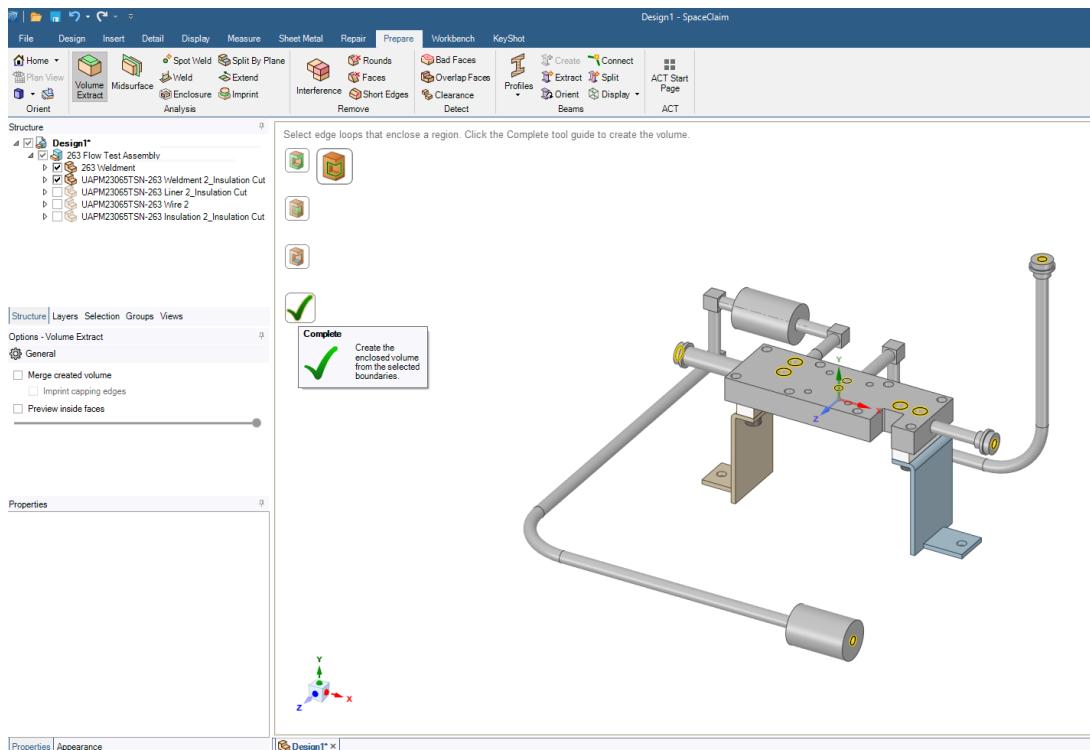


Figure 1.28:

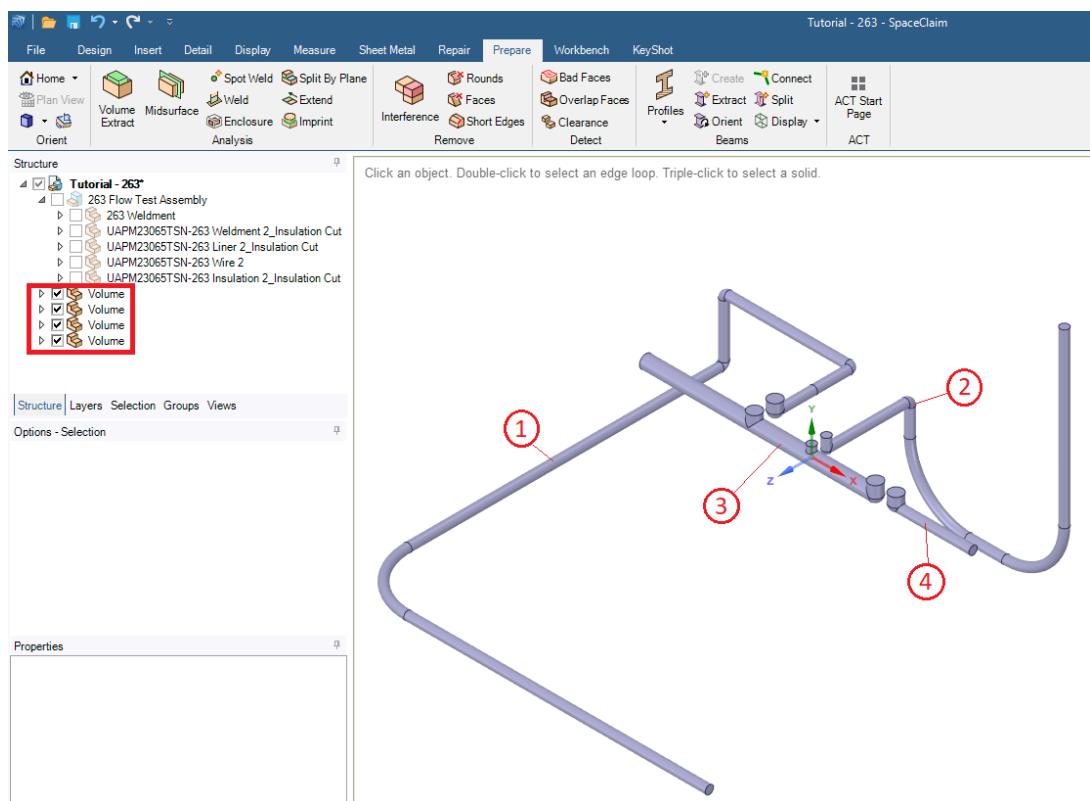


Figure 1.29:

- With our fluid volumes extracted we can move our focus to our Heating Element & Liner. In this case we are using Heating Wire so we will be using a two-step method –

- (i) Imprint the Liner edge onto our Heating Wire
- (ii) Create a groove in our Liner where the Heating Wire intersects it

**Something to remember when processing models in SpaceClaim is that it is best to hide all the other bodies you are not trying to modify when performing an action.

- (i) For this step since we are looking at the Liner and Heating Wire, we will be hiding all other bodies to make sure any unwanted operations/modifications are not performed. For a Heating Wire Layout, we will be using the Combine Tool under the Design Tab - to imprint the Liner onto our Heating Wire as shown in Figure 1.30.

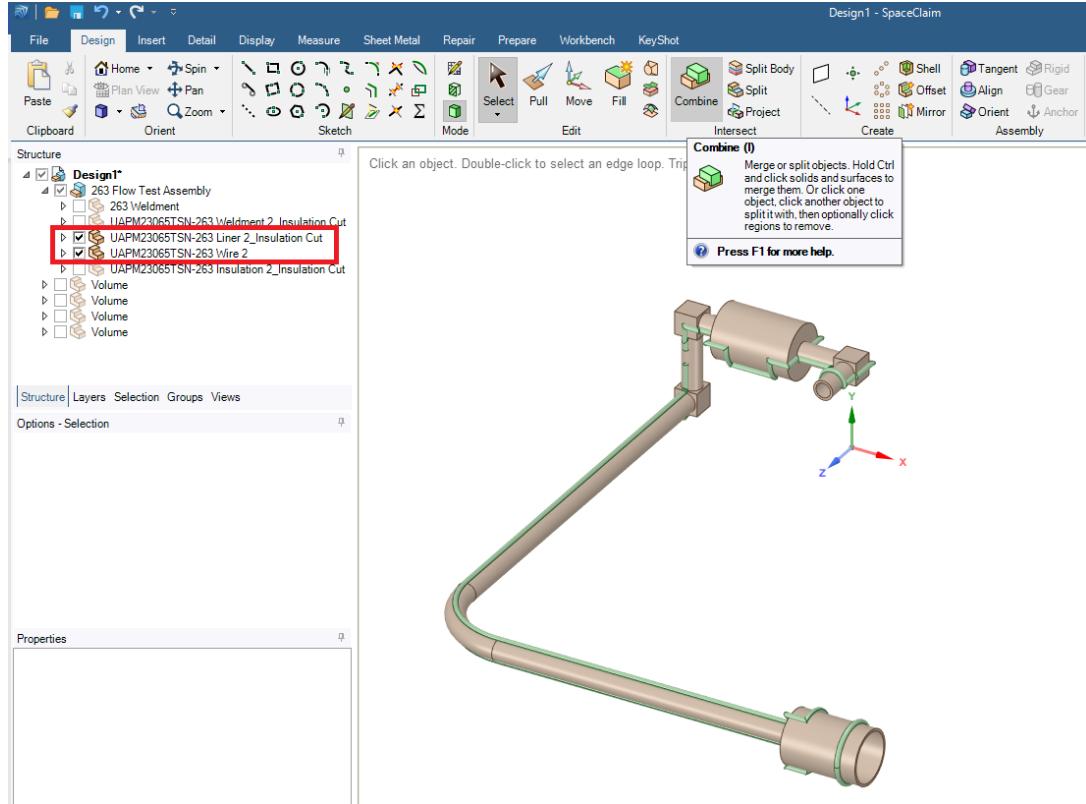


Figure 1.30:

Under the Options on the left of the screen under the feature tree, we want to make sure we have the “Make Curves – Imprint as edges” option selected (Figure 1.31). Once that option has been checked we will proceed with first selecting the Wire and then the Liner. This will result in imprinted edges being shown on the wire like in Figures 32(a) and 32(b).

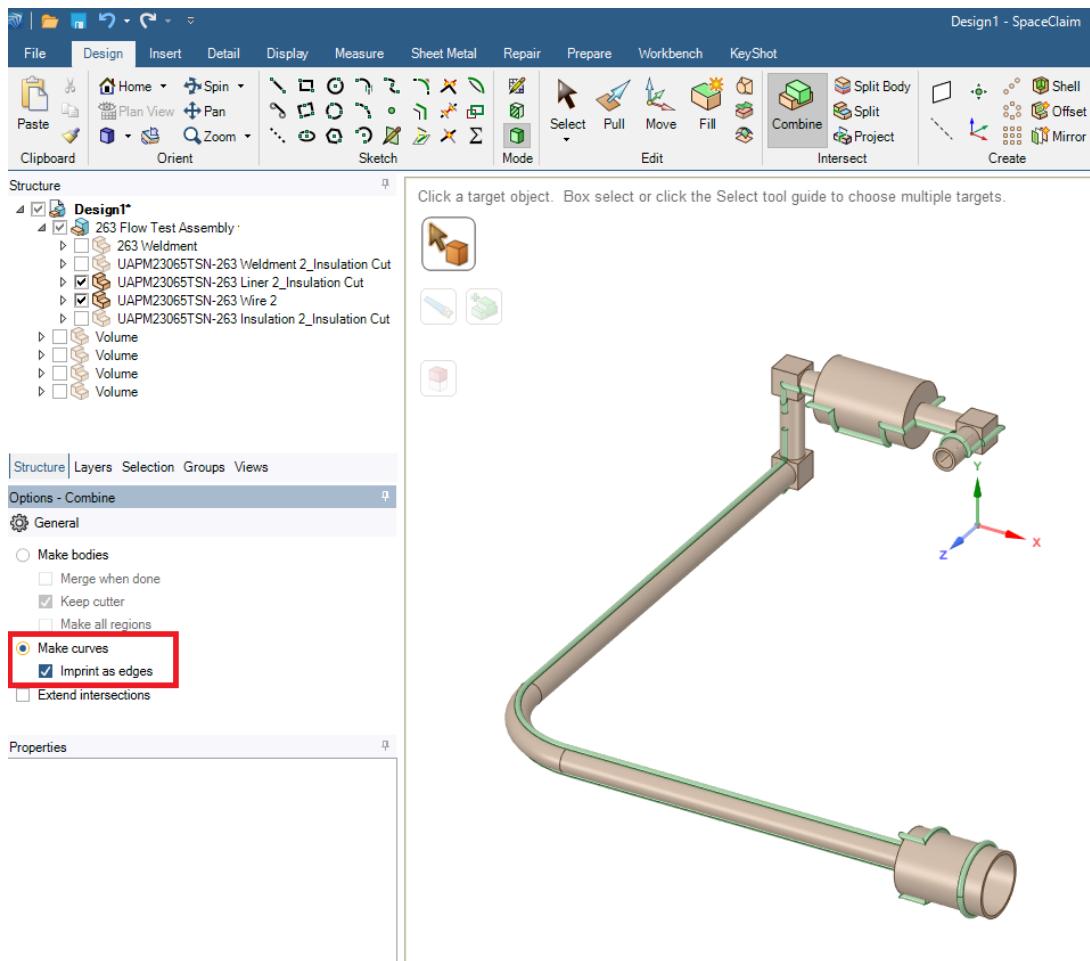


Figure 1.31:

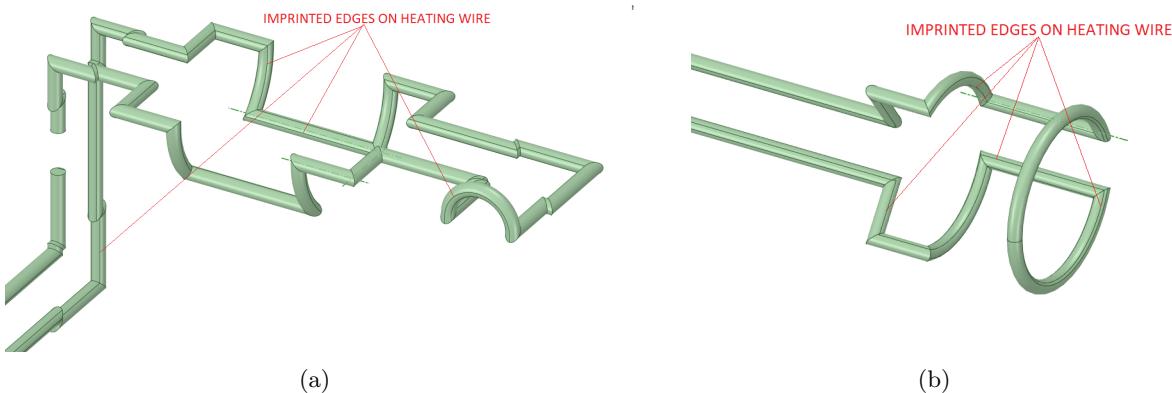


Figure 1.32:

- (ii) With the edges now imprinted we need to create a “groove” or “channel” in the liner that the Wire will lay in/contact our liner with. This can be done by using the Interference Tool under the Prepare Tab (Figure 1.33) to remove the areas where the Liner intersects our Wire. After selecting this function, the area should be highlighted in red where the Wire/Liner intersect as shown in Figure 1.34.

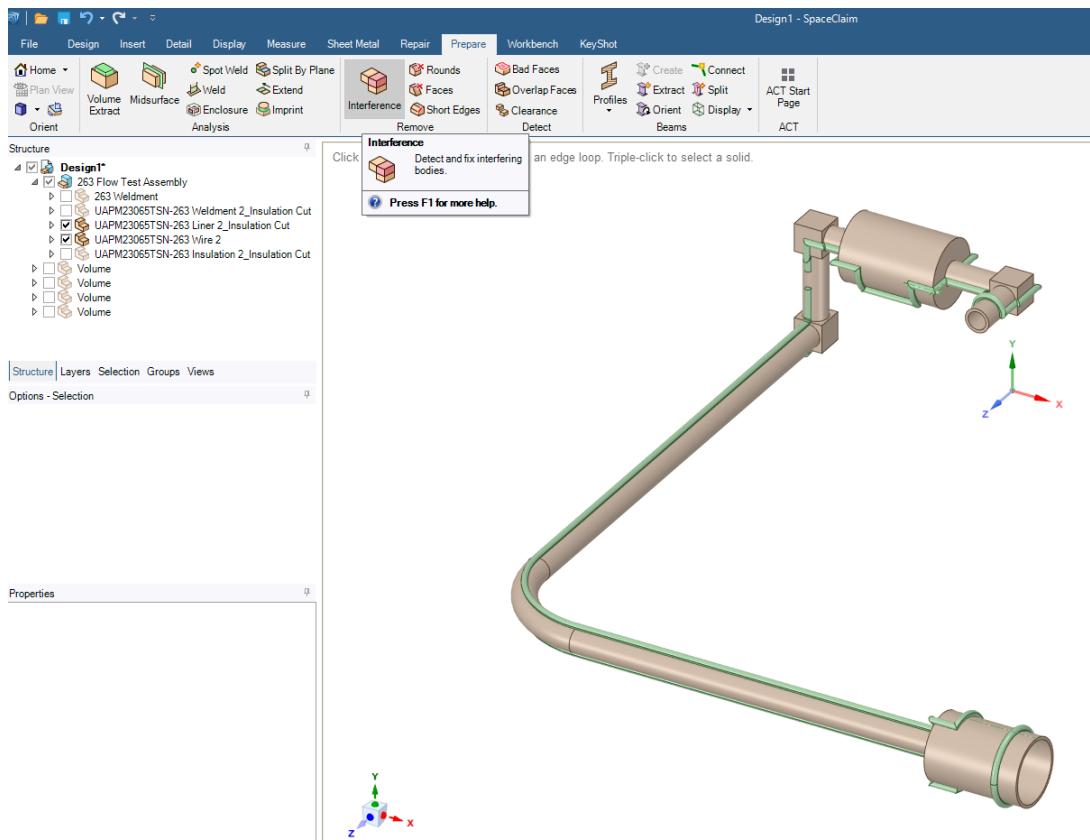


Figure 1.33:

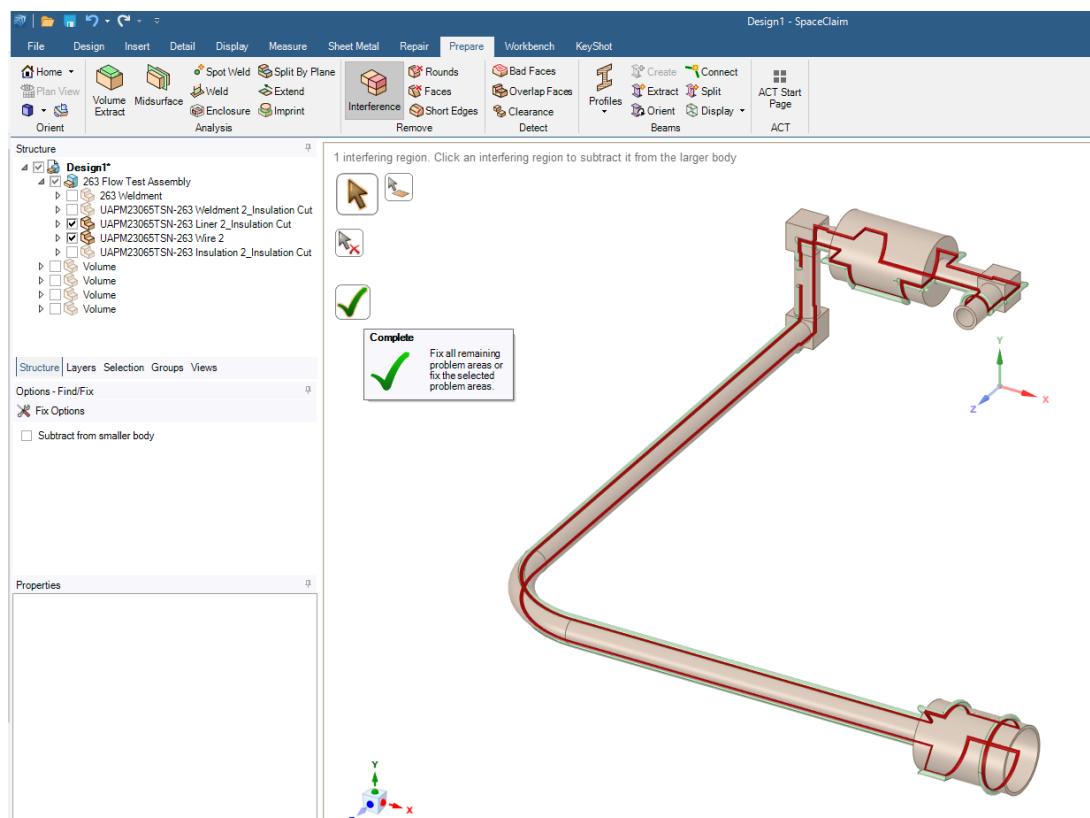


Figure 1.34:

After pressing the green check mark the red highlighted area should be subtracted from the Liner creating the groove/channel desired (depicted in Figures 1.35 and 1.36).

