Gerald Recktenwald
gerry@pdx.edu
January 26, 2022

1 Overview

Imagine that you are part of a team developing a medical diagnostic device. The device has a millimeter scale flow channel designed to allow a sample fluid to be mixed with a carrier fluid. Figure 1 is a top-view schematic of the base case design for the flow channel. Figure 2 is a three-dimensional model created in Solidworks. The dimensions of the channel in mm are

$$L = 50,\ L_i = 6,\ L_s = 5,\ W = 8,\ w_s = 2,\ h = 2 \qquad \text{all in } mm$$

where h is the depth of the channel into the page.

In the test apparatus, water is used for both the carrier fluid and the sample fluid. The sample fluid is marked with red food coloring to provide a visual indication of how well the sample fluid mixes with the carrier fluid. The carrier stream has a velocity of $V_{\text{in}} = 0.1\,\text{m/s}$ and the sample stream has an inlet velocity of $V_s = 0.05\,\text{m/s}$. The properties of water used by STAR-CCM+ are

$$\rho = 997.56\ \frac{\text{kg}}{\text{m}^3} \qquad \mu = 8.887 \times 10^{-4}\ \text{Pa} - \text{s}$$

The Reynolds number based on hydraulic diameter, $D_h = 2Wh/(W + h) = 3.2$ mm for the main inlet duct is

$$\text{Re}_{Dh} = \frac{\rho V_{\text{in}} D_h}{\mu} = 359$$

Measurements show that despite the intention of the designers, the channel does a poor job of mixing the sample fluid with the carrier fluid. Your job is to propose design changes that will improve the mixing. Your design changes cannot alter the overall dimensions L, W, w_s, L_s or h. You cannot change the

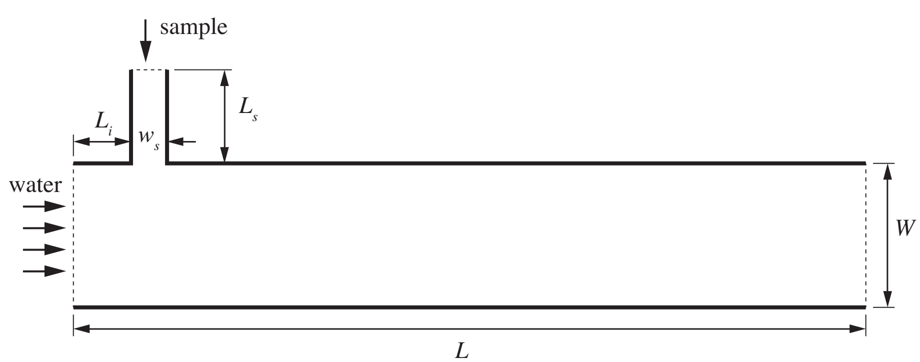


Figure 1: Device for testing flow mixing strategies. Depth into the page is h

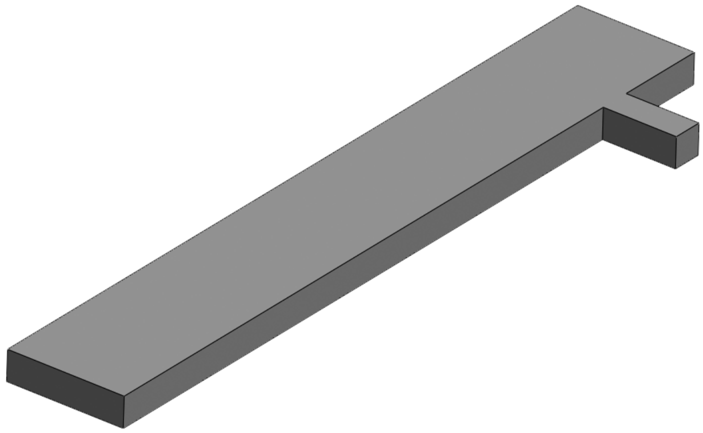


Figure 2: Solidworks model of the flow channel.

fluid velocities V_{in} and V_s. You *can* alter the geometry of the channel, for example, by adding obstructions or objects to increase mixing. Any modifications should be geometrically simple so that the new design is easy to manufacture.

The three-dimensional geometry of the channel was created in Solidworks model and saved as a *Parasolid* XT CAD file[1] that is imported into STAR-CCM+ as a surface mesh.

2 Import the Model Geometry

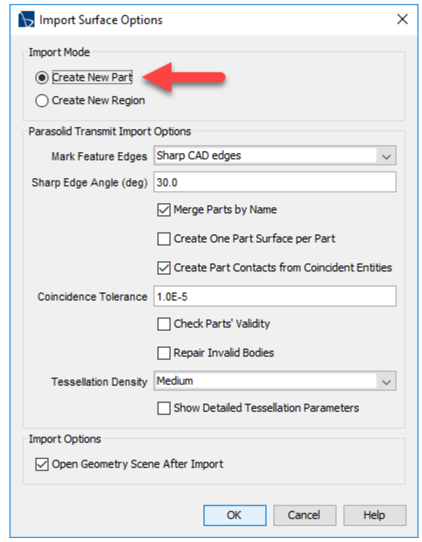
The model geometry is defined in a *Parasolid* file stored as `microchanel_mixer.x_t`. Download that file from `http://web.cecs.pdx.edu/~gerry/class/ME448/tutorial/` and store it in a convenient place.

2.1 Import the Parasolid Surface Mesh

The Parasolid model defines a geometry in a form that STAR-CCM+ can understand, but not work with directly.

[1]Parasolid is a 3D geometric modeling engine owned by Siemens. See `http://www.plm.automation.siemens.com/en_us/products/open/parasolid/index.shtml`.

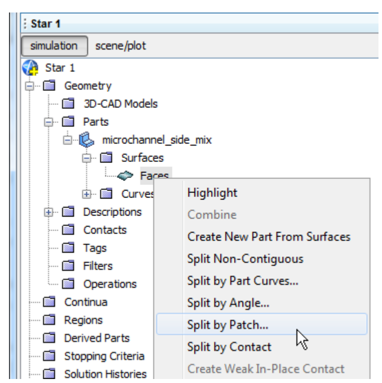
1. Launch STAR-CCM+ and open a new simulation.
2. Import the surface mesh:
   a. File ▷ Import ▷ Import Surface Mesh.
   b. Select the `microchanel_mixer.x_t` file from your computer's drive.
   c. In the *Import Surface Options* dialog box, make sure the *Create New Part* option is selected.
   d. Click OK.



