

RSNA 2016

Introduction to Computational Fluid Dynamics from Medical Images: A Step-by-Step Demonstration (Hands-on) Training Guide

Dimitris Mitsouras
Course Director
Applied Imaging Science Lab
Boston, MA USA

Course Faculty
Marc Horner
Xingshi Wang
Todd Pietila
Elizabeth George



Introduction

Computational fluid dynamics (CFD) refers to using a computer to solve the Navier-Stokes equations, that is, the basic equations governing the motion of a viscous, heat-conducting fluid. These equations were derived by Navier, Poisson, Saint-Venant and Stokes between 1827 and 1845. They are derived by applying Newton's Law of Motion to a small fluid element, i.e., the relationship between forces acting upon it and its motion in response to those forces (referred to as the *momentum equation*), supplemented by a mass conservation equation, i.e., the requirement that fluid molecules cannot be created or destroyed (referred to as the *continuity equation*), and finally the conservation of energy equation representing the first law of thermodynamics, i.e., the requirement that the total energy in an isolated system is constant (referred to as the *energy equation*). Discretizing and computationally solving these equations in a collection of (typically) small finite "control" volumes whose collection fully defines the entire problem domain (e.g., the coronary artery tree lumen) is referred to as a CFD simulation. This simulation is iterative, with the computer choosing a potential solution to the Navier-Stokes equations (pressure, velocity, etc.), calculating any resulting imbalances in the equations given this solution, and correcting the solution by a small amount toward reducing the imbalances at each step.

CFD has been in use for nearly 3 decades toward enhancing our understanding of hemodynamics and pathophysiology. It emerged in the early 1970's in engineering, as high-speed computers became available. Applications in medicine, involving simple models such as two-dimensional "cross-section" models, began to appear in the medical literature in the 1980's (1). Interest in such "*in silico*" assessment of hemodynamics *via* CFD grew in the early 1990's following the recognition of endothelial shear stress (the frictional force exerted on the endothelium by flowing blood) as an important determinant of cardiovascular development and function (2). Today, CFD is employed to study numerous problems in cardiovascular medicine, including evaluating and optimizing prosthetic valves, stents and ventricular assist devices, assessing the significance of findings and predicting disease progression in aortic, cerebrovascular and coronary pathologies, including dissections, aneurysms, and atherosclerotic plaques, and in heart failure, pulmonary hypertension and congenital heart disease for earlier diagnosis, estimating response to surgical treatment, or better risk stratification, as appropriate. A recent thorough review can be found in (3).

In 2015, in what can be considered a major shift in CFD use from research to clinical practice, the US Food and Drug Administration approved the marketing of a CFD-based technique to estimate coronary fractional flow reserve (FFR) from computed tomography (CT) angiography (4). FFR is the ratio of pressure distal to a coronary stenosis to that proximal of the stenosis at maximum hyperemia (5). It is interpreted as the ratio of flow through a stenosed coronary artery in hyperemia, to the theoretical flow through the same artery if the stenosis were absent (6). FFR measured invasively in the cardiac catheterization lab using a pressure wire after administration of a pharmacologic stress agent such as adenosine is the gold standard to establish lesion-specific myocardial ischemia and guide the decision for percutaneous coronary intervention (7).

Given the large growth of research investigations employing CFD, and the onset of CFD to direct patient management, it is likely that radiologists will benefit from a basic understanding of CFD. Our goal is thus to educate participants about the capabilities of CFD, and, through a hands-on exercise, provide a working knowledge of how to perform a flow analysis. Armed with this knowledge, participants may be better equipped to recognize the requirements, and limitations, of the technology toward optimizing imaging protocols to successfully apply existing and future CFD-based diagnostic tools, or, to apply CFD toward new discoveries in cardiovascular diseases.

In this hands-on course we will perform a CFD simulation of blood flow in the coronary arteries starting with a patient's 3D angiography images stored in DICOM (digital imaging and communications in medicine) format. A high-level overview of the process is illustrated in Figure 1. Specifically, in this course we will calculate the pressure drop across a right coronary artery using hyperemic conditions estimated based on a simplified physiologic model, starting from a patient's coronary computed tomography (CT) angiogram.

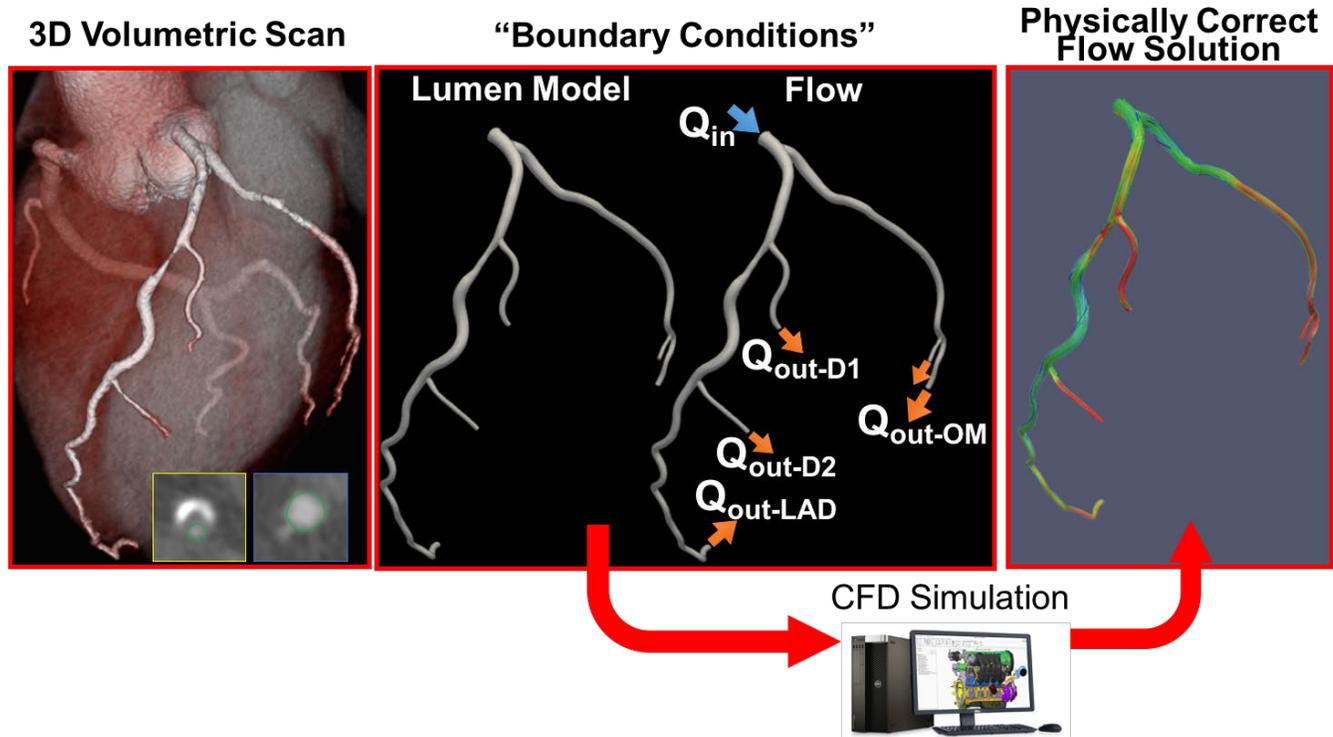
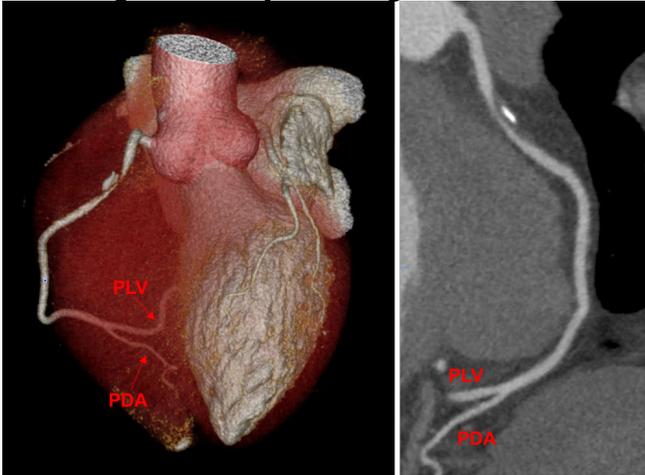


Figure 1. Overview of the CFD process starting from a 3D image dataset; the arterial lumen and the amount of inflowing and outflowing blood are estimated from available patient data. The CFD solver uses these patient-specific “boundary conditions” along with a model of blood (e.g., Newtonian fluid with fixed viscosity) to form a solution to the Navier-Stokes equations satisfying those boundary conditions for that fluid. Pressure, velocity, and surrogate properties of the solution (e.g., endothelial shear stress, vorticity, helicity, etc.) can then be interrogated.

Participants in the course will learn to use two commercial software packages: Mimics (Materialise NV, Belgium) and Fluent (ANSYS, Inc, Canonsburg, PA USA). Mimics is an advanced DICOM image-processing software package widely used in academia, hospitals, and industry that enables segmenting tissues and organs to create 3D models of the segmented anatomy for use in applications such as 3D printing, finite element analysis (FEA) and CFD. Fluent is an extensive CFD software tool that offers validated physical modeling capabilities for a wide range of CFD and multiphysics applications. Other softwares can be used for segmentation and CFD, including open source packages. This information, as well as more advanced topics than covered in this training guide, are discussed in the educational exhibit “Computational Fluid Dynamics for Patient-Specific Assessment of Blood Flow”, IN116-ED-X available in the informatics community at the Lakeside Learning Center.

CT-FFR Case

In this exercise, we will be working with the contrast-enhanced coronary CT angiogram (CTA) of a 48-year-old male patient with hypertension and dyslipidemia who presented with atypical chest pain and that had no personal or family history of CAD. Quantitative coronary CTA demonstrated a 59% stenosis of the proximal RCA (AHA segment 1). The patient then underwent elective catheter angiography, which demonstrated a 61% stenosis of the corresponding segment and an FFR measurement of 0.85, indicating no hemodynamic significance of this obstructive ($\geq 50\%$) lesion.

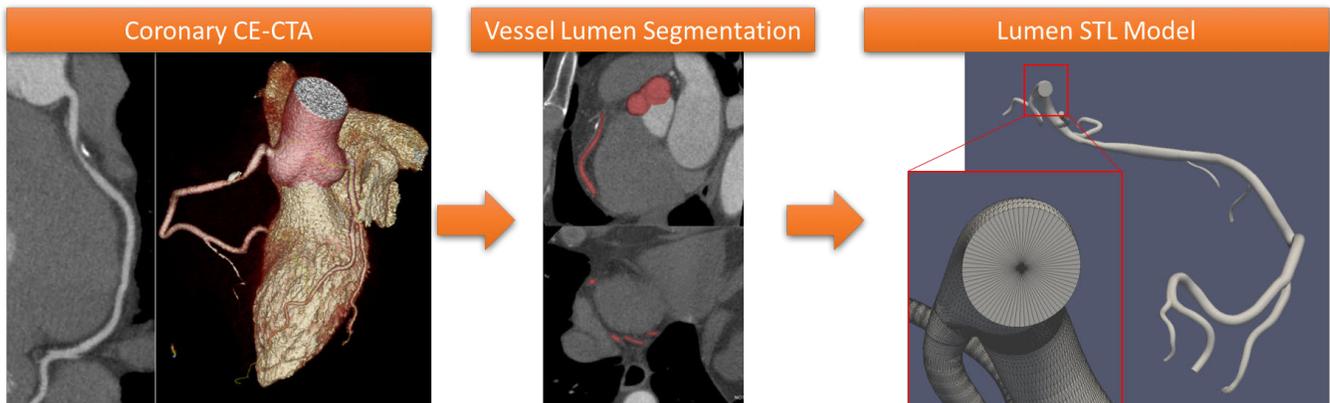


We will first use a semi-automated coronary segmentation tool in Mimics to segment the right coronary artery and its two terminal branches, the posterior descending artery (PDA) and posterior left ventricular branch (PLV) from the CTA and create a 3D model. We will then export the model in the Standard Tessellation Language, or STereo Lithography, (STL) file format. The STL file will then be imported into the CFD software (ANSYS Fluent) and we will generate a finite volume mesh to fill the lumen defined by this STL. We will finally solve the Navier-Stokes equations in this mesh simulating blood flow at hyperemic conditions, and we will interrogate the solution for pressure and CT-FFR after setting the coronary pressure at the ostium to that measured in the patient using a sphygmomanometer.

Actions that must be taken by the user are indicated in **bold text with a yellow background** below. Additional information is provided in *italicized text with a gray background*.

Step 1: Coronary Segmentation and STL Model Design

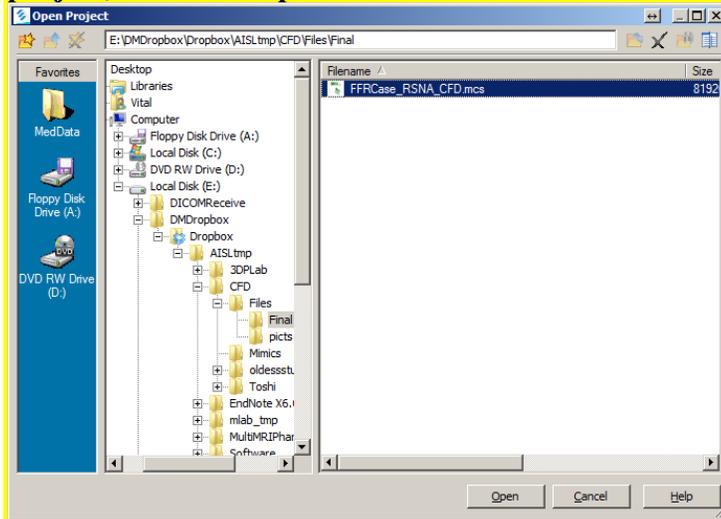
The first step involves segmenting the coronary artery blood pool from the contrast-enhanced images and creating an STL model of the endoluminal surface that has flat surfaces at the inlets and outlets.