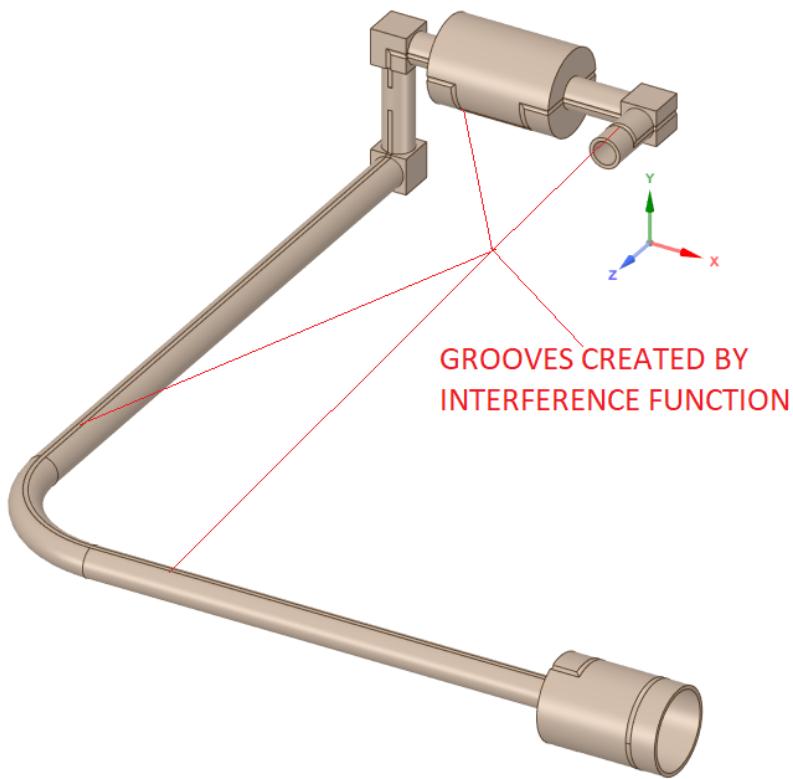


Figure 1.35:

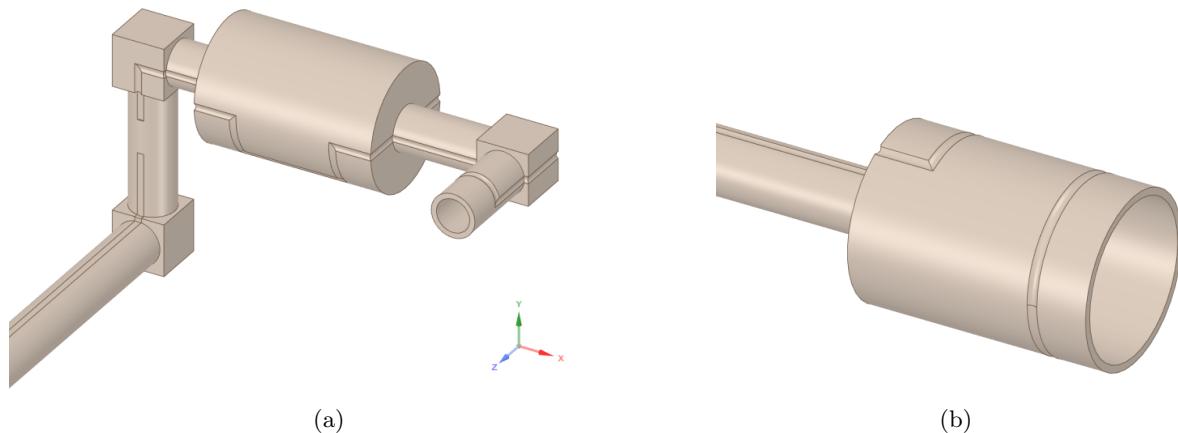


Figure 1.36:

3. Our Liner is now ready to go leaving just the Insulation to be modified before we should be simulation-ready. We will accomplish this by once again using the Interference Tool to subtract the area from the Insulation that is intersecting with the wire. (Remember to only have the Insulation and Wire shown when performing this operation.) Like before the area we are removing should be highlighted in red (Figure 1.37). Once the check mark has been selected the intersecting area will be subtracting from the Insulation resulting

in a cavity/channel for the wire as shown in Figure 1.38.

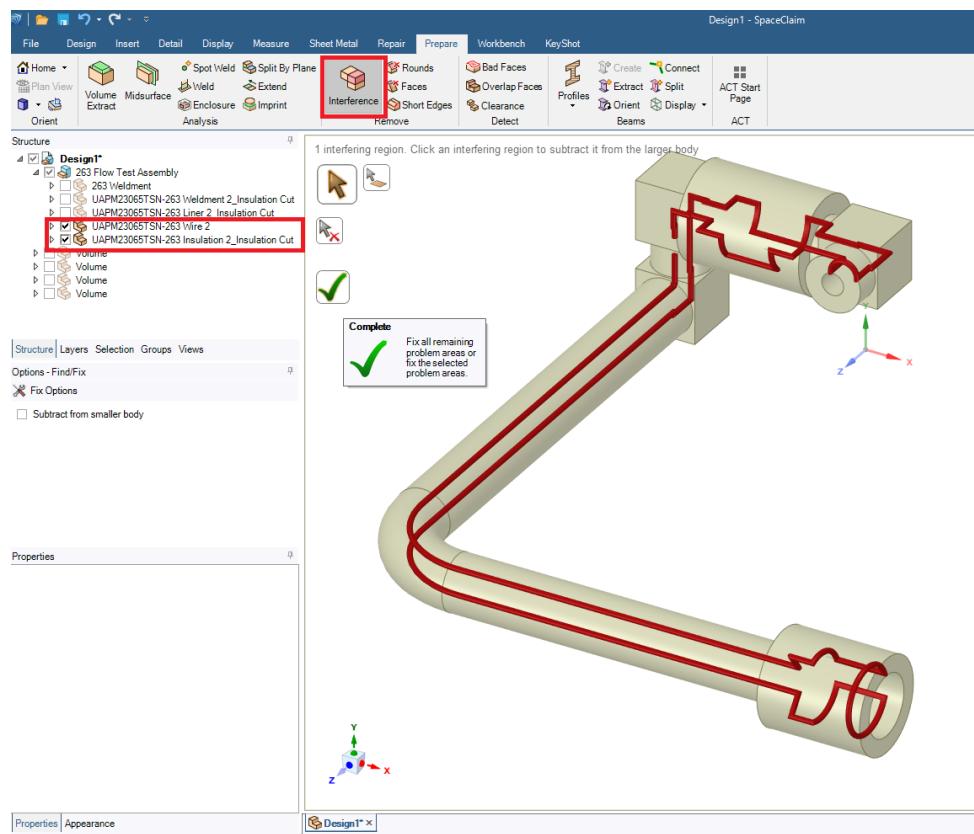


Figure 1.37:

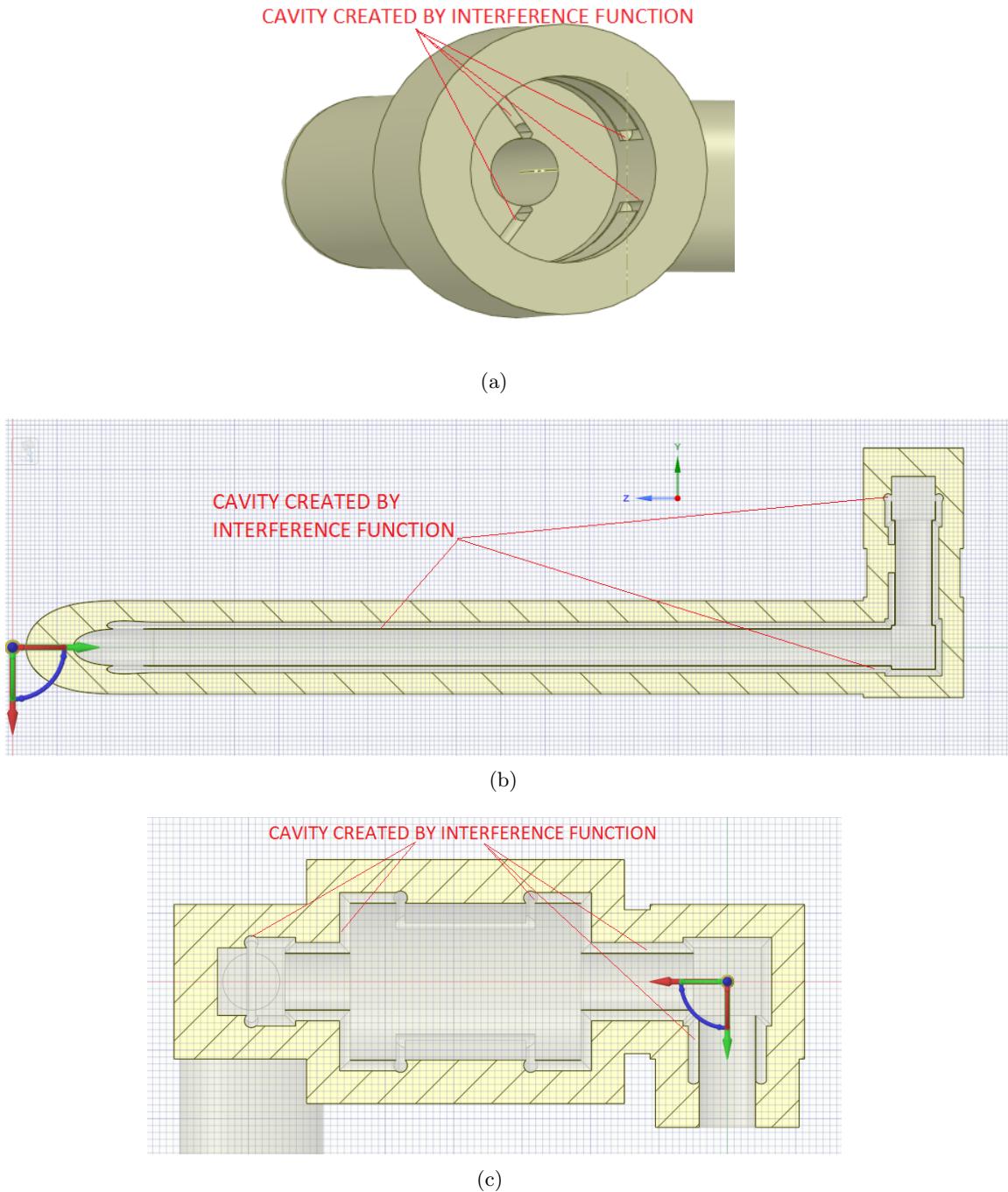


Figure 1.38:

4. The final stage of any simulation model preparation is to use the Imprint function one last time with all components (except the Heating Element shown in Fig. 1.39) to make sure that all necessary edges are captured. This is a very important step because in the simulation software everything needs to be very strictly defined and if an edge was not imprinted then it could cause an error, make defining that region difficult if not impossible, or even produce inaccurate results.

**The reason for hiding the Heating Element before this step is because there are usually some small edges that have “point” imprints which do not need to be altered and in some cases can cause errors/issues running through the simulation software.

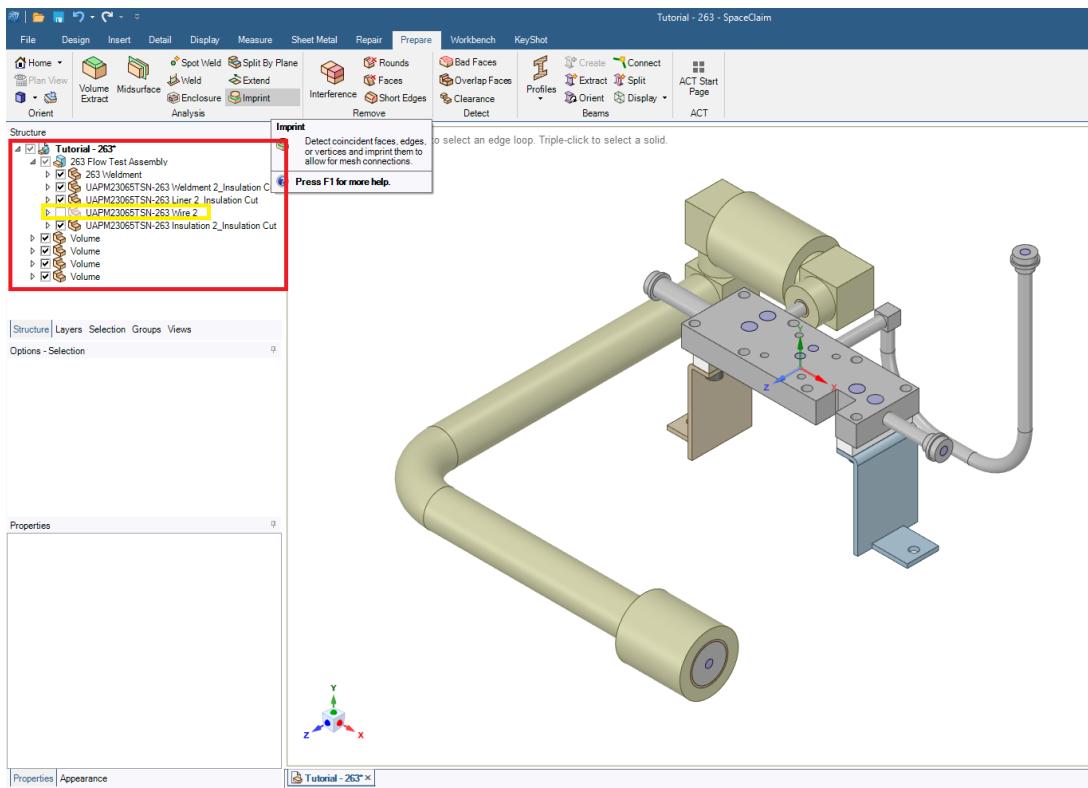


Figure 1.39:

After hiding the Heating Element simply select the Imprint function under the Prepare tab. This will then show all the remaining edges that need to be imprinted by highlighting them in red (Figure 1.40) – press the green check mark and then those edges should be imprinted on their respective components.

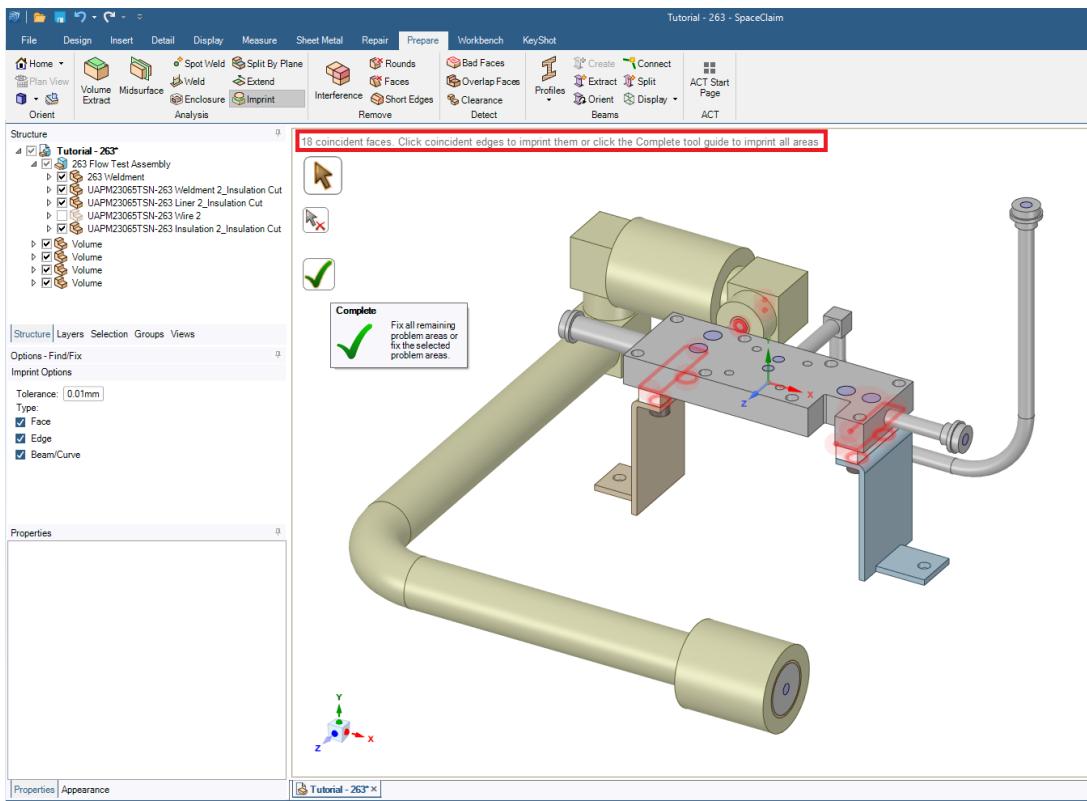


Figure 1.40:

With this final step complete we can show all our components and should have a completed simulation model as shown below in Figure 1.41 –

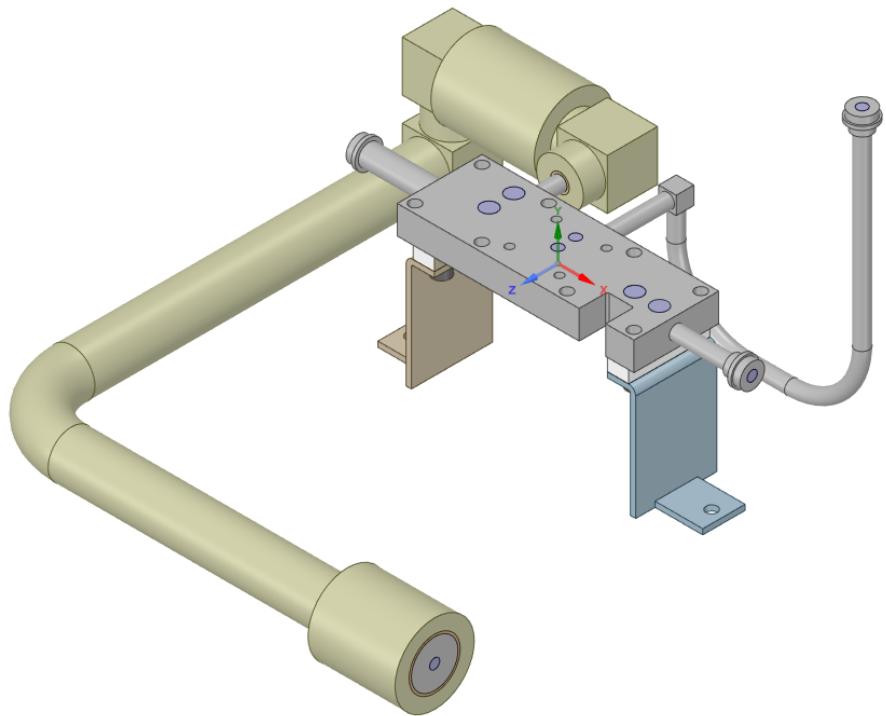


Figure 1.41:

Make sure to save your model! The file format is a .scdoc file as shown in Fig. 1.42. This is SpaceClaim file format. This will be the file you import into ANSYS to run your simulation.

