ICEMM
SIMULATION ENGINEERING

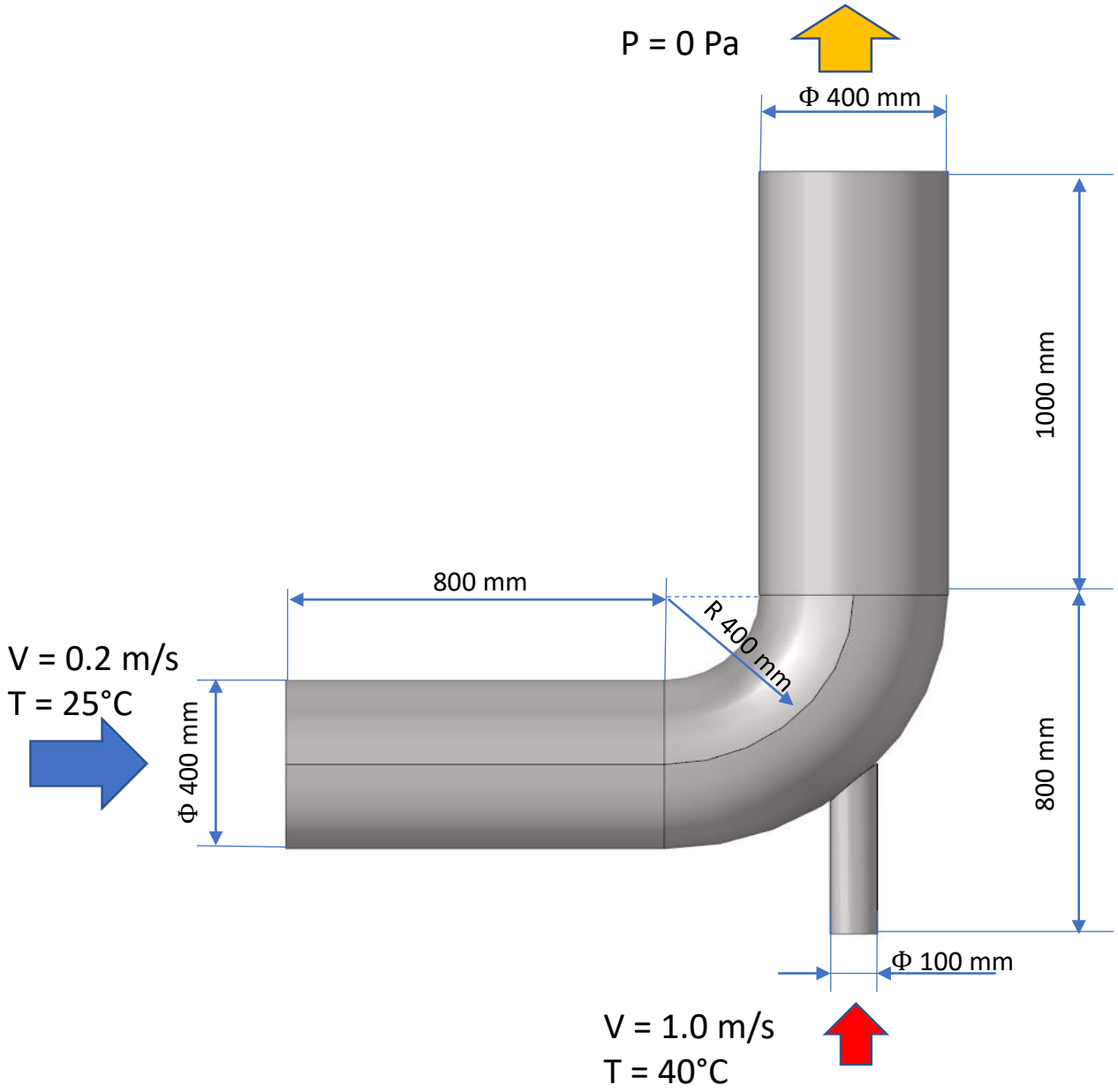
CFD Tutorial 01

Mixing Elbow: Basic Tutorial. Interface.

CERTIFICACIÓN DEL SISTEMA DE CALIDAD
DNV·GL
ISO 9001
PYME INNOVADORA

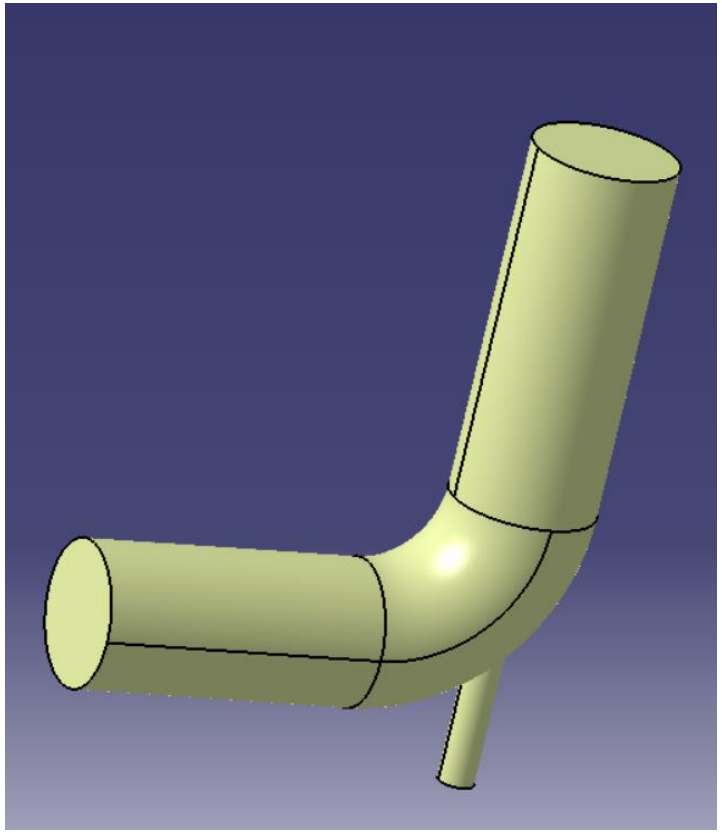
Mixing elbow

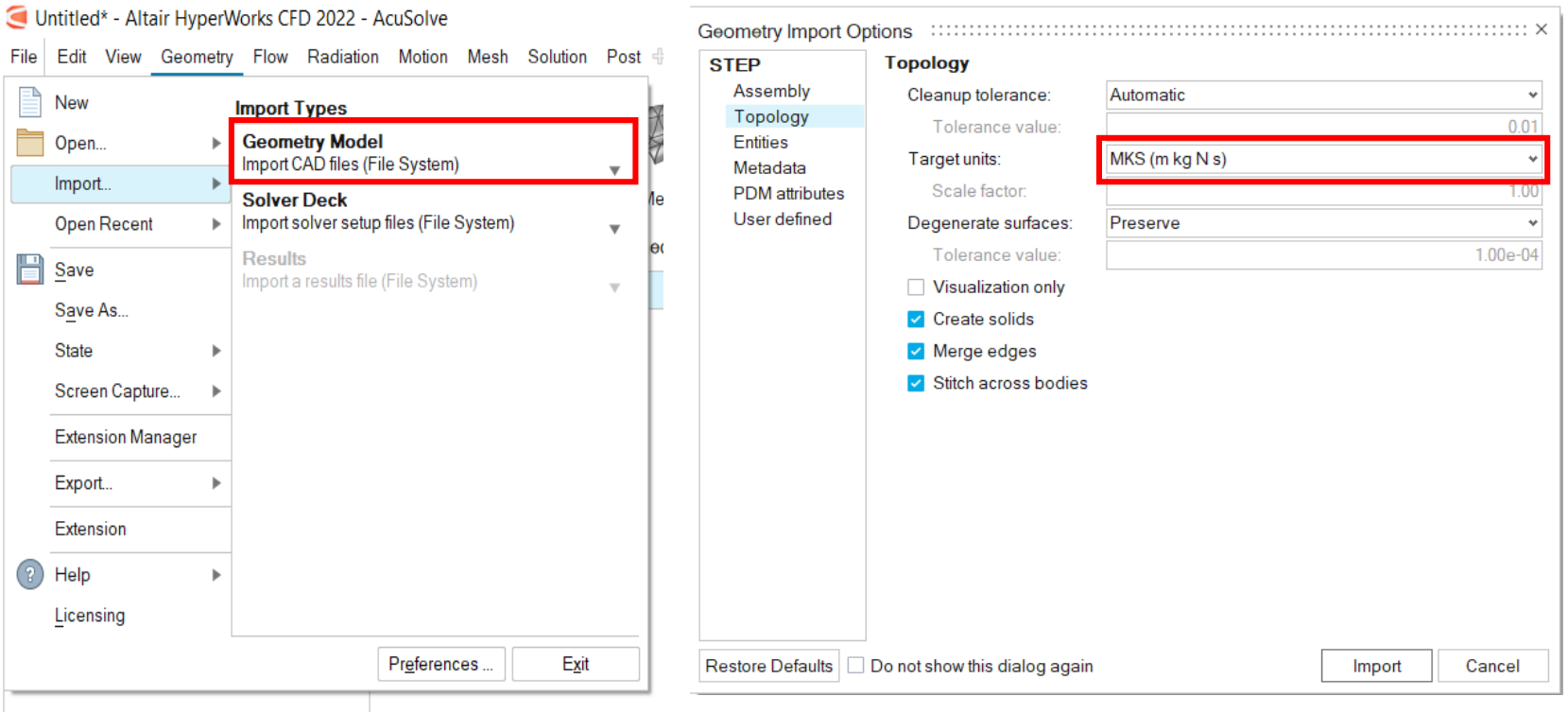
- The aim is to simulate the thermal and flow behavior of a mixing elbow.
- A cold flow is coupled with a high temperature inlet according to the diagram.
- This example is used to see the basic Hyperworks CFD workflow and the handling of its interface.



Import Geometry

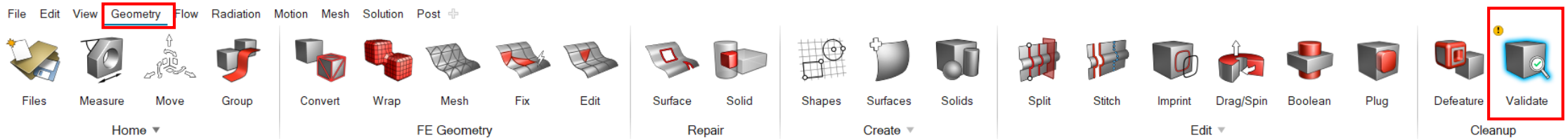
- The geometry is generated in a CAD software such as CATIA.
- It is exported as .stp file and then imported into Hyperworks CFD: File>>Import>>Geometry Model
- In Import Options, check that Topology>>>Target Units is in the units of the analysis (MKS by default in Acusolve).





Repair and validate geometry

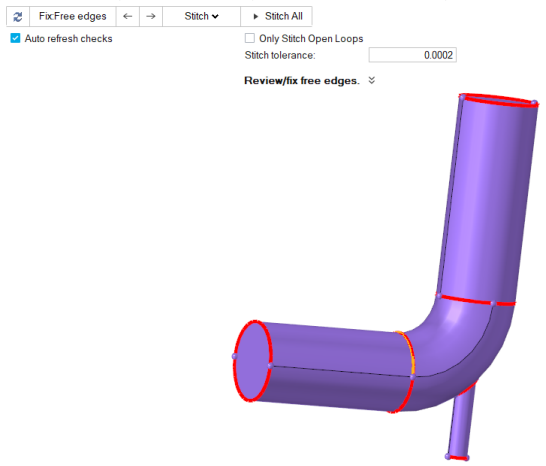
- In the Geometry ribbon, select Clean-up>>Validate

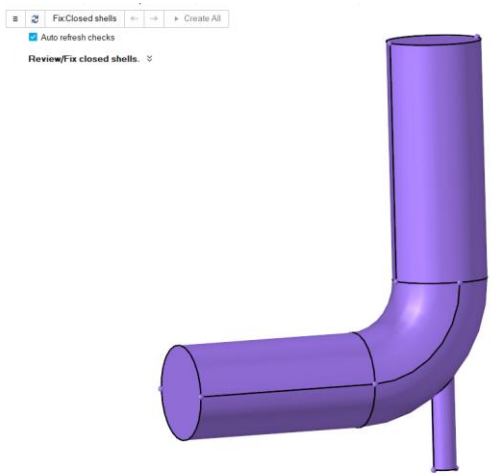


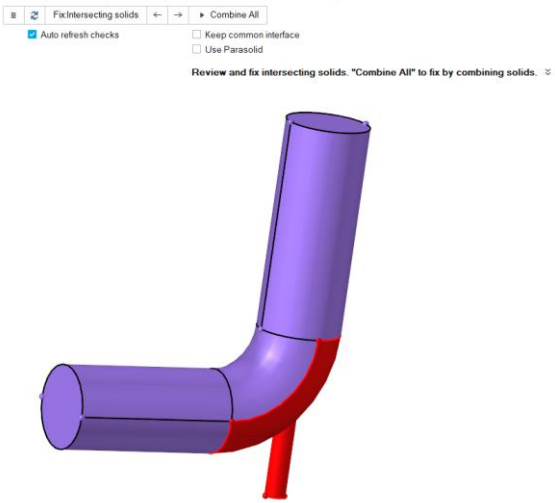
- From this menu you can solve the usual geometry problems to generate the volumes of the solids and fluids present in the problem.



- For simple surface-based geometries, such as the mixing elbow, it is usually sufficient to use Stitch All under Free Edges, Create All under Closed shells and, if required, Combine under Intersecting solids.