We will perform these steps in Mimics. Our goal is to segment the right coronary artery from the ostium and past the PDA-PLV bifurcation as shown below. We will begin by:

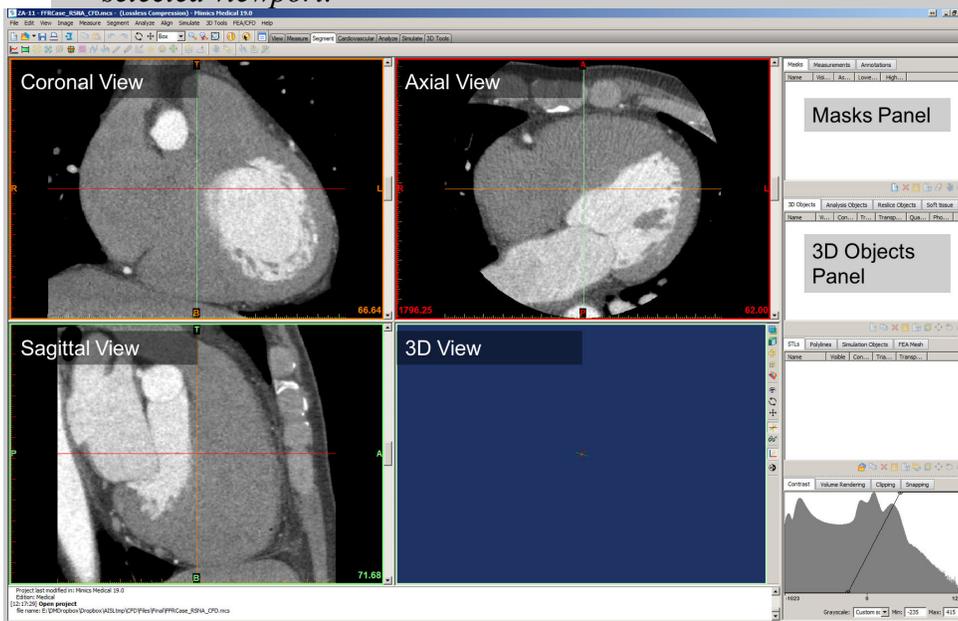
1. Open Materialise Mimics from the Start Menu or desktop icon ().

2. Select Open Project from the File menu () and navigate to the FFRCCase_RSNA_CFD.mcs project, and select open.

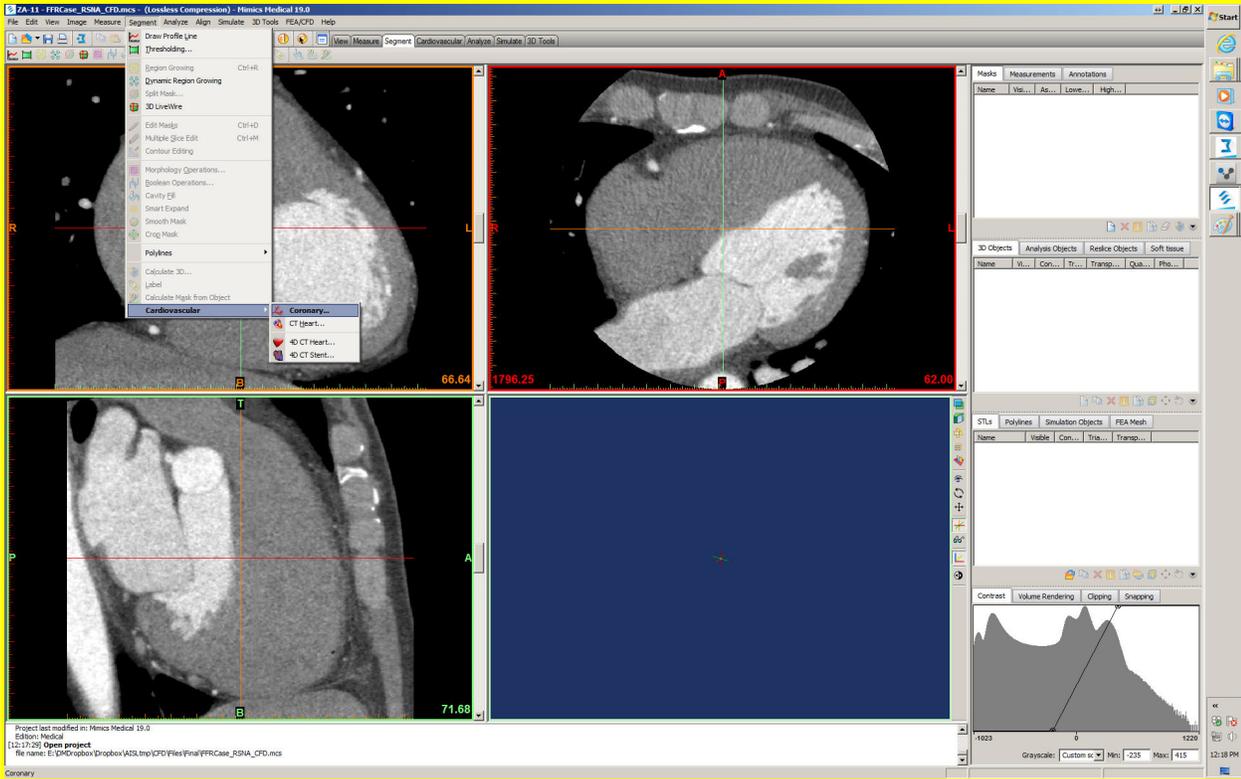


Note: You have now opened the pre-saved project we created from the DICOM images of the patient's CTA. If you are instead starting with the DICOM images, use the "New project wizard" from the file menu and go through the steps of importing & confirming the DICOM images. First, you should familiarize yourself with the layout of the Mimics program, as well as mouse and keyboard shortcuts.

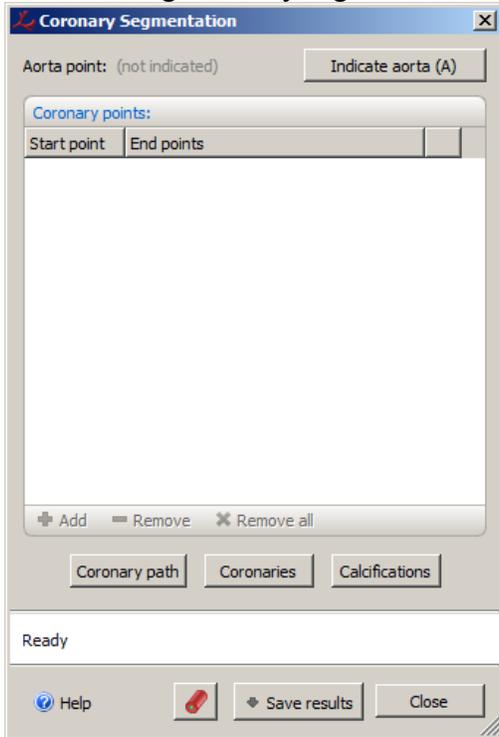
- There are four viewports in the program. Clicking inside any viewport and hitting space bar will "zoom" that viewport to full screen. Hitting space bar again will return to the 4-viewport layout.
- The mouse scroll wheel allows scrolling through the CT slices.
- Right mouse-button click and drag in the image viewports adjusts the window/level
- Right mouse-button click and drag in the 3D viewport rotates the view.
- Holding down the Control (Ctrl) keyboard button and right-clicking and dragging the mouse allows zooming in/out the image in the viewport.
- Holding down the middle mouse button (scroll wheel) and dragging will "pan" the image in the selected viewport.



3. In the Segment menu click the Cardiovascular sub-menu, and choose the Coronary tool ().



The following coronary segmentation tool window should now appear:



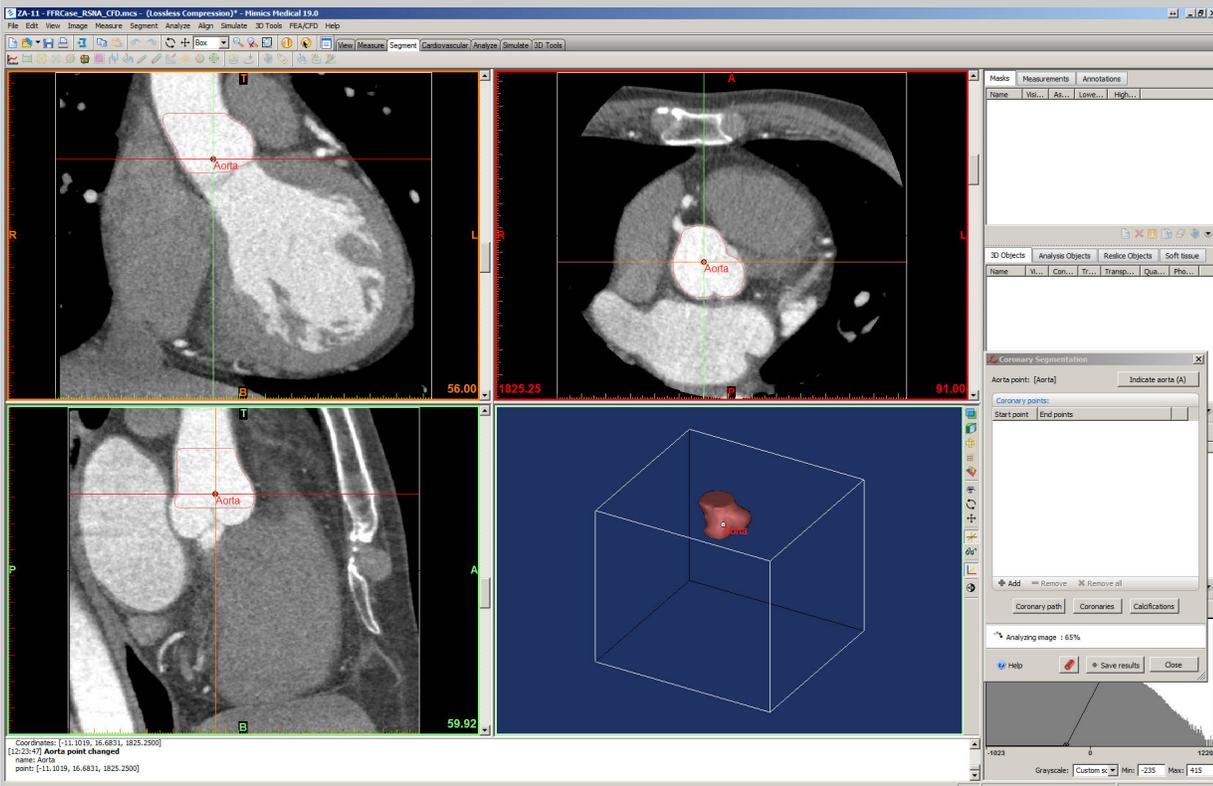
Note: The coronary segmentation tool is designed for semi-automatic segmentation of coronary arteries on contrast enhanced CT images using user-provided start and end points. The output of the tool is a mask of the vessel lumen of the coronary artery. To use this tool efficiently, it is recommended that a region of interest box is used to define the volume used for the calculations (this box can be resized if the dataset is large). Then, a point will be used to indicate the location of the aorta at the level of the coronary ostia. Next, end points will be indicated along the path of the coronary artery. The tool will automatically create a coronary path connecting the start and end points, and then, the lumen of the vessel will be segmented automatically along the path.

4. In the axial viewport (top left viewport surrounded by red outline), scroll up to slice 91.50 (the slice position is shown in the bottom left corner of each viewport).

5. In the coronary segmentation window, select the Indicate Aorta button. Then move your pointer in the axial viewport, and *left click* to indicate the approximate center of the aorta on axial slice 91.50.



Note: In the Coronary Segmentation window, you will see the status update as the tool segments the aorta. Once finished, a preview of aorta will be displayed in the 3D viewport, and contours of the 3D preview will be overlaid on the 2D DICOM images.



6. In the Coronary Segmentation tool window, click the Add button. In the axial viewport, scroll to slice 91.00 and left click to indicate the location of the right coronary artery starting point. Be sure to indicate the starting point just outside of the aorta area identified by the contour line.

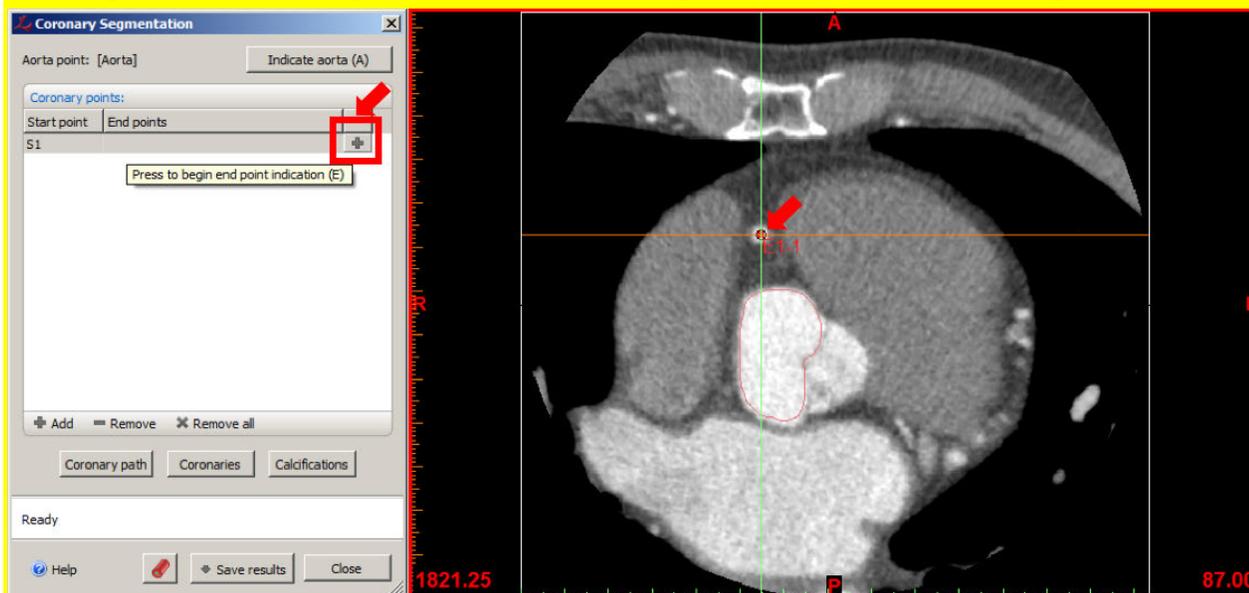


Next, we must indicate the endpoints of the coronary artery.