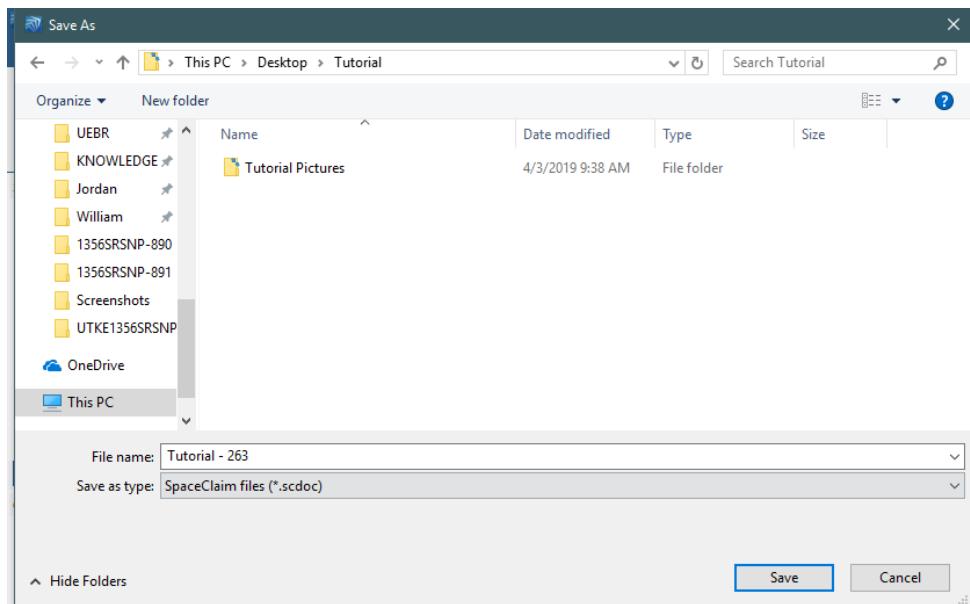


Figure 1.42:

2 Start of Simulation

1. Open the ANSYS AIM (or Discovery AIM) Software and Select the Fluid-Solid Heat Transfer Card to start the Simulation Set-Up as shown in Fig. 2.1. This example can be found with a file name of “Example_AIM.wbpj.”

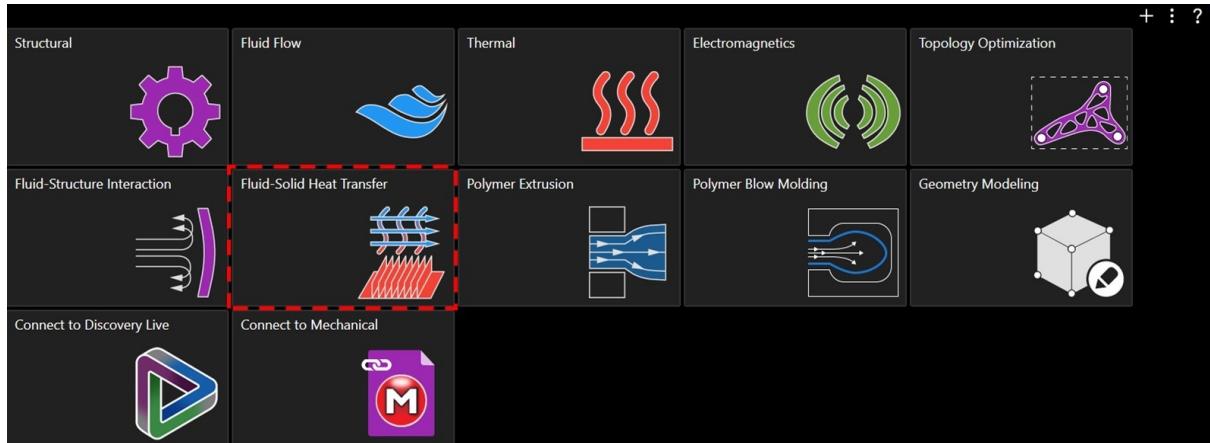


Figure 2.1:

2. Import your geometry file by clicking next and selecting the desired SpaceClaim File, as shown in Fig. 2.2, (Check the “**Define mesh manually**” and leave default boxes checked)

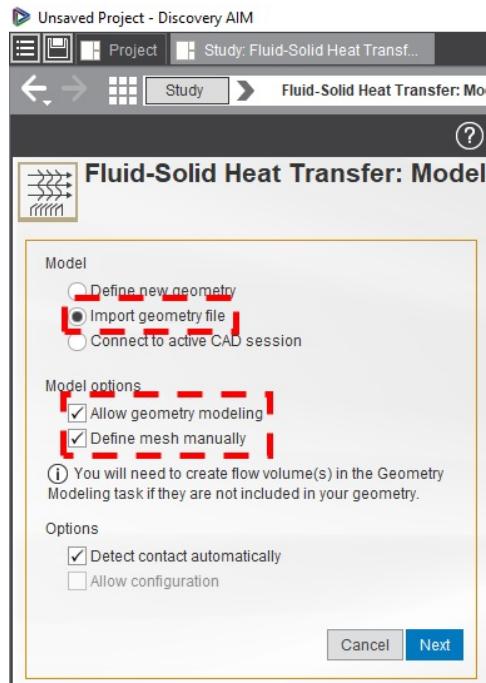


Figure 2.2:

3. Once the model is imported you will begin the set-up wizard using the prompts in the top left of the screen, as shown in Fig. 2.3. The **Swirling flow** needs to be checked to handle

flow in any elbow(s) of the weldment(s). We will be leaving the analysis as a Steady/Static study.

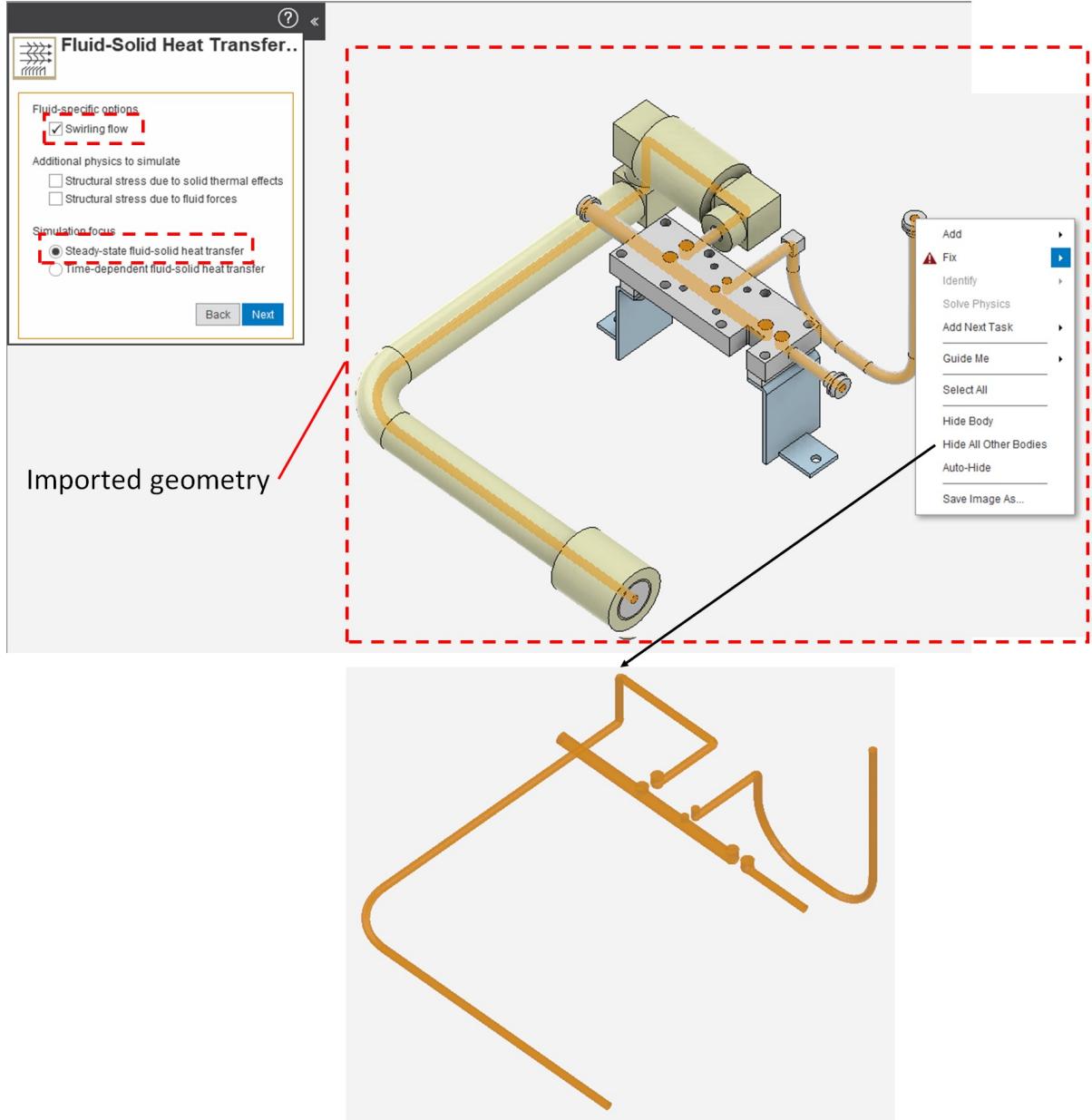


Figure 2.3:

4. Next you will be prompted to select the number of Fluid regions in your model and define what material those regions are. Using the imported geometry, shown in Fig. 2.3, as an example there are 2 fluid regions of Air. As shown in Fig. 2.4, first select the volume(s) that represent **Fluid Physics Region 1** and assign it the Material of Air. (You may click fluid volume(s) simultaneously and select “Hide All Other Bodies”, as shown in Fig. 2.3, to get to your fluid(s)). You can show the other components afterward by right clicking in the white space and selecting “show all”) Repeat this process to define you **Fluid Physics Region 2**. (Note: You can type into the box to see if a Material exists in the Ansys Library or select “Other” for a custom material you can set up the properties for later).

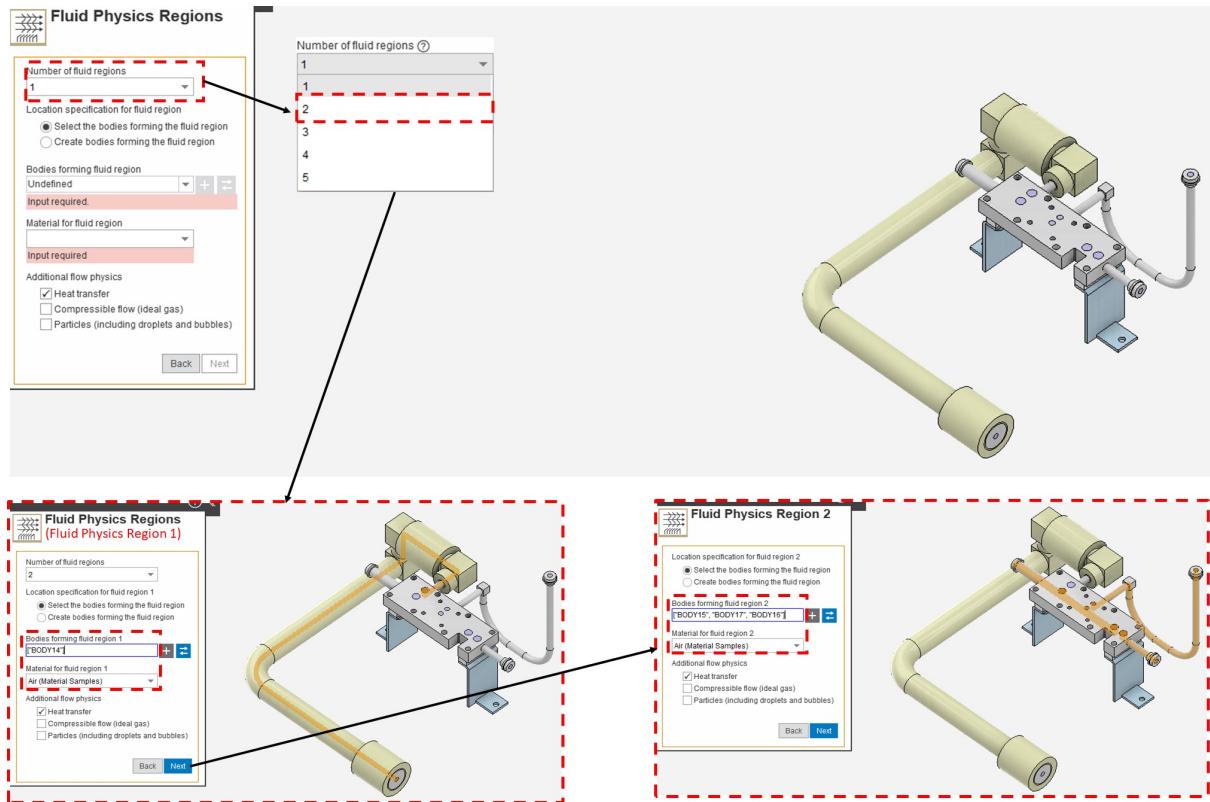


Figure 2.4:

- Once your Fluid Physics Regions (e.g., 2 fluid regions of Air) have been defined the next step is to define your Solid Physics Regions. As shown in Fig. 2.5, select the number of Solid Physics Regions you have in your geometry and apply the appropriate material (e.g., 4 Solid Physics Regions: Insulation in **Solid Physics Region 1**, Heating Wire in **Solid Physics Region 2**, Liner in **Solid Physics Region 3**, and Weldment in **Solid Physics Region 4** in this example). For the application of our Heat Sources later, a thing to remember is to make each heat source its own Solid Physic region (if they will have different wattages applied). Click next and repeat this process until all your Solid physics regions have been defined. Use the “hide body” function to access desired volumes where necessary. (Note: You can have as many Fluid and Solid regions as desired, however the setup has pre-defined caps in the drop downs. If more are desired, they can be added later.)

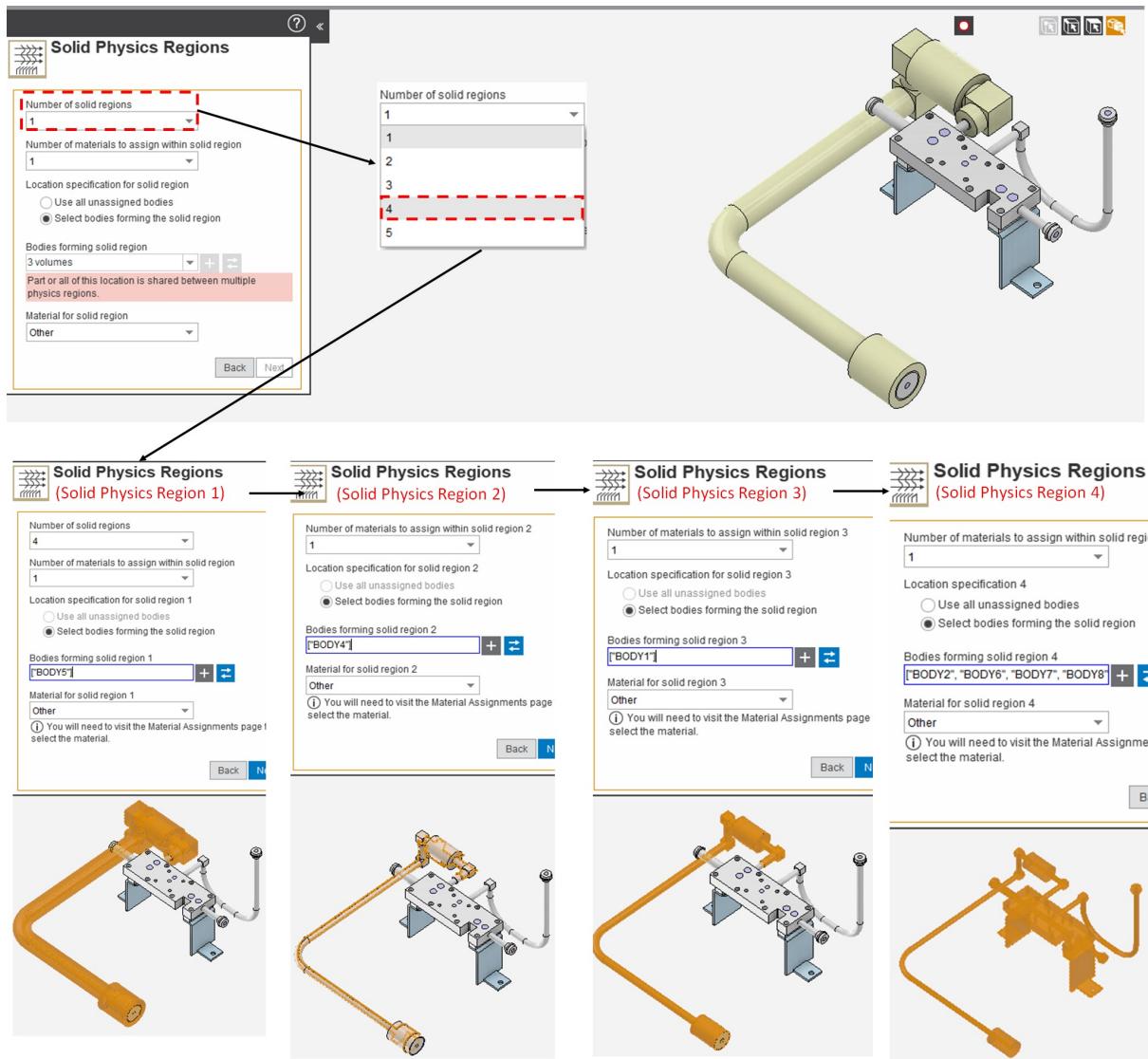


Figure 2.5:

- Once all your Solid Physics Regions have been defined you will be prompted to finish the setup. Press the Finish button and let the Simulation Initialize. (Note: If anything was input incorrectly or requires to be changed we can go back and do so if necessary, clicking finish does not permanently lock everything in.)