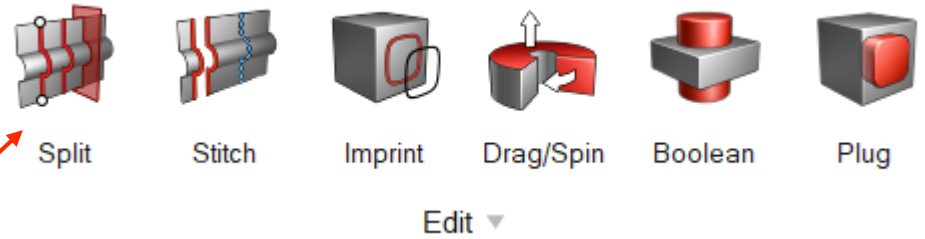


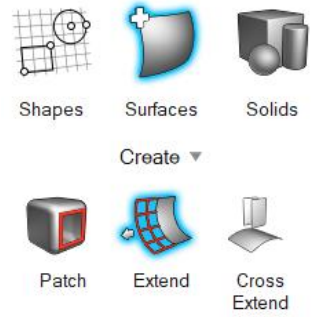


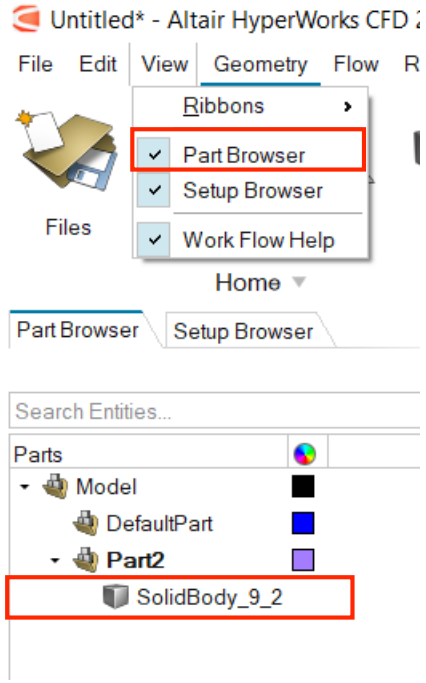


Repair and validate geometry

- In the Geometry ribbon there are several other tools for arranging the geometry or making modifications.





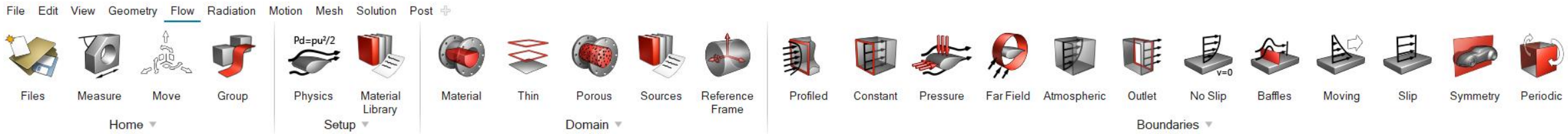


- The Split tool using planes for partitioning allows you to create internal surfaces with nodes, useful if you want to visualize results in certain planes inside the domain.
- Finally, check that the geometry is valid for meshing (Blue Tick on Validate) and that the modelled solids are correct (View>>Part Browser (Tick activated) and check them there).

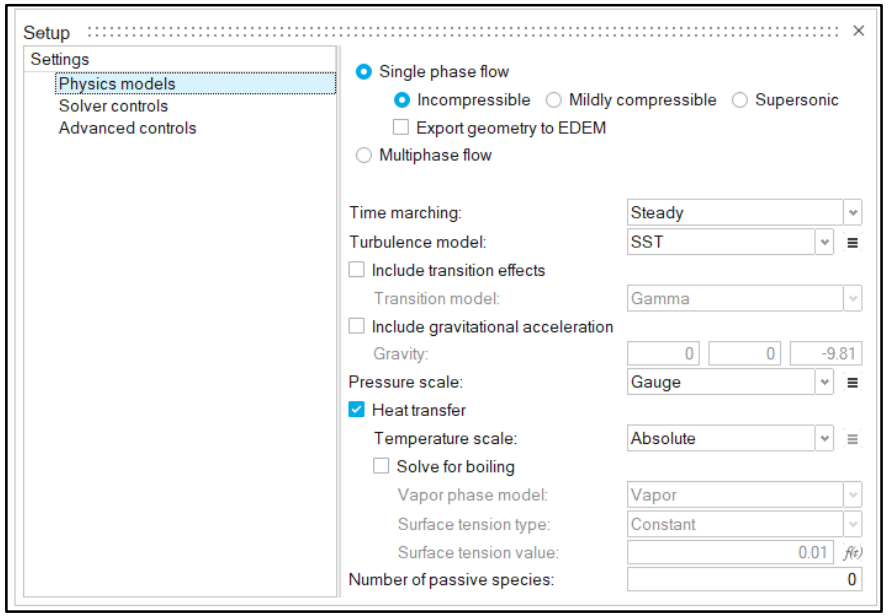


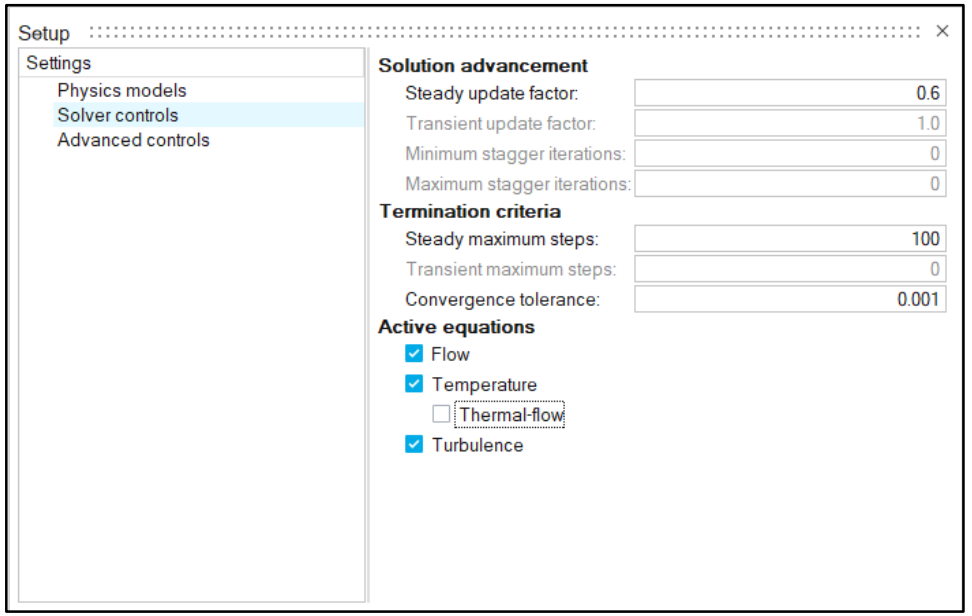
Flow: Physics

- Materials and boundary conditions are created and assigned in the Flow ribbon.



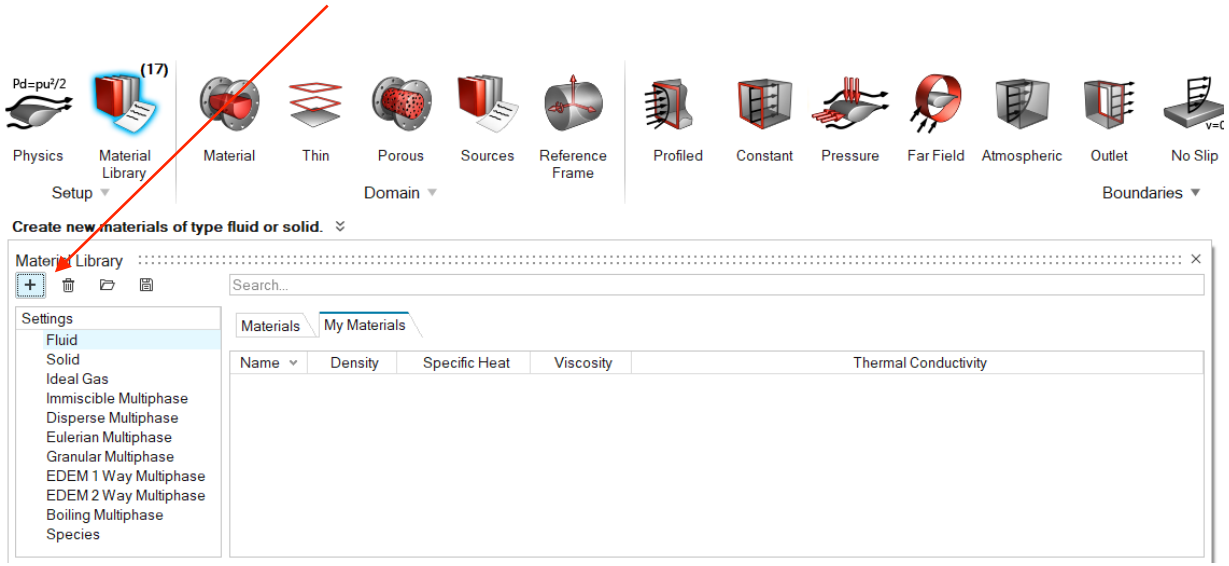
- In Flow>>Physics, in the Physics models section, select the type of fluid analysis (Incompressible, multiphase...), time marching (Stationary or transient), the turbulence model, whether heat transfer is to be considered and other model options.
- In the Solver controls section, select the active equations for the analysis and the solution acceptance parameters.
- For the Mixing elbow, the following options are selected:

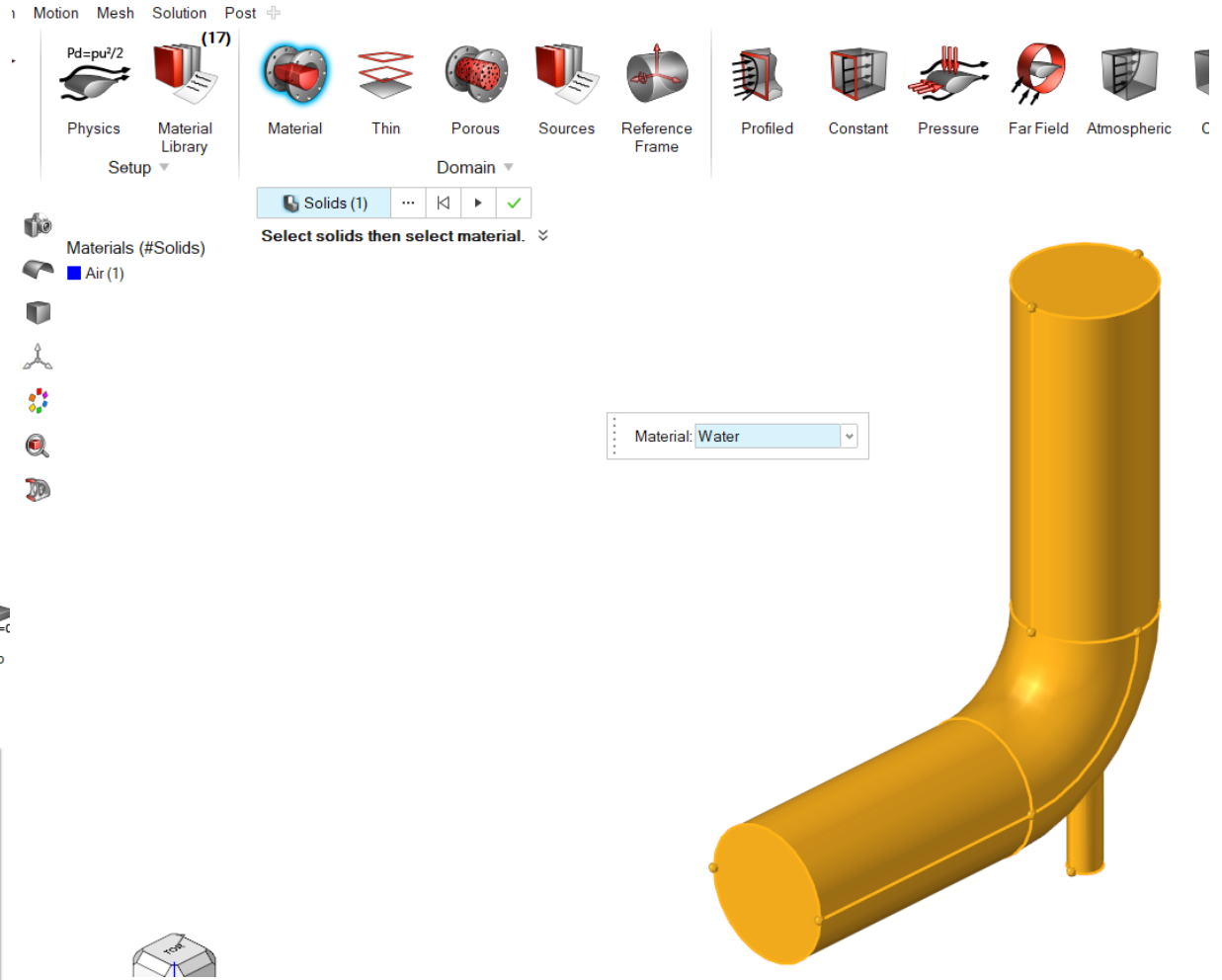




Flow: Materials

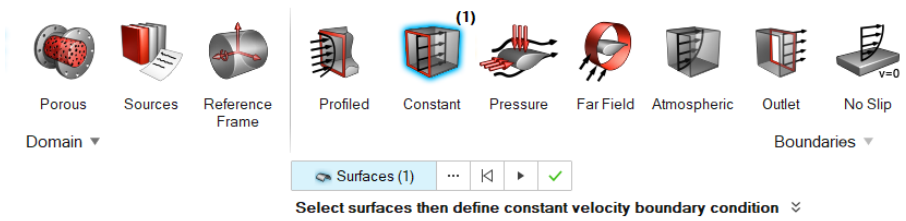
- In Flow>>Materials, the materials are assigned to the volumes.
- In this case, all the elbow volume is made of water.
- Defining a material that is not included in the Acusolve default library can be done from Flow>>Material Library by selecting My Materials, clicking in the + and entering the necessary data (Density, Specific Heat, Viscosity and Thermal Conductivity).