Note: The coronary segmentation algorithm will find the coronary path connecting the start point with the end points (note that endpoints can be as many midpoints as you need to guide the coronary segmentation algorithm to identify the correct path of the coronary artery). In this exercise we will set a number of these endpoints. As you add endpoints, the detected coronary path is shown in the 3D viewport as a green line, and the segmented coronary is lumen is shown as a deep red 3D rendering.

We will first add one “way point” early along the course of the coronary artery.

7. Scroll to axial slice 87.00. In the Coronary Segmentation window, click the + button to initiate the endpoint indicator. Left click in the axial viewport to indicate the coronary lumen.



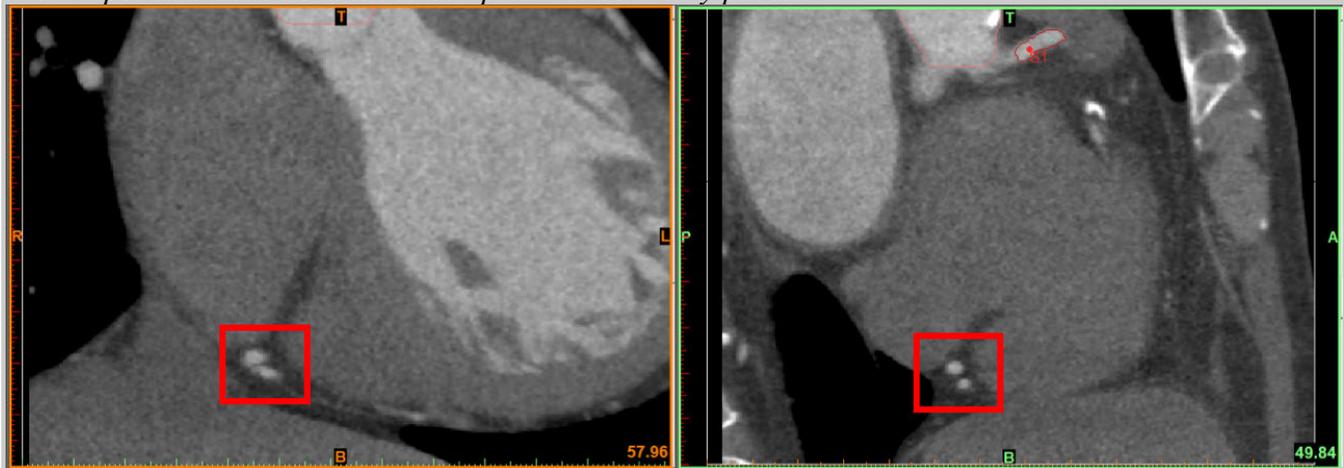
As we scroll through the axial slices, you will see that there is a significant stenosis with both soft and calcified plaque. The lumen is still visible on all the slices, but with reduced Hounsfield units due to beam hardening and partial volume effects due to its small size. Since this is a difficult region for the algorithm, we will mark one coronary end point in this stenosed region.

8. Scroll to axial slice 77.25. In the Coronary Segmentation window, click the  button to initiate the endpoint indicator. Left click in the axial viewport to indicate the endpoint location in the lumen.

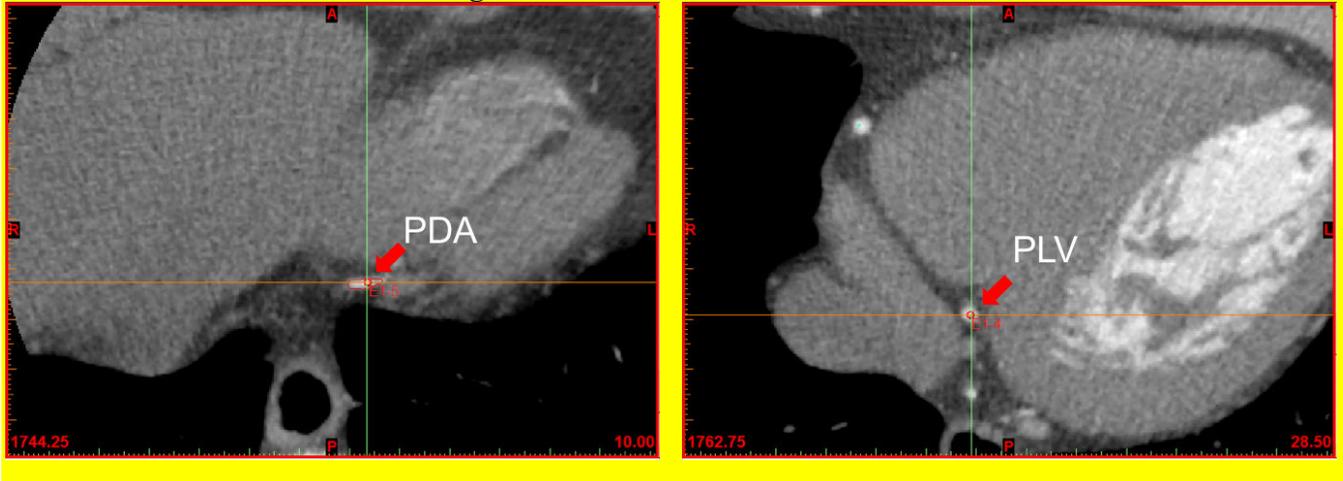


Note: Do not worry if the calcium is picked up in the coronary segmentation. In a future step, we will manually remove these areas of calcium from the coronary artery segmentation.

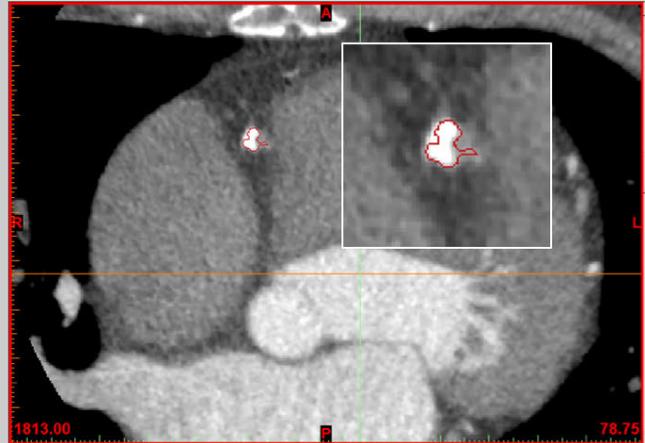
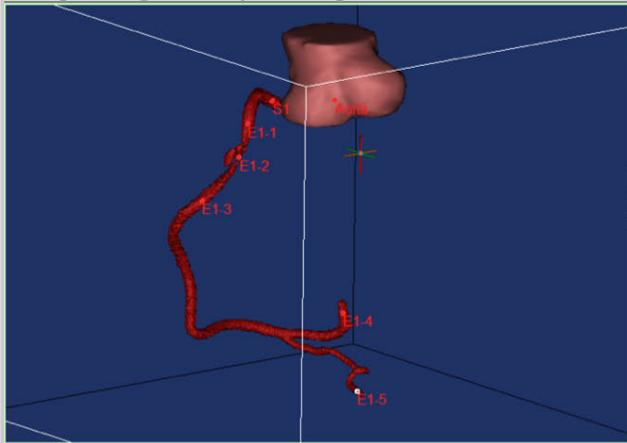
Note: The remainder of the artery is largely free of coronary artery disease, and we can expedite the segmentation by continuing to the location after the PDA-PLV bifurcation (figure), marking the final two endpoints on each branch to complete the coronary paths.



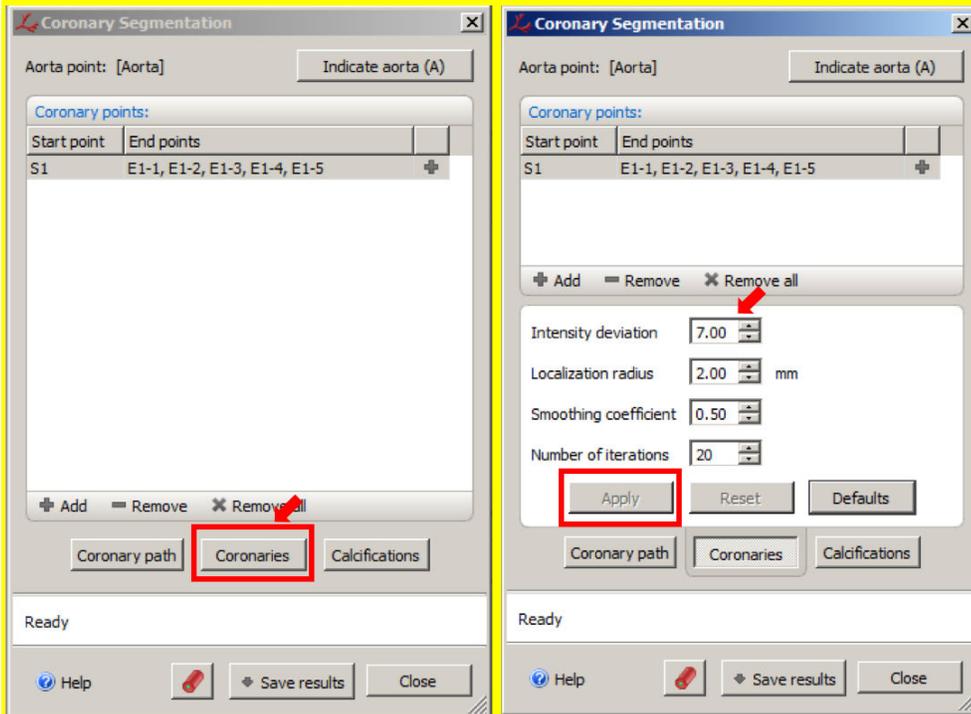
9. As before, add two more endpoints, one for each branch on axial slices 10.00 for the PDA and one at slice 28.50 for the PLV using the **+** button.



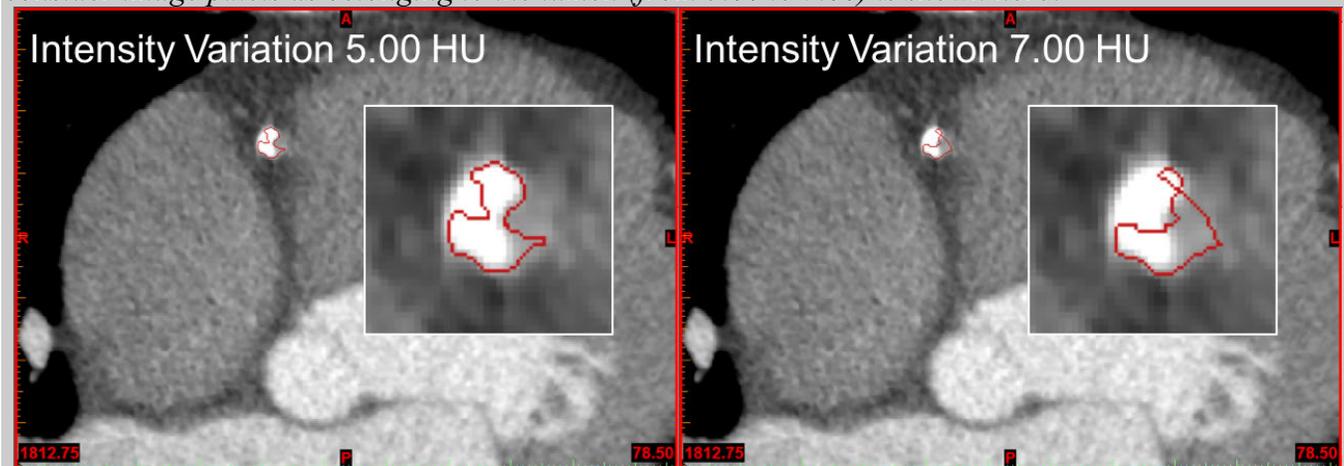
Note: As mentioned earlier, the coronary segmentation tool calculates the coronary path and lumen as the user is placing the endpoints. At every step a 3D preview of the coronary segmentation is displayed in the 3D viewport. If the above steps have been completed, the figure below on the left should be the 3D visualization you see in the 3D viewport. Furthermore, examining the 3D preview, it is clear that the algorithm “picked up” the calcium as coronary lumen, and missed the lumen where the HU is significantly reduced due to partial volume effects. We can partly (but not completely) adjust for this using the options of the algorithm.



10. Click on the Coronaries button in the Coronary Segmentation tool window. Change the Intensity deviation to 7.00 to increase the gray values in the images that are included in the coronary segmentation. Click Apply.