2.2 Split the Single Surface into Patches

The surface mesh in the Parasolid model is a network of triangles that define the geometric features of the surface that encloses the domain volume. This single surface must be separated so that boundary conditions can be applied to distinct areas on the boundary of the domain. The first step in that process is to split the one bounding surface into patches, where each patch is a area that defines a smaller piece of the surface geometry. After the dividing the surface into patches, we rename those patches, and in some cases regroup the patches to form boundaries. When the patches are renamed and regrouped, we are ready to assign those patches to the boundaries of the region in Section 3.

1. Select the new part in the Geometry tree
2. Expand the *Surfaces* node
3. Right-click on *Faces* and select Split by Patch. . . .
   Notice these features of the user interface as evident in Figure 3:
   - The Geometry Scene now has different color surface patches.
   - Each surface patch corresponds to a number listed in the *edit* panel in the upper left.
   - Surface patches are selected by either clicking on their number or clicking on the patch in the *Geometry Scene*.



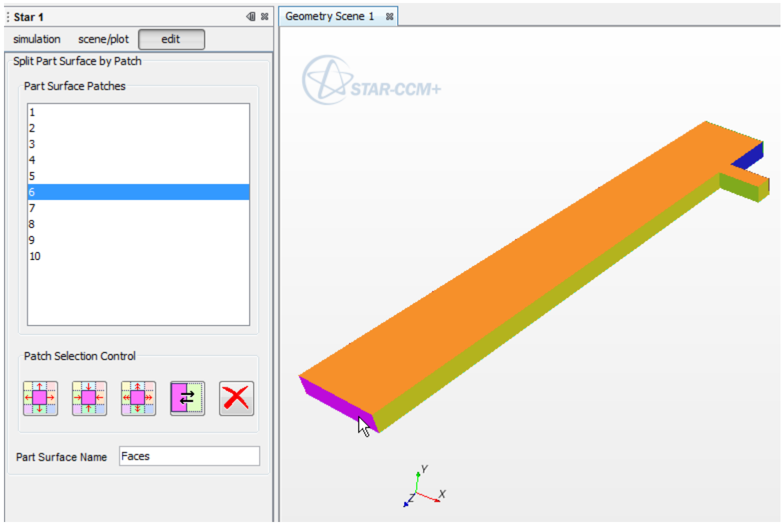


Figure 3: Appearance of the Geometry Scene and the *edit* panel used to select and name patches. Note that the numbers of the *Part Surface Patches* may be different in your model.

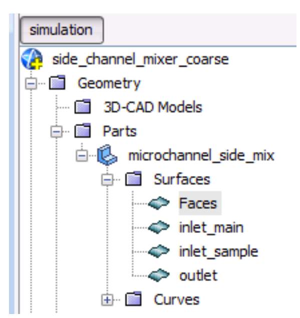
4. Orient the model in the *Geometry Scene* so that you can select (click on) the outlet face as shown in Figure 3.

5. In the middle of the *edit* panel, type "`outlet`" into the *Part Surface Name* box and click *Create*. This will rename the surface patch and remove it from the display in the *Geometry Scene*. It is possible to select and name multiple patches at a time with this procedure, which has the effect of combining those patches into an object with the same name. For the current model, we will only work with one patch at a time.

6. Repeat the two preceding steps for the two inlets. Choose `inlet_main` and `inlet_sample` for the names of the main inlet surface and sample inlet surface, respectively.

7. Click Close to stop selection of the patches.

   Notice that under the *Surfaces* node (a sub-node of the *Parts* node) that there are separate surfaces: the three newly created and named surfaces (`inlet_main`, `inlet_sample` and `outlet`, and one remaining surface called `Faces`.



8. Select the `Faces` surface, right-click to select Rename... and change the name to `duct_wall`.

2.3 Save the Model

Select Save from the file menu to save the model in a `.sim` file on the hard drive of your computer. It's a good idea at this point to create a *directory* for the simulation results as well as a meaningful file name. As you build the model and run different cases, you will accumulate alternate `.sim` files for the same physical problem. You will also generate external graphics files to be incorporated into reports. Therefore, creating a directory to group the files related to a single simulation will help you stay organized later.

Click the *Save* icon.

3 Create Fluid Region and Boundaries

The volume occupied by the part must be assigned to a *Region*. This can be accomplished (at least) two ways. Choose *only one* of the following

1. Select *Assign Part to Regions. . . .*

   or

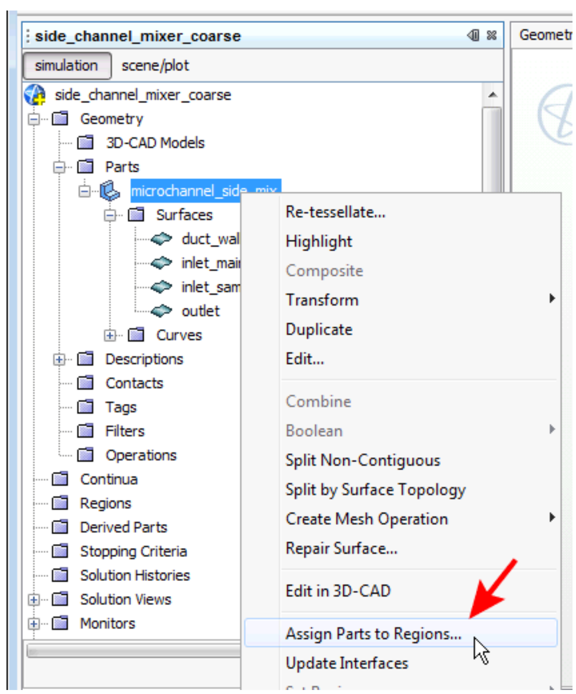
2. Select *Regions* ▷ *New*, and then select the parts that belong to it.