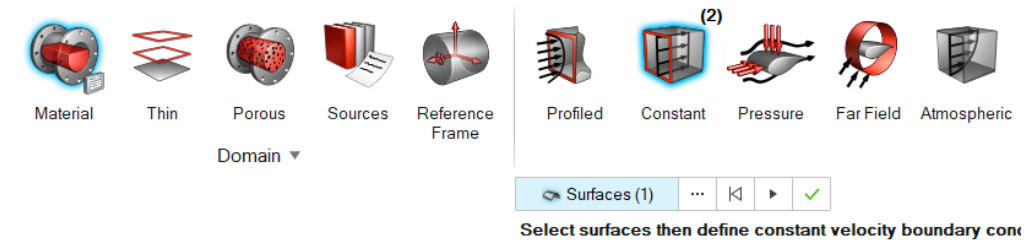


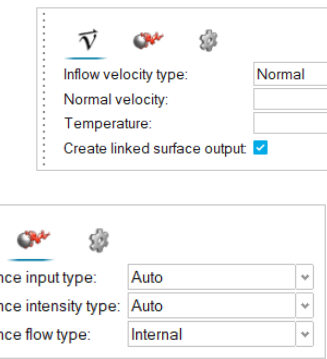


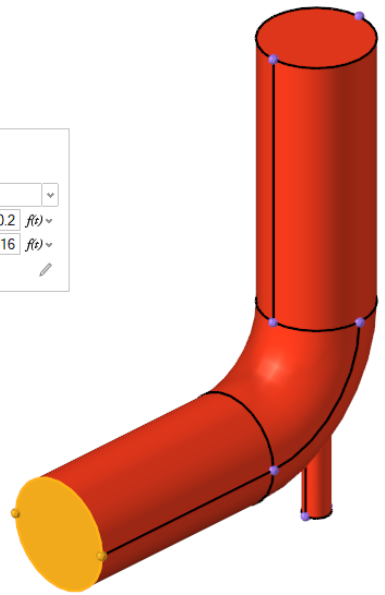
Flow: Boundary Conditions. Inlets.

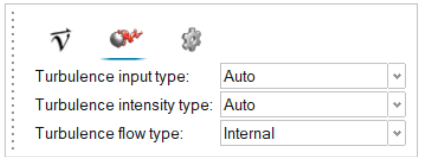
- To define the flow inlets, the constant inlet option is selected from the Boundaries section in the flow ribbon.
- Select Inflow velocity type: normal and enter the inlet velocities and temperatures specified in the first slide.
- At the turbulence tab, you can enter the turbulence intensity (https://www.cfd-online.com/Wiki/Turbulence_intensity) by selecting value in the second drop-down menu or directly by selecting automatic for internal/external flow. In this case internal is selected for both inlets.

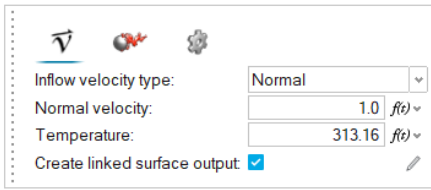


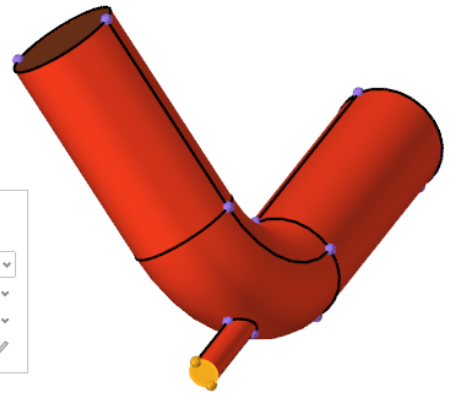




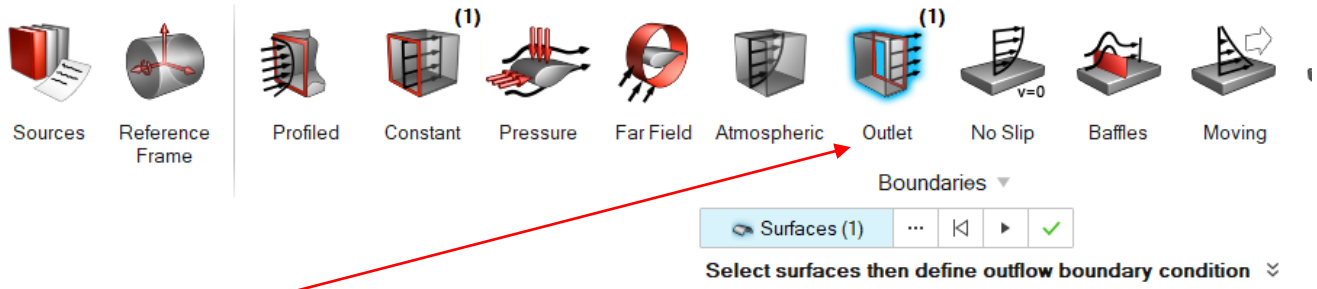


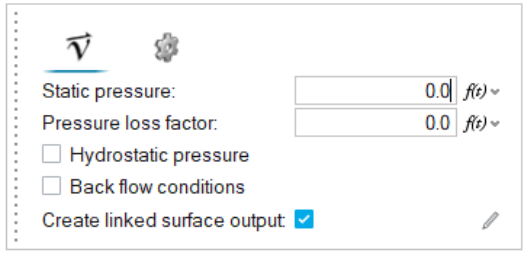


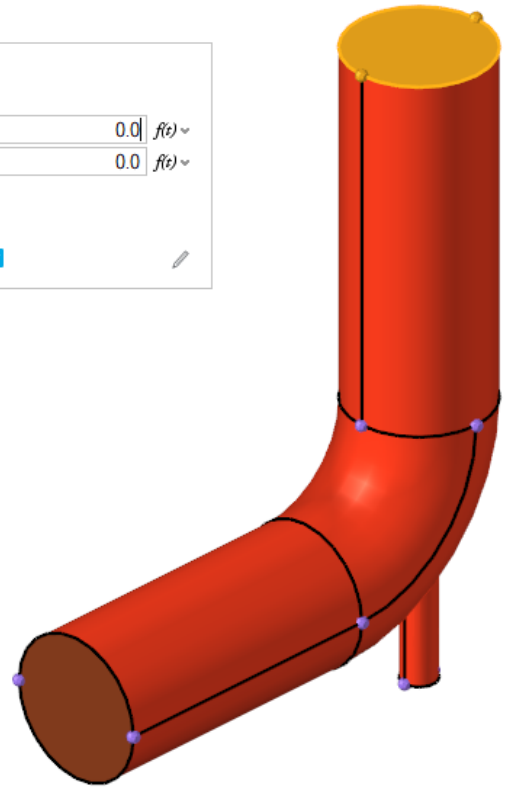




Flow: Boundary Conditions. Outlet.



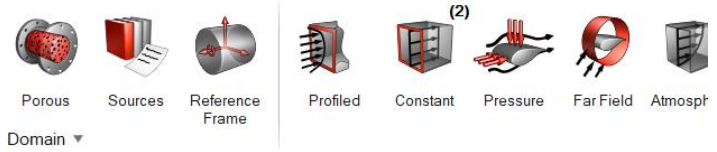


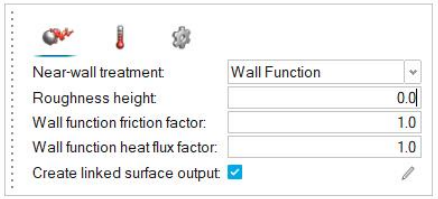


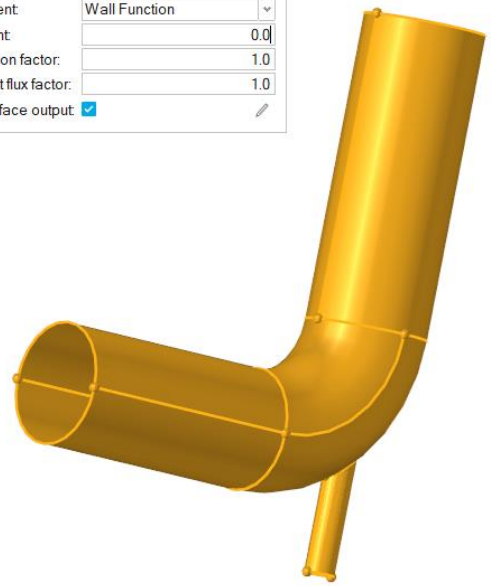
- To define the outlet, this option is selected from the Flow ribbon and the corresponding surface is chosen.
- As stated in the first slide, static pressure of 0 Pa is set to this outlet.

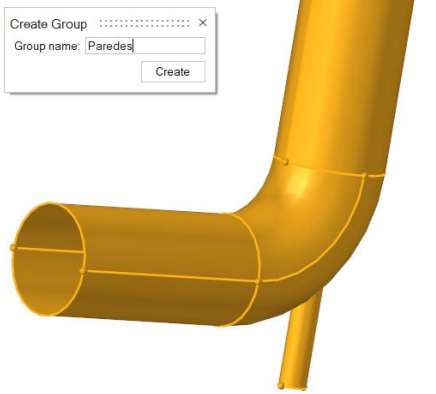
Flow: Boundary Conditions. Walls.

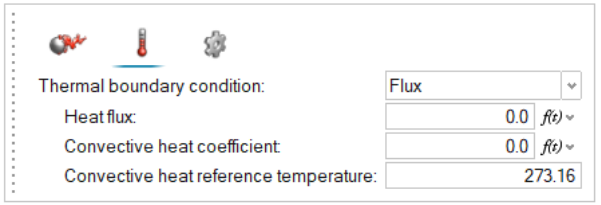
- To define the last boundary conditions, it is useful to hide the inlets, outlets, etc. and select the remaining surfaced by dragging the mouse on the screen with the left mouse button clicked.
- For the example, No-Slip Walls are used as BC.
- The surfaces are hidden by selecting them directly or in their groups and pressing the "h" key. To show them again, select them and press "a".
- In the No-Slip Wall properties menu, the flow and thermal conditions are selected.
- For the Mixing Elbow, these are left as shown in the right picture.
- With the surfaces selected, it is convenient to do Right click>>Create Group and give it an appropriate name to select these surfaces more easily in later stages.





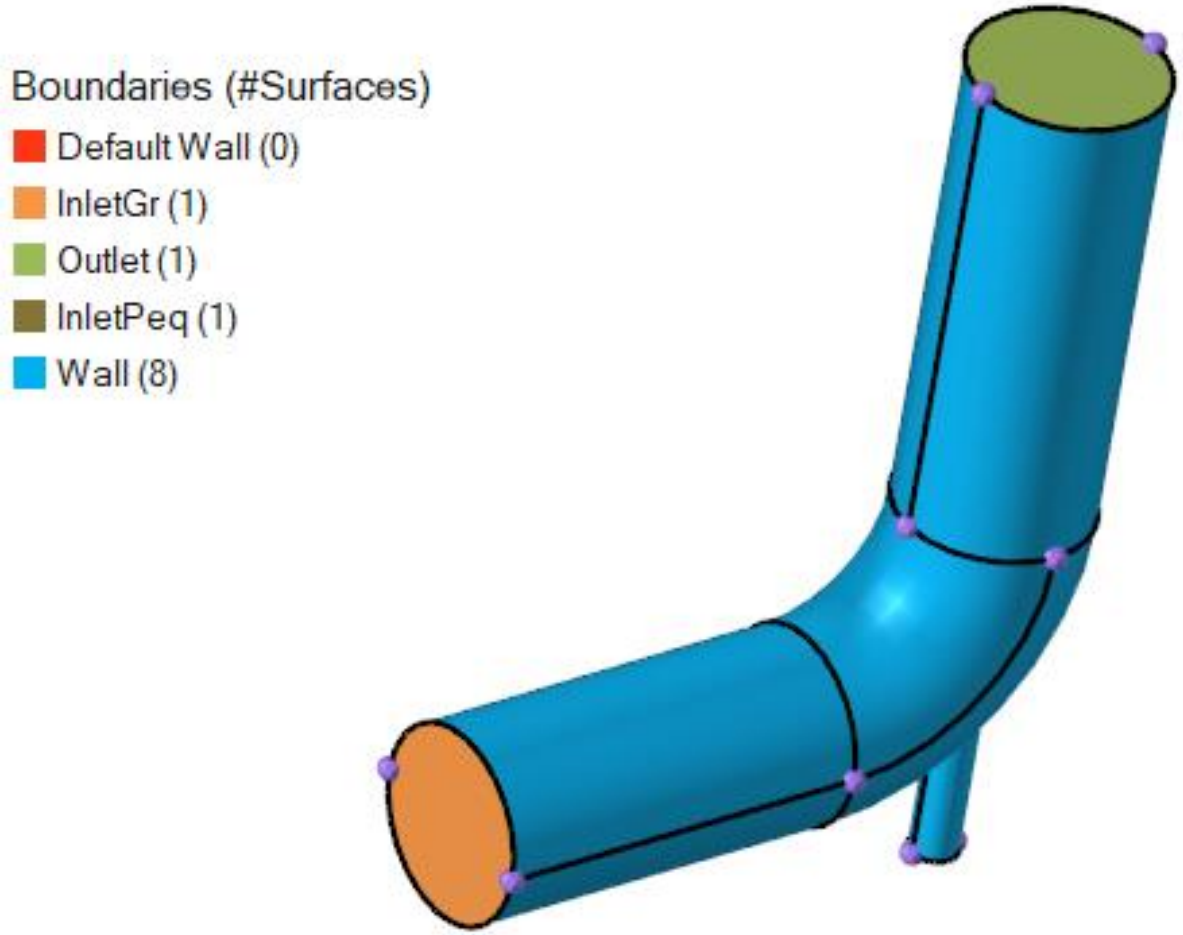






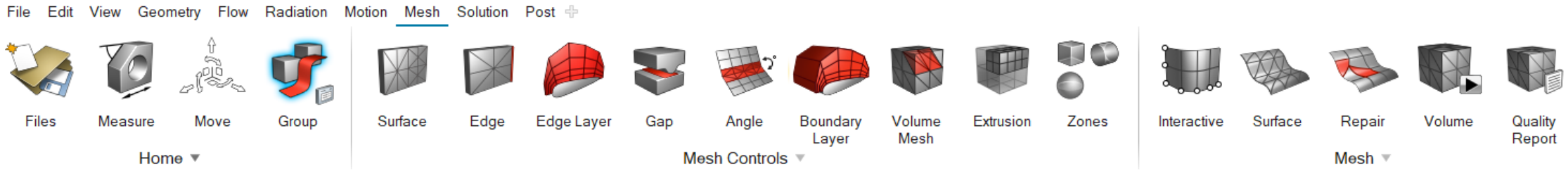
Flow: Boundary Conditions

- From the Setup Browser (View>>Setup Browser with Blue Tick) you can see the assigned boundary conditions.
- They can be modified by right clicking on them and selecting Edit. They can also be renamed by pressing Rename.
- Pressing Edit and then Esc once you get the color scheme on the right, useful for checking the selection.
- If any surface has no assigned condition it is in the Default Wall collector.
- By default Hyperworks CFD treats exterior walls as Adiabatic No-Slip and interior walls as non existing (Flow ignores them as they are not a real boundary).