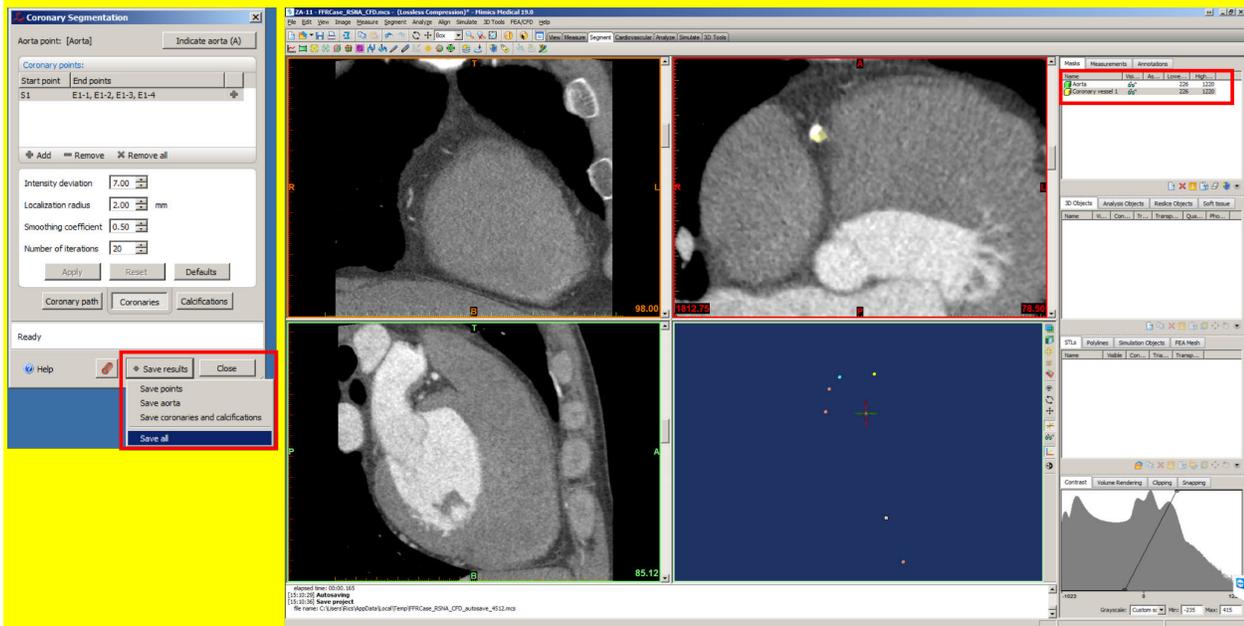


Note: The effect of performing this change for the acceptable intensity variation for the algorithm to consider image pixels as belonging to the lumen (from 5.00 to 7.00) is shown here.



We are now done with the semi-automated coronary segmentation.

11. In the Coronary Segmentation window, click the Save results button and choose to Save all. The results, a mask of the right coronary artery lumen and a mask of the aorta, will appear to the Masks tab of the Project Management area. Finally, close the coronary segmentation tool.



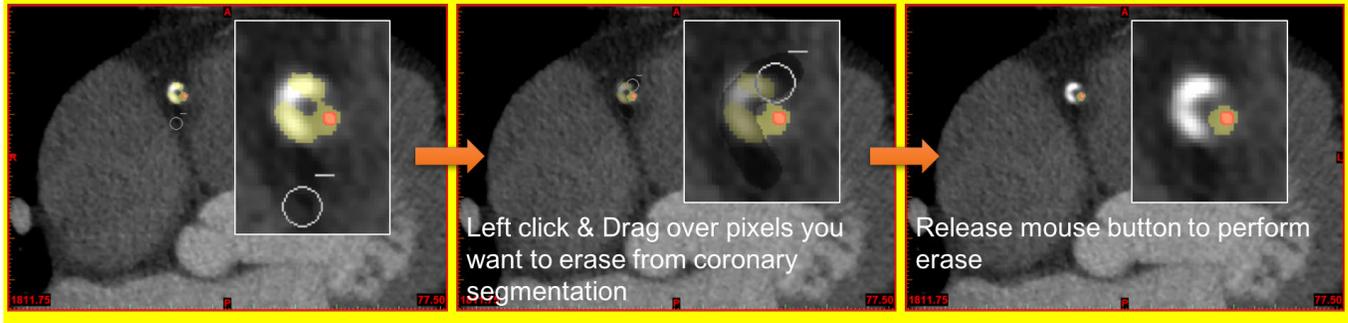
Next we will clean up the coronary artery segmentation by removing the calcifications that were picked up by the Coronary Segmentation tool.

12. From the Segment menu, choose the Edit Masks tool (). In the Edit Masks window, be sure the type is set to circle (i.e, brush tool) and that the Erase operation is selected. We recommend a width/height of 12 for this exercise.



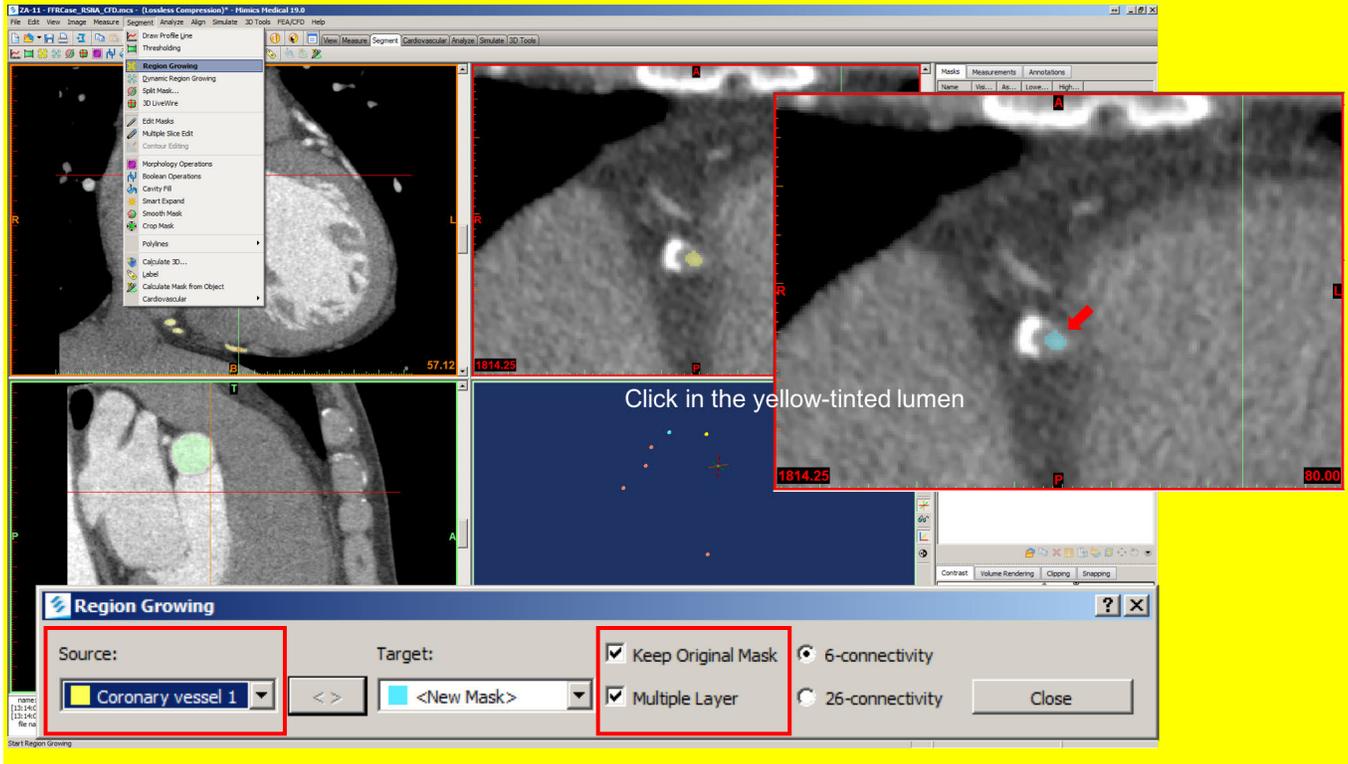
Note: You can also perform free-hand editing using the “Lasso” rather than “Circle” tool, and you can edit the lumen to add pixels rather than remove them from the lumen segmentation.

13. Use the brush tool to erase the calcium portions of the yellow-tinted mask by *left clicking* in the axial viewport. Any location you click (and drag) the brush tool over will be erased from the segmentation, and will no longer appear with the yellow tint. Be sure to edit every slice between axial slice 75.00 and 85.00.



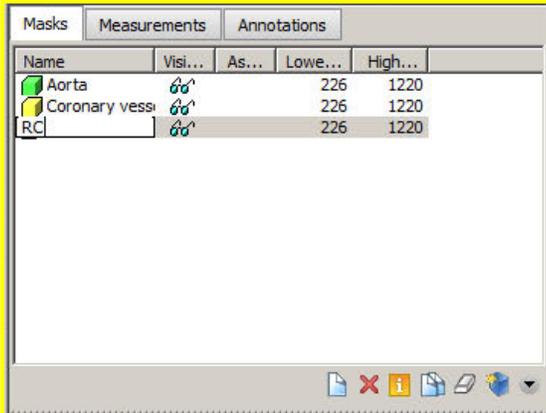
When we edited the segmentation, it is possible that some calcium pixels that we missed with the erase tool remain, but are most likely disconnected from the main segmentation (if for example our brush wasn't large enough to cover all the calcium on the far side of the lumen). We will now remove any such “dangling” portions of the segmentation.

14. To remove any portions from the segmented coronary mask that are not connected to the coronary artery lumen, choose the Region Growing tool (🌱) from the Segment menu. Make sure the “Coronary vessel 1” is selected in the “Source:” of the Region Growing Tool. Then, *left click* on any area of the yellow-tinted mask that is in the coronary artery lumen in the axial viewport. This will create a new mask named “Cyan” in the Masks tab of the Project Management area.

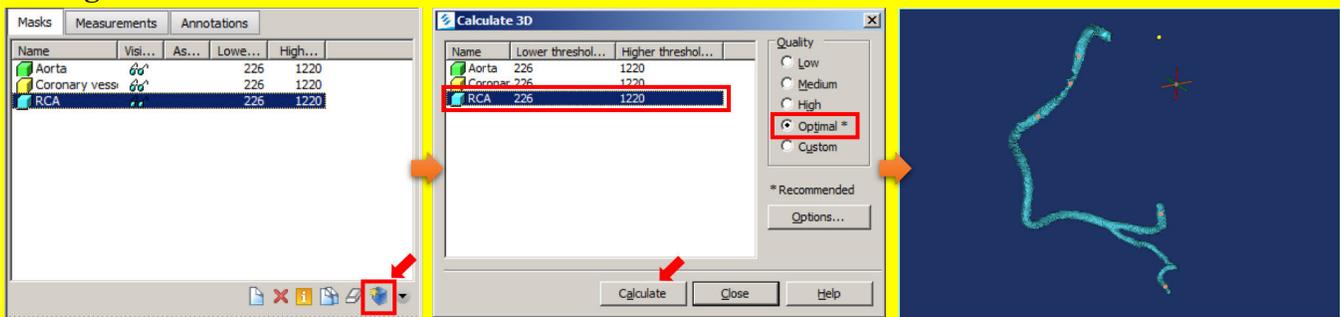


We are now finished with segmentation and are ready to make our 3D model to use for CFD! Before proceeding to CFD however, we will perform a few additional steps to smooth and trim the 3D model.

15. First, let's rename the Cyan mask to "RCA".

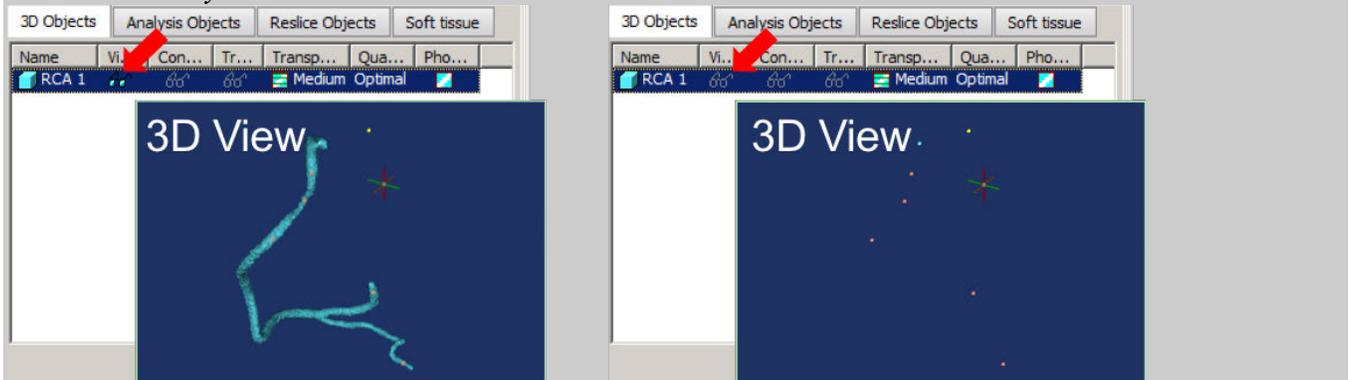


16. To generate the 3D model of the coronary artery, in the bottom left corner of the masks panel (or alternatively from the Segment menu-> Calculate 3D), click the Calculate 3D tool (). Check that the "RCA" mask is selected, and that Quality is set to Optimal and click Calculate. The new 3D object will be displayed in the 3D viewport and listed in the 3D Objects tab of the Project Management area.

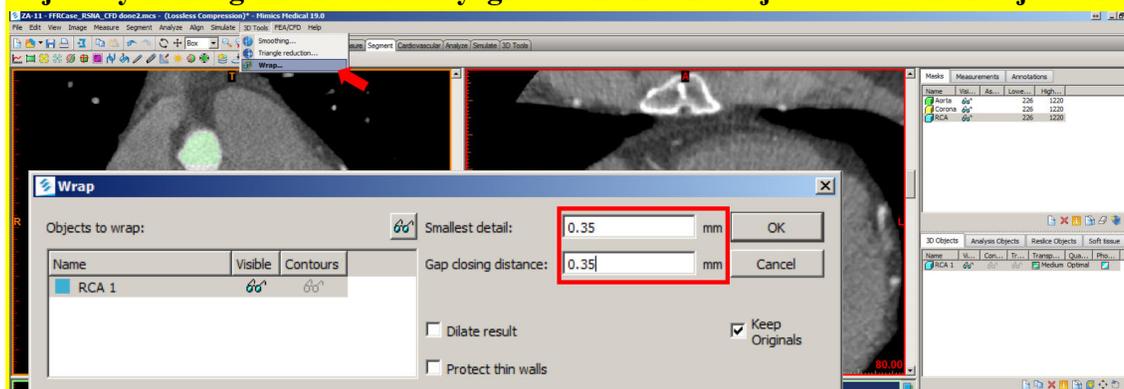


We will now clean up the 3D model to smooth out noise in the luminal surface. The surface is noisy because of imperfections in segmentation and because the intrinsic resolution of the CT scanner is ~0.35 mm in-plane and 0.5 mm through-plane (detector size). Variations in the model less than those are due to image noise causing individual pixels to flip between considered as being in the lumen or not.

