# Tutorial

# Blood flow simulation in a coronary bifurcation using Ansys Fluent Software

Dr. Adelaide De Vecchi and Dr. Hamed Keramati
Kings College London

Edited and curated by Ansys Academic Development Team

education@ansys.com

## Ansys Software Used

This resource uses Ansys Fluent® fluid simulation software, Mechanical™ structural finite element analysis software, Ansys Discovery™ 3D product simulation software, and Ansys CFD-Post tool.

## Summary

This tutorial will model blood flow in a bifurcation of coronary arteries using Ansys Fluent Software, a fluid simulation software that is used to solve various problems related to fluid flow, heat and mass transfer, chemical reactions, and more. Ansys Fluent software uses advanced physical models like turbulence modeling, multiphase modeling, battery modeling, combustion, and fluid- structure interactions to solve the given problem to high level of accuracy.

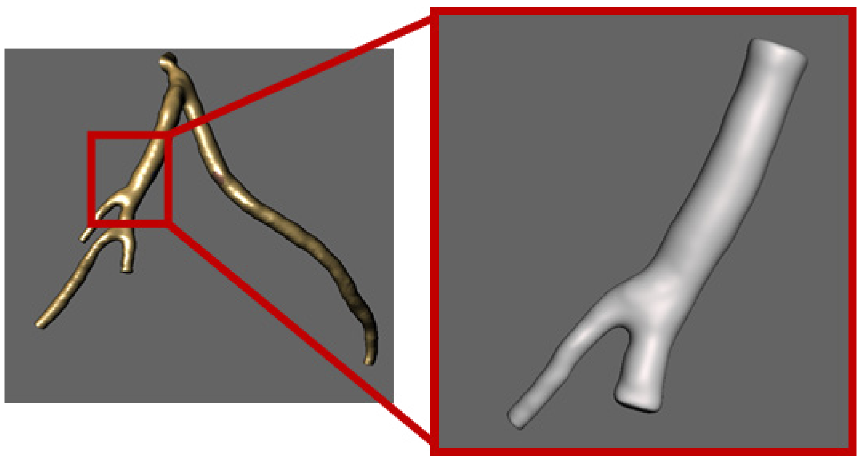
This tutorial illustrates the setting up, simulating and results visualization of steady state blood flow in a coronary bifurcation. Software uses include Ansys Discovery, Ansys Mechanical meshing, Ansys Fluent and Ansys CFD-Post software clusters.

## 1. Background

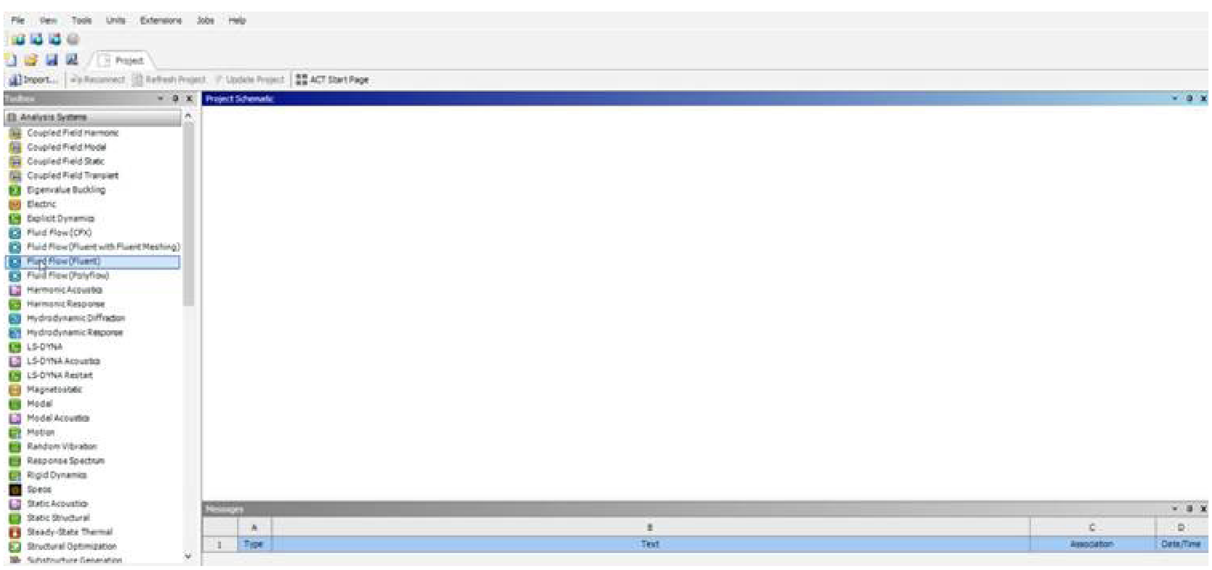
We perform computer simulations of blood flow in the coronary arteries to better understand the dynamics of blood circulation within the heart and to assess the effects of different conditions, such as blockages or narrowing of the arteries. These simulations help to predict how blood flow might be altered by factors like plaque buildup, stents, or surgical interventions, allowing for more accurate diagnostics and treatment planning. By modeling blood flow computationally, we can explore various scenarios in a controlled environment, providing valuable insights without the risks of invasive procedures.