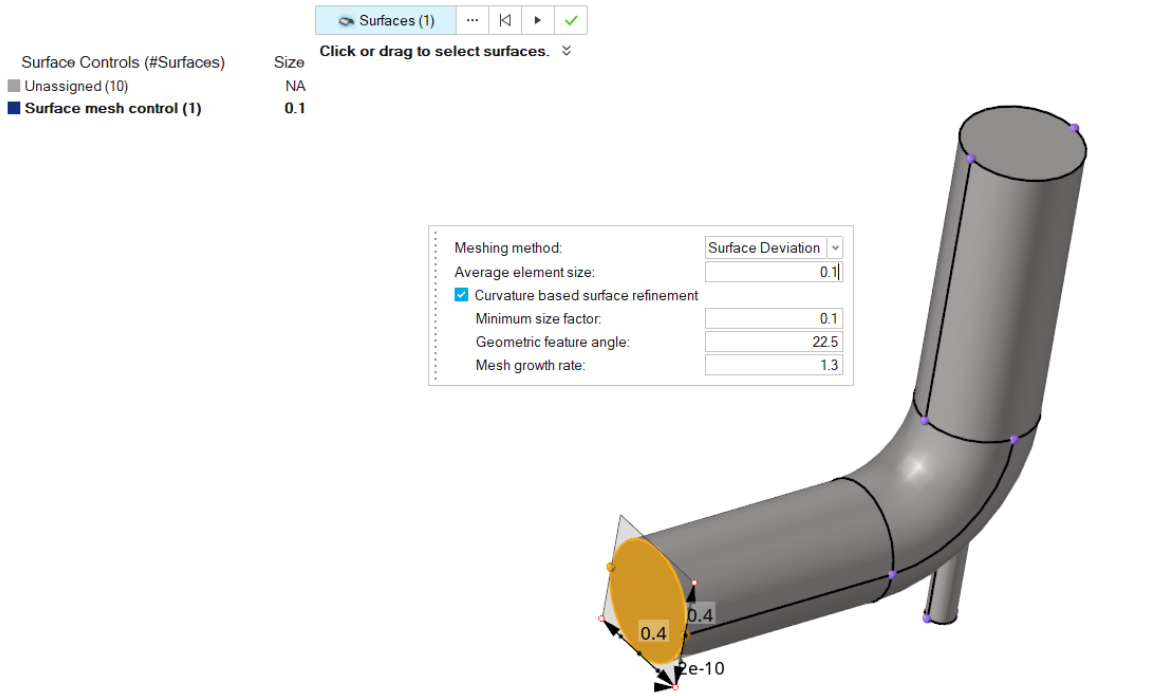


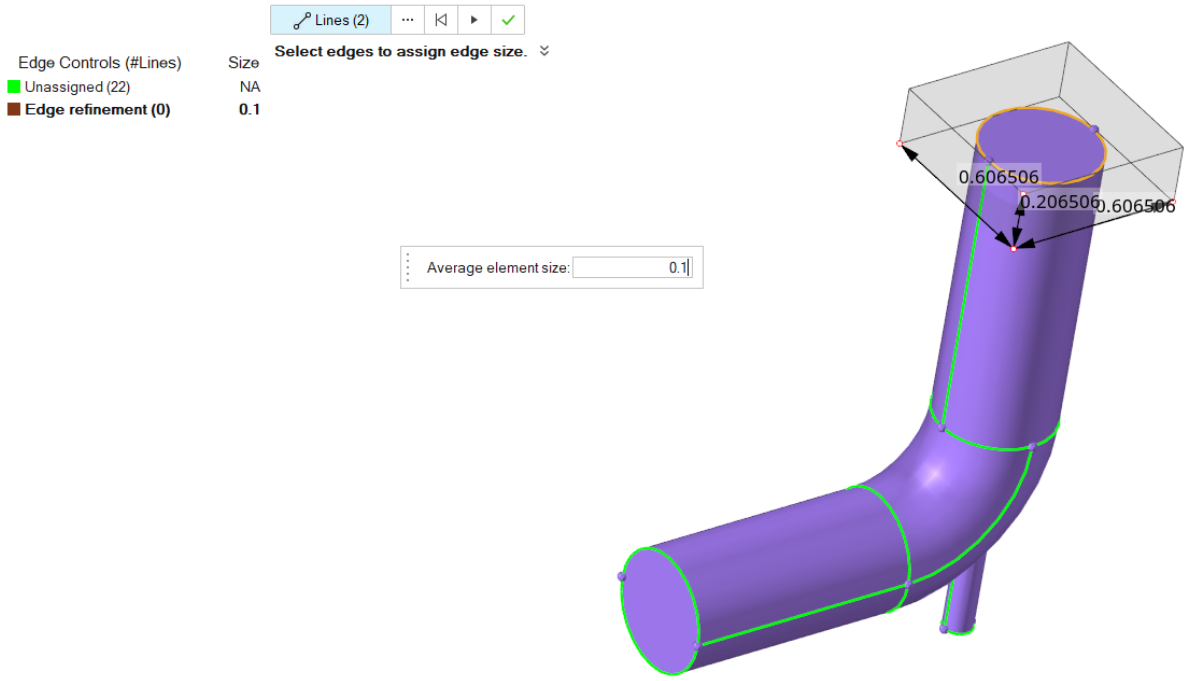
Mesh: Mesh controls.

- In the Mesh ribbon, the mesh controls are defined and the 3D mesh is generated.



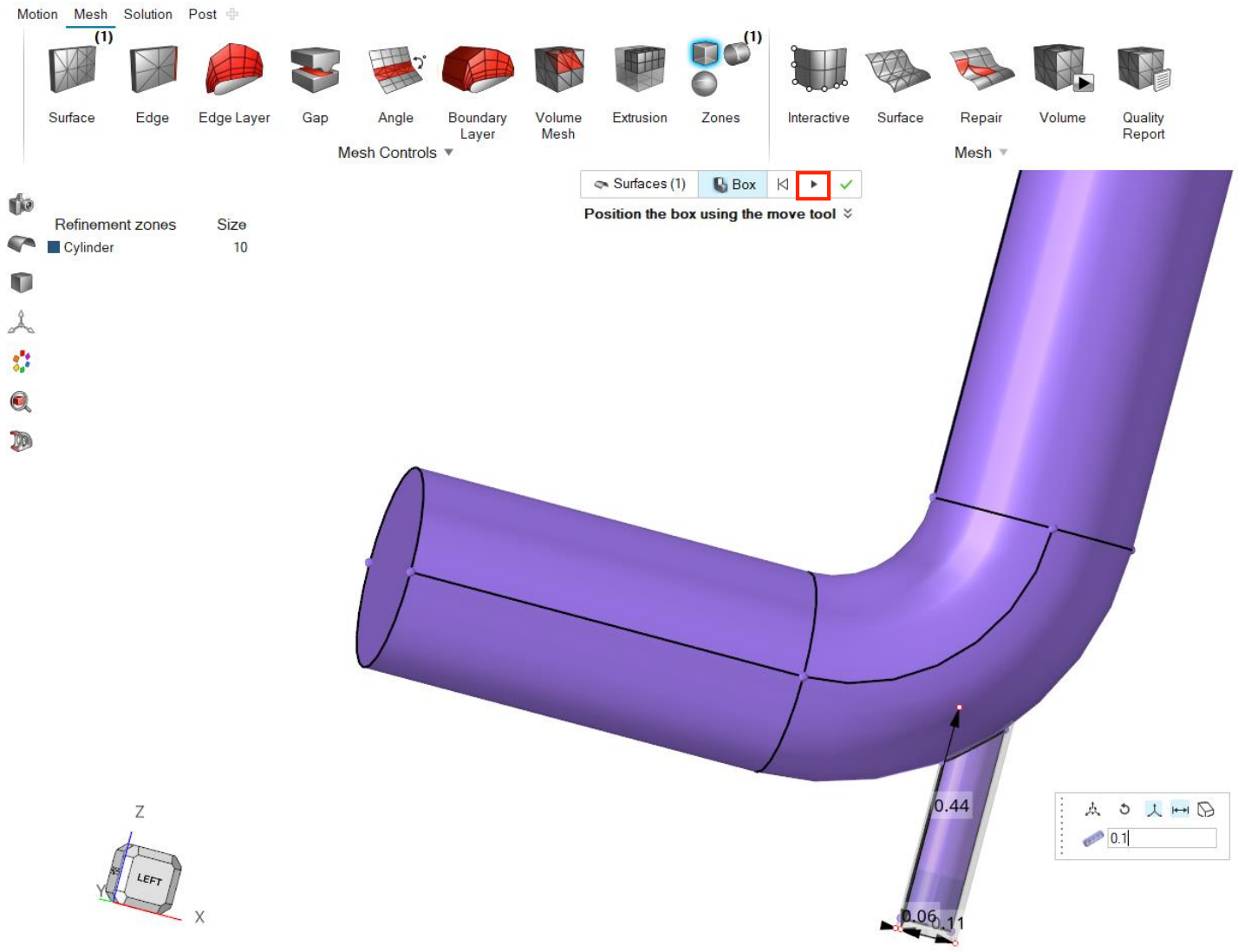
- The Surface and Edge tools in Mesh Controls are used to define the average element size on surfaces and edges where finer or coarser meshes are needed.





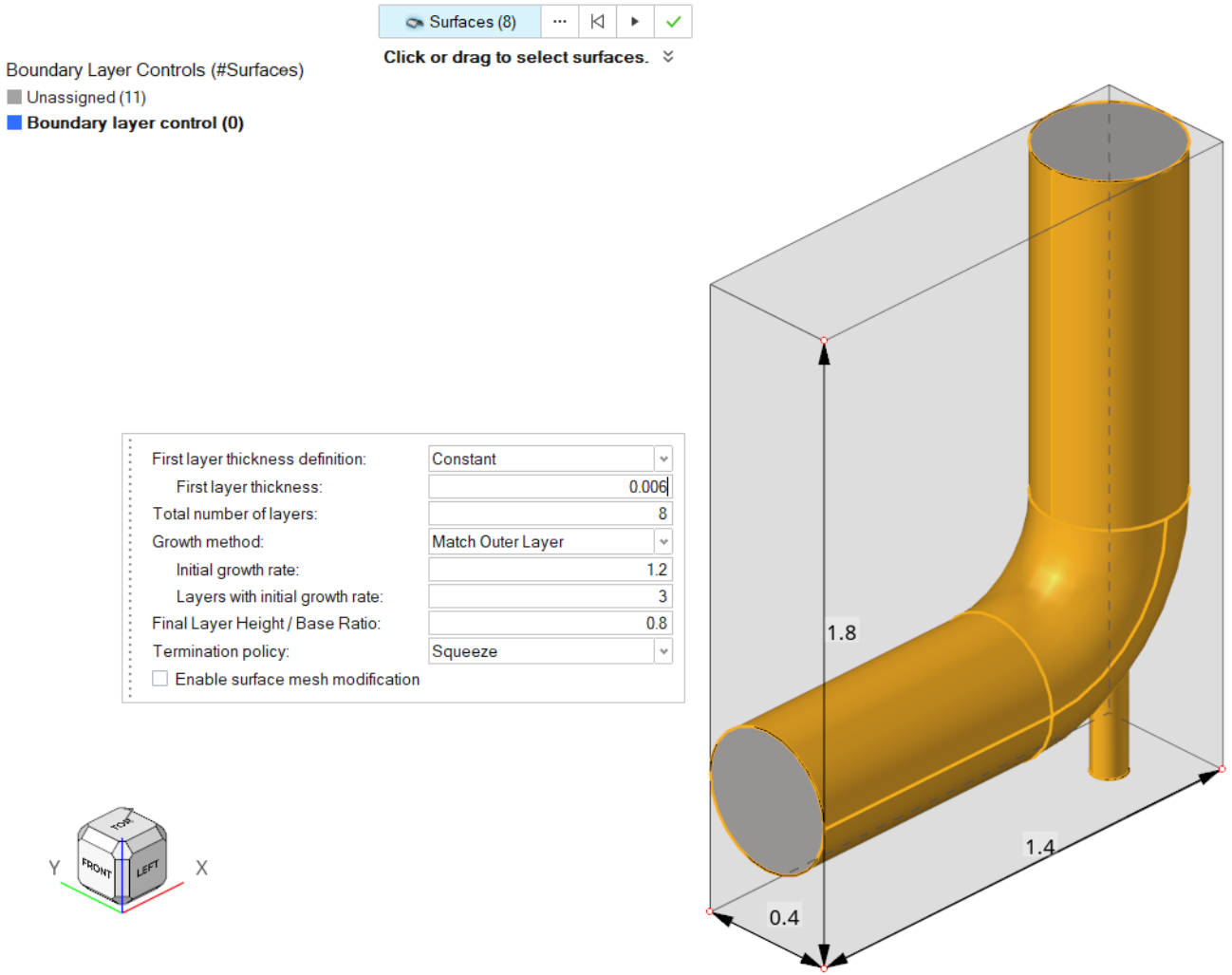
Mesh: Refinement zones

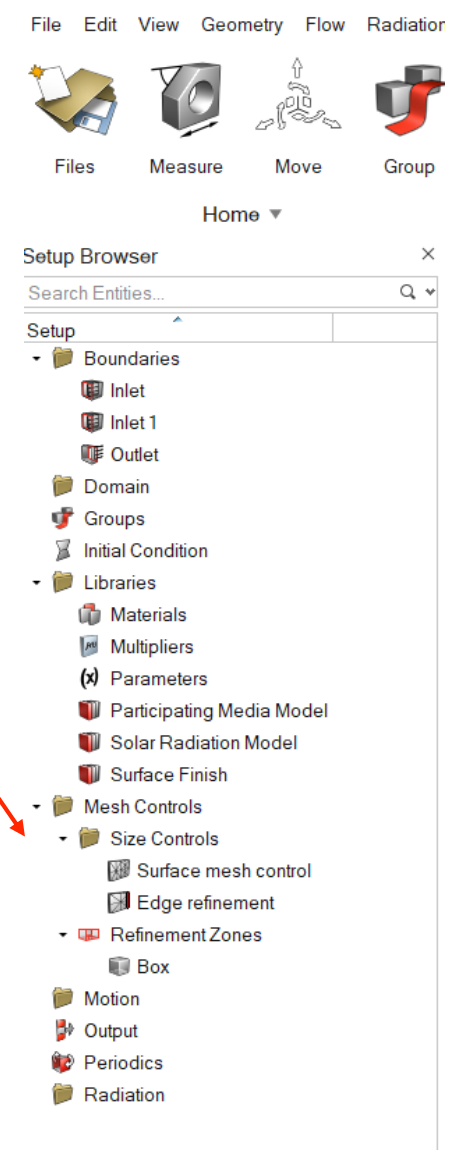
- The Zones tool in Mesh controls is used to define volumes where a finer mesh is required. The element size in this region can be set and edited in this menu.
- The refinement zone can be defined by its limiting surfaces or by setting the limits manually.
- In the current interface, to create the zone, click on the black arrow besides the box selector before clicking on the green tick.
- By double clicking under size, in the left menu, the size of the element in the region can be changed.