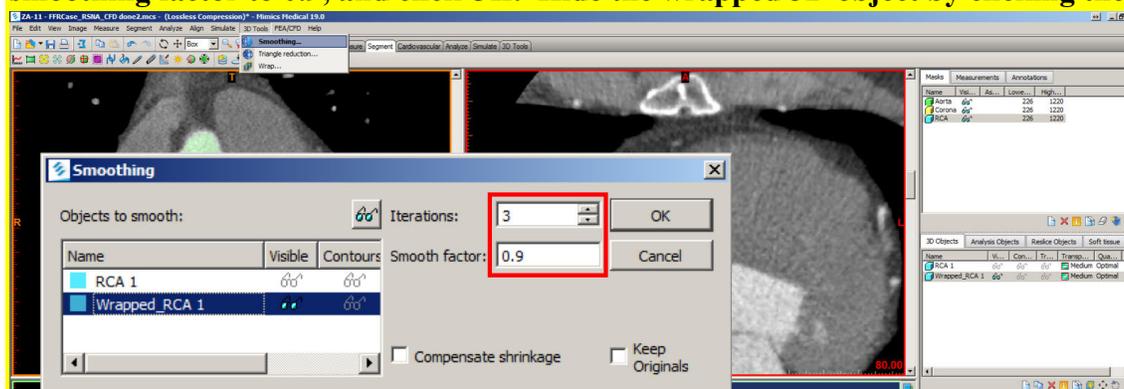
Note: In the 3D Objects panel (middle panel on left side of Mimics window), you can click the visibility eyeglasses to show or hide the 3D model from the 3D viewport. In the following steps we will hide prior 3D models every time we create a new one.



17. We will first “wrap” the model. Go to the 3D Tools menu, and choose the Wrap tool (). Make sure the “RCA” 3D object is highlighted in the Objects to wrap list that appears. Set the smallest detail and the gap closing distance to 0.35 mm, and click OK. Hide the original RCA 3D object by turning off the visible eyeglasses in the 3D Objects tab of the Project Management area.

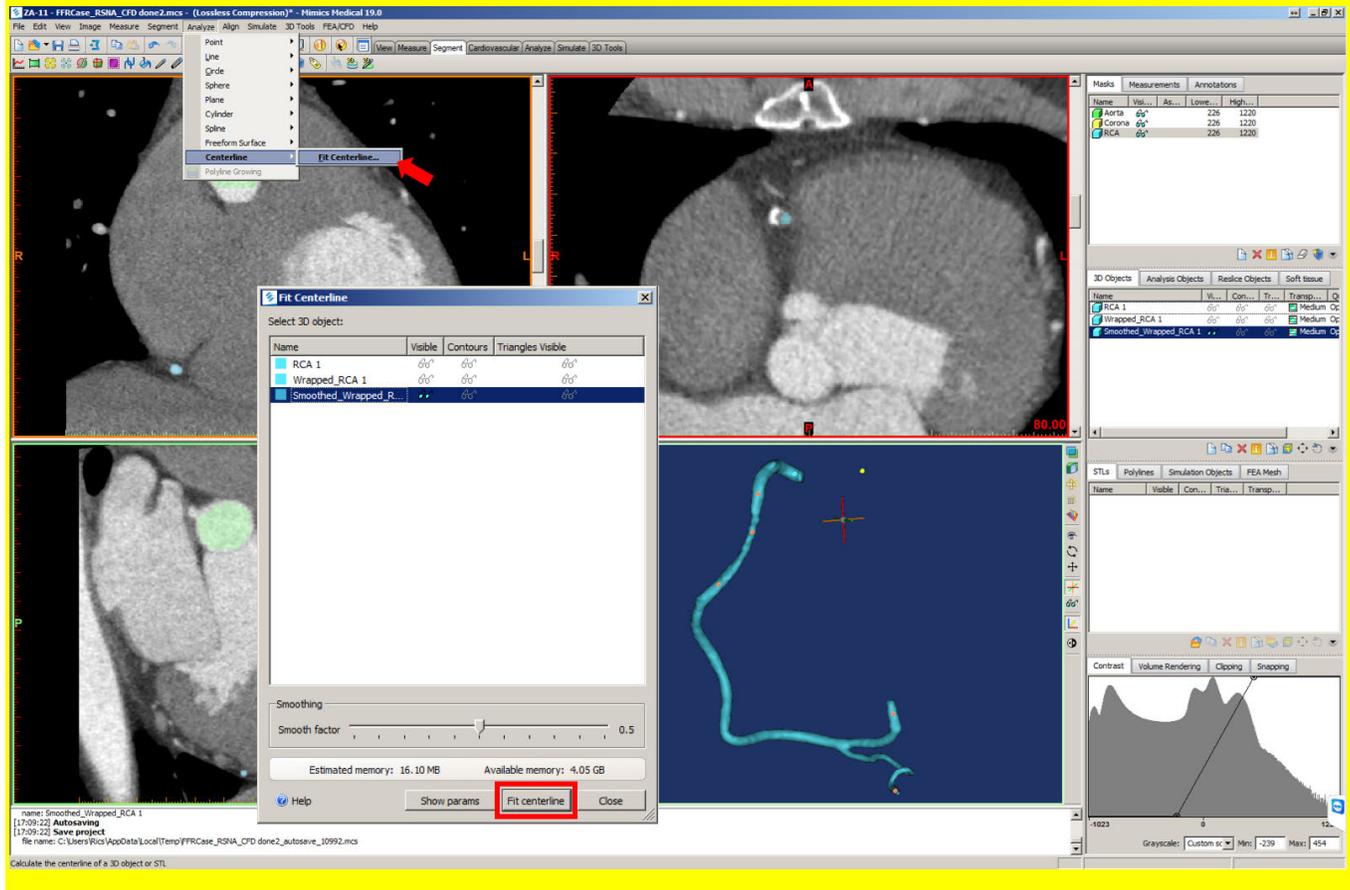


18. Next, we will smooth the model. Go to the 3D Tools menu, and choose the Smoothing tool (). Select the Wrapped_RCA object from the Objects to smooth list. Set the iterations to 3 and the smoothing factor to 0.9, and click OK. Hide the wrapped 3D object by clicking the eyeglasses.

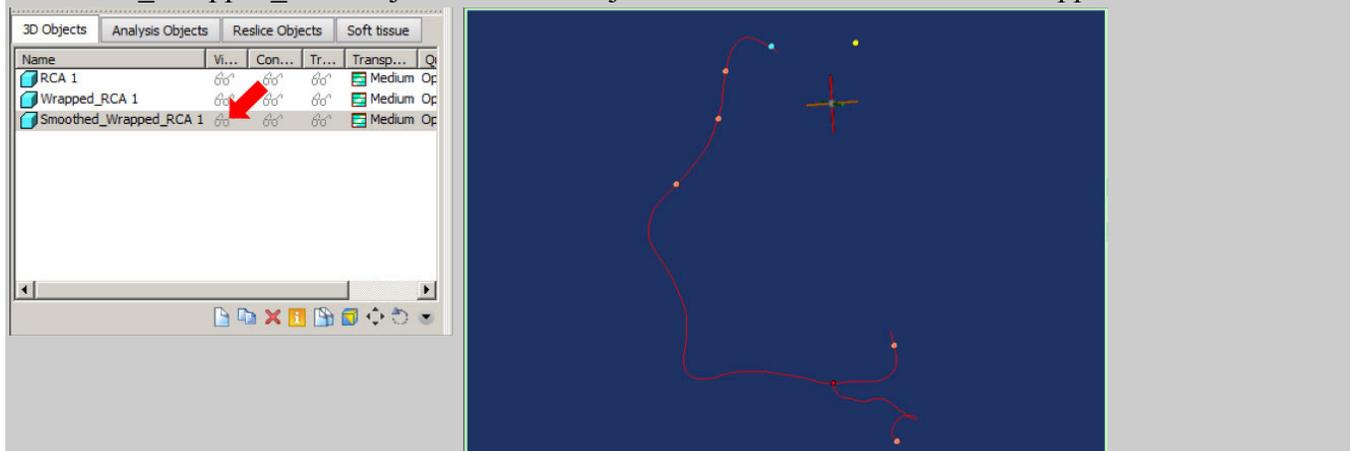


Note: The final modification required for the luminal STL model is to make flat surfaces at the inlet and outlets of the coronary tree. Flat surfaces are required in order to be able to easily tell the CFD software where flow is entering and exiting.

19. To accurately trim the inlets and outlets of the 3D model orthogonal to the vessel course, we will first fit a centerline. Go to the Analyze menu, click the Centerline sub-menu, and choose the Fit Centerline tool. Select the Smoothed_Wrapped_RCA object from the Select 3D object list and click *Fit Centerline*. Once the centerline has been created, close the Fit Centerline window.

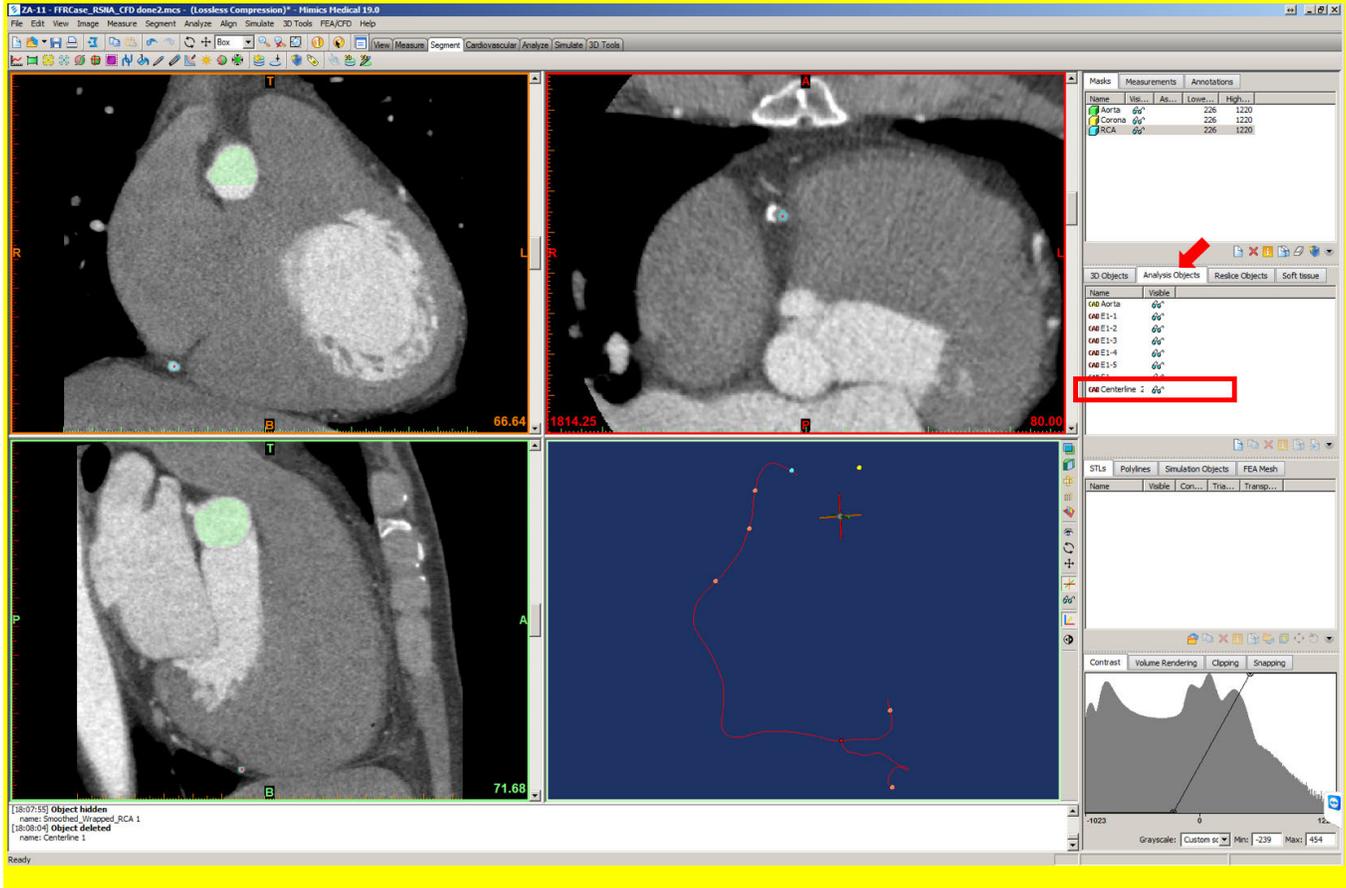


Note: to show the centerline in the 3D viewport, click to check off the Visible eyeglasses of the Smoothed_Wrapped_RCA object in the 3D Objects list. The centerline will then appear in the 3D view.