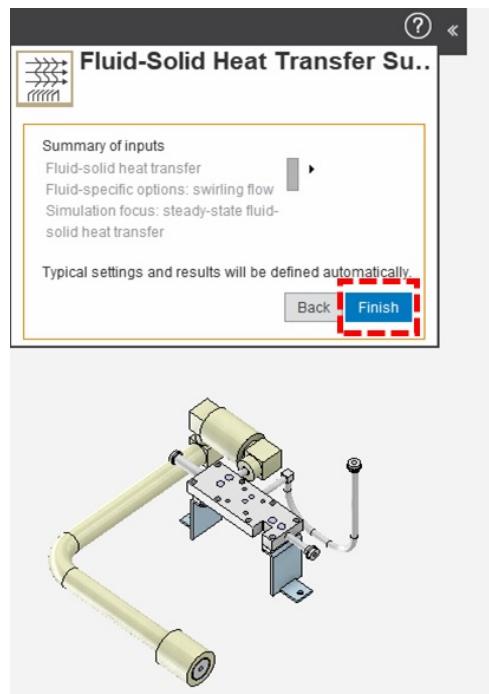


Figure 2.6:

3 Workflow

After your simulation has finished its initial setup, the Workflow, including Geometry, Mesh, Flow, and Results, should resemble the image below in Fig. 3.1 –

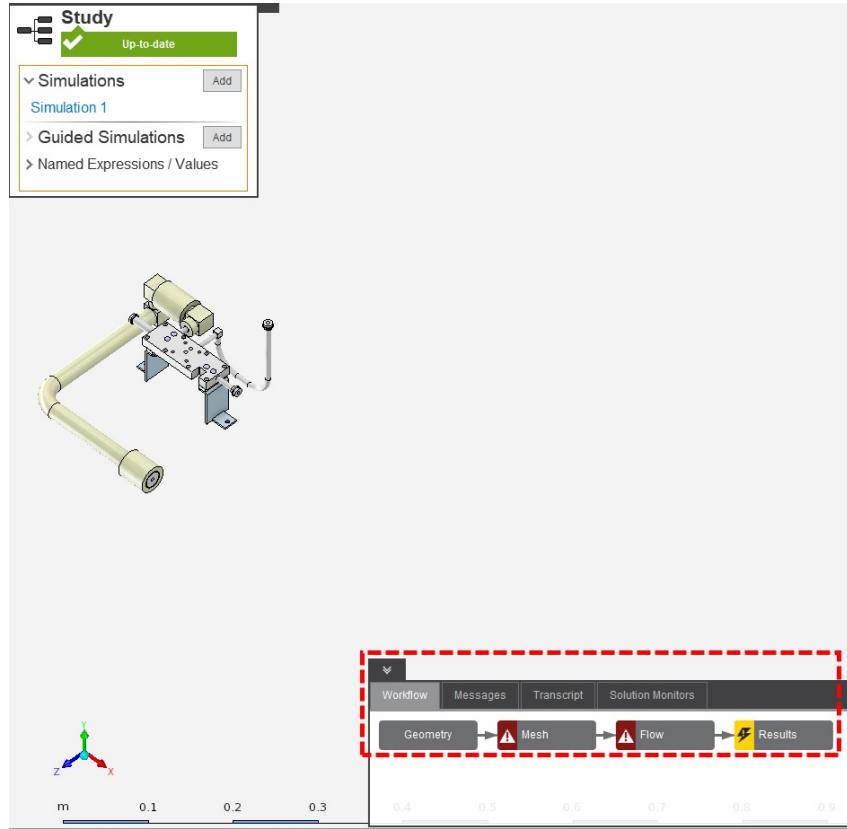


Figure 3.1:

Each bubble represents a different stage in your simulation. The Triangular figure indicates something requires attention before that stage can be completed. The Lightning bolt means you can evaluate/run through that stage. Also, in the Window are the Messages and Solution Monitors tabs. The Messages tab can provide feedback/a reason why something is not running properly and can be used to troubleshoot problem areas. While the Solution Monitors tab can allow you to watch your simulation solve in real time.

The detail of each stage in the Workflow is introduced in the following sections, respectively.

3.1 Geometry

The Geometry bubble is the where to enter SpaceClaim. Please find detail in [SpaceClaim](#). If no operation is needed in SpaceClaim, please skip this section and proceed into [Mesh](#) section.

3.2 Mesh

1. The next step is to select the Mesh bubble and to define the boundary layer of fluid volume(s) for our problem. After selecting the Mesh bubble, press the “Mesh Controls” link shown in Fig. 3.2. (Leave all the other settings at the default)

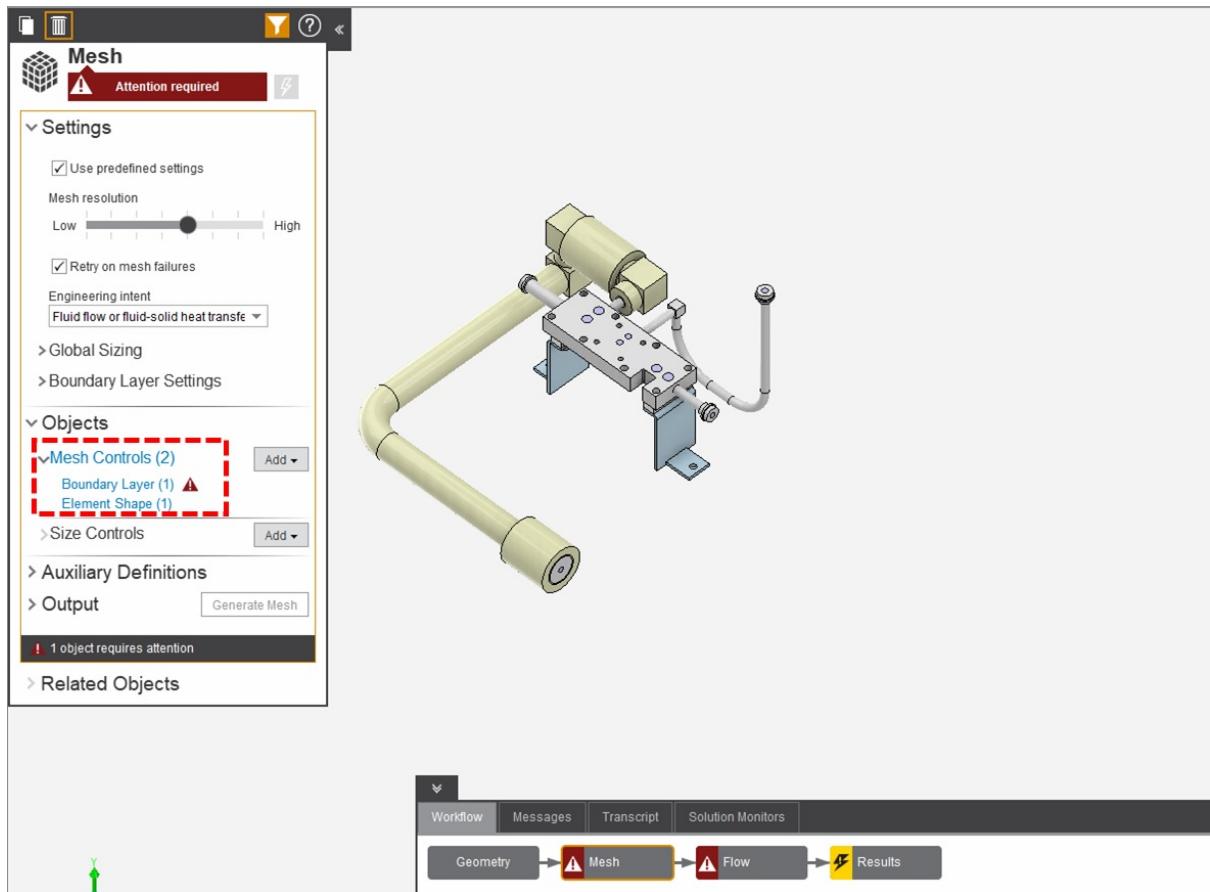


Figure 3.2:

- Once the Mesh Controls link is selected, we will need to define our boundary layer as shown in Fig. 3.3. We will be defining the boundary layer by select all the faces of both fluid volumes except the inlet and outlet faces where the fluid will be entering and exiting the system. Select all those faces and the press the + sign next to the location drop down to add them.

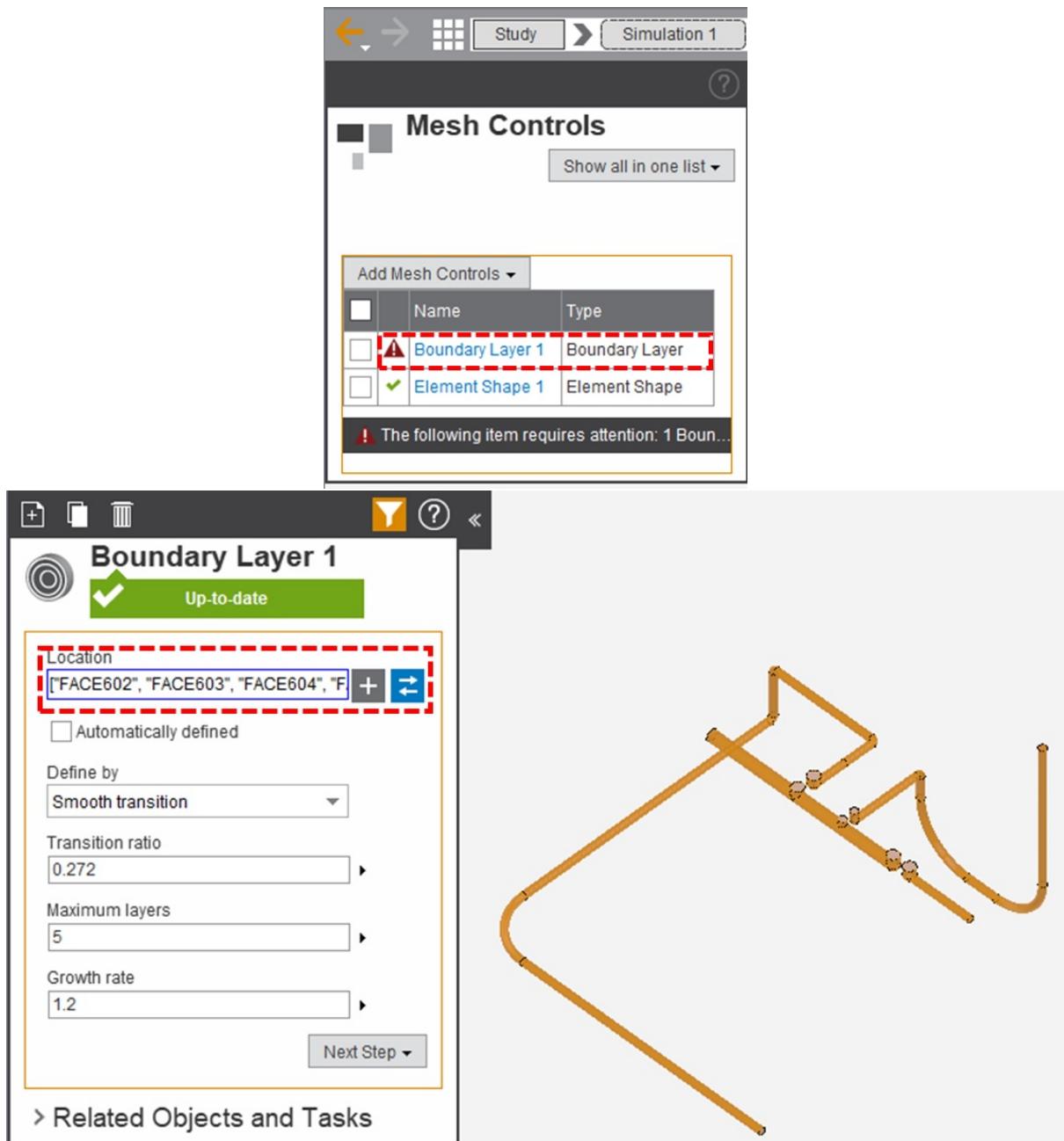


Figure 3.3:

3. Once the Boundary Layer has been defined, element shape can be accessed as shown in Fig. 3.4, and the shape we will select is **Automatic**.

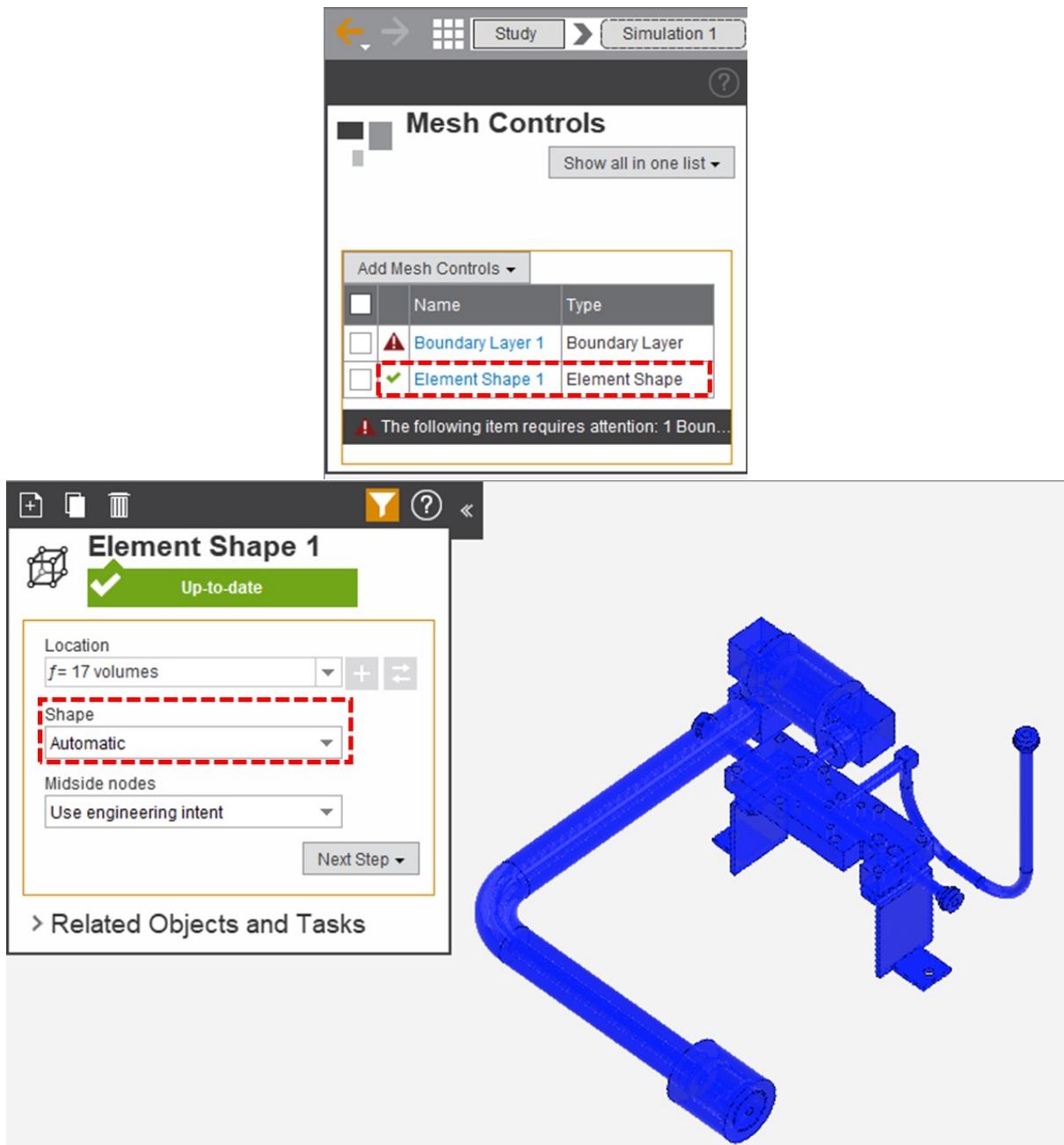


Figure 3.4:

4. The Mesh bubble will now have a Lightning Bolt next to it, as shown in Fig. 3.5. This indicates that we are now able to generate the Mesh. Click the Lightning Bolt in the Dialog area next to “Out-of-Date” to do so.