Mesh: Boundary Layers

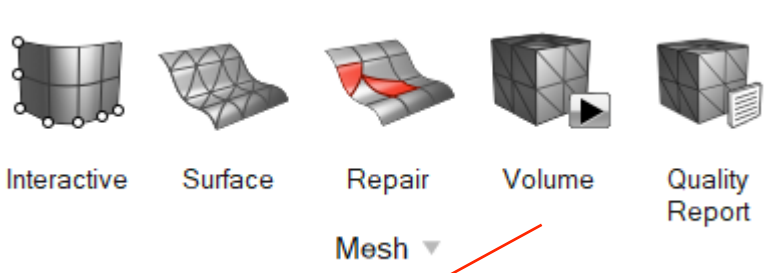
- The Boundary Layers tool in Mesh Controls allows to define the boundary layer mesh with a thickness and number of layers. In this example it can be left undefined.
- The set of mesh controls defined for the analysis can be checked in the Setup Browser.

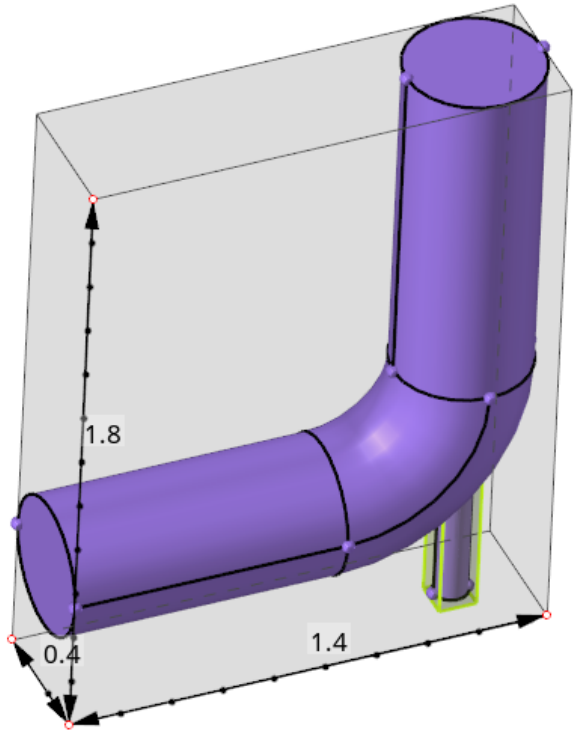


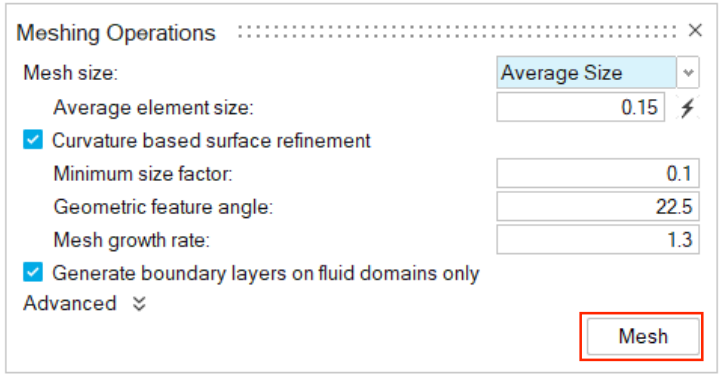


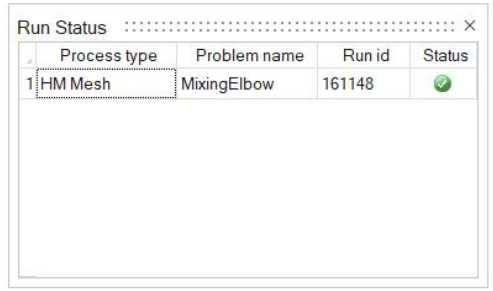
Mesh: Volume mesh

- After defining the appropriate mesh controls, select Volume in the Mesh ribbon and choose the average element size for the regions without prior definition.
- Press Mesh to generate the mesh.
- **Important:** To modify the mesh after creating it, it is advisable to delete the previous mesh by clicking on the red X that appear above the Volume and/or Surface tools.



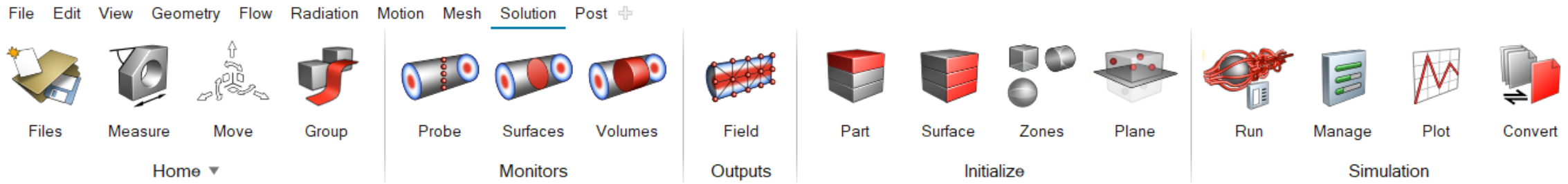




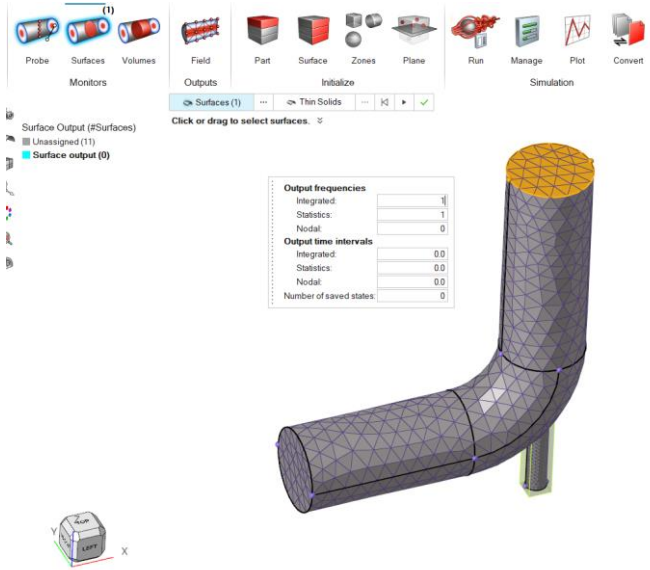


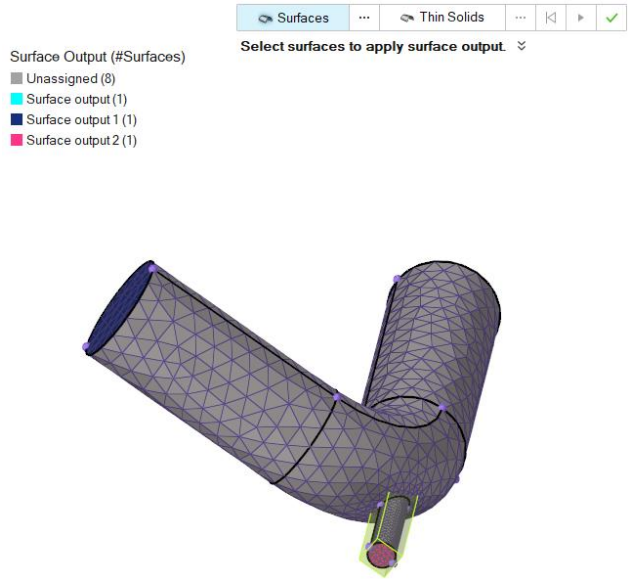
Solution: Monitors

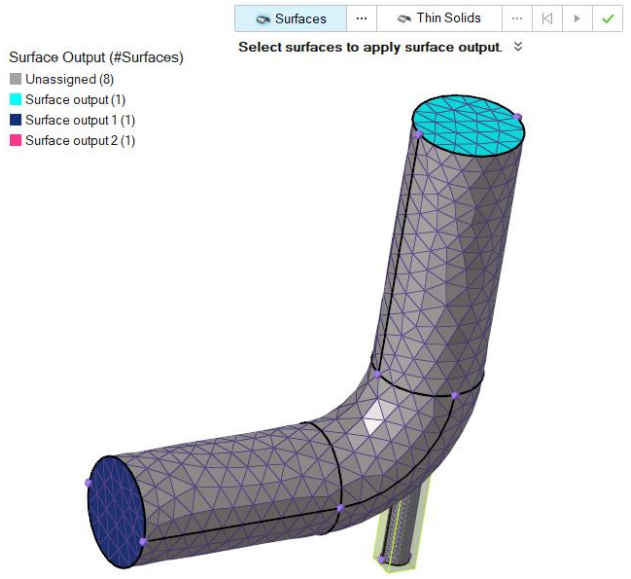
- On the Solution ribbon, the initial conditions of the simulation are set and the output results are requested.



- The tools in Monitors can be used to define points, surfaces and volumes where averaged and/or integrated results are required. In the example, the inlets and outlet surfaces are selected. These can be edited in the Setup Browser.