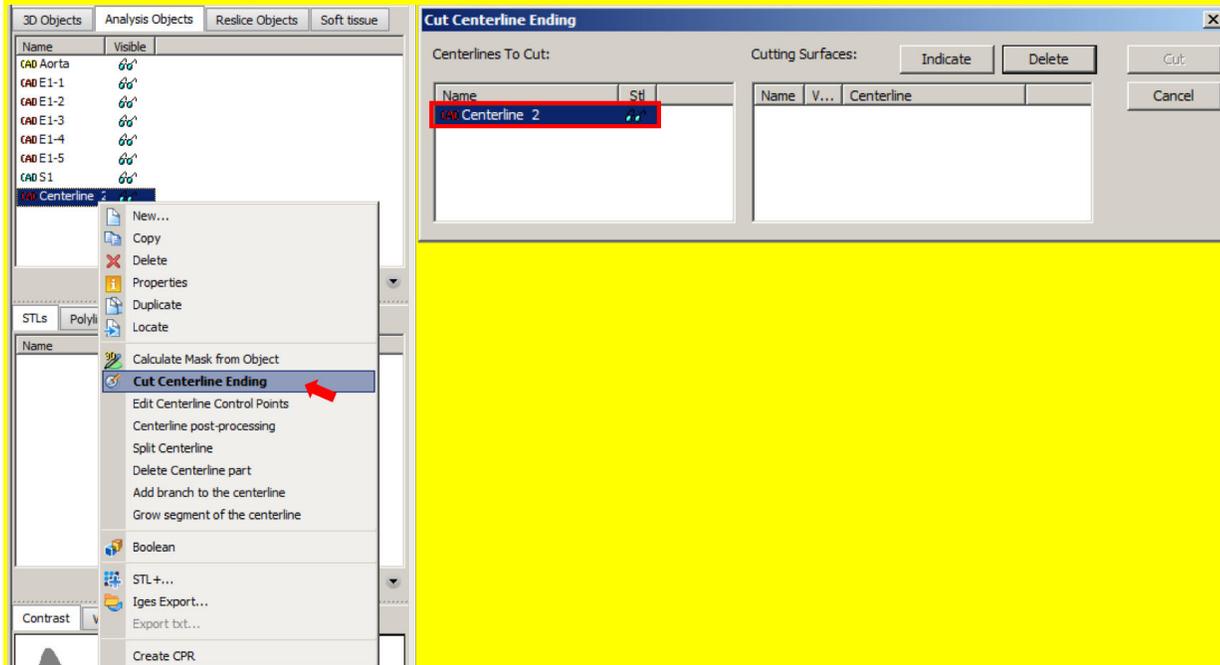


Next, we will cut the endings of the coronary 3D object *perpendicular* to the calculated centerline.

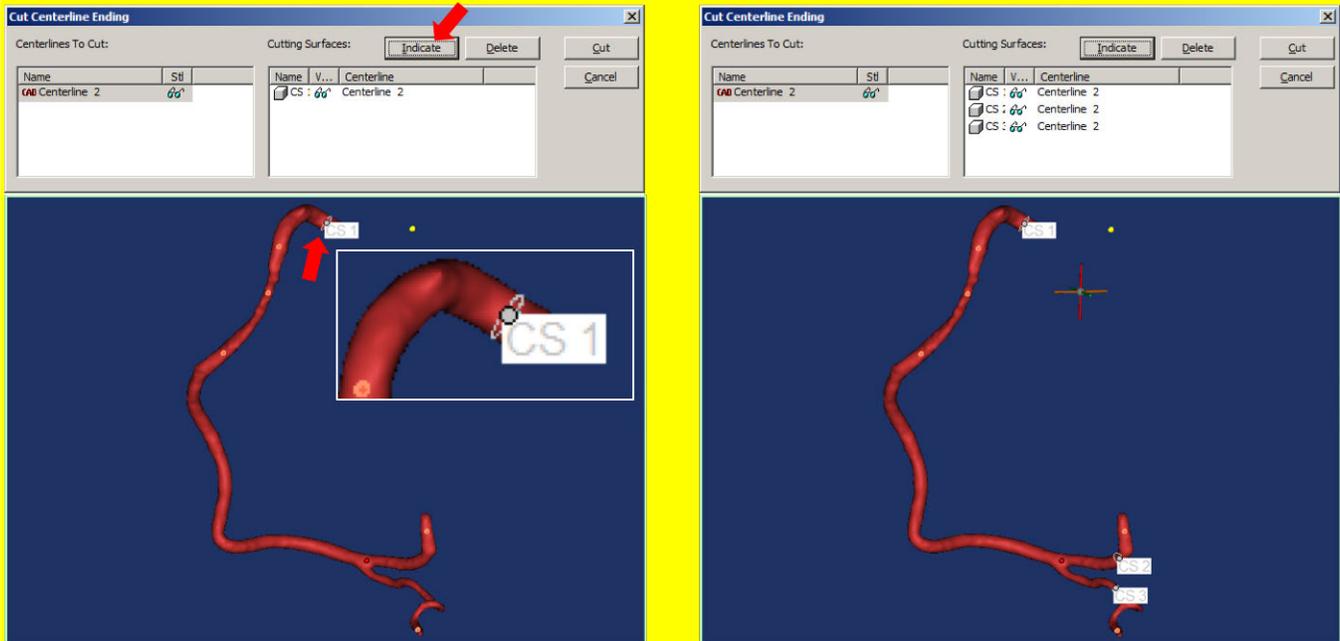
20. Switch to the “Analysis” tab in the Objects Project Management area. You should see the “Centerline” object at the bottom of the list.



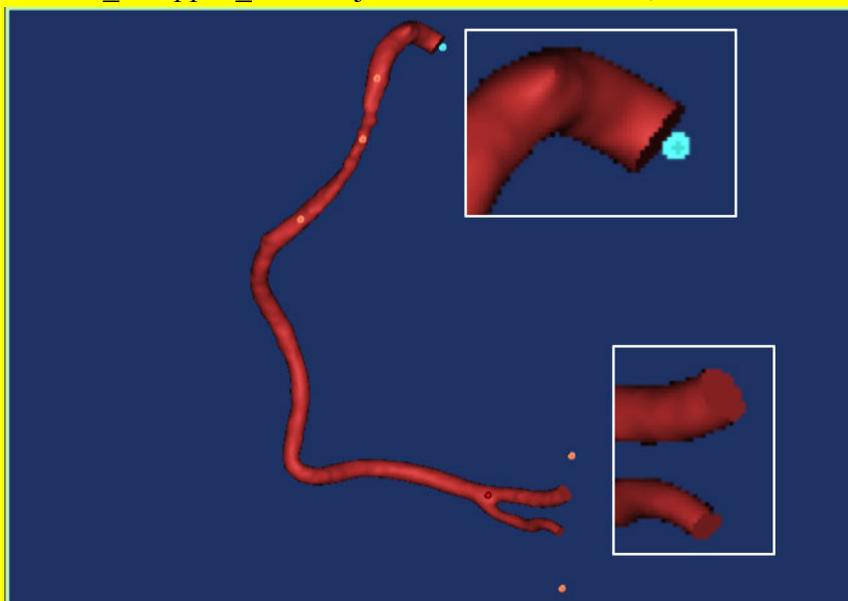
21. Right click on the Centerline and choose Cut Centerline Ending. This will bring up the Cut Centerline Ending window. Make sure that the centerline is selected in the Centerlines to Cut list.



22. In the Cut Centerline Ending window, click the *Indicate* button, and move the mouse pointer in the 3D viewport and over the vessel ostium; an indicator in the form of a pale white circle will appear on the surface of the vessel which will move across the vessel along with your mouse pointer. When you have selected an appropriate location to cut the vessel, *left click* to fix the cut location. Note cutting is not yet performed; a listing of the selected cut appears in the window. Add 2 more cut points at the PDA and PLV branches to have a total of 3 cuts.



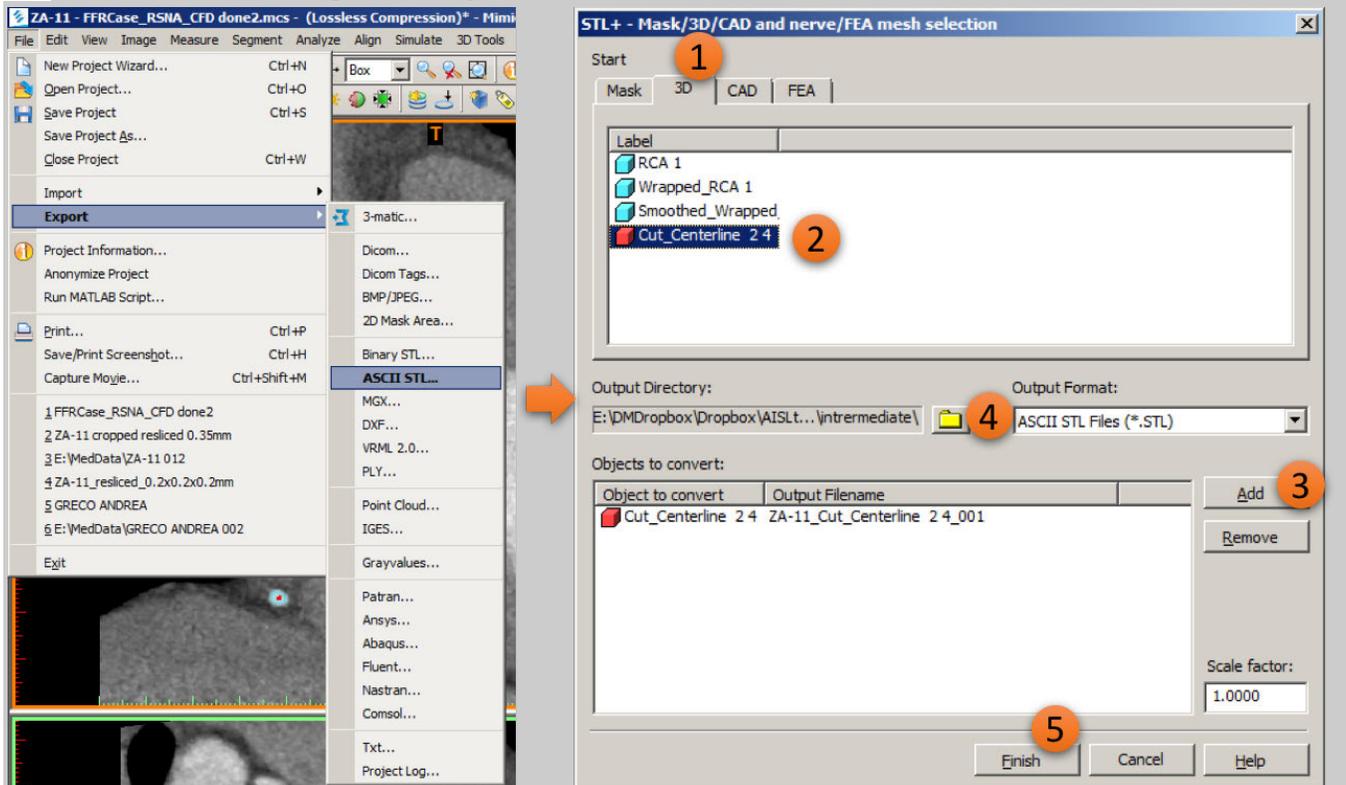
23. Once all 3 cutting points have been selected, click the *Cut* button to perform the operation and produce the final 3D model with flat surfaces at inlets and outlets. A new 3D object will be created after the cut operation; returning to the 3D Objects tab of the Project Management area and hiding the Smooth_Wrapped_RCA object will show the final, cut model.



The segmentation portion of the exercise is now complete!

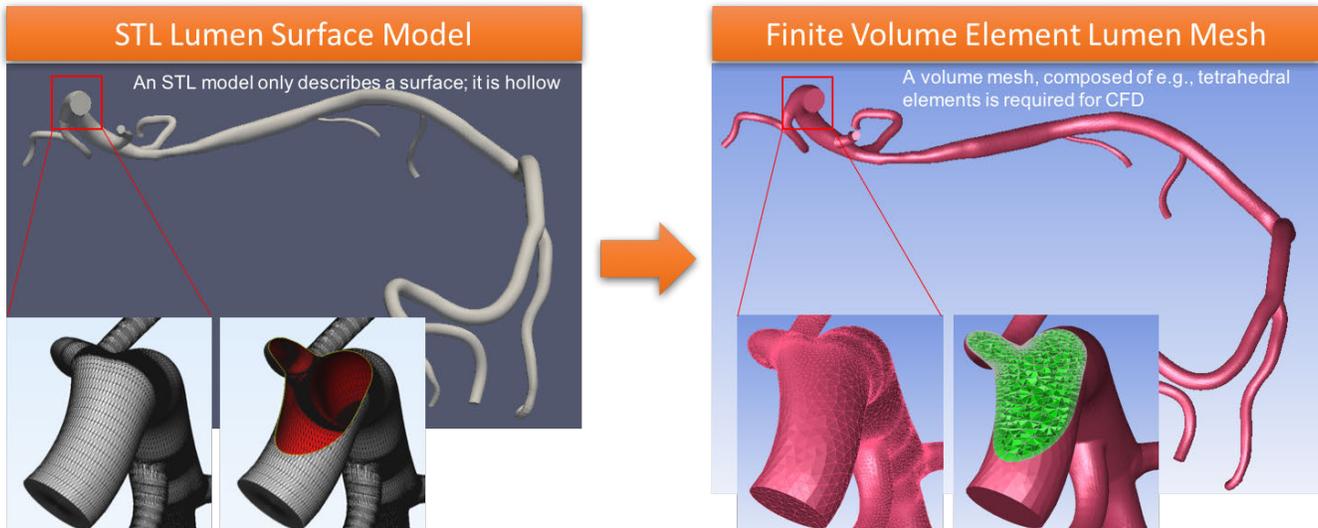
Note: Now that the segmentation is completed, the next step is to export the 3D model as an STL file so that it can be used for CFD. We will not do this step since we have already saved an STL file on the computers in case you encountered any issues while segmenting the case. In order to export the coronary model you segmented, you would go to the Export sub-menu in the File menu, and choose **ASCII STL**. Once the export window appears, you would follow the below steps to export:

1. Click the “3D” tab at the top
2. Highlight the final 3D object you want to export.
3. Click the *Add* button to select the selected (blue highlighted) model for export.
4. Choose an output directory where you would like to save the STL file and
5. Click *Finish* to export the object as an STL file.