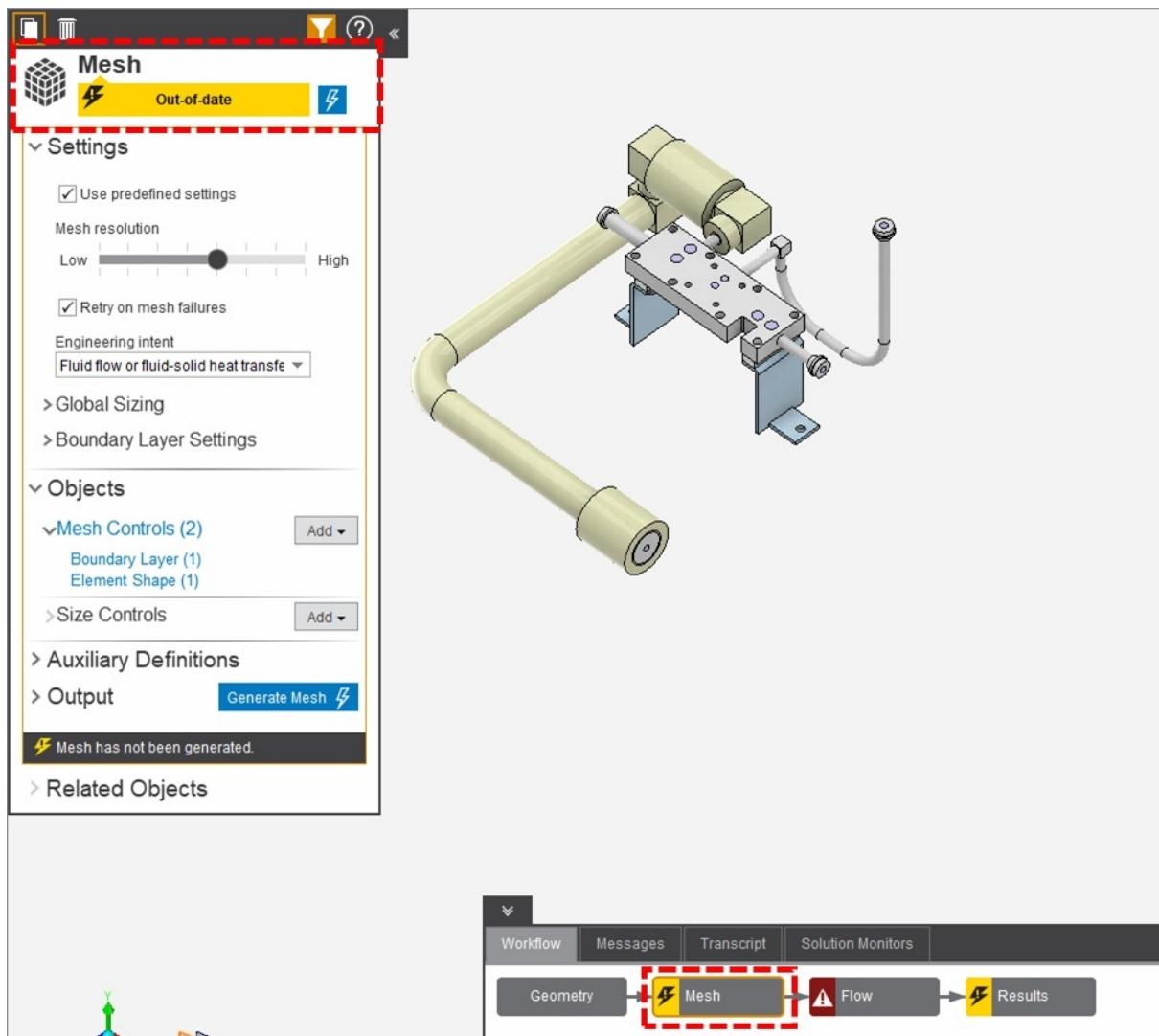


Figure 3.5:

- Once the Mesh has been generated as shown in Fig. 3.6 it is recommended to check the quality and skewness of the mesh.

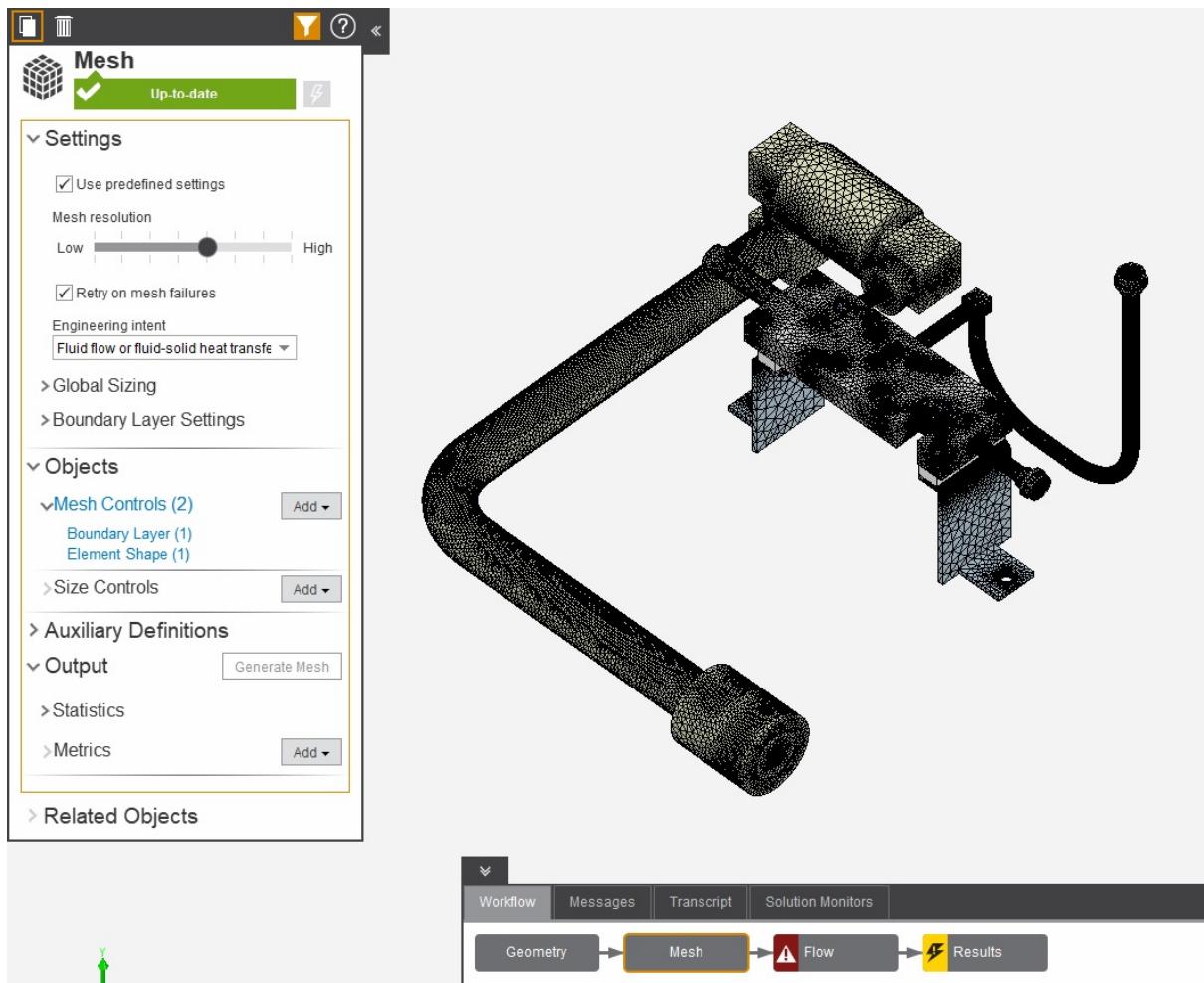


Figure 3.6:

6. The quality and skewness of the mesh are found in Metrics and are evaluated using Mesh Diagnostics as shown in Fig. 3.7. Average element quality higher than 0.8 and average skewness lower than 0.25 are fairly good to ensure quality results. If these two values are unsatisfactory, adjustment to a higher Mesh resolution can improve them before proceeding to [Flow](#).

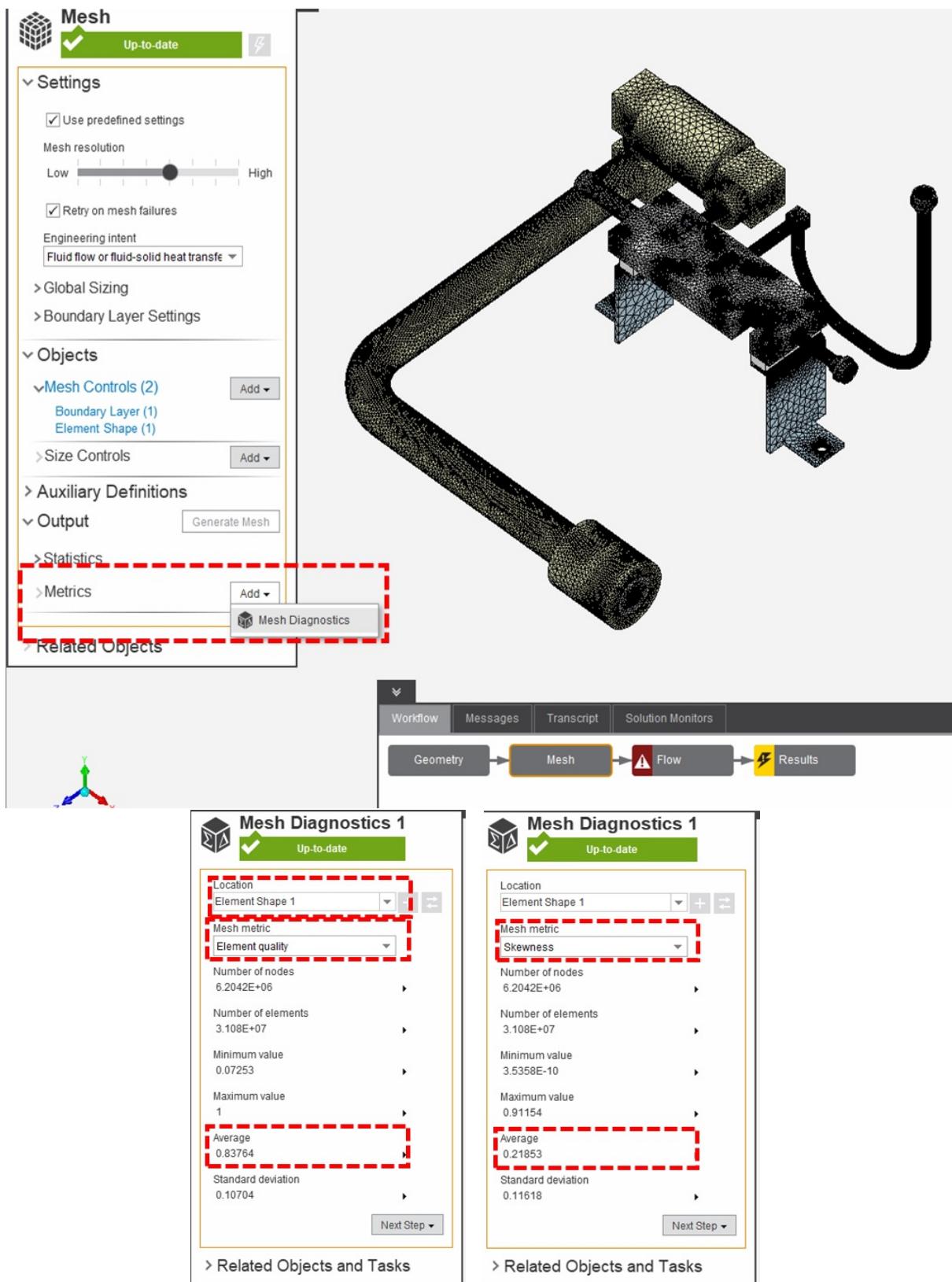


Figure 3.7:

3.3 Flow

At this moment, we are ready to define the flow criteria of our simulation. This can be done by selecting the Flow bubble in the Workflow area at the bottom of the screen. Once we are in the Flow Dialog Box, we need to address the circled items as shown in Fig. 3.8. Note those with a red Triangle  next to it indicates any setting(s) to be completed.

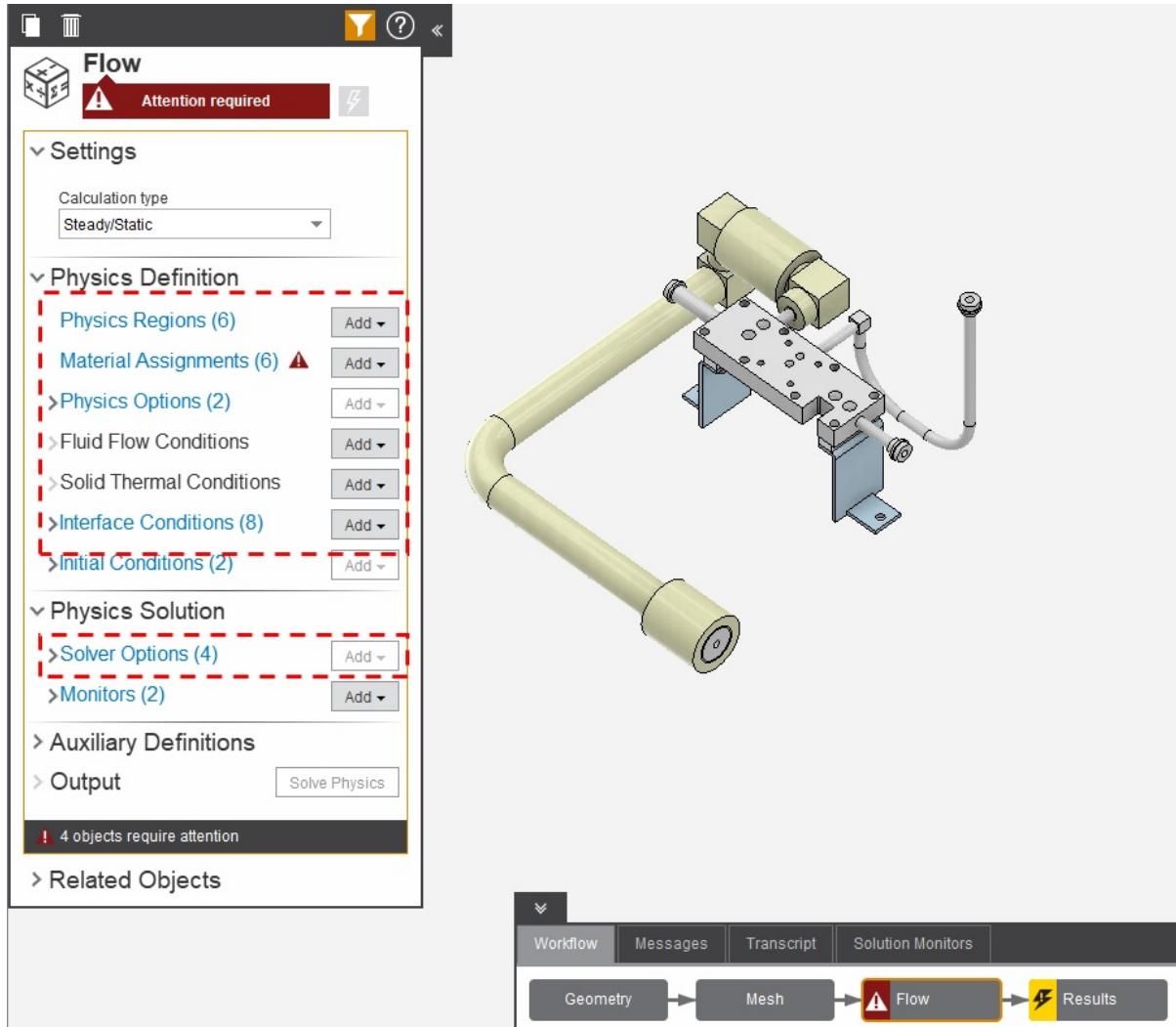


Figure 3.8:

Though there is no specific order to set up each item in the Flow, here is a recommended order for the listed items.

1. Solver Options
2. Physics Regions
3. Material Assignments
4. Interface Conditions
5. Physics Options
6. Fluid Flow Conditions

7. Solid Thermal Conditions

3.3.1 Solver Options

To speed up the simulation speed, the number of processors used can be changed from 2 as default to 4 as shown in Fig. 3.9 (Note: for computers that cannot use 4, try 3 instead).

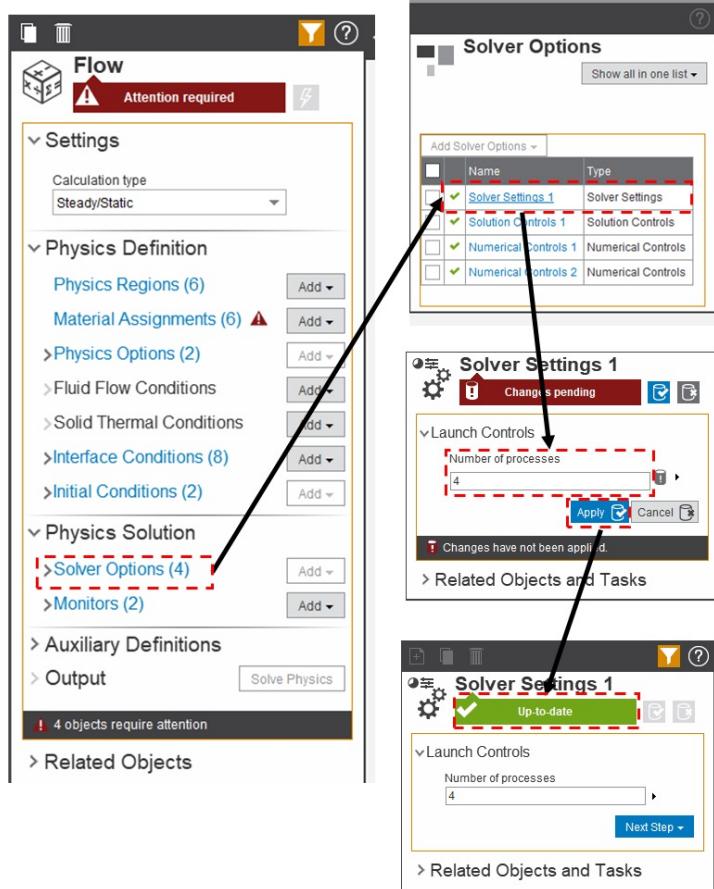


Figure 3.9:

3.3.2 Physics Regions

In Physics Regions, we can assign different layer(s) (e.g., insulation) to specific Physics Regions. If these were assigned already, the name of each Physics Region can be changed accordingly as shown in Fig. 3.10, naming Fluid_ext, for instance. It is recommended to re-name each Physics Region accordingly to aid in the setup of the simulation.