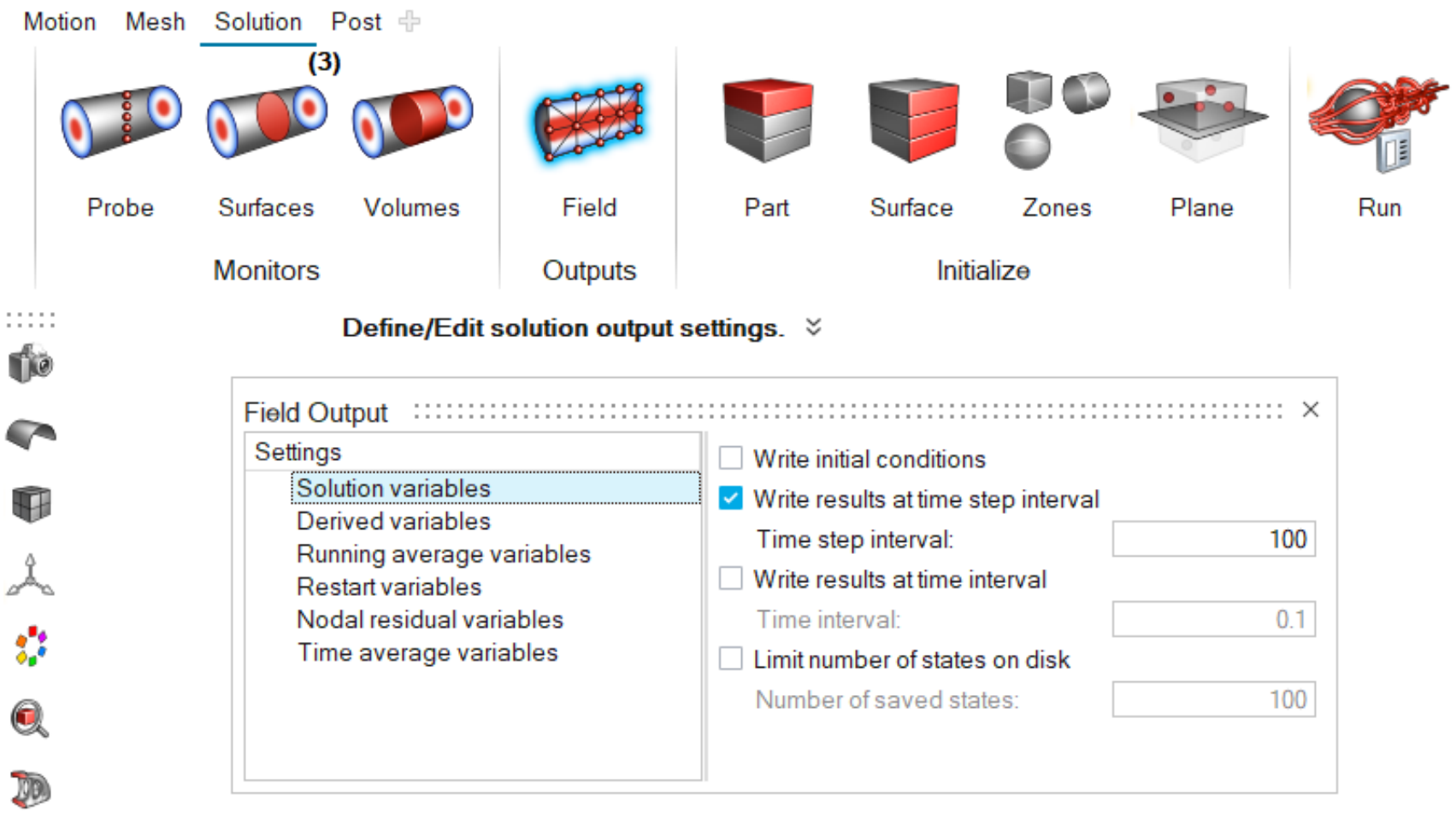






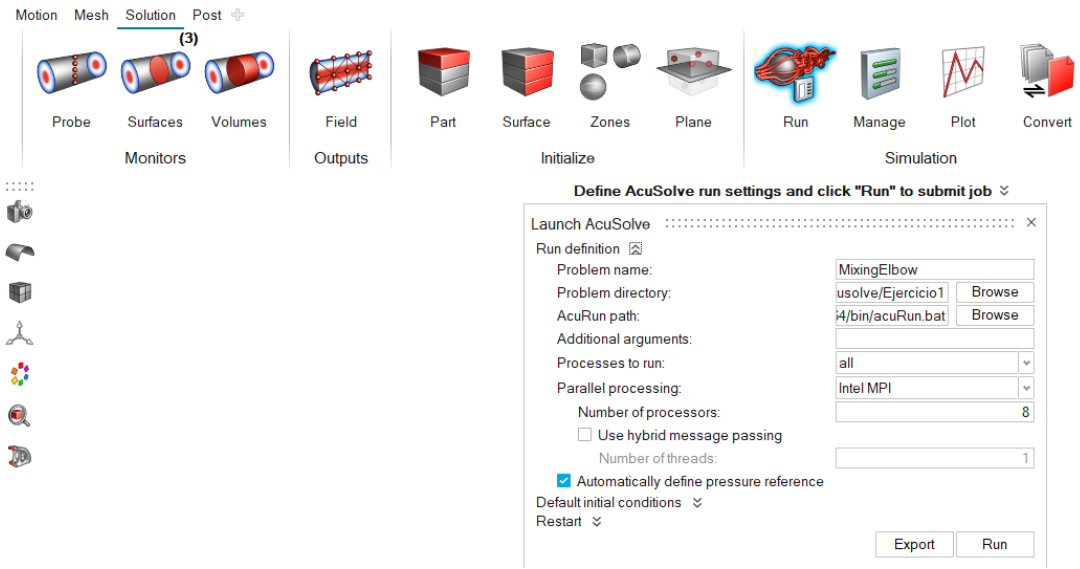
Solution: Field Outputs

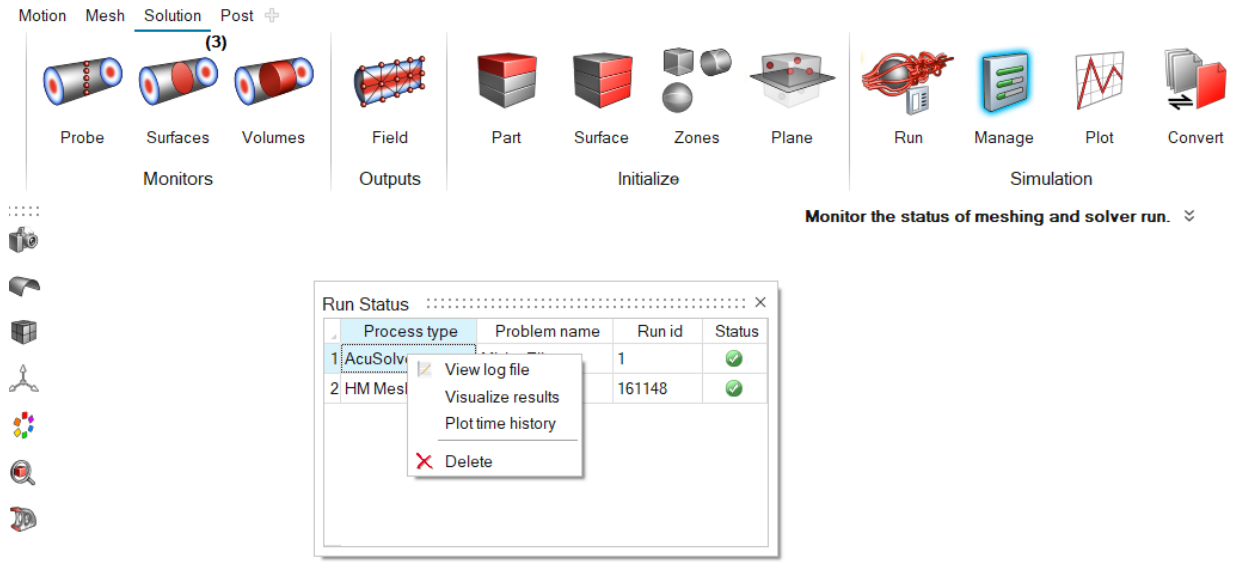
- The tool Outputs>>Field is used to define the writing intervals for the different outputs.
- For the example, the default values can be left as follows



Solution: Run & Export

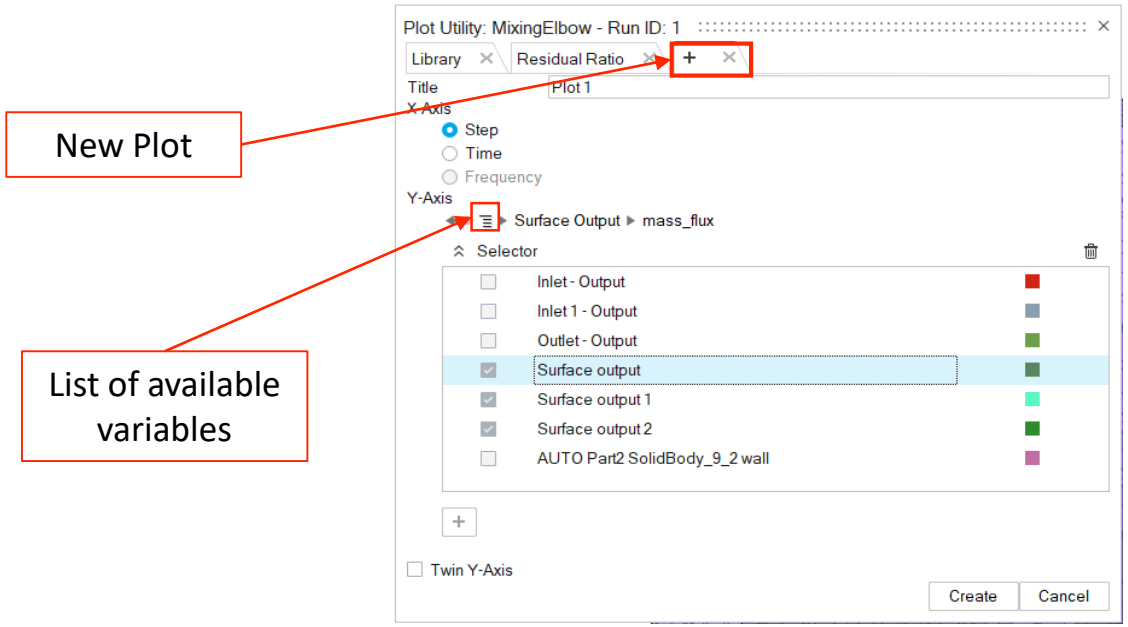
- Select the Run tool to access the launcher interface. In Problem Name the name of the files generated in the simulation is selected.
- The Problem Directory is the folder where the results will be generated and Acurun Path is the absolute path to the acuRun.bat file of the Hyperworks CFD installation.
- Under Parallel processing, Intel MPI and then the number of processors to use are selected. In this menu it is possible to define the default initial conditions on the set of nodes where they have not been previously specified.
- Pressing export generates the .inp file and the MESH.DIR folder needed to launch the simulation from the Acusolve CMD Window. Press Run to start the analysis. When the analysis is finished, the results can be loaded in the Post tab of the job manager by right clicking on the simulation>>Visualize results.

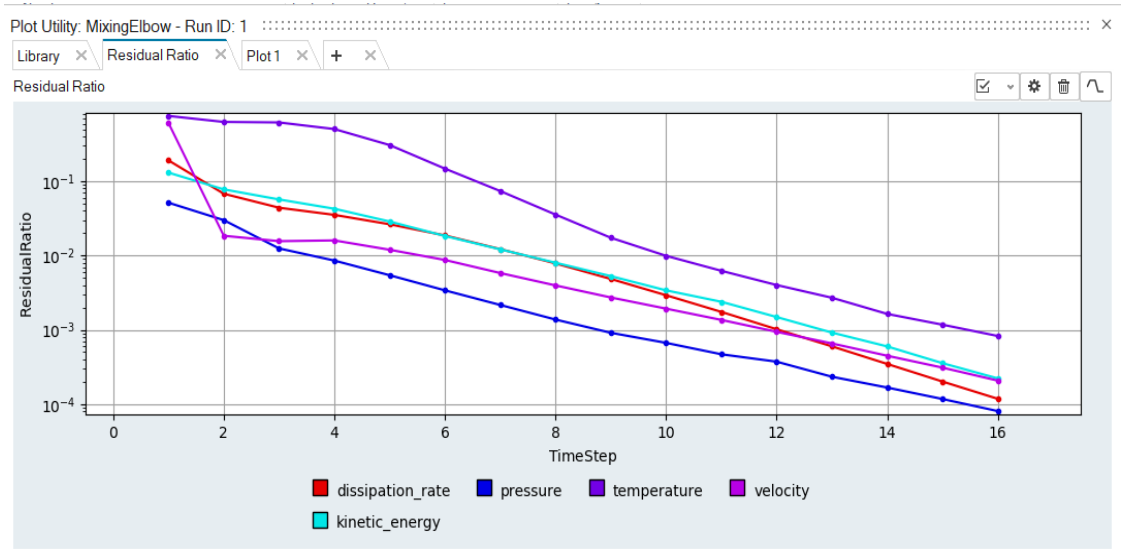


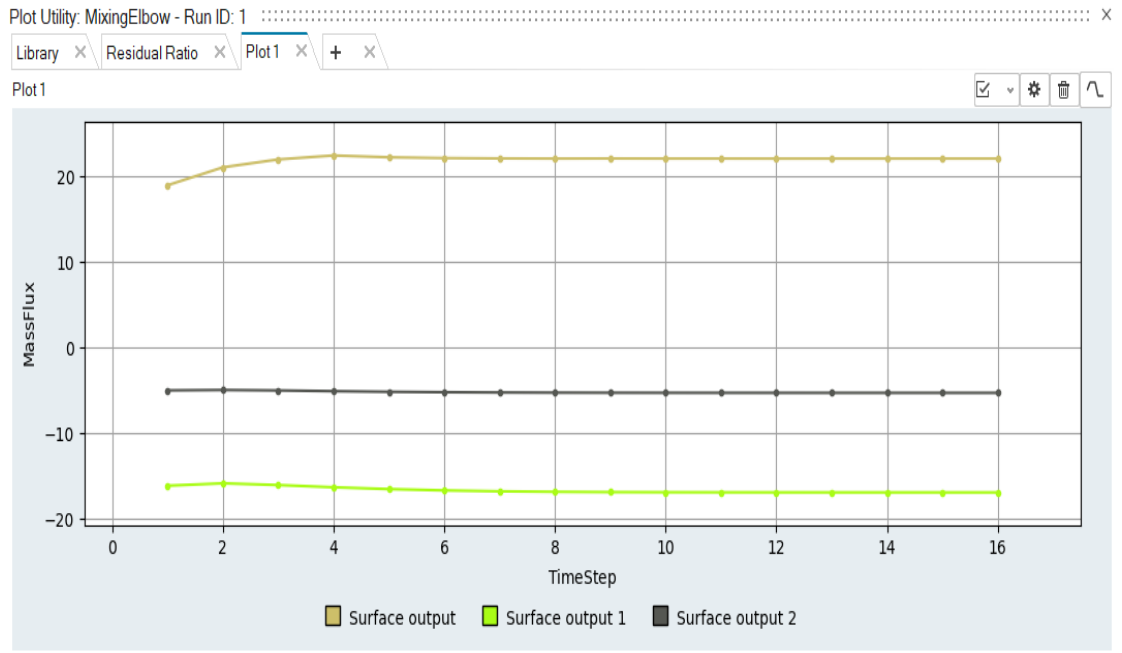


Solution: Plot

- The Plot tool can be used to visualize the convergence and error graphs during the simulation.
- Residual Ratio shows the deviation of the solutions with respect to the problem equations and Solution Ratio marks the relative variation of the solution at each time step, to check its stability.
- From this tab it is also possible to generate graphs of average and integrated magnitudes in monitored surfaces, for example the mass flow in inlets and outlets to check their stability and the mass balance of the whole domain.

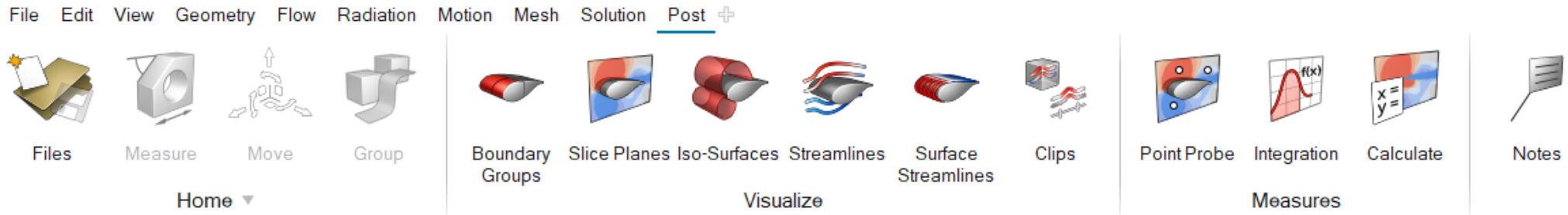






Post: Load results

- The Post ribbon contains the tools used to visualize the results of a simulation:



- To load results, select the folder icon on the ribbon or go to File>>>Open>>>Results and choose the .log file generated in the simulation. This calls other files generated in the simulation folder and in the subfolder ACUSIM.DIR, which must be in the same file storage location.
- When opening the .log file, you can directly visualize the results on the surfaces by right clicking on them in the Post Browser and selecting Edit. You can make contour plots and vector plots among other possibilities. These options are the same as in the Slice Planes and are explained in the following slides.