## 2. Geometry

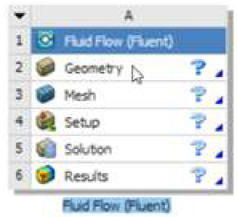
First, we need to determine the domain, physics and the boundary conditions constraining the simulation. The anatomy of the coronary bifurcation has been obtained by segmenting Computer Tomography (CT) data of the heart and vessels (not shown in this tutorial). This allowed us to delineate the patient-specific anatomy which will form the domain for the simulation. The segmented anatomy is presented below.



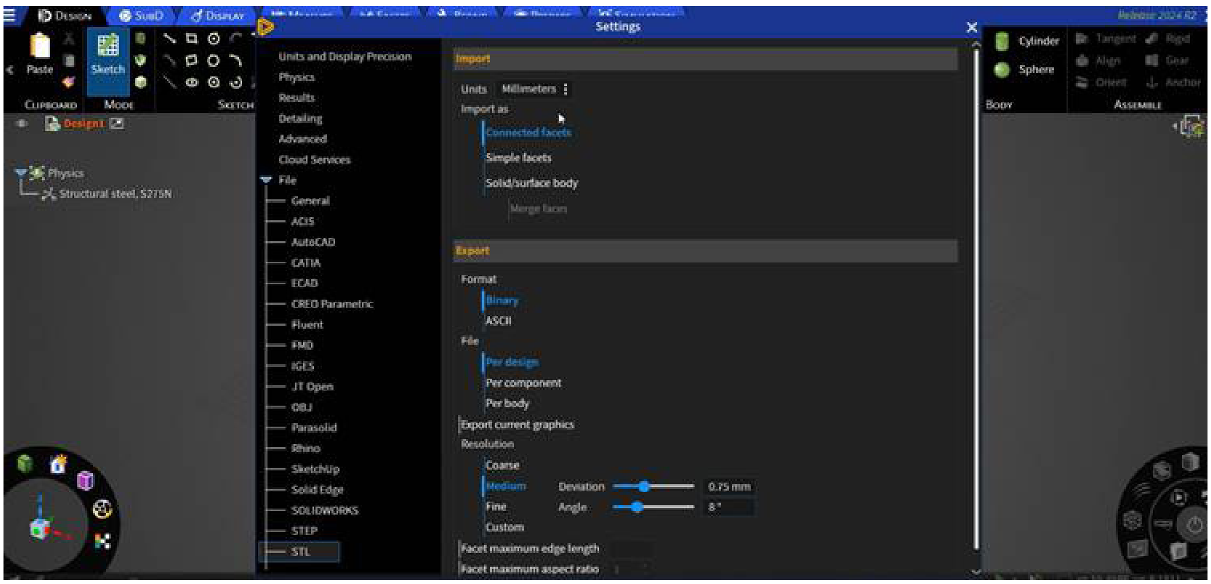
First, open the Ansys Workbench tool, drag and drop the Fluid Flow (Fluent) module into the working environment of Workbench, as shown in the figure below. Fluid Flow (Fluent) has a complete package for pre-processing, meshing, simulation (using Ansys Fluent Software) and results visualization.



We can then use Ansys Discovery software to prepare the geometry. Double click on the geometry section of the panel to open Ansys Discovery software.

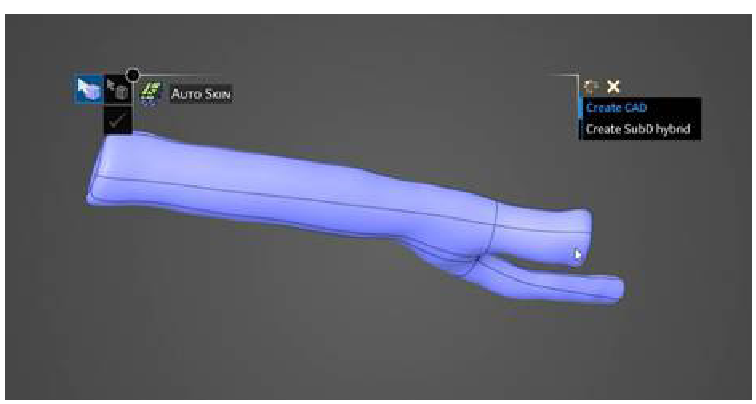



To import the geometry with appropriate dimensions, we must ensure that the length unit is set to millimeter. This can be done by clicking the three-line button and selecting Setting. As shown in the figure below, under File, select STL and ensure “Millimeters” is selected in the Units box.