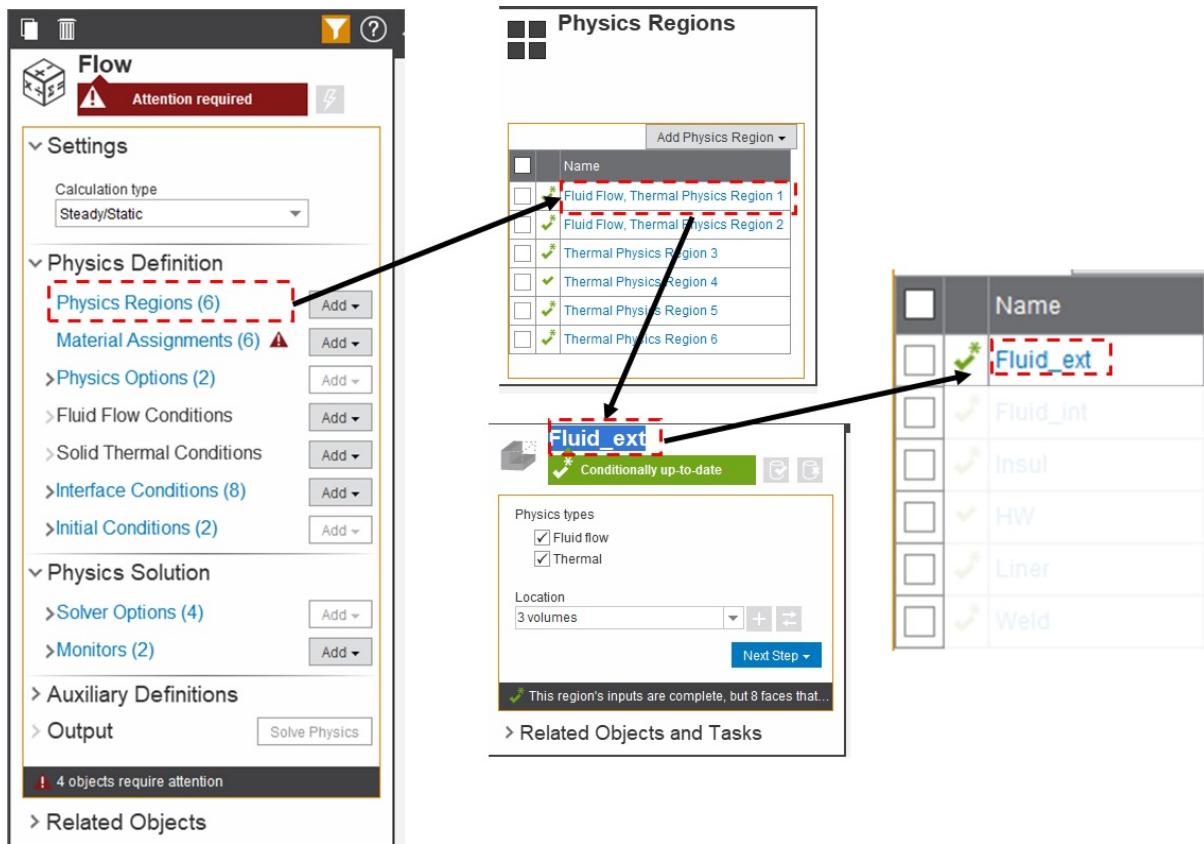


Figure 3.10:

3.3.3 Material Assignments

In Material Assignments, material(s) are assigned to each specific location, which is exactly linked to each specific Physics Region. As shown in Fig. 3.11, for instance, “Insul” is selected as Location, and “Fiberglass insulation (slit)” is selected as the Material.

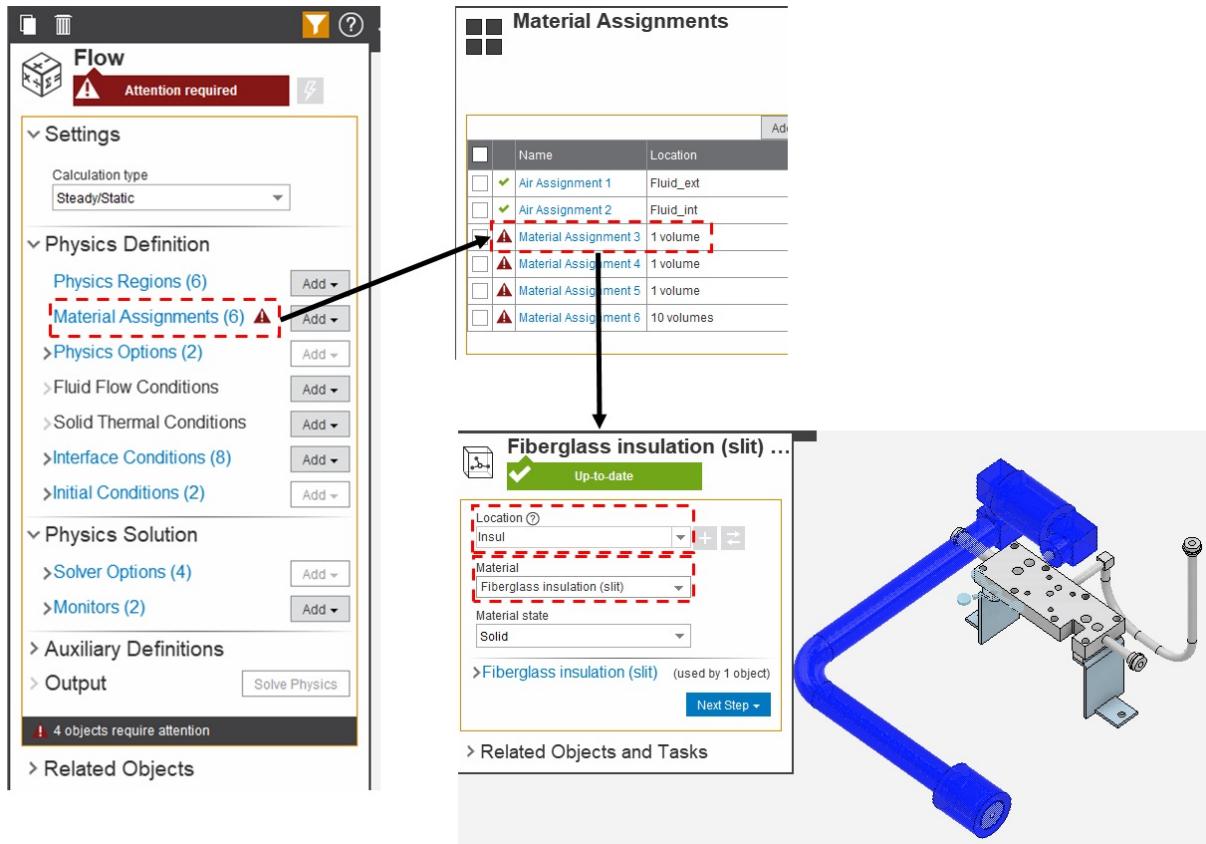


Figure 3.11:

3.3.4 Interface Conditions

An interface is the surface between layers (e.g., a surface between weldment and liner). The Interface Conditions provide the setting of interface thermal conductance (ITC) in the unit of $\text{W}\cdot\text{m}^{-2}\text{ K}^{-1}$. As shown in Fig. 3.12, for example, “Liner/HW” is the interface between Liner and HW Physics Regions, and “Energy interface model” in Interface Models is set None, which physically means no temperature jump in this interface. For another interface “Weld/Liner”, Thermal conductance is selected in “Energy interface model” with a value of $25 \text{ W}\cdot\text{m}^{-2}\text{ K}^{-1}$. This value may vary for different geometries and systems, please check with ext. 1118 for the latest update of these settings.

Also, it is recommended to rename each interface based on the names of adjacent layers (e.g., interface between Liner and HW is renamed as Liner/HW), as shown in Fig. 3.12. This will aid in post-processing/results.

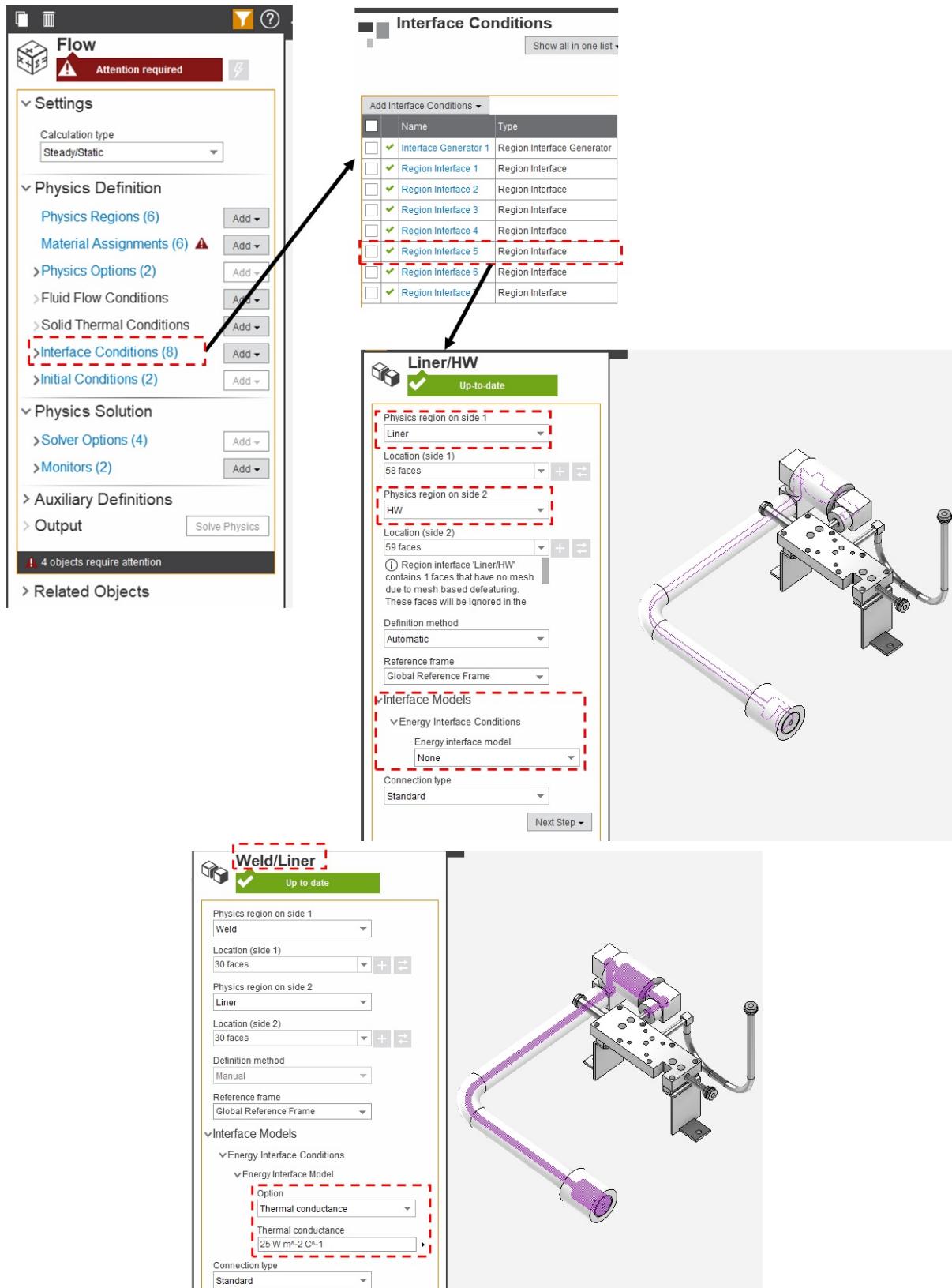


Figure 3.12:

3.3.5 Physics Options

This setting is needed only for flow region(s) as static fluid(s) (e.g., no flow case). The buoyancy setting is turned “On” in the Option, as shown in Fig. 3.13. Then items in need of definition, such as Gravity definition, Buoyancy model, etc., are setup for air in this example: Gravity definition with -9.81m/s^2 in Y direction (Designers should confirm the direction of gravity in each case), Boussinesq as the Buoyancy model, $27\text{ }^\circ\text{C}$ as Operating temperature, and $0.00382\text{ }^\circ\text{C}^{-1}$ (for air) as Thermal expansion coefficient. Please check thermal expansion coefficient for different fluids.

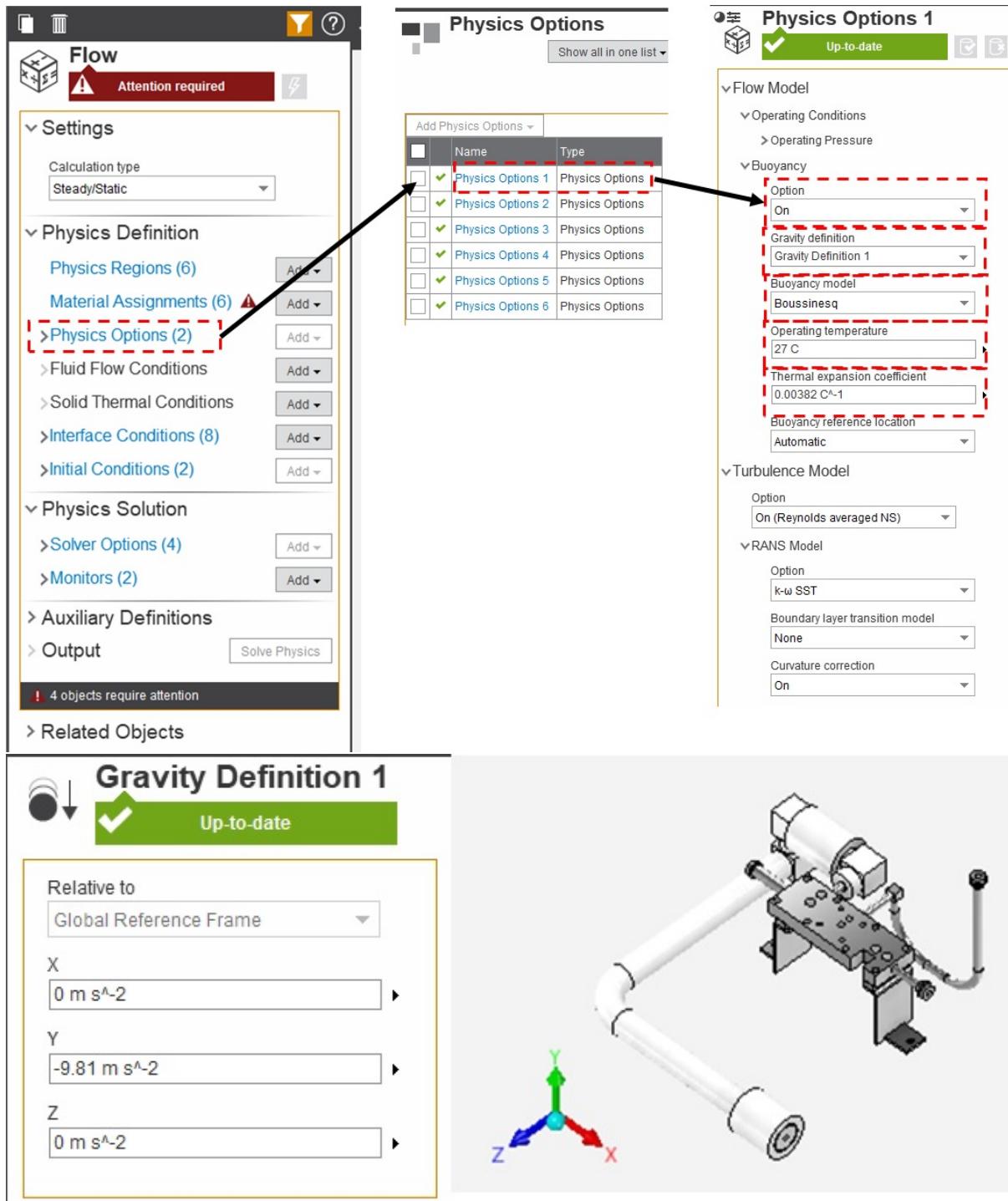


Figure 3.13:

3.3.6 Fluid Flow Conditions

Fluid flow conditions provide options, such Inlet, Opening, Outlet, etc. Here, an inlet is set up for demonstration shown in Fig. 3.14. Click add, select Inlet and then choose Fluid_int (Add → Inlet → Fluid_int). Accessing Inlet menu, the surface of Inlet can be selected, and click the Add icon near Location. We type a command “20 [litre/min]/Area(GetBoundary(“@ Inlet”))” (Note Inlet in the command needs to match Name of this setting shown in Fig. 3.14) for the velocity magnitude of the flow. For temperature, 20 °C is set up.

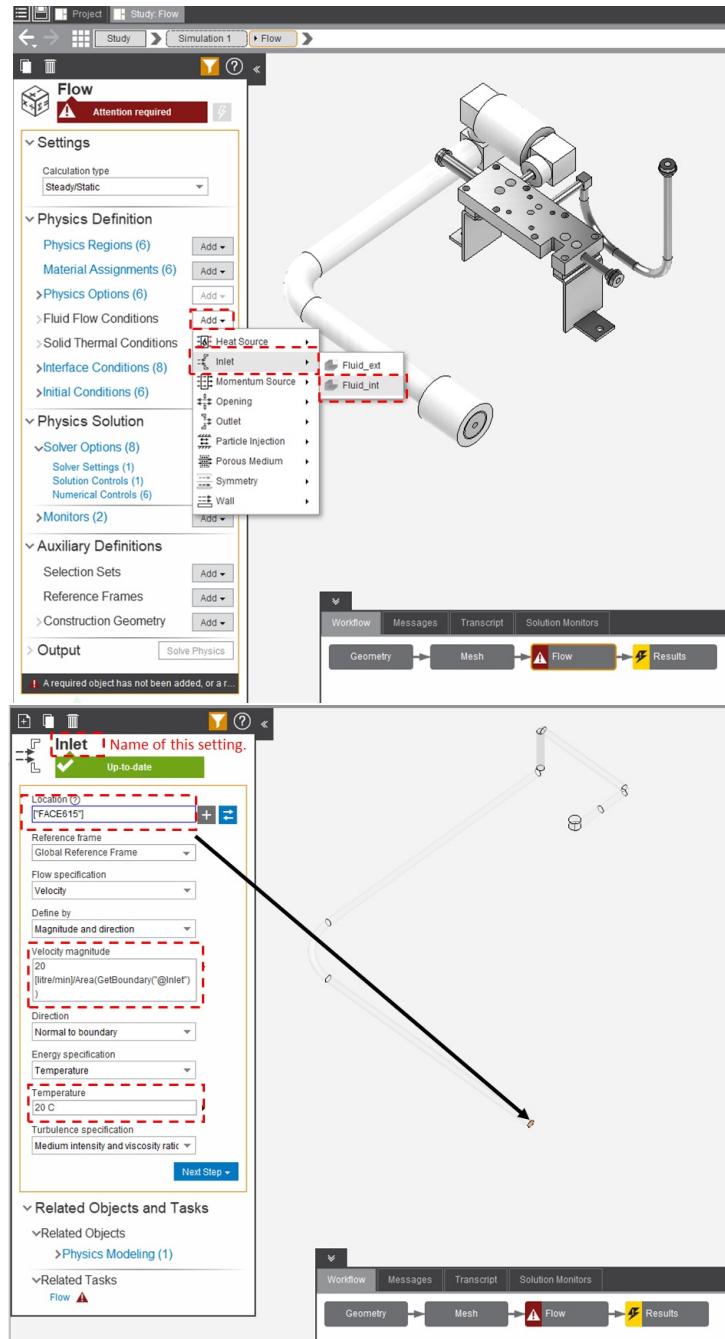


Figure 3.14:

For an outlet, as shown in Fig. 3.15, click add, select Outlet and then choose Fluid_int (Add → outlet → Fluid_int). For the Gauge static pressure of the outlet, 0 Pa is set here (see more detail in [Pressure Settings](#)).