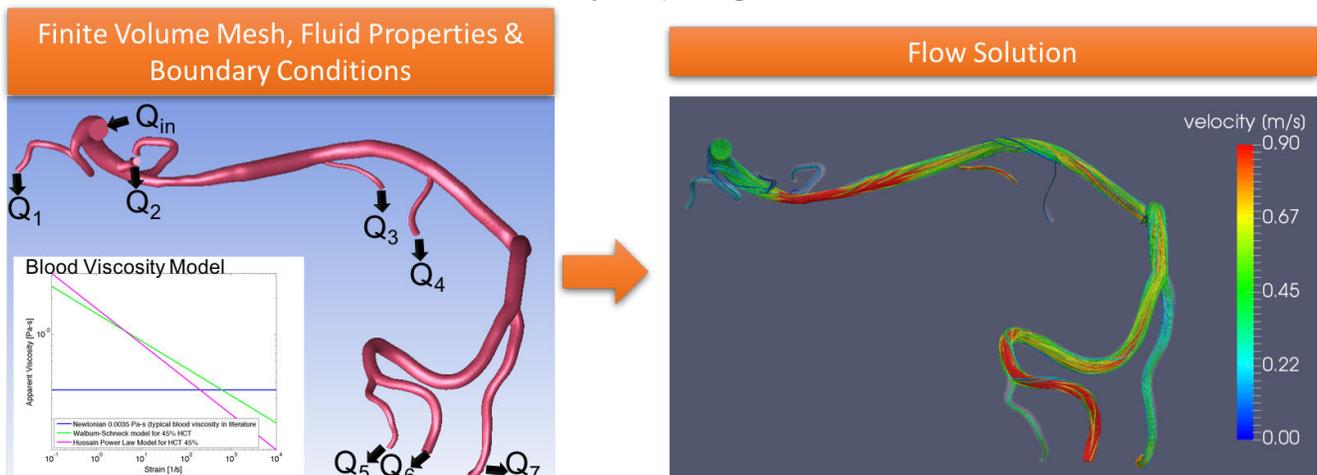


Step 2: CFD Simulation using the Segmented Coronary STL

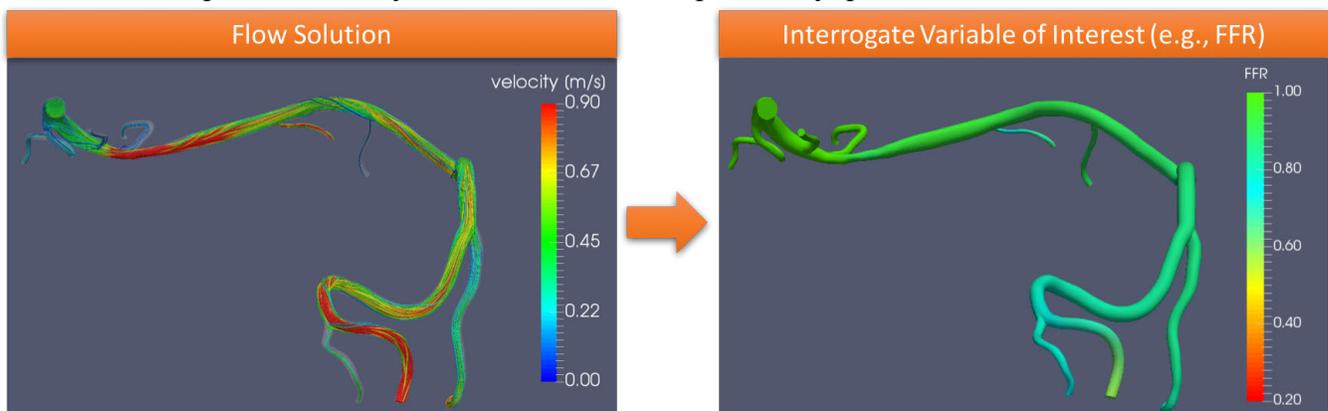
Any 3D model of the lumen that has been created and stored as an STL can be used for CFD. In order to do so however, the luminal volume encompassed by the STL surface needs to first be populated with small “finite volume” elements in which the CFD solver can calculate the Navier-Stokes equations. This will be the first step we will perform in the CFD software package Fluent.



After we have created the finite volume mesh, we will set up the characteristics of the fluid that we will simulate (i.e., blood) along with the flow boundary conditions (e.g., how much blood flows into the inlet and how that blood is distributed in the coronary tree) and perform the CFD simulation in Fluent.



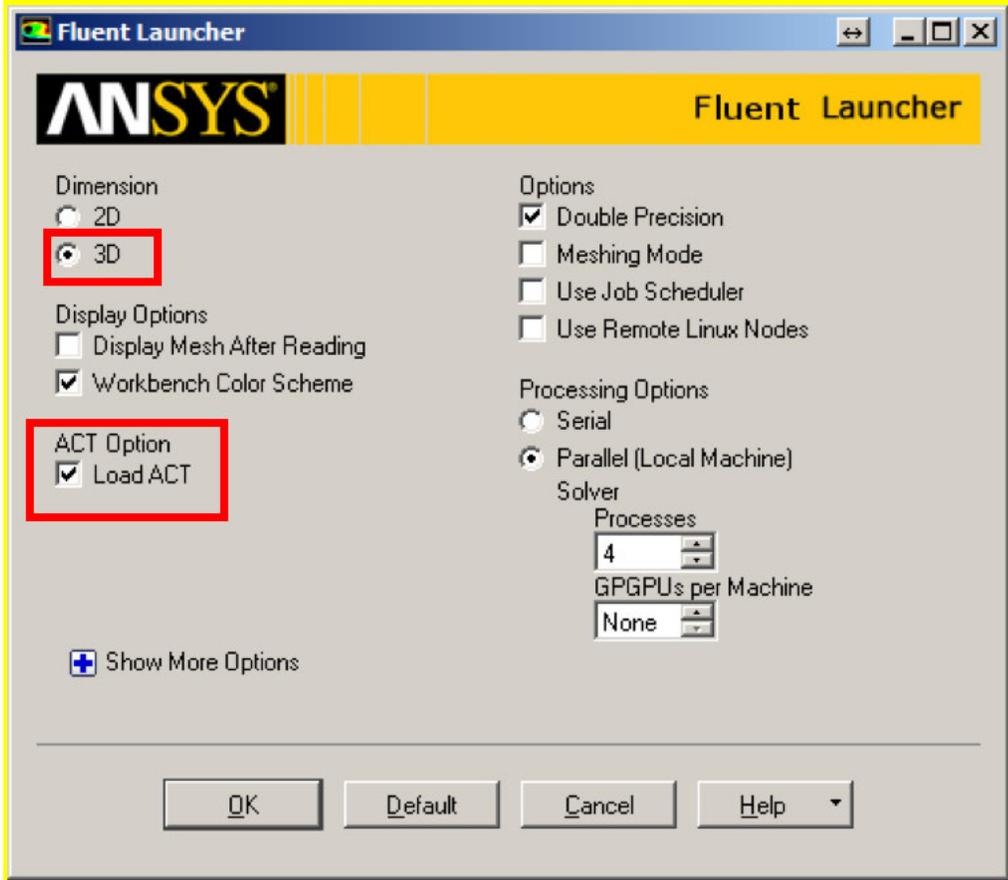
Once the CFD solution has been obtained, we will use Fluent’s post-processing options to interrogate the solved flow problem for any variables of interest, specifically, pressure and FFR.



For the purposes of this exercise we will use a “toolkit” in Fluent that simplifies these steps specifically for blood flow simulations. The user who wishes to perform each step with full control of the techniques and parameters needed to perform CFD is referred to the educational exhibit “Computational Fluid Dynamics for Patient-Specific Assessment of Blood Flow”, IN116-ED-X.

We will begin by:

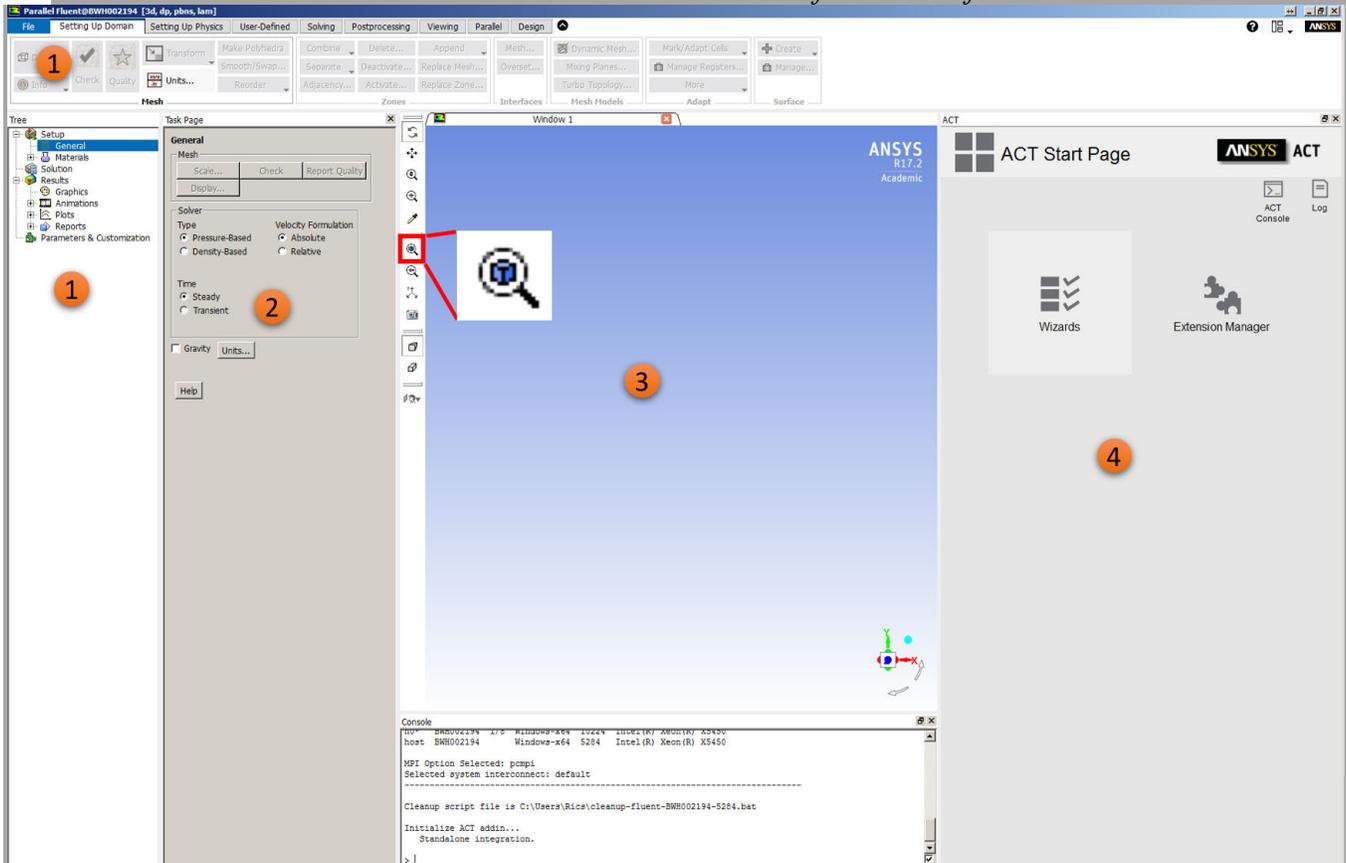
24. Opening ANSYS Fluent from the Start Menu or desktop icon (). Ensure that the “Load ACT” option is checked to enable using the blood flow toolkit, and that 3D mode is selected, and click OK. This will launch the Fluent software.



Note: You have now started Fluent. The Fluent window is separated into 4 zones (3 if you do not use the ACT. First, you should familiarize yourself with the layout of the Fluent program (numbers correspond to the figure):

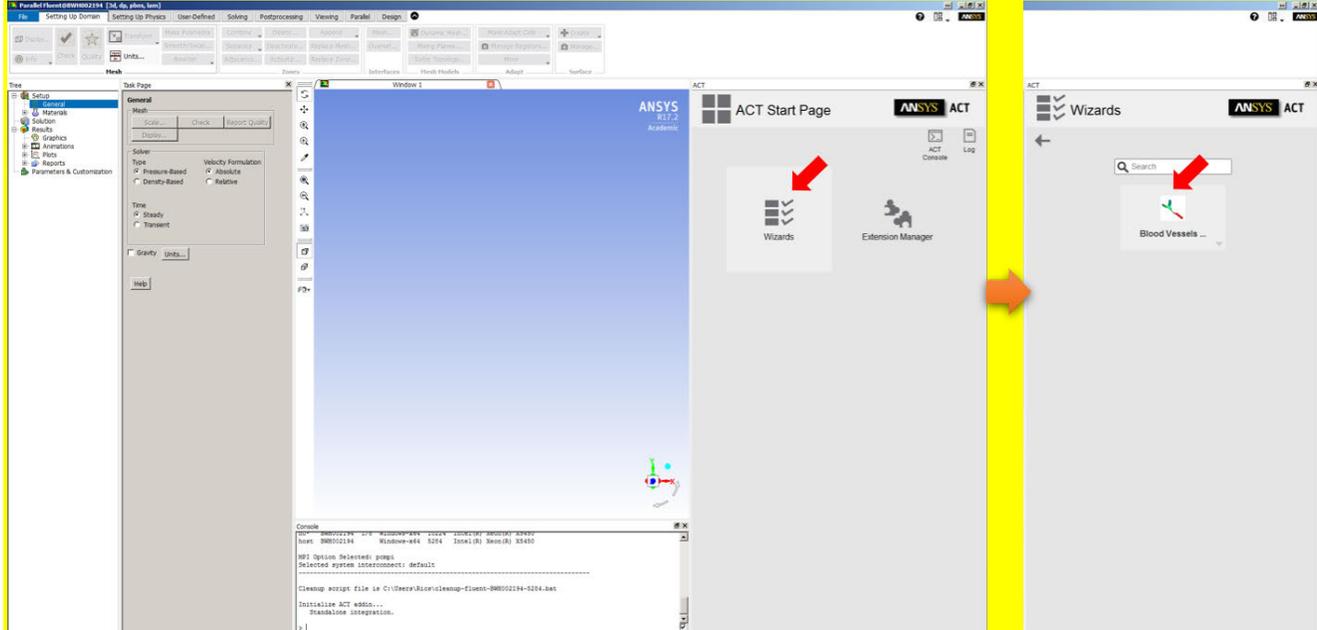
1. The banner at the top of the window and the tree list on the left-hand side of the window provide access to all functions required for a Fluent workflow. We will not use these for the exercise except in few occasions; you can see how to use them to perform all the steps of the workflow individually in the aforementioned educational exhibit.
2. The Task Page allows viewing and editing the parameters for each item in the workflow. Again, we will not use this here.
3. The main viewport. In this viewport a chosen item will be visualized; this can be selected to be the STL, the volume mesh, and plots of solution results. A few shortcuts that are useful are:
 - Left mouse button click & drag rotates a 3D visualization if one is shown.