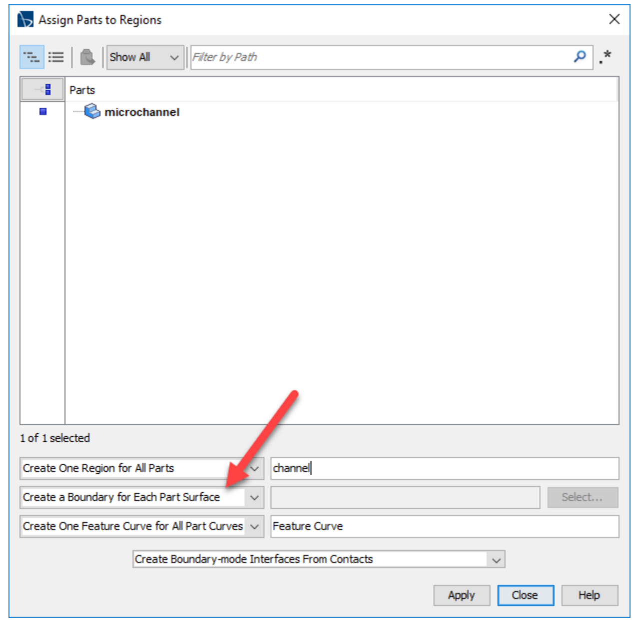
The procedures are equivalent. We will demonstrate both.

> When you build your model, **follow the steps in either Section 3.1 or Section 3.2, but not both.**

3.1 Assign Parts to Regions

1. Right click on the `microchannel_mixer` part node (first level under *Parts*) and select *Assign Parts to Regions. . .*





2. In the *Assign Parts to Regions* dialog box, make sure the Part is selected (blue dot next to the `microchannel` part).

3. **Important**: Select *Create a Boundary for Each Part Surface*. This selection will simplify the steps in § 3.1.1, below.

4. Change the name of the region from "Region" to "fluid" or "channel" or some other descriptive name.

5. Click the Create Regions button

6. Click Close

3.1.1 Assign Boundary Types to Surfaces of the Region

It's important to assign the types of boundary condition before the mesh is created. We will first assign the boundary types, and then set the boundary values later in Section 5.

Create the outlet boundary

1. Expand the *Boundaries* sub-node of the *fluid* Region

2. Select `outlet`

3. In the *Properties* pane, select *Pressure Outlet* from the *Type* property

Create the main inlet boundary

1. Select `inlet_main` sub-node under *Boundaries* sub-node of the *fluid* Region

2. In the *Properties* pane, select *Velocity Inlet* from the *Type* property

Create the sample inlet boundary

1. Select `inlet_sample` sub-node under *Boundaries* sub-node of the *fluid* Region

2. In the *Properties* pane, select *Velocity Inlet* from the *Type* property

Note that the icons for the outlet and the two inlets reflect the type of boundary condition applied to the surfaces of the region.

Skip to Section 4.

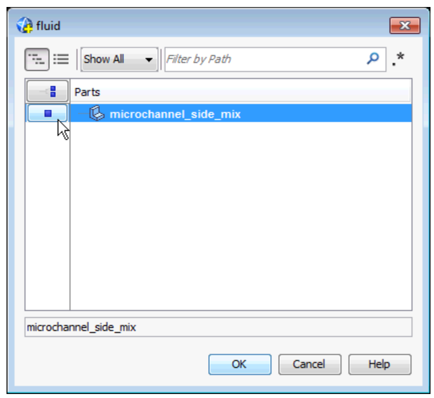
3.2 Create a Region and Assign Parts to It

The following steps are an alternative to the steps in Section 3.1.

1. Select *Regions*, right-click New
2. Right-click on the newly-created region, and select Rename. . . .
3. Enter `fluid` as the fluid name. Click OK.

Assign the *Part* to the fluid *Region*.

1. Click on the `fluid` node to select it.
2. In the *Properties* pane (lower left corner), click the [. . .] icon to initiate selection of the parts for the *Region*.
3. In the dialog box, select the Part corresponding to the fluid by clicking the blue dot to the left of the part name.
4. Click OK.



3.2.1 One outlet boundary

Separate the `outlet` surface and assign its boundary type.

1. Expand the *Boundaries* node under the `fluid` Region.
2. Right-click on *Boundaries* and select New.
3. Right-click on the newly-created *Boundary 1* node and select Rename. . .
4. Enter `outlet` in the *New Name* field. Click OK.
5. Select the newly created `outlet` boundary surface.
6. In the *Properties* pane (lower left corner), Click the [. . .] icon to initiate selection of part surfaces for the `outlet` boundary.
7. In the dialog box, expand the *Part* node and select the sub-Part corresponding to the outlet by clicking the blue dot to the left of the part name.
8. In the *Properties* pane (lower left corner), Click in the box to the right of *Type*. Select *Pressure Outlet* from the pop-up menu of options.

3.2.2 Two inlet boundaries

Separate the `inlet_main` and `inlet_sample` surfaces and assign the boundary type to those surfaces. The procedure is the same as that used for the `outlet` boundary, with the exception, of course, that a different boundary names and boundary types and applied to different part surfaces.

1. Right-click on *Boundaries* and select New.
2. Right-click on the newly-created Boundary (currently *Boundary 1*) and select Rename. . . .
3. Enter `inlet_main` in the *New Name* field. Click OK.
4. Select the newly created boundary surface (`inlet_main`)
5. In the *Properties* pane (lower left corner), Click the [. . .] icon to initiate selection of part surfaces for the `inlet_main` boundary.
6. In the dialog box, expand the *Part* node and select the sub-Part corresponding to the main inlet by clicking the blue dot to the left of the part name.
7. In the *Properties* pane (lower left corner), Click in the box to the right of *Type*. Select *Velocity inlet* from the pop-up menu of options. We will specify the value of this velocity later.

Repeat the preceding steps, substituting `inlet_sample` for `inlet_main` while selecting the sample inlet surface instead of the main inlet.

3.3 Save the Model

Click the *Save* icon. 

4 Create the Fluid Continua

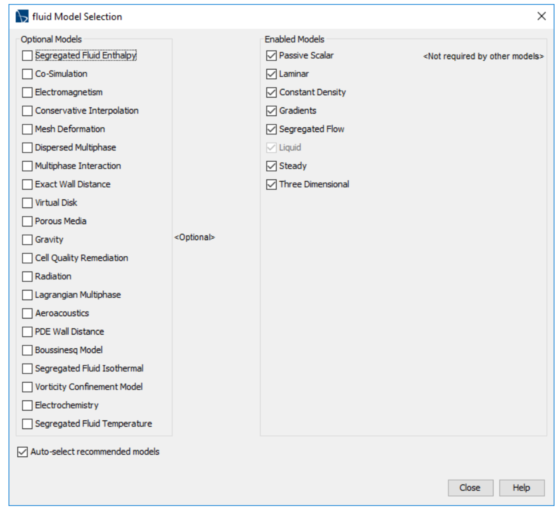
This problem introduces the passive scalar, which will be used as a marker to indicate the degree of mixing. A passive scalar is a quantity of interest that is transported by convection and diffusion. It is passive in the sense that its presence does not affect the motion of the fluid. We will call the passive scalar "red dye" to reinforce the idea that the passive scalar could be red food coloring that is uniformly mixed into the sample stream.