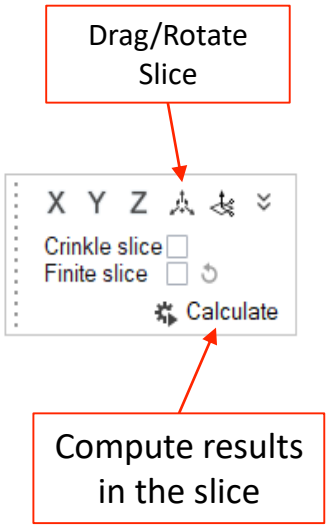


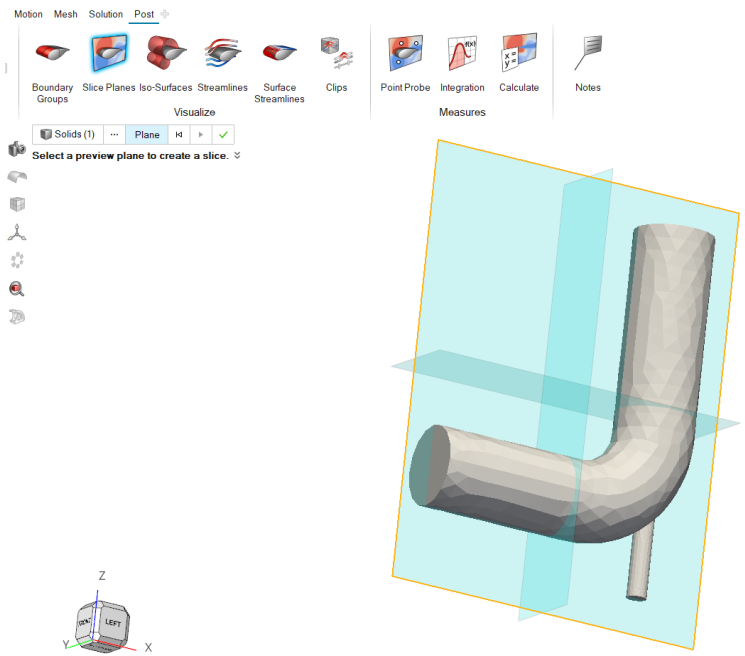


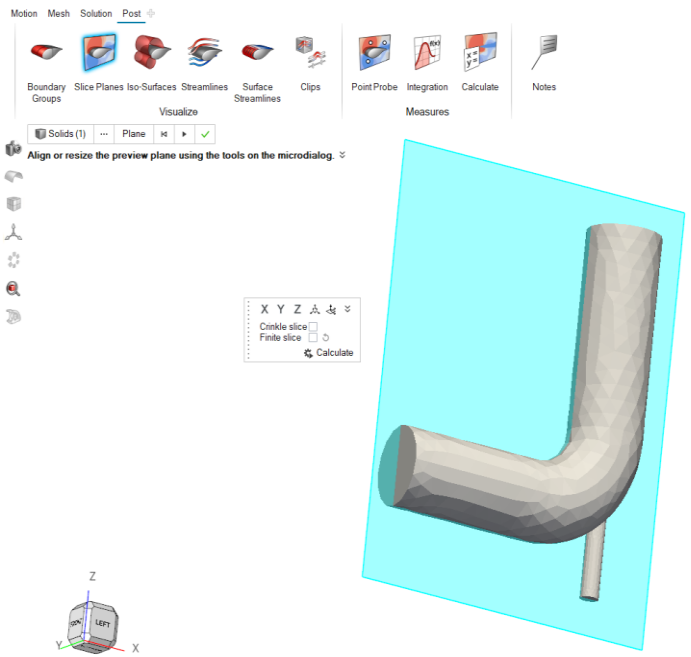
- To save display results associated with the .log, go to File>>State>>Save State. The option Open State can be used to load them later. Keep this file in the same folder as the simulation .log file.

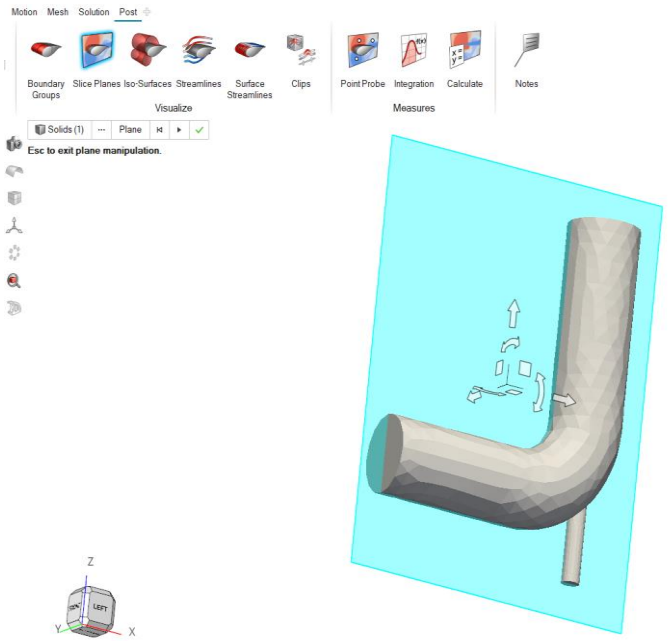
Post: Surfaces & Slice Planes

- The Slice Planes tool allows you to create slices to visualize results on surfaces not defined in the pre-process. For the example, select the middle plane shown in the images.
- The planes are defined starting from the ones proposed by the software when selecting the Slice Plane option in the Post ribbon. This planes can be dragged and rotated as desired. The results are computed by pressing the Calculate button (to exit the plane transformation mode, press Esc twice).
- As with the surfaces, by clicking Edit on them in the Post Browser you can change the displayed results.
- Selecting over the Slice Plane: Right click>>Animate generates an animation of the plane sweeping the volume in the direction of its normal.



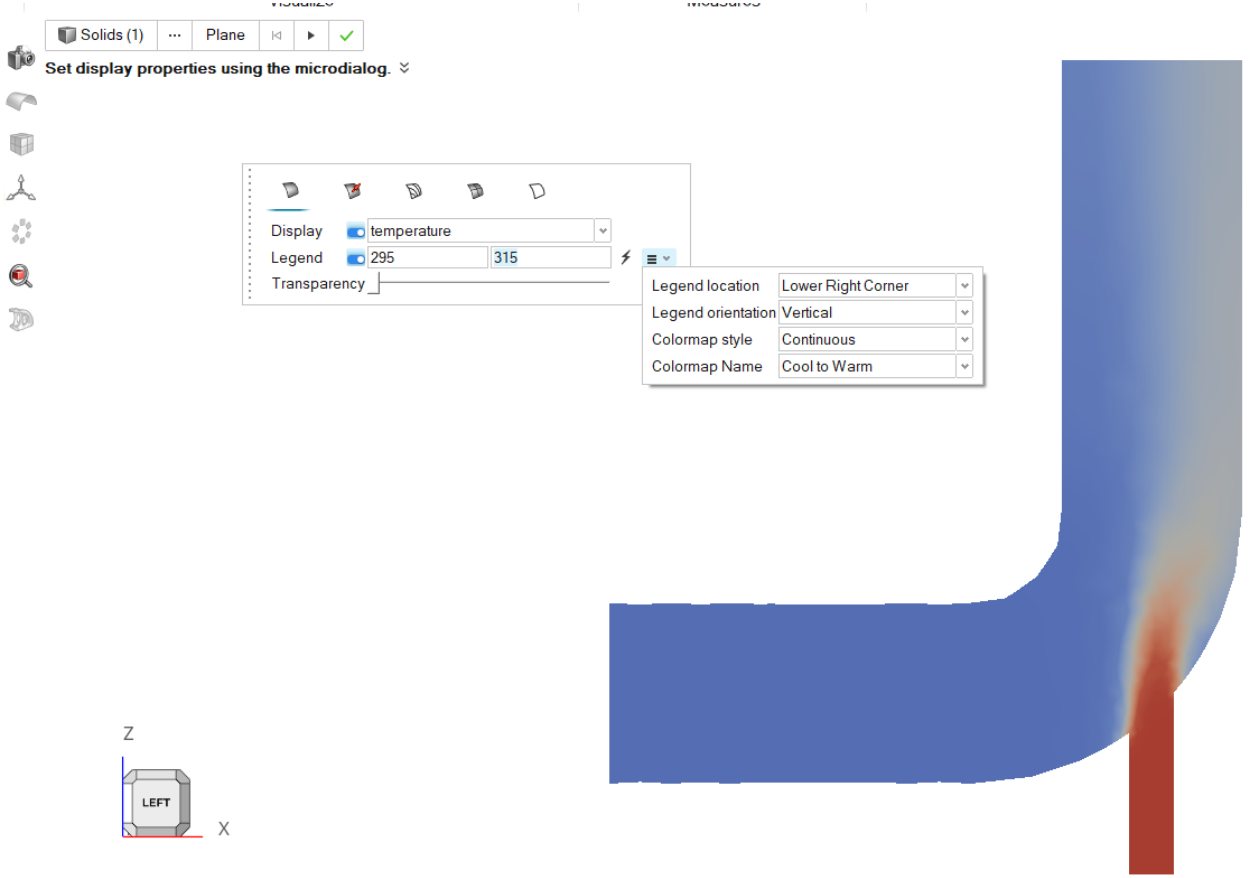


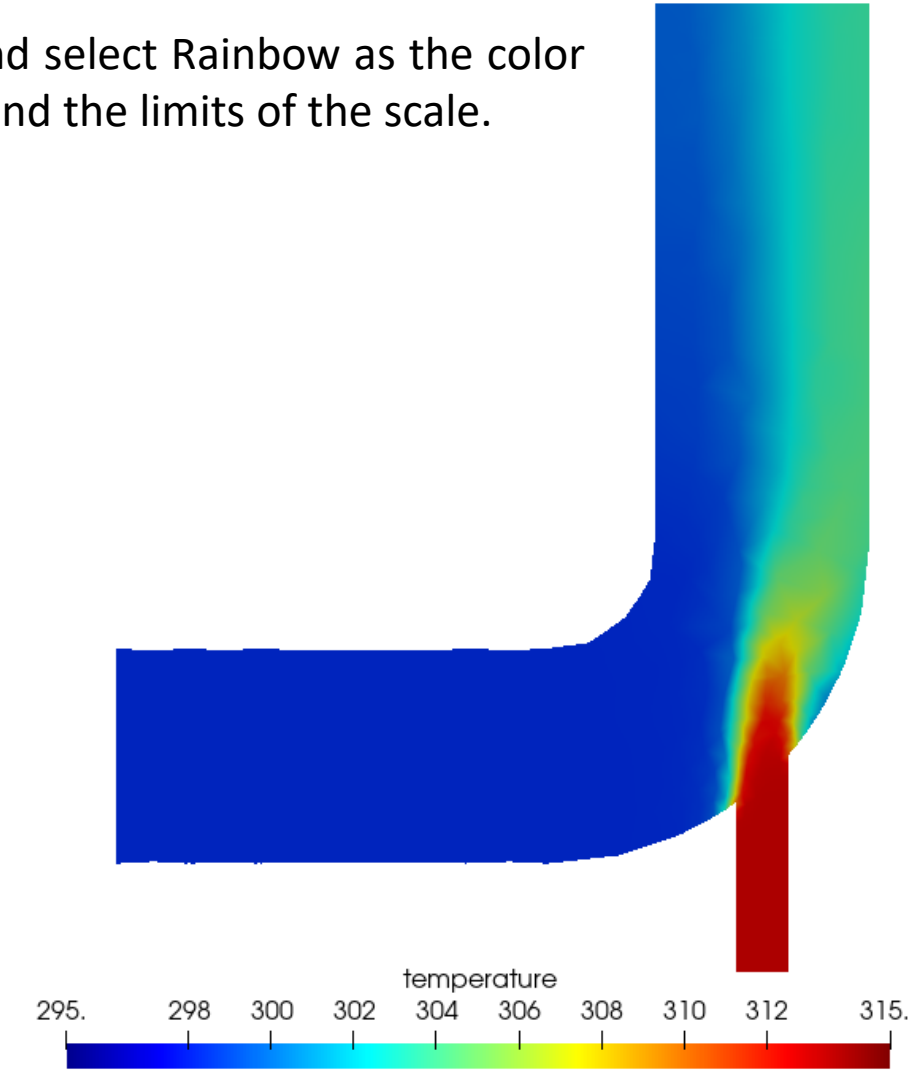




Post: Surfaces & Slice Planes

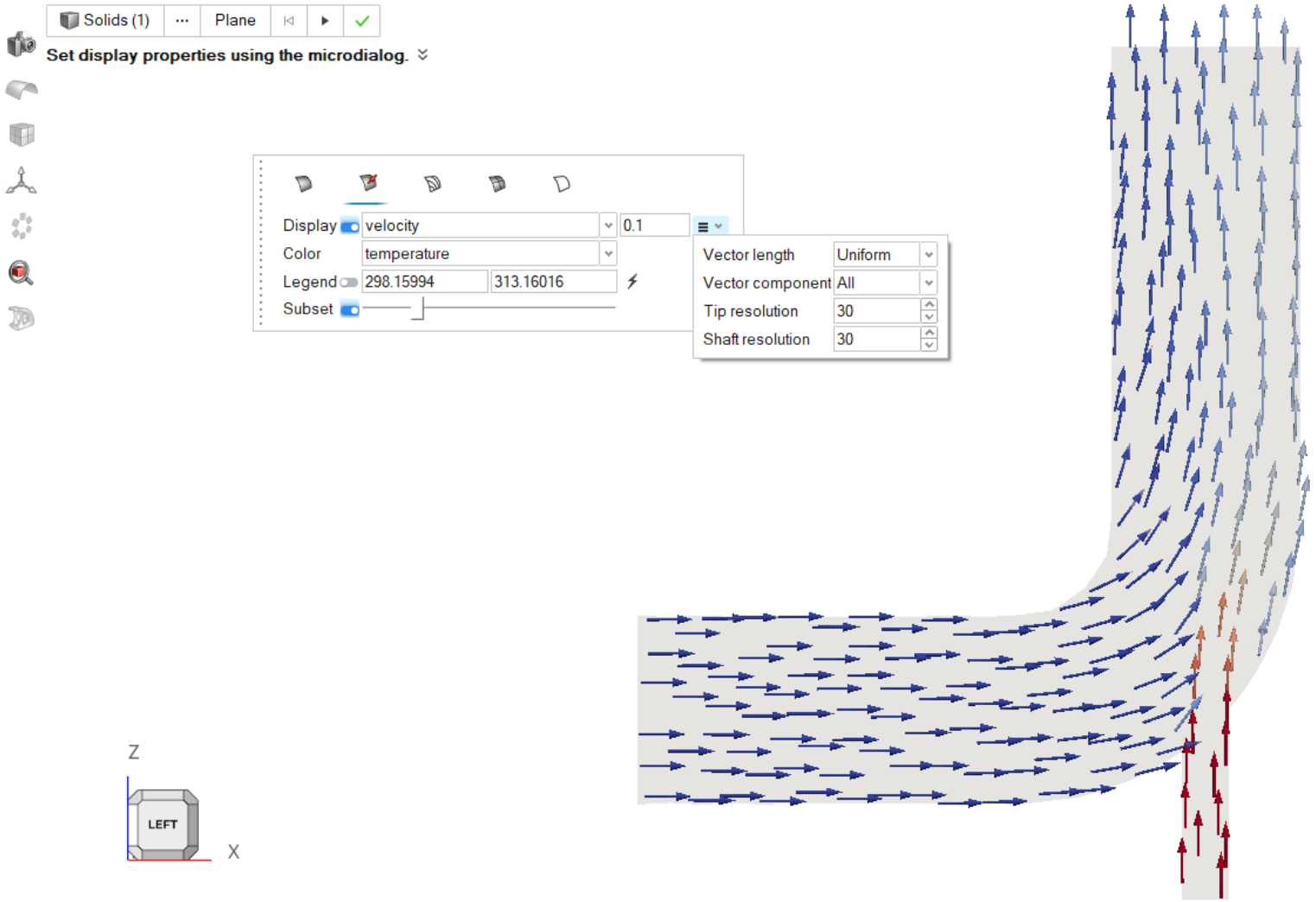
- On the created Slice Plane, choose Temperature as variable under Display and click on the selector next to legend.
- To choose the contour format, open the drop-down next to legend and select Rainbow as the color map. The other options are used to define the location of the legend and the limits of the scale.