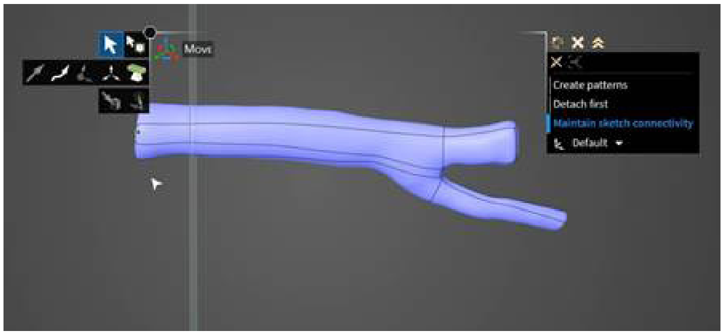

Click the three-line button and select Open, navigate to the STL file location and select the domain geometry file (vessel_geo_1.stl).

The next step is to convert the STL geometry to a more suitable format for the simulation. To do so, click the FACETS tab and then click Auto Skin. Select the geometry and confirm the selection by clicking on the tick sign. The STL geometry will be covered by curved surfaces as shown in the figure below.

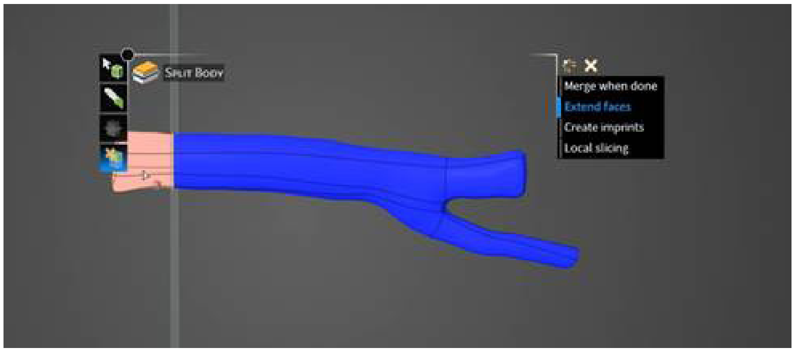


The next step is to prepare the inlet and outlets. We need to create planes as our cutting tool. To create planes, we use a point (DESIGN > CREATE > Point), create a plane (DESIGN > CREATE > Plane) passing through the point and then adjust the orientation of the plane.

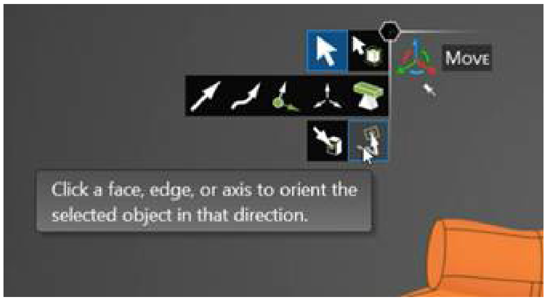
To adjust the orientation of the plane, we use the Move tool (DESIGN > EDIT > Move) and select the plane and moving the reference coordinate to a suitable location (the created point). By grabbing and moving the x-y-z axes icon, we can move the plane to the region where we want to perform the cut. The orientation of the plane can be controlled by clicking on the colored curved arrows near the x-y-z axes icon.




Next the DESIGN > INTERSECT > Split Body tool will be used to cut the geometry. Begin by selecting the Split Body option from the toolbar, then choose the geometry you want to cut and select the plane to serve as the cutting tool. After specifying the plane, identify the side of the geometry that is not needed and ensure it is properly defined for removal. This process allows you to accurately split and modify the geometry as required.




We repeat the process for the two outlets. If the cutting plane splits the geometry in an unintended location, you can use the DESIGN > INTERSECT > Combine tool to merge the parts back together. Finally, we move the body to the origin. Again, we use the Move tool. The Move tool has two options that are useful for our purpose: Align the orientation and Move to a given destination. Please see the images below for clarity.