4.1 Turn on the features of the Physics Continuum

Except for the turning on the passive scalar feature, all other fluid continuum features are the typical ones for steady, laminar flow.

1. Right-click Continua, select New ▷ Physics Continuum.
2. Right-click on *Physics 1* and select *Rename. . . .* Choose a suggestive name like "fluid"

3. Make the following choices for the phyisics models
    - Three-dimensional
    - Steady
    - Liquid
    - Segregated Flow
    - Constant Density
    - Laminar
    - Passive Scalar
4. Click Close



4.2 Create the `red dye` passive scalar

Checking the *Passive Scalar* box in the *Physics Continuum* model causes the *Passive Scalar* feature to be enabled. We must also define that Passive Scalar, i.e., we must identify the red dye as a passive scalar

1. Expand the *Passive Scalar* node in the *Physics 1 Models* tree.
2. Right-click on Passive Scalars ▷ New
3. Right-click on the newly created *Passive Scalar* node and select Rename. . .
4. Enter `red dye` as the new name of the passive scalar

4.3 Verify fluid (water) properties

The default liquid is water. No changes are necessary, but it is instructive to inspect the properties used by StarCCM+

1. Expand the *Liquid* node in the *fluid Models* tree.
2. Expand the *H2O* sub-node, and expand the *Material Properties* sub-sub-node
3. Expand the *Density* sub-node, click on the *Constant* sub-sub-node and notice the value of 997.561 kg/m^3 assigned to the *Value*
4. Expand the *Dynamic Viscosity* sub-node, click on the *Constant* sub-sub-node and notice the value of 8.8871E-4 Pa-s assigned to the *Value*
5. After inspecting the values, collapse the nodes under the *fluid* Continua

4.4 Save the Model

Click the *Save* icon. 

5 Specify values of boundary properties

The type of boundaries were specified when the surfaces of the *Region* were created. After *Physics Continuum* is created, the values of the fluid properties on the surface can be specified. Expand the *fluid* sub-node of the *Regions* node in the simulation tree.

1. Expand the `inlet_main` node under *Boundaries*
2. Expand the *Physics Values* node and make the following changes to the sub-nodes
   - Select *Passive Scalar* and confirm that the value is `[0.0]` in the *Properties pane*
   - Select *Velocity Magnitude* and set the value to `0.1 m/s` in the *Properties pane*
3. Expand the `inlet_sample` node under *Boundaries*
4. Expand the *Physics Values* node and make the following changes to the sub-nodes
5. On the `inlet_sample` surface
   - Select *Passive Scalar* and set that the value to `[0.50]` in the *Properties pane*. The value of the passive scalar is arbitrary in this model, as long as it is non-zero. The value set here will be the maximum value achieved in the domain.
   - Select *Velocity Magnitude* and set the value to `0.05 m/s` in the *Properties pane*

The default values on the `outlet` and `wall` boundaries are correct.

6 Create the Mesh

The mesh can be created with two different processes: region-based meshing and parts-based meshing. Parts-based meshing is a newer approach (circa 2017) and is recommended because the region-based meshing approach will be phased out in a future version of STAR-CCM+.

6.1 Dimensions controlling the mesh size

The smallest dimensions in the physical problem are $h = 2\,\mathrm{mm}$ and $w_s = 2\,\mathrm{mm}$. Keep these values in mind when specifying the mesh base size and the thickness of the prism layer. Both of base size and the thickness of the prism layer should be smaller than the smallest macroscopic feature in the domain.

6.2 Create the Parts-based Mesh

- Under the *Geometry* node, right-click the *Operations* node and select New ▷ Mesh ▷ Automated Mesh.
- The *Create Automated Mesh Operation* modal pane opens
  - ▷ At the top left of the pane, check the small box to the left of `microchannel_mixer`.
  - ▷ In the bottom half of the pane, make three selections: Surface remesher, Trimmer Mesher, and Thin Mesher.
  - ▷ Click OK.
- Expand the newly created *Automated Mesh* node, and expand the *Default Controls* sub-node.
  - ▷ Select Base Size, and in the Properties panel change 1.0 m to 0.5mm. Note that when you change values in the *Properties pane*, there is no "OK" or "Close" button. Simply type in the new value and press Return or Enter on your keyboard.
  - ▷ Click on the *Number of Thin Layers* node. In the *Properties* pane, the *Number of Layers* value is 2, which is fine for now. Leave this value unchanged.
- Execute the mesh: Right click on the *Automated Mesh* node and select Execute.
- View the mesh in a new *Mesh Scene*

The *Mesh Scene* pane will show one of four representations of the mesh: Geometry, Initial Surface, Remeshed Surface, Volume. Figure 4 shows three of those representations. The Geometry and Initial Surface representations show the triangles (sometimes very long and skinny triangles) used by the CAD tool to define the surface of the domain. The *Remeshed Surface* is also a geometry-defining mesh created by the *Surface Remesher*. The *Volume Mesh* depends on the kind of volume mesher selected for the *Mesh Continuum*. Figure 4 shows the volume mesh at the outlet when the *Prism Layer* and *Polyhedral* volume mesh options are selected.

There are many ways to control the size of the volume mesh elements. The primary control is via the *base size* parameter. Figure 5 shows the mesh on the outlet surface for two choices of base size: 0.5 mm and 0.2 mm.

6.3 Save the Model

Click the *Save* icon. 

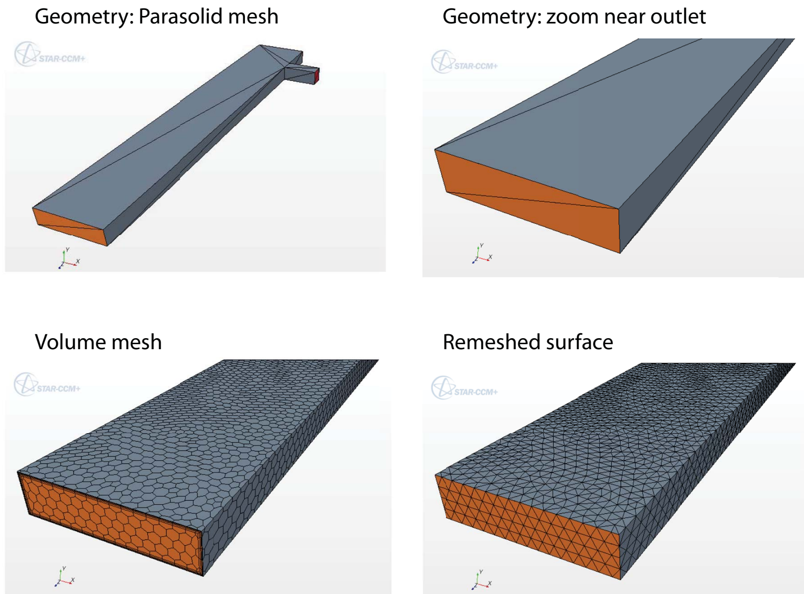


Figure 4: Three representations of the mesh in the Mesh Scene: *Geometry*, *Remeshed Surface*, *Volume Mesh*.