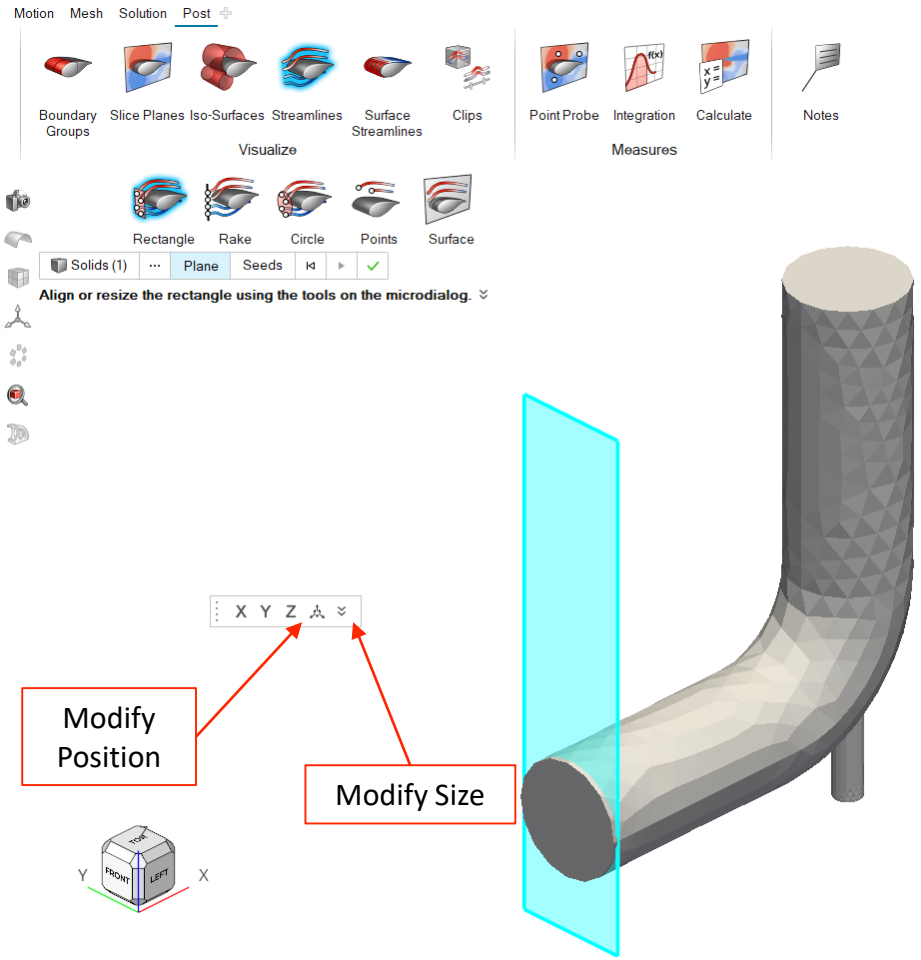
Post: Surfaces & Slice Planes

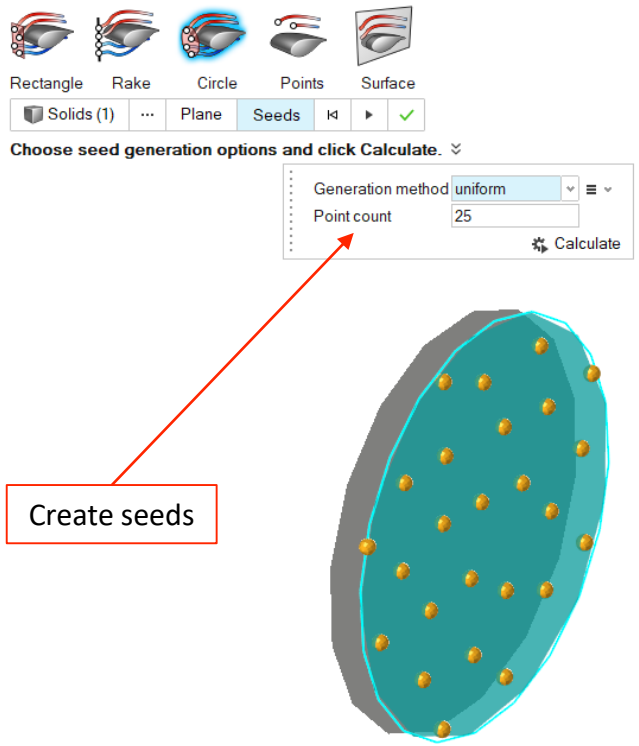
- Within the menu that appears when selecting Edit, the second tab creates the vector plots.
- The most common one is the velocity vector plot. A suitable scale must be chosen with the factor next to velocity. Selecting vector length: Uniform helps the correct display.
- Color sets the variable used to color the vectors. Temperature is chosen in this example.
- The subset option is very useful to change the number of arrows represented, which is often excessive by default.

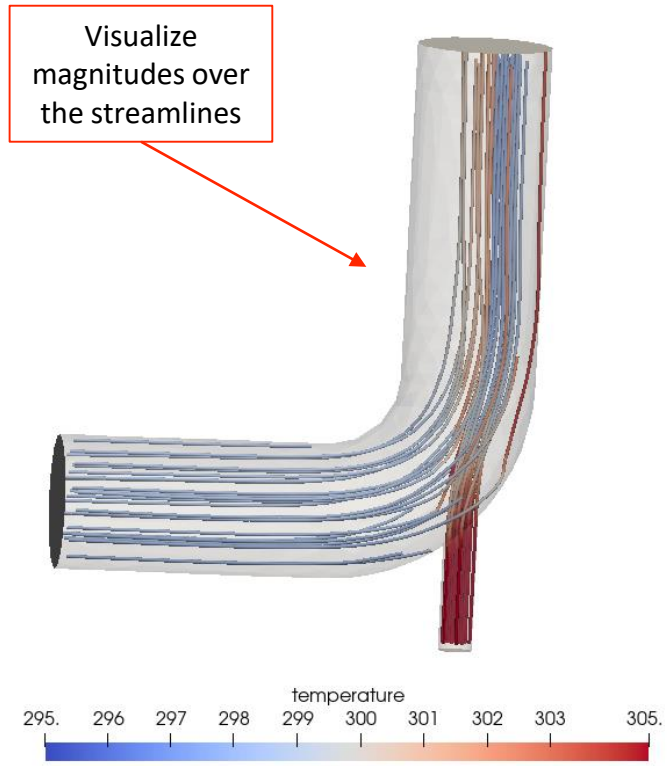


Post: Streamlines

- A useful option to visualize and understand the flow is the streamlines representation. From its option in the Post ribbon you can choose the geometry where to place the seeds, from which the lines start. The position and dimensions of the region can be modified. The distribution of seeds can be random, structured or manual.
- The variables represented and the color map can be changed clicking on the Edit button in the Post Browser.

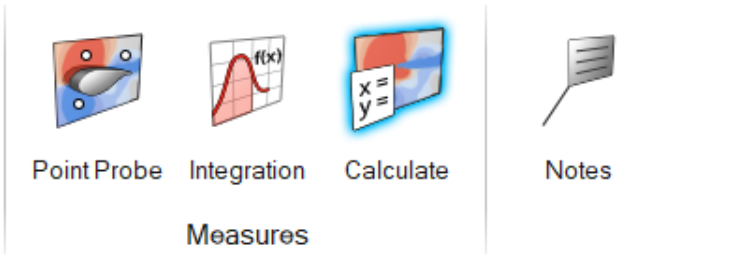


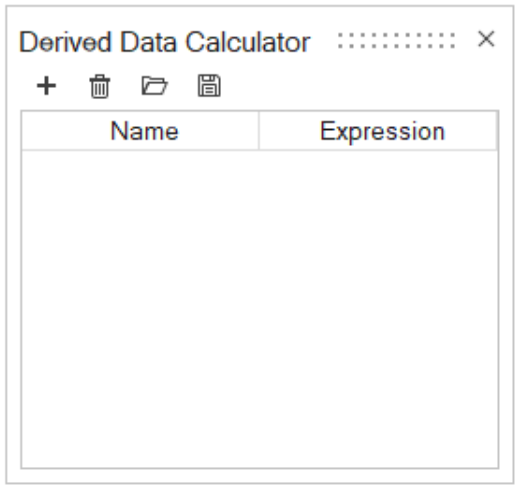


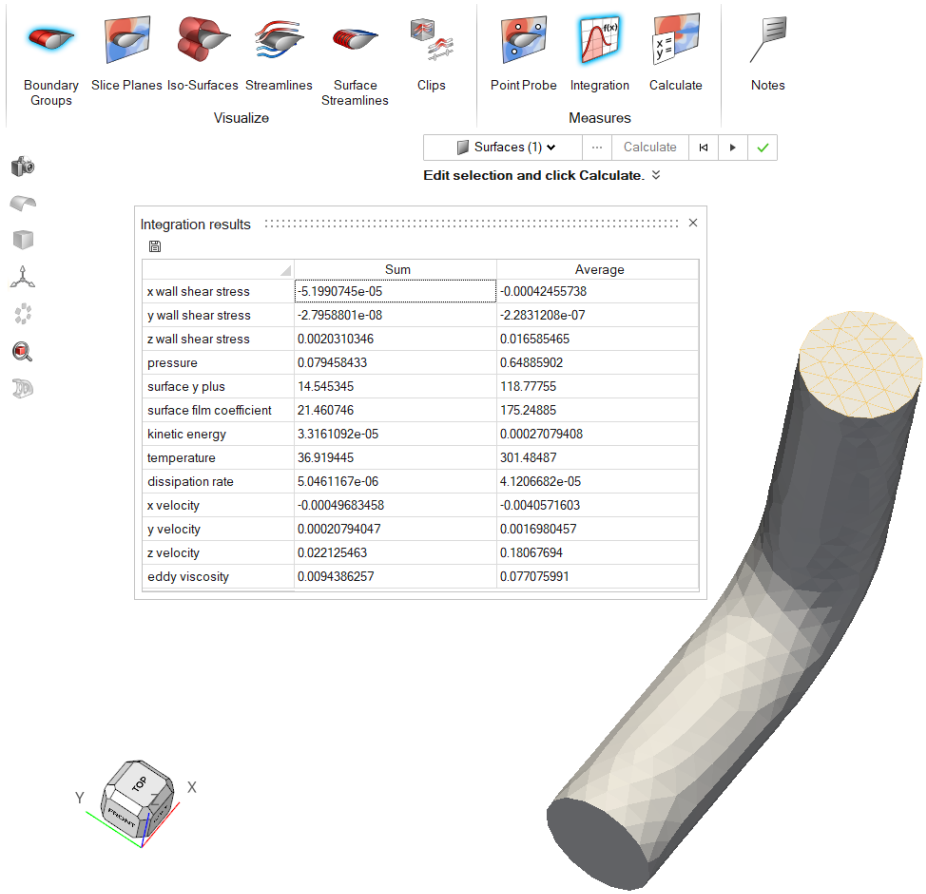


Post: Calculate & Integration

- The Calculate tool can be used to define new variables with Python expressions which can be displayed just like the ones provided by the program. Once defined, they can be selected in the same menus as the default ones.
- The Integrate tool provides results of average and integrated magnitudes on the model and monitored surfaces. For example, the average temperature output in the example can be obtained.







File Structure

- The model containing the geometry and conditions of the simulation is saved as an .hm file.
- When exporting the model an .inp file and the MESH.DIR folder necessary to launch the simulation are generated. The .inp could be edited externally to make changes not allowed by the interface.
- Simulating the .inp file generates the .cnf, .echo, .log files and the results folder ACUSIM.DIR.

Nombre	Fecha de modificación	Tipo	Tamaño
ACUSIM.DIR	27/09/2022 16:20	Carpeta de archivos	
MESH.DIR	27/09/2022 16:20	Carpeta de archivos	
Acusim.cnf	27/09/2022 16:20	Archivo CNF	1 KB
HMMeshLog_161148	27/09/2022 16:11	Documento de texto	7 KB
MixingElbow.1.echo	27/09/2022 16:20	Archivo ECHO	23 KB
MixingElbow.1	27/09/2022 16:20	Documento de texto	74 KB
MixingElbow	27/09/2022 15:03	CATIA Part	150 KB
MixingElbow	27/09/2022 15:50	Altair Simulation Produ...	112 KB
MixingElbow	27/09/2022 16:20	Archivo INP	19 KB
MixingElbow.ss.inc	27/09/2022 16:20	Archivo INC	15 KB
MixingElbow.stp	27/09/2022 15:03	Archivo STP	23 KB

EDIFICIO ANTARES

C/ INDUSTRIAS, 8

28923 – ALCORCÓN (Madrid) – España

Tel.(+ 34) 910 569 228 – (+ 34) 629 87 94 37

. . . . .

info@icemm.es

www.icemm.com