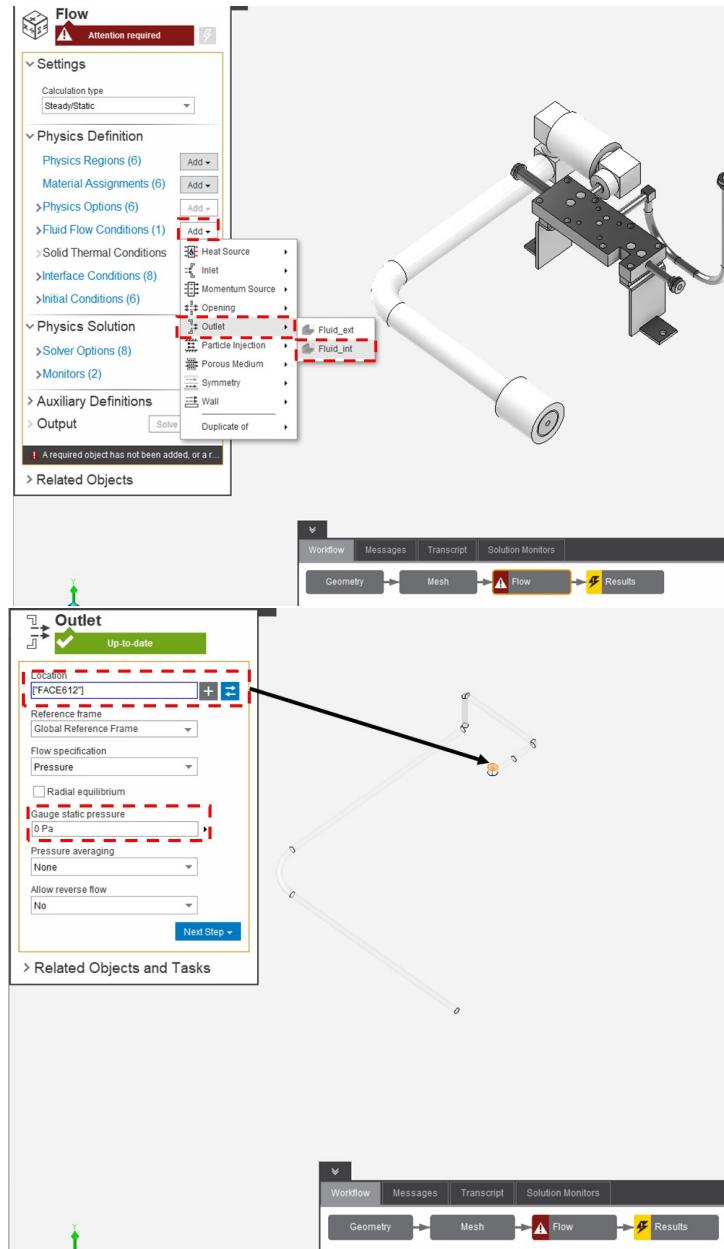


Figure 3.15:

3.3.7 Solid Thermal Conditions

Two types of Solid Thermal Conditions are applied here for demonstration - 1.) Convection and Radiation and 2.) Heat Source.

Convection and Radiation is the main heat loss from the Insulation, Insul. As shown in Fig. 3.16, we select Add → Convection and Radiation → Insul. Then, by default, AIM selects all undefined surfaces in Location (or type command “DefaultWalls()” in Location). For convection, the heat transfer coefficient is $20 \text{ W}\cdot\text{m}^{-2}\cdot\text{K}^{-1}$ with 25°C as the convection temperature in BH lab. For radiation, emissivity is 1 with the radiation temperature of 25°C .

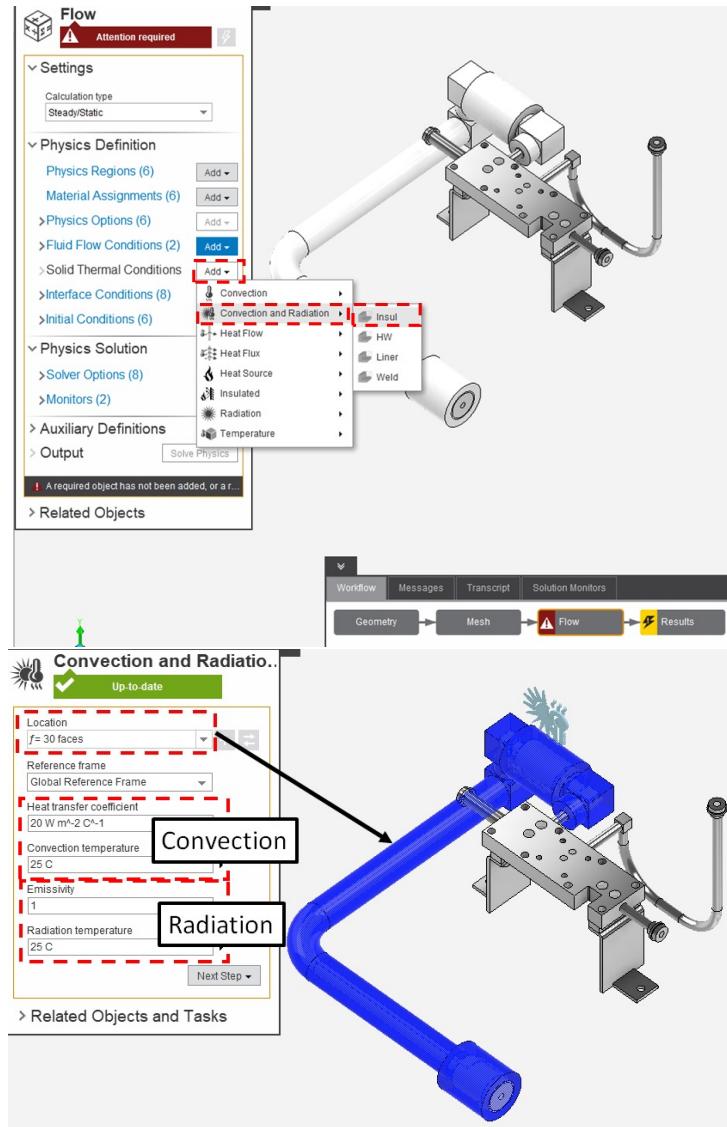


Figure 3.16:

For Heat Source, we first select Add → Heat Source → HW, as shown in Fig. 3.17. The volume of the Heating Wire is selected first. Total source is chosen in Source specification with 62 W as the Total source.

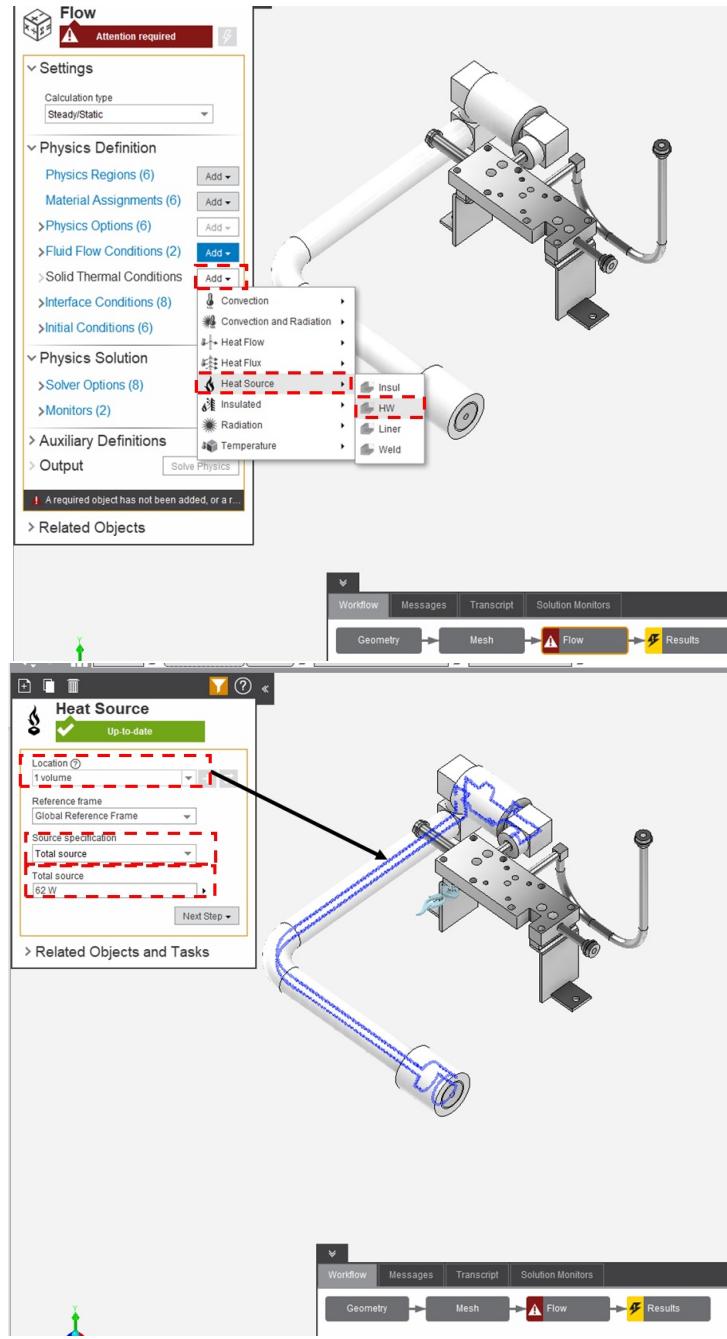


Figure 3.17:

After properly defining all our Materials, Fluid Regions, and Solid Regions the Flow portion of our Simulation should be ready to be evaluated. This is indicated by the yellow “Out-of-date” box and the blue lightning bolt appearing, .

If this does not happen, you may have an issue with your geometry. A quick check that can be done is to look at your Interface Conditions under the Flow section of the Simulation. Ansys will create an interface region for every surface touching/mated together. An example is shown below where an interface was generated for the outside surface of the fluid and then inside surface of the piping it is running through. If a region is not created after Meshing, there is most likely something that needs to be geometrically fixed before you can progress. (Note: If you do change your Geometry, you will have to either re-mesh or potentially run through the setup from the start depending on the severity of the geometry change.)

However, if you did not have any issues, we can proceed with running the Flow portion of

our simulation by pressing the blue lightning bolt next to the “Out-of-Date” text. The amount of time the simulation will take to run will depend mainly on the number of different Fluid/Solid regions you have, the complexity of your geometry, and how refined your mesh is.

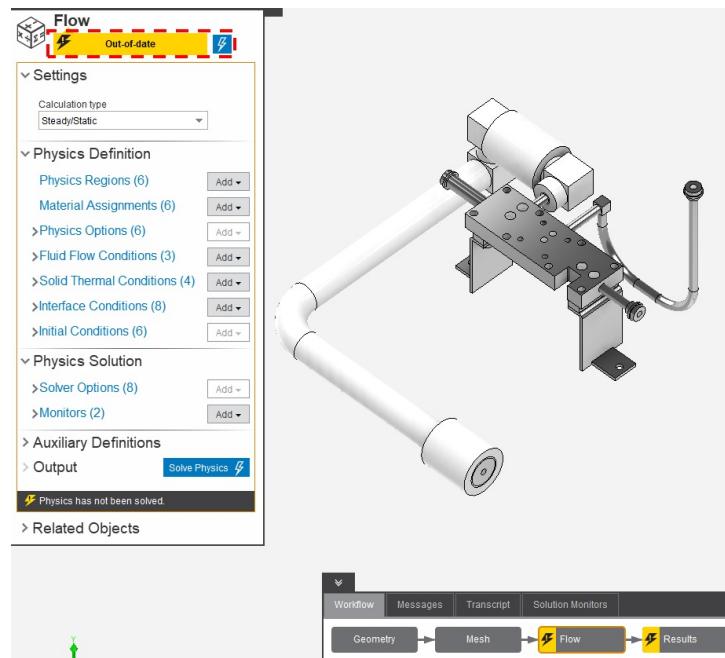


Figure 3.18:

3.4 Results

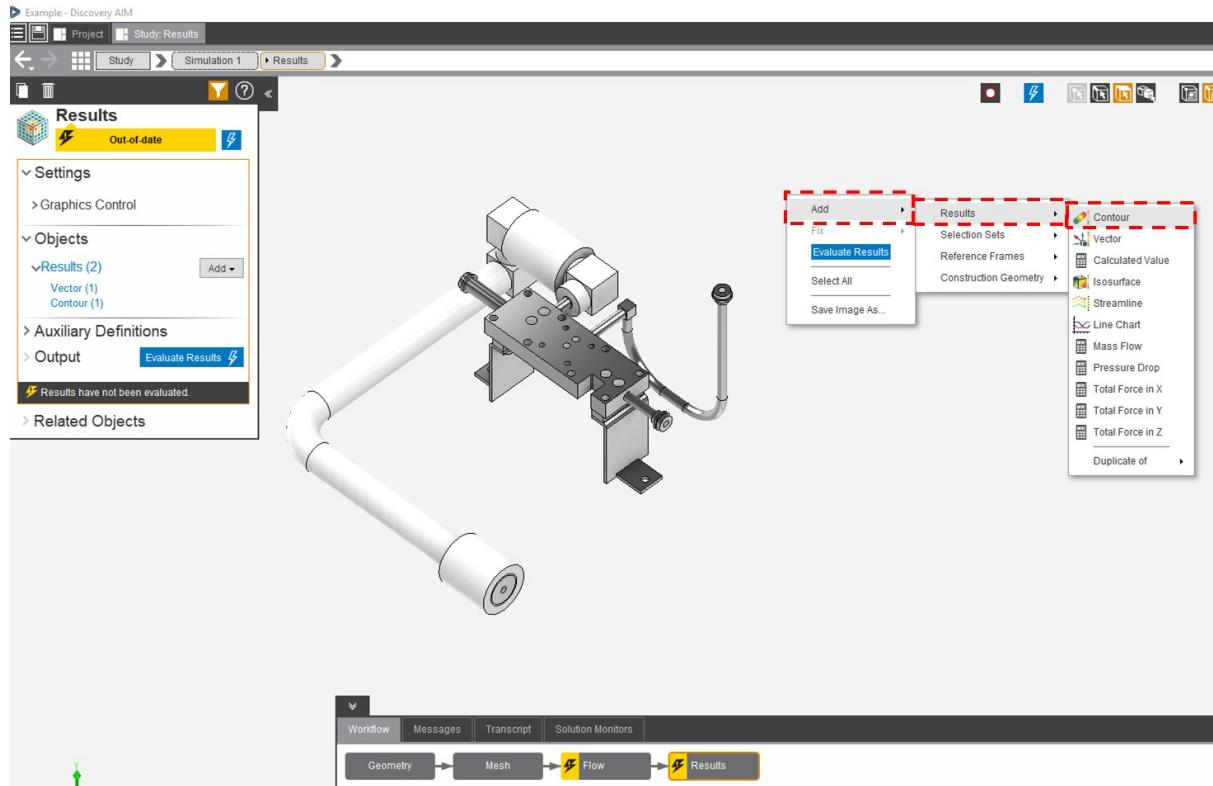


Figure 3.19:

Once the simulation has finished running, we can look at the results. By clicking on the Results bubble in the workflow we can add different values, contours, and more. You can do so by using the right click in white space method going to Add → Results → Contours, Calculated Value, Streamline, etc., as shown in Fig. 3.19.

After selecting any of the above you will be prompted to either select a face (e.g., Calculated Value of average temperature of outlet shown in Fig. 3.20), a volume (e.g., Streamline of fluid_int shown in Fig. 3.20), or interface condition (e.g., Contours of temperature of weld/liner shown in Fig. 3.20).

Selecting a face and adding a streamline (or alternatively select “Volumetric grid sampling” with 0.001 m as the Grid spacing) is useful for seeing how the flow travels throughout the system, using a face and assigning a calculated value is good for looking at the exit temperature and velocity of a fluid leaving the geometry. Using the contour result with a plane is more useful for analyzing how the heat is transferred throughout the geometry while picking an interface condition helps with looking at the surface temperature of things within the geometry.

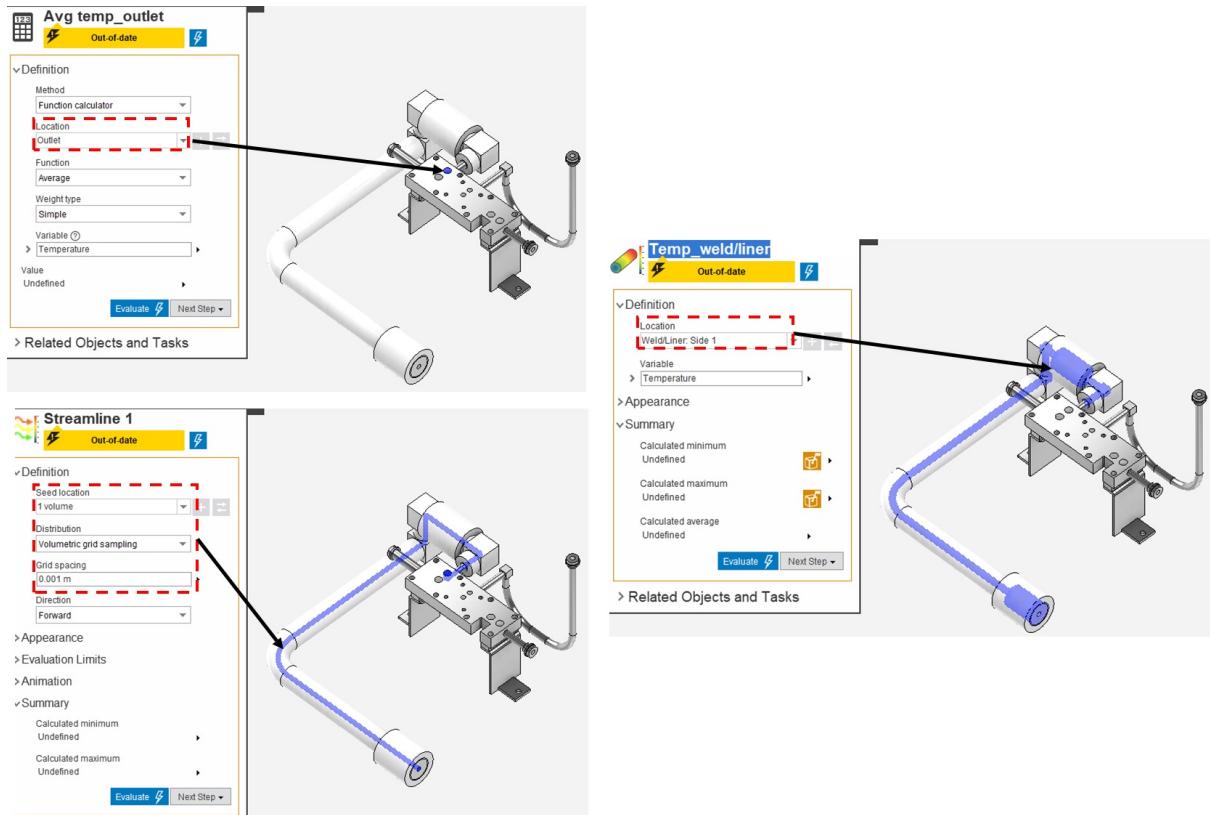


Figure 3.20:

The temperature contour between liner and weldment, average outlet temperature, and temperature contour of streamline are results as shown in Fig. 3.21.