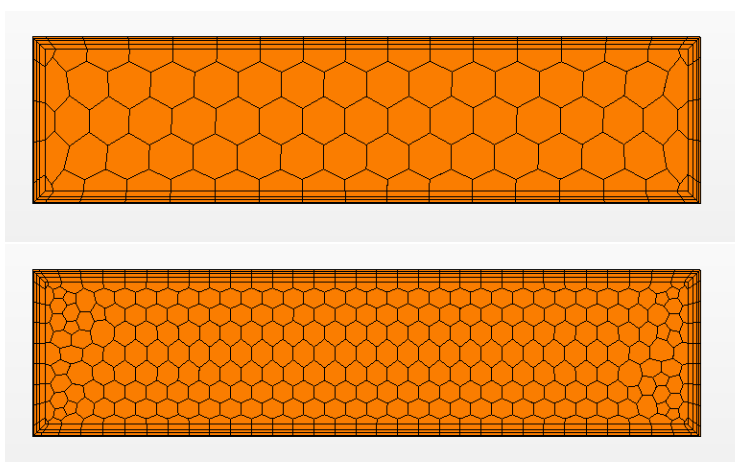


Figure 5: Mesh on the outlet surface for base size of 0.5 mm (top) and 0.2 mm (bottom). Both meshes have an absolute prism layer thickness of 0.15 mm.

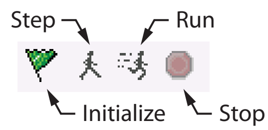


Figure 6: Icons in the menubar for running, stepping and stopping a simulation.

7 Run the model

Before running the model, limit the number of iterations and set a stopping criteria on the residuals of the continuity equation. You could skip these steps and manually intervene during the solution by clicking the *Stop* icon. However, it makes sense to set these standard stopping controls because it will allow you to do other work (on long runs) while the iterations procede toward convergence.

7.1 Limit the maximum number of iterations

1. Expand *Stopping Criteria* node
2. Click Maximum Steps and set to 300

7.2 Add stopping criterion for continuity residuals

1. Right-click on the *Stopping Criteria* node
2. Select Create New Criterion ▷ Create from Monitor ▷ Continuity
3. Click on the *Minimum Limit* sub-node
4. Notice that the *Minimum Value* in the *Properties Pane* is `1.0E-4`. Leave this value for now.

7.3 Start the solver

Figure 6 shows the four buttons in menu bar that are used to run, step and stop a solution. The start button is frequently used to begin the simulation. Clicking the *run* button automatically initializes the solution if it hasn't already been initialized. Clicking the *run* button will continue a simulation that has already taken some iterations.

If you click on the *stop* button, the solver completes the current iteration before stopping. Therefore, clicking *stop* will not risk damage to a successful simulation. The *step* button causes the solver to complete one more iteration.

1. Click the *Run* icon 
2. Watch the residual plot.

   If the residual plot does not automatically open in the *Scene* pane, expand the *Plots* node in the simulation tree, then right-click on the *Residuals* node and select Open.

   The residual plot in Figure 7 shows that the model converges to a continuity equation criterion of 1×10^{-4} in about 220 iterations.

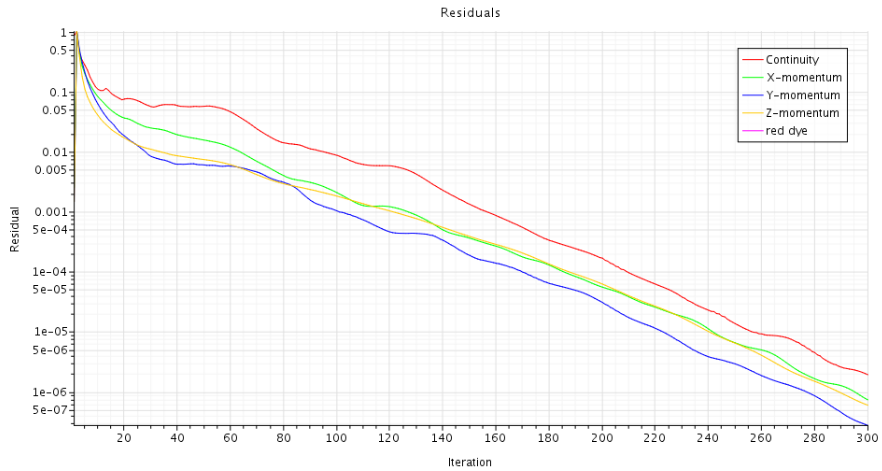


Figure 7: Residual history for the base case problem: mesh base size of 0.5 mm.

8 Visualize the Results

8.1 Red dye concentration

Red dye from the sample inlet is convected downstream after it enters the main channel. The dye also slowly diffuses into the water from the main channel flow.

8.1.1 Dye concentration a horizontal plane in middle of the duct

Create a plane for visualizing the $x - z$ distribution of red dye in the channel.

1. Bring the *Geometry Scene* tab to the foreground
2. Right-click on *Derived Part* ▷ New ▷ Section ▷ Plane This opens an edit panel in the upper left corner.
3. Click on *Select...* in the *Input Parts* area at the top of the edit panel.
4. Select the `channel` fluid region (or whatever you named the Region)
5. Enter the values to the right into the *Plane Parameters* table. Any x and z values will work in the `origin` column.

	origin	normal
x	0 mm	0 mm
y	1 mm	1 mm
z	0 mm	0 mm

6. Click Create
7. Click Close
8. In the list of *Derived Parts*, right-click on the newly created *plane section* and select *Rename....* Change the name to `y-normal midplane` or some other meaningful name.
9. Click OK.

Create a scalar viewer to display the concentration on the horizontal midplane.