After adjusting the location and the orientation of the geometry, we must suppress all unwanted geometrical entities except Patch Body11 under the tree (see figure below for reference).

## 3. Meshing

Open the Mesh section from the Workbench and double-click on Ansys Mechanical Meshing Software. Once the simulation domain appears in the meshing environment, check Geometry under the project tree and suppress any unwanted geometrical entity (image below).

To create the mesh, begin by right-clicking on the Mesh node, navigating to Insert, and selecting Method. Choose the body (as shown in the figure) and click the Apply button.

To perform the meshing process, we need to change the method to Tetrahedrons. Next, right-click on the Mesh node again and insert Sizing. Select the body and set the Element Size to 0.0002 m. To create the boundary layer mesh, right-click on the Mesh node, insert Inflation, and select the body under the Geometry section. In the Boundary section, select all surfaces on the vessel wall while ensuring that the inlet and outlet surfaces are excluded. These selected surfaces will define where the boundary layer mesh begins to grow. Leave all other settings at their default values. Finally, right-click on the Mesh node and select Generate Mesh. After a short processing time, the body mesh will be generated.

To define the boundaries and assign conditions in Ansys Fluent Software, start by right-clicking on the surface that you want to name as the inlet. From the dropdown list, select Named Selection and assign the name inlet to the selected surface. This process ensures that the boundary is correctly identified and labelled for subsequent setup steps. Repeat the same procedure for defining the two outlets as outlet_1 and outlet_2.

Once the meshing is complete, update the meshing node at the Ansys Workbench model and open the "Fluent" module.

## 4. Simulation Setup

In the Ansys Fluent Launcher window, select Double Precision and the number of Solver Processors according to your machine hardware. Then, click Start.
In the software, under the General section, keep the Time adjustment as Steady as we are intended to perform a steady-state simulation.

Under the Models, change the Viscous model to Laminar.
Now, we need to select the Material properties. Under the Material>Fluid, change the name and then set the viscosity and density to the those of the blood. And then click Change/Create.

Now, we should set the boundary conditions. This part is one of the most important in any simulation. Double click on the Boundary Conditions. You will see the named surfaces.

We select the inlet as velocity-inlet and then edit the value to 0.2 m/s. For the outlets, we select pressure-outlet at the boundary condition and keep the value to 0 Pa, as the only pressure difference is important in the CFD with a rigid body. Please ensure the arterial wall boundary condition is set as wall. Under the Solution node in the Outline View section, expand Monitor and double click on the Residual. We should select the convergence criteria in the section. Change the value to 0.0001 for all the residual thresholds. For some simulations, one can det the threshold lower, but 0.0001 is good enough for our tutorial.

The next step is to initialize the solution as a starting point. Double-click the Initialization node, select Hybrid Initialization and click Initialize. After the simulation is initialized, double-click the Run Calculation node. Select 500 as the Number of Iterations. This number is the number you decide as the maximum number of iterations if the solution is not converged. Click the Calculate button to start the simulation. After the simulation is finished, save the project and double-click the Results section in the Workbench to start the post-processing.

## 5. Post-Processing

We are now going to visualize the velocity using different tools, e.g. streamlines, velocity vector plot and velocity contour plot in CFD-Post.

### 5.1 Velocity contour plot

To create contour plots, we should create planes. We create the planes based on the following points and the normal vectors.

In CFD-Post, click Location and select Plane.

Select the Method as Point and Normal. Enter the point coordinates and the normal vector components and then click Apply.
Then click the contour icon (). In the contour detail section, select the plane of interest as the location and velocity as the variable. You can also change the number of contours for a smoother contour. Click Apply to finalize the contour plot. Repeat the process for all the planes.

### 5.2 Vector plot

Click the vector icon (). In the vector detail section, select the plane of interest as the location and velocity. Select the Face Center as Sampling. Then apply to create the vector plot on the plane of interest.

### 5.3 Streamline

Click the streamline (). In the streamline detail section, select inlet plane as the Start From. Change the number of points to 100 and apply.

## 6. Conclusion

Through this tutorial, we have seen the complete process of simulating blood flow in a coronary bifurcation, from importing and preparing the geometry in Ansys Discovery, to setting up the mesh in Ansys Mechanical Meshing, defining boundary conditions and material properties in Ansys Fluent, and finally visualizing results in Ansys CFD-Post. By applying steady-state, laminar flow assumptions, we demonstrated how to model the fluid dynamics of blood flow in a patient-specific artery. This methodology can be expanded upon for more complex analyses involving turbulence or transient flow studies supporting improved clinical decision-making and device design.