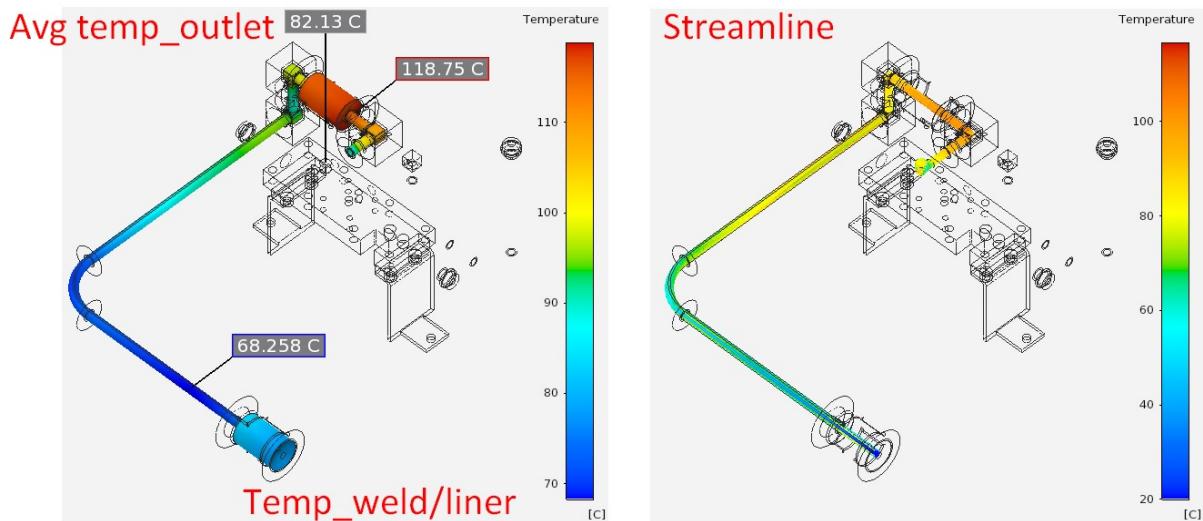


Figure 3.21:

4 Appendix

4.1 Pressure Settings

Pressures can be measured on different basis and defined as absolute pressure (P_{abs}), atmospheric pressure (P_{atm}), and gauge pressure (P_{gauge}), where $P_{abs} = P_{atm} + P_{gauge}$, as shown in Fig. 4.1. For instance below, in a semiconductor tool chamber, P_{abs} is 10 torr, and in AIM P_{abs} needs to be converted to $P_{gauge} = -99992$ Pa for the input of Gauge static pressure in Fig. 4.1.

$$P_{abs} = P_{atm} + P_{gauge}$$
$$10 \text{ torr} = 760 \text{ torr} + P_{gauge}$$
$$\rightarrow P_{gauge} = -750 \text{ torr}$$
$$= -99992 \text{ Pa}$$

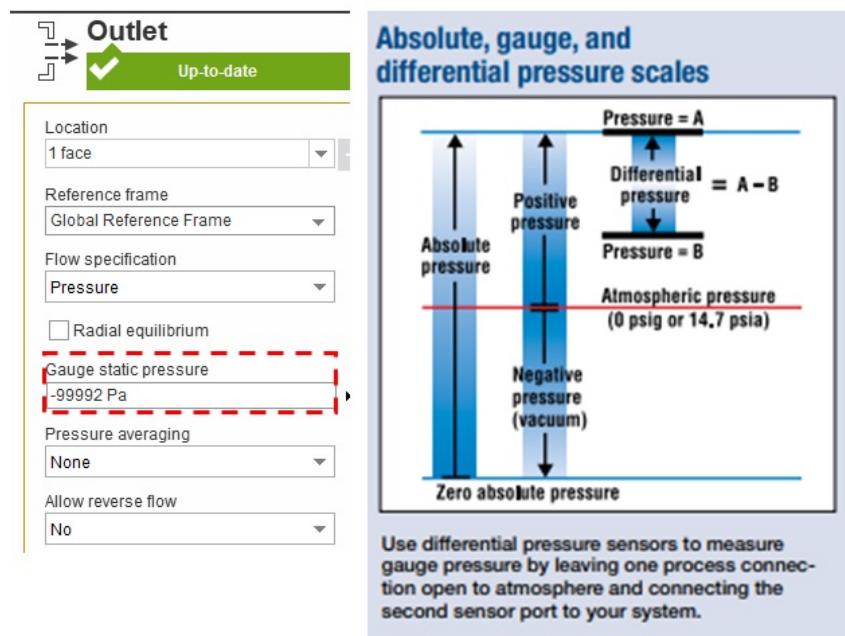


Figure 4.1:

4.2 Filter Settings

Filters are devices in which flows pass porous mediums, with larger resistance to flow. The settings for filters can be accessed via Fluid Flow Conditions → Add → Porous Medium as shown in Fig. 4.2. Once in Porous Medium menu (e.g., Porous Medium 1), fluid volume of interest needs to be selected. In Option, Orthotropic is selected. Internal resistances are set zero in X, Y, and Z directions. The values of viscous resistances in X, Y, and Z directions depends on the main flow direction. For instance, the main flow direction is in X direction, and thus $2.13E+09 \text{ m}^{-2}$ is the viscous resistance in X, while $1.17E+10 \text{ m}^{-2}$ is the viscous resistances in Y and Z directions. The streamline of flow passing the porous medium is shown in Fig. 4.3, where the main flow direction is in X direction.

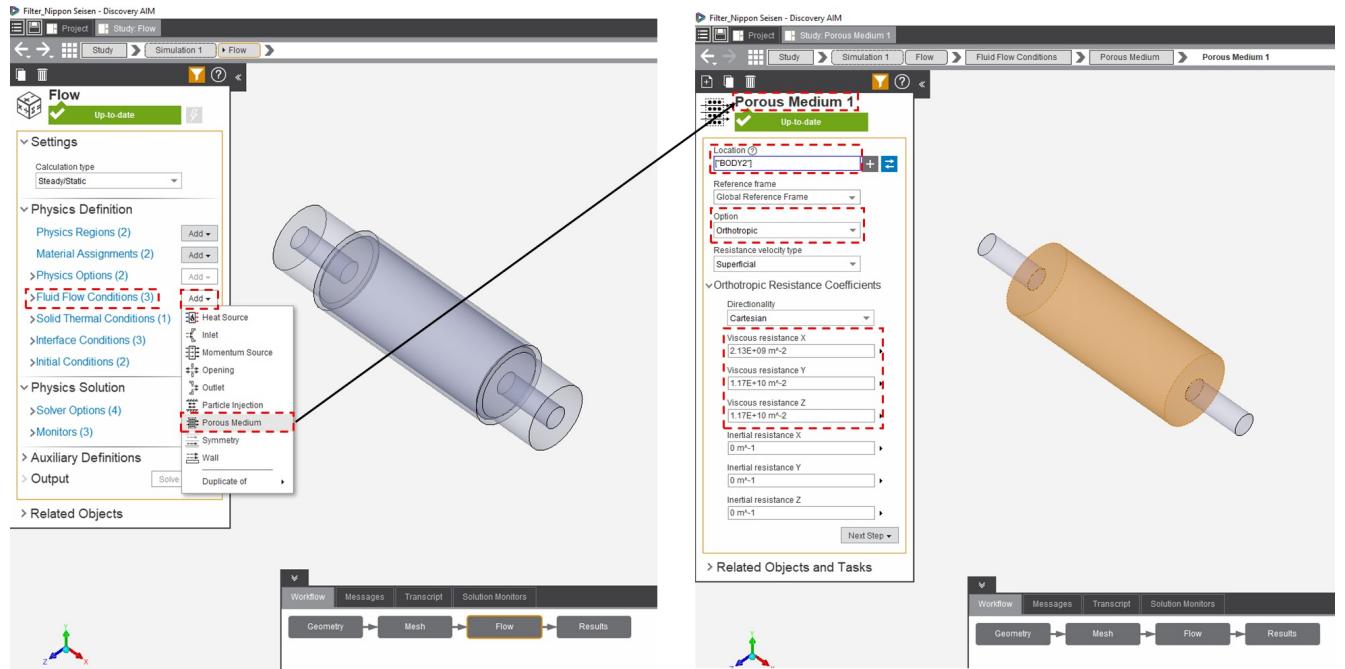


Figure 4.2:

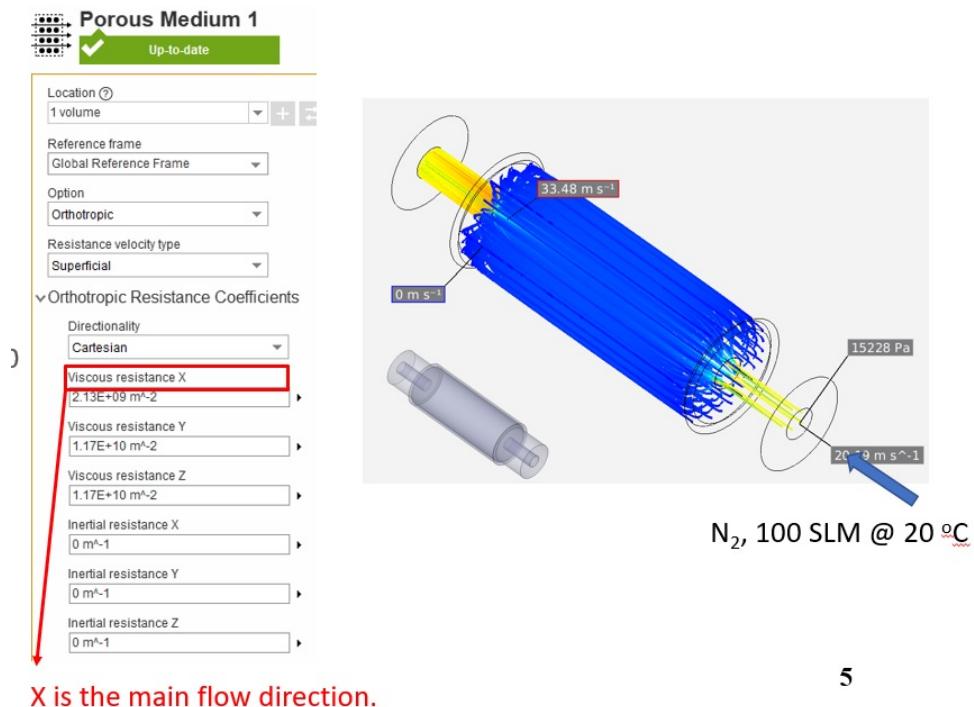


Figure 4.3:

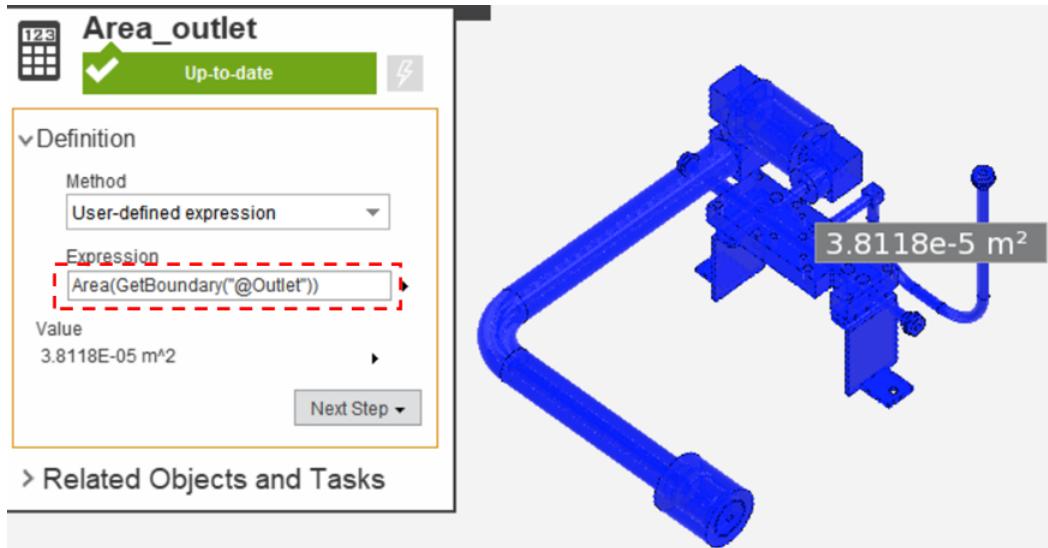
4.3 Expressions

Quantity (unit)	Expression, $f(\textcolor{red}{x}_1, \textcolor{blue}{x}_2, \dots)$	Arguments, $\textcolor{blue}{x}_1, (\textcolor{blue}{x}_2, \dots)$
Area (m ²)	Area(GetBoundary("@" $\textcolor{blue}{x}_1$))	$\textcolor{blue}{x}_1 = \text{Inlet}$
Velocity magnitude (m/s)	$\textcolor{red}{x}_1$ [litre/min]/Area(GetBoundary("@" $\textcolor{blue}{x}_2$))	$\textcolor{red}{x}_1 = 20, \textcolor{blue}{x}_2 = \text{Inlet}$
Volumetric flow rate (SLM)	Average(Velocity.mag, GetBoundary("@" $\textcolor{red}{x}_1$), "Area")*Area(GetBoundary("@" $\textcolor{red}{x}_1$)) * 60000 [s m ⁻³]	$\textcolor{blue}{x}_1 = \text{Outlet}$

- Area (m^2), the cross-sectional area of the **Outlet** is **estimated** using the expression

$$\text{Area}(\text{GetBoundary}("@\text{Outlet}))$$

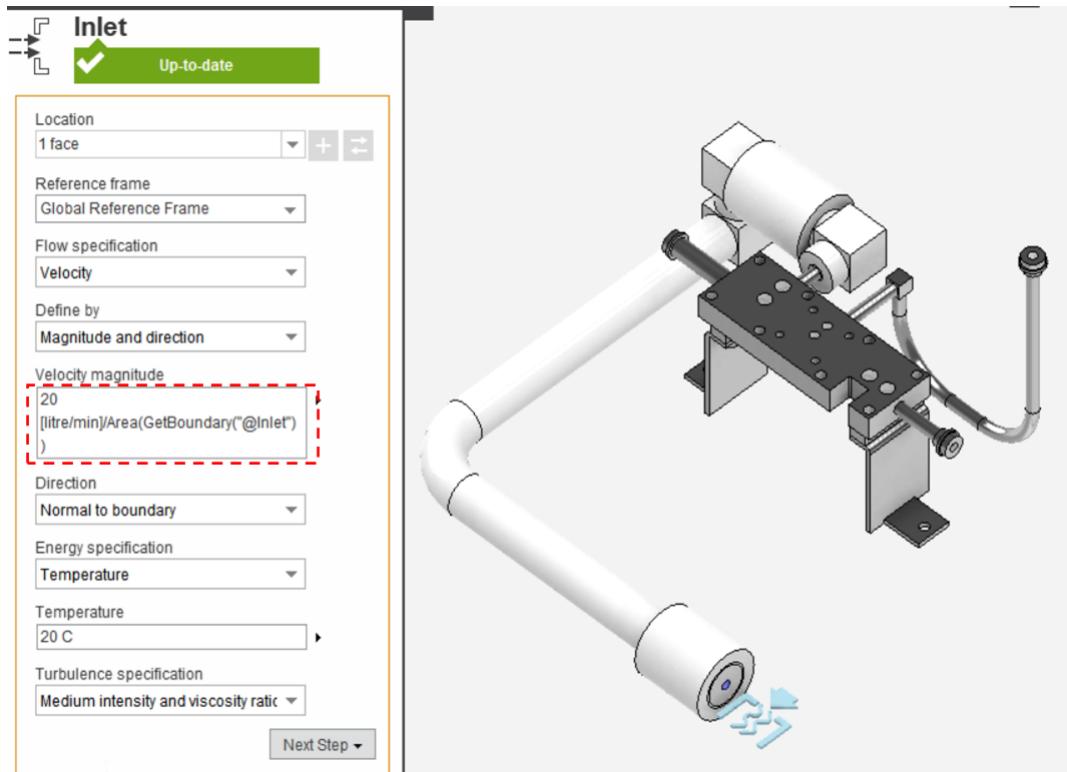
, as shown below.



- Velocity magnitude (m/s), the magnitude of **20 SLM** at the **Inlet** is **defined** using the expression

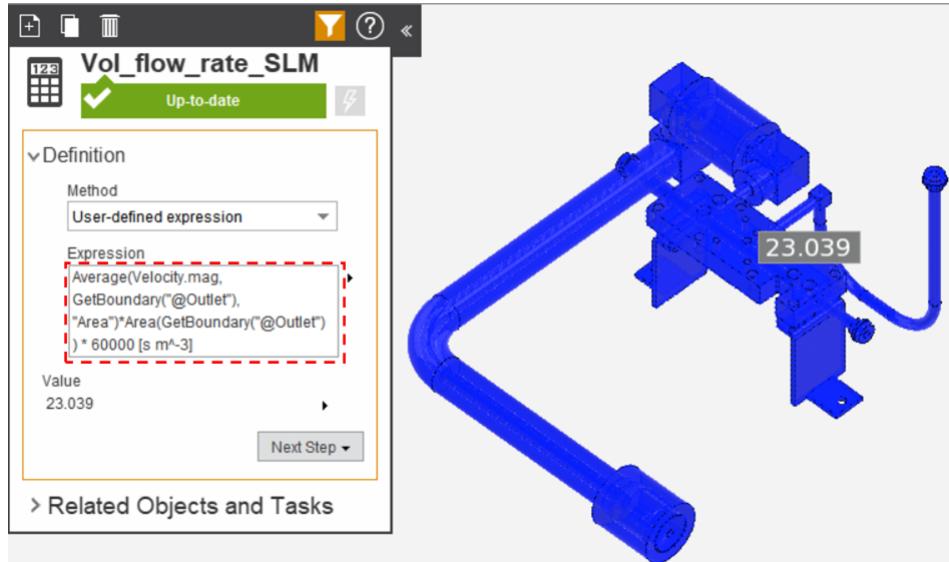
$$20 \text{ [litre/min]} / \text{Area}(\text{GetBoundary}("@\text{Inlet}))$$

, as shown below.



- Volumetric flow rate (SLM), the value of the volumetric flow rate at the *Outlet* is estimated using

Note, the unit "SLM" is NOT shown along with the estimate.



4.4 Design of experiments

To be added.