1. Right-click on *Scenes* ▷ New Scene ▷ Scalar
2. Expand the newly created *Scalar Scene 1* node. (At this point you could rename the scene to make it easier to identify at later stages in model development and visualization.)
3. Click on the *Parts* node under the *Scalar 1* node of the newly created *Scalar Scene*.
4. In the *Properties* pane, click on the [. . .] icon to the right of the *Parts* item.
5. Select the `y-normal midplane` from the list of *Derived Parts* (Be sure to click on the blue square, not the name of the part.)
6. Click OK to close the parts selection dialog box
7. Click on the *Scalar Field* node under the *Parts* sub-node
8. In the *Properties* pane, click on the `<Select Function>` field in the *Function* row. Select Passive Scalar ▷ red dye
9. Click on the *Scalar 1* node
10. In the *Properties* pane, click on the `Filled` field in the *Contour Style* row. Select *Smooth Filled* from the pop-up menu.
11. Click on the *Outline 1* node under the *Displayers* node of the Scalar Scene.
12. In the *Properties* pane, check the *Feature lines* box

Figure 8 shows two view of the image created by the preceding steps. The views are created by the rotating the model in the *Scalar Scene* viewer. Standard views and user-define views can be defined and recalled by clicking the `Save-Restore-Select views` button  and choosing views `+Y ▷ Up +X`.

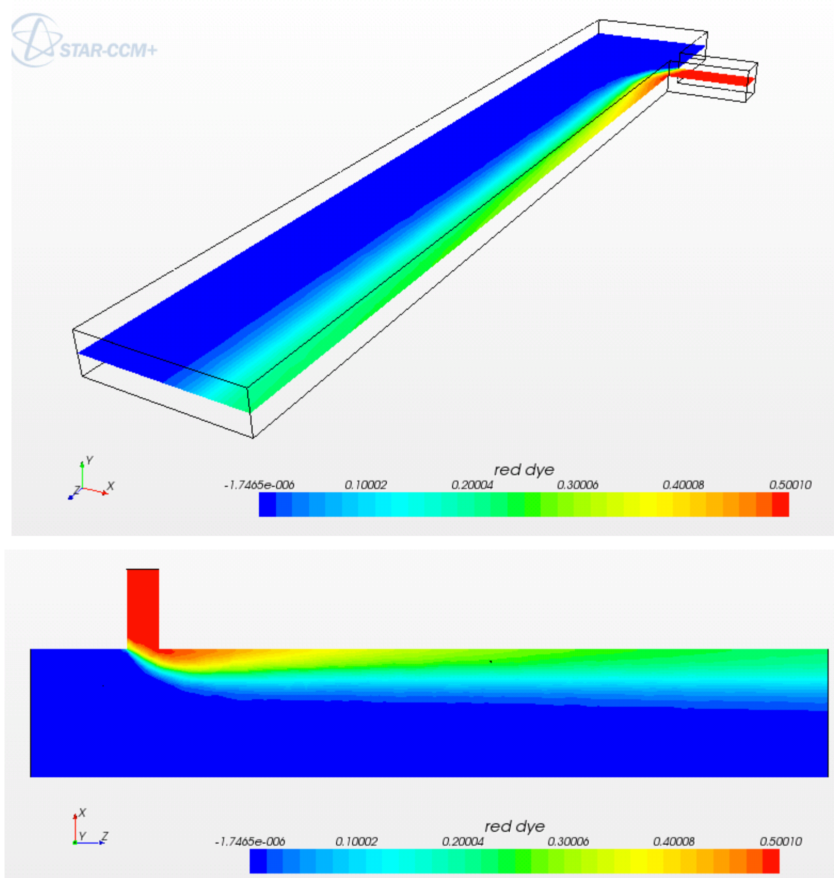


Figure 8: Two views of the concentration of red dye on the mid plane for the coarse mesh solution.

8.2 Concentration profile at the exit mid-plane

Creation of a concentration profile at the exit is a two-step process.

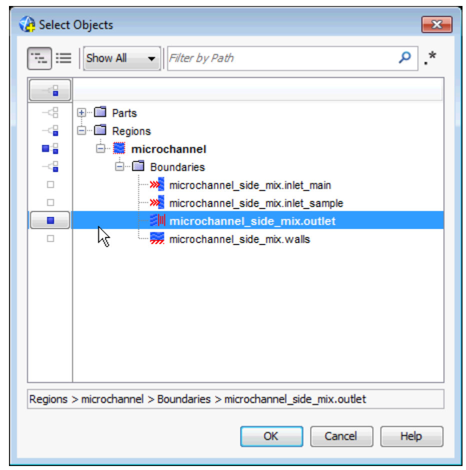
1. Create a line probe to extract data from the field values on the mesh
2. Create an X-Y plot to display the profile data

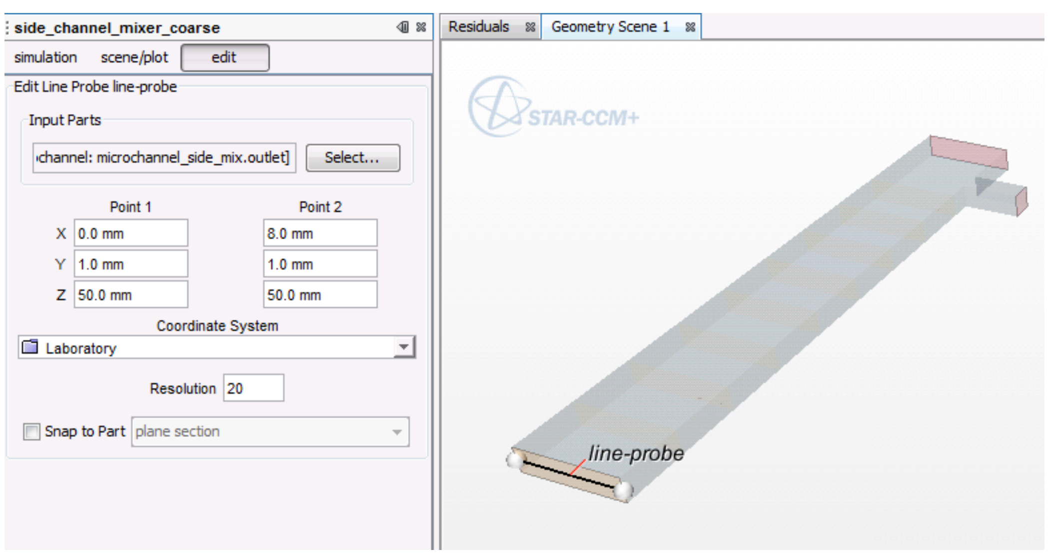
8.2.1 Create a line probe

1. Open the Geometry Scene tab
2. Right-click on *Derived Parts* ▷ New Part ▷ Probe ▷ Line. . . .
   This opens an edit panel in the upper left corner.
3. Click on *Select. . .* in the *Input Parts* area at the top of the edit panel.
4. Select the `outlet` boundary in the fluid region
5. Enter the following values for the coordinates of *Point 1* and *Point 2*

	Point 1	Point 2
x	0 mm	8 mm
y	1 mm	1 mm
z	45 mm	45 mm

Inspect the image in the *Geometry Scene* to verify that the line probe is correctly located.