- Rotating the scroll wheel zooms in and out. Click and hold & drag the scroll wheel selects the area dragged over to zoom in. **If you accidentally do this or otherwise lose the model in the viewport and need to re-center the object, click the small loupe with the blue cube in it.**
 - **When displaying a 3D object in the viewport, a right mouse button click will select the entity pointed at by the mouse, or a specific point in the entity.** We will use this to both set up the CFD problem, and to interrogate the solution.
4. The ACT toolkit work area where we will run the blood flow toolkit from.

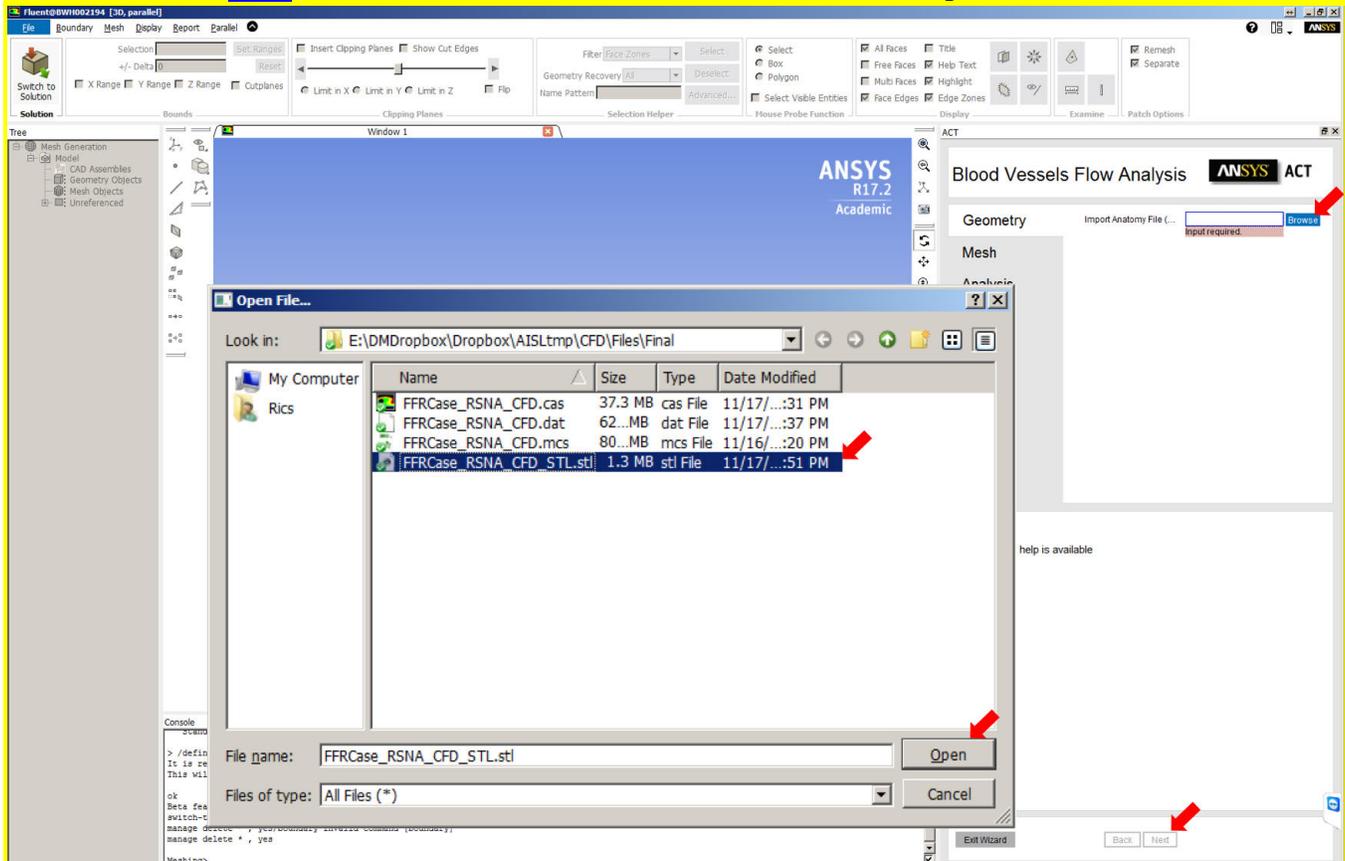


Step 2A: Generating the Finite Volume Lumen Mesh

25. In the ACT work area, click on the “Wizards”, and then click on “Blood Vessels”. The ACT work area will now show the first step of the Blood Vessels Flow Analysis Toolkit.

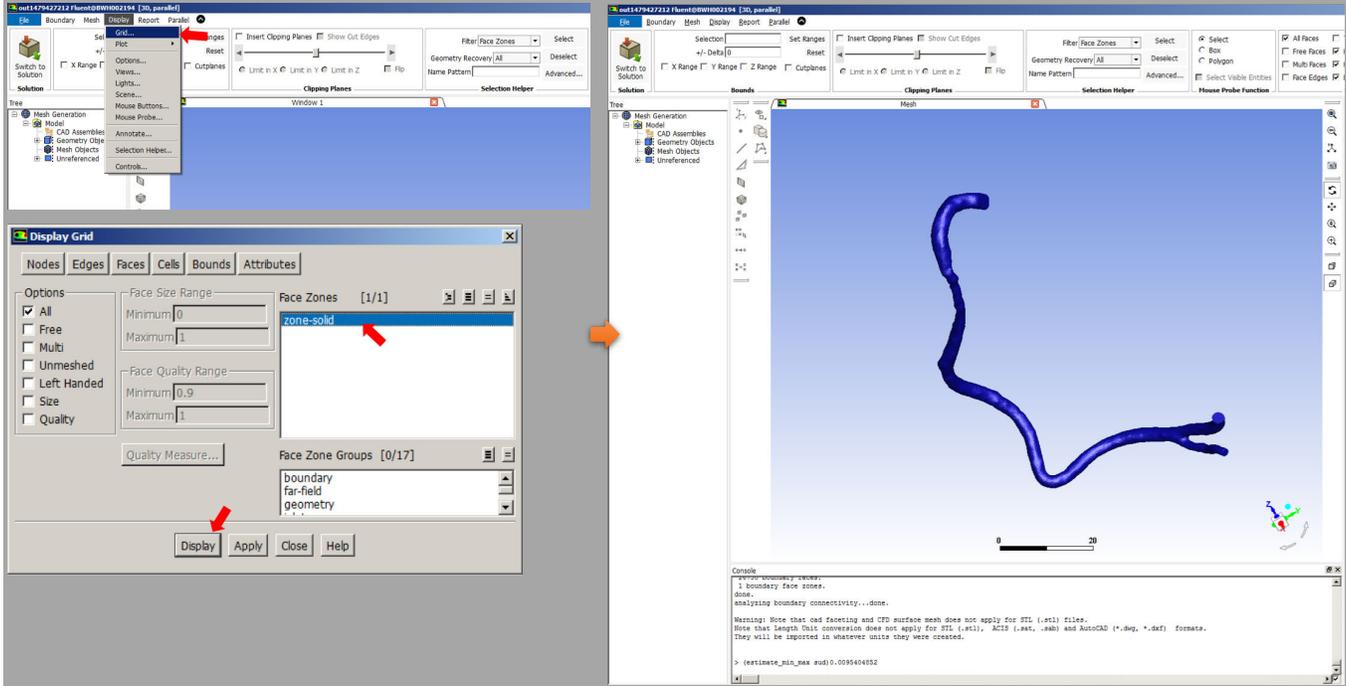


26. Click the “Browse” button to Import the coronary lumen anatomy STL file made in the Mimics step. In the open file window, navigate to the course folder, select the **FFRCCase_RSNA_CFD_STL.stl** file, and click Open. Once this is done, in the bottom of the ACT work area, the “Next” button has been filled in blue color. Click it to proceed.



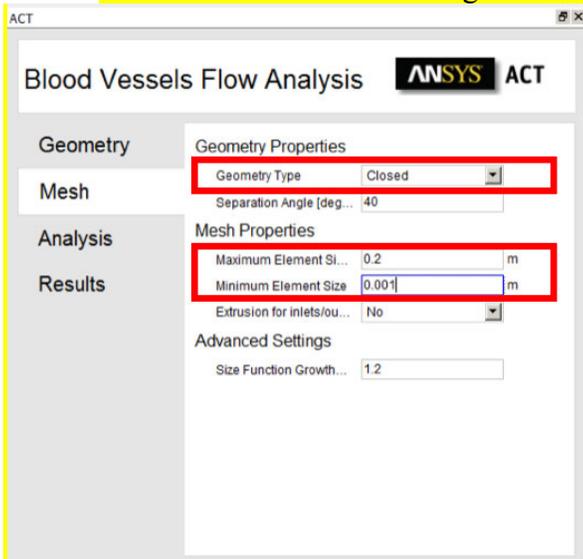
Once the import operation is complete, the ACT work area is populated with the volume meshing options. Before proceeding to mesh the case, we will take a side step to view the 3D STL model we just imported. This step is not necessary and is thus provided as a note below.

Note: To visualize the STL model, go to the Display menu, select “Grid...”. In the Display Grid window that appears, highlight all Face Zones shown (e.g., zone-solid), and click Display.



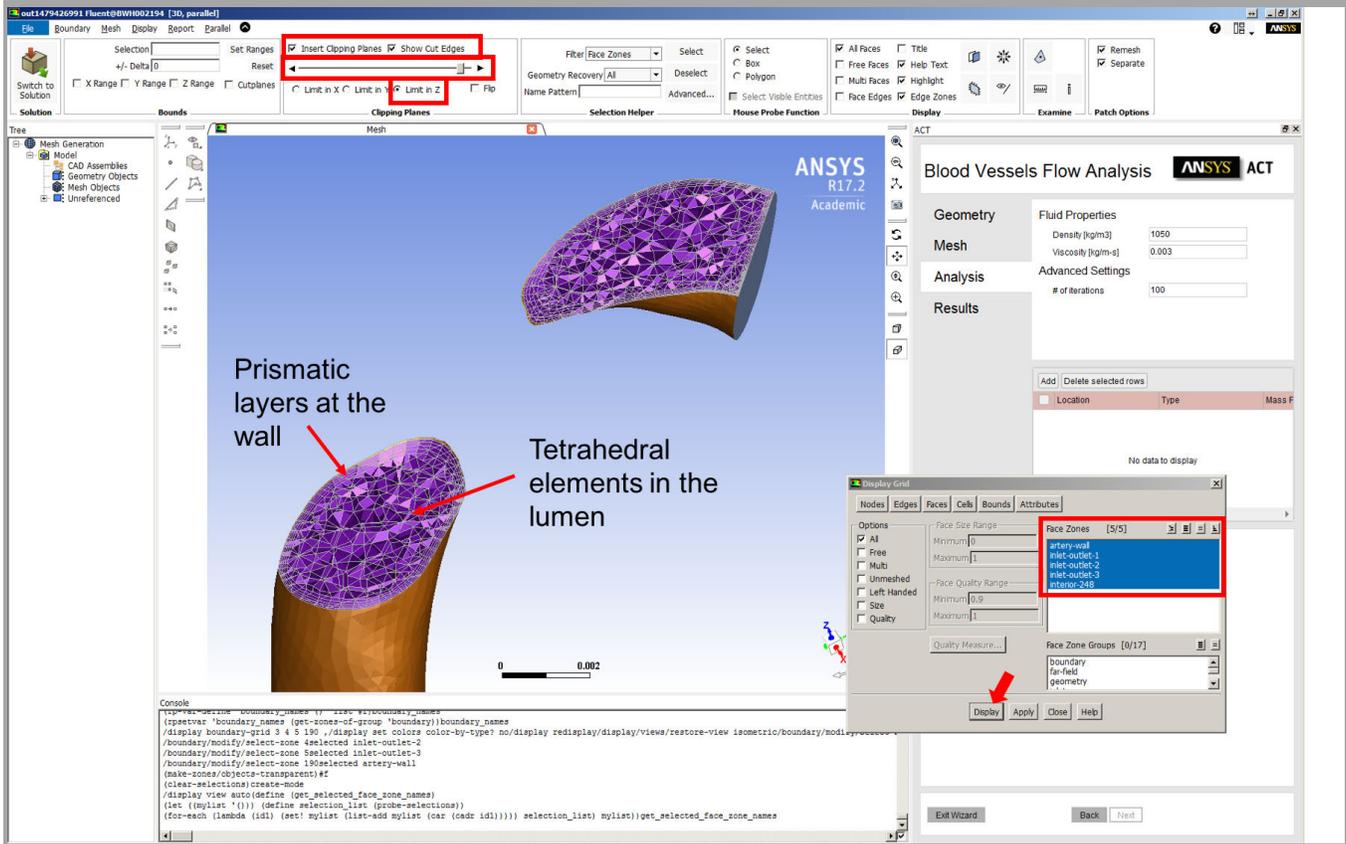
27. Once the import operation is complete, the ACT work area is populated with the volume Meshing options. Set these options as follows:

- Closed geometry type (our STL has surfaces at the inlet & outlet) & 40° separation angle
- 0.2 Maximum Element and