6. Click Create
7. Click Close.
8. In the list of *Derived Parts*, right-click on the newly created line probe and select *Rename...*. Change the name to `exit profile probe`. Click OK.

8.2.2 Plot the concentration data on the line probe

Now that the probe line has been created, we can construct the plot

1. Right-click on the *Plots* node.
2. Select New Plot ▷ X-Y
3. Right-click on the new *XY Plot 1* node and select Rename.... Give the plot a new name, like `exit profile`
4. In the *Properties* pane, Click on the [...] icon for the *Parts* property
5. Select the `outlet` boundary as the *Part* to be used for the plot data.
6. Expand the *X Type* node and click on the *Position* sub-node
7. In the *Properties* panel, note that the *Direction* value is `[1.0, 0.0, 0.0]` which corresponds to having the independent (*X Type*) value equal to the x axis. This is correct for the profile we want to create.
8. Expand the *Y Types* node
9. Expand the *Y Type 1* sub-node
10. Click on the *Scalar* sub-node
11. In the *Properties* pane, select Passive Scalar ▷ red dye.
12. The appearance of the plot can be improved by adjusting the symbol type, x axis range, and x-axis label spacing.

    First, change the axis range.

    a. Expand the *Axis* node. Expand the *X Axis* sub-node. Expand the *Title* sub-node.
    b. Click on the *Labels* sub-node.
    c. In the *Properties* pane, change the *Maximum* property to 0.008, and change the *Label Spacing* to 0.0010.

    Next change the symbol type

    a. Expand the *Y Types* node. Expand the *Y Type 1* sub-node. Expand the *exit profile probe* sub-node. (*exit profile probe* is the name of the line probe derived part.
    b. Click on *Symbol Style*.
    c. In the *Properties* pane, change the *Size* to 8. Change the *Shape* to *Dot*. Change the *Color* to *Black*.

    The result of these operations is the plot in Figure 9.

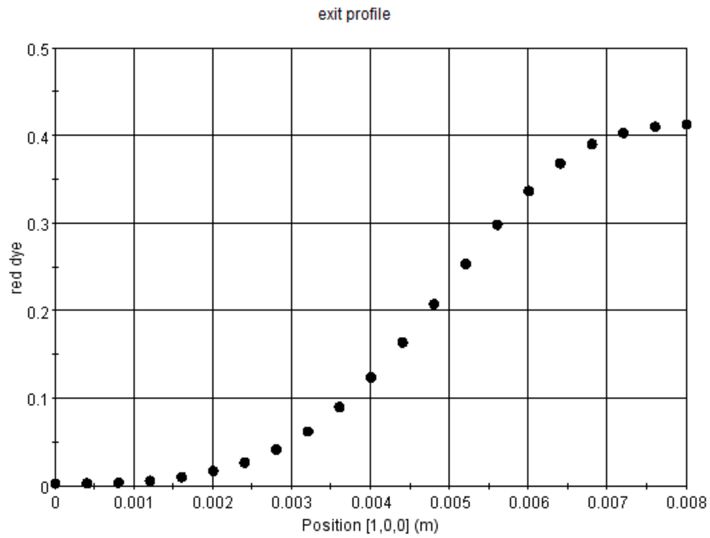
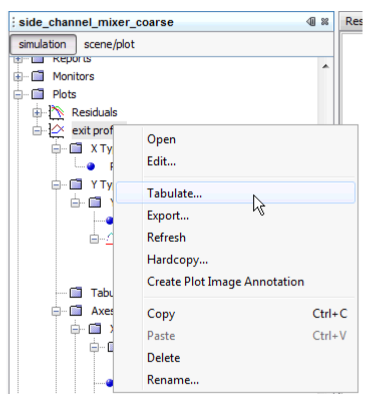


Figure 9: Profile of red dye concentration across the x direction at the duct exit

8.2.3 Export the profile data to a CSV file

To post-process the data in MATLAB or Excel, export the profile data to a CSV file.

1. Right click on *exit profile* – the name of the X-Y plot under the *Plots* node.
2. Select Tabulate...
3. Click *Export...* in the *Tabular Data* dialog box
4. Navigate to a good place in your file system and enter a name like `dye_exit_profile.csv` in the *File name:* field and click Save.



8.3 Save the Model

Click the *Save* icon. 

8.4 Streamlines

8.4.1 Create Streamline Derived Parts

Streamlines are geometric features derived from the velocity field. The *Derived Part* interface is used to specify the seed point (or points) of the streamlines. We will create two separate line seeds. After the seeds are defined, and after the velocity field has been obtained, the streamlines are visualized in a *Scene*.

1. Open the *Geometry Scene* tab.

2. Right-click on *Derived Parts* ▷ New Part ▷ Streamline. . . .
   This opens an edit panel in the upper left corner. It also opens up a pop-up window that a

3. Make sure that the *Input Parts* selection is for the fluid domain, e.g., it is not empty or referring to a boundary

4. Make sure *Vector Field* is set to Velocity

5. In the *Seed Mode* pop-up menu, select Line Seed.

6. Enter the following values for the coordinates of *Point 1* and *Point 2*.

	Point 1	Point 2
x	7.5 mm	7.5 mm
y	0 mm	2 mm
z	0 mm	0 mm

   If the model includes a velocity field (i.e., if the flow solver has been run and there is a velocity field in the current model) the streamlines should be visible in the *Geometry Scene*.

7. Set the resolution (number of points) to 5.

8. Click Apply. You should now see faint streamlines in the *Geometry Scene*.

9. Enter the following values for the coordinates of *Point 1* and *Point 2*.

	Point 1	Point 2
x	12.99 mm	12.99 mm
y	0 mm	2 mm
z	7.75 mm	7.75 mm

   The x coordinates are just shy of the 13 mm that defines the far edge of the domain. Due to round-off, the streamline seed may fall outside of the domain if $x = 13$ mm is used to define the seed line. This may or may not be an issue in your model. Using $x = 12.99$ mm guarantees that the see line is *inside* the domain.

10. Click Apply. A new set of streamlines should be visible in the *Geometry Scene*.

11. Click Close.

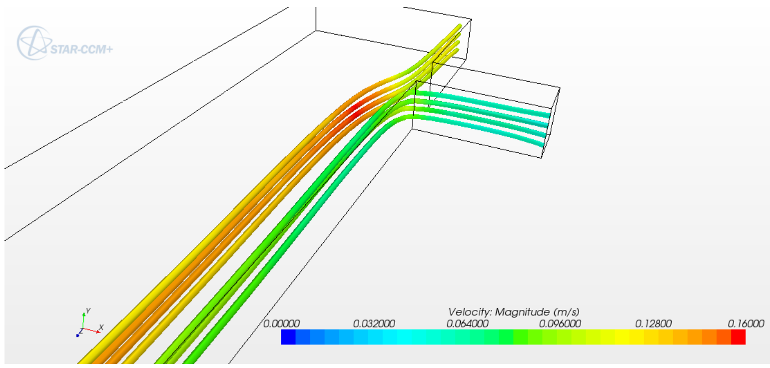


Figure 10: Stream tubes emanating from the two inlets for the coarse mesh solution with base size $= 0.5$ mm.

8.4.2 Display the Streamlines

Once streamlines are created with the preceding steps, those streamlines can be added to any *Scene*.

1. In any *Scene*, right-click on the *Displayers* node and select New Displayers ▷ Streamline.
2. Expand the *Streamlines 1* node.
3. In the *Properties* pane, change *Mode* to Tube. This changes the streamlines to stream tubes, but the scale of the tubes is so large that they obliterate the other features of the Scene.

   Change the *Width* parameter in the *Properties* pane to `1e-4` to give the stream tubes an appropriate scale.
4. Click on the *Scalar Field* sub-node of the *Streamline 1* node. In the *Properties* pane, click on `<Select Function>` adjacent to the *Function* property, and select Velocity ▷ Magnitude.

Repeat the preceding steps for the second streamline part.

When scalars are displayed on more than one set of streamlines, it is *very important* that the scalars use the same scale. Figure 10 shows two sets of stream tubes (created with the preceding steps) and a single color scale to indicate the velocity magnitude.

1. Select *Scalar Field* sub-node of the *Streamline 1* node
   a. In the *Properties* pane, set the min value to 0.
   b. Set the max value to a value somewhat larger than the default value. In this example a maximum value of 0.16 is sufficient.
2. Select *Scalar Field* sub-node of the *Streamline 2* node
   a. In the *Properties* pane, set the min value to 0.
   b. Set the max value to the *same value* as the maximum for the velocity scale used in the other streamline object.

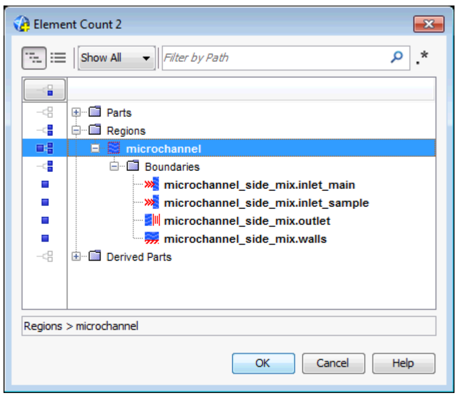
c. Click on the *Color Bar* sub-node for that *Scalar Field* and uncheck the Visible box.

The preceding steps should yield a stream tube plot similar to that in Figure 10.

9 Quantitative Reports

9.1 Element Counts

1. Right-click *Reports* ▷ New Report ▷ Element Count
2. Click on the just-created *Element Count 1* sub-node
3. In the *Properties* pane, click on the [. . .] icon in the *Parts* row.
4. Select all of the nodes under the fluid region (under the *Regions* node). Make sure you click the blue squares, not the names of the region or boundary.



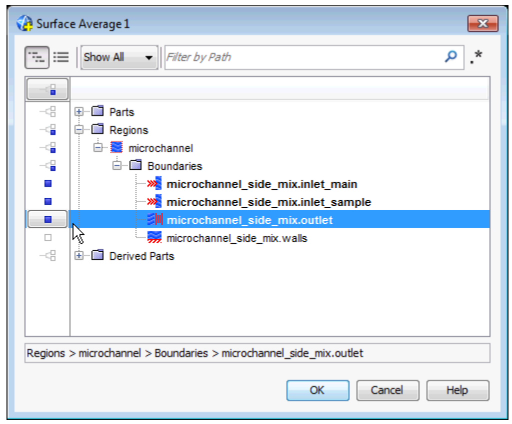
5. Right-click on *Element Count 1* ▷ Run Report. The results of the report are printed to the *Output* pane.

```
            Element Count

Part                              Element Count
------------------------------    -------------
channel                           2.976200e+04 cells
channel_side_mix.inlet_main       2.590000e+02 faces
channel_side_mix.inlet_sample     1.090000e+02 faces
channel_side_mix.outlet           2.590000e+02 faces
channel_side_mix.walls            5.353000e+03 faces
                                  -------------
                       Total:     3.574200e+04 element
```

9.2 Average Pressures on inlets and outlet

1. Right-click *Reports* ▷ New Report ▷ Surface Average
2. Click on the just-created *Surface Average 1* sub-node
3. In the *Properties* pane, click on the [...] icon in the *Parts* row.
4. Select two inlet and one outlet surfaces under the fluid region (under the *Regions* node). Make sure you click the blue squares, not the names of the region or boundary.



5. In the *Properties* pane, click on the `<Select Function>` field in the *Scalar Function Row* row. Select Pressure
6. Right-click on *Surface Average 1* ▷ Run Report. The results of the report are printed to the *Output* pane.

```
        Surface Average of Pressure on Volume Mesh

Part                                       Value (Pa)
------------------------------            -------------
channel: channel_side_mix.inlet_main      2.865981e+01
channel: channel_side_mix.inlet_sample    2.793480e+01
channel: channel_side_mix.outlet          0.000000e+00
                                          -------------
                          Total:          1.584156e+01
```

9.3 Save the Model

Click the *Save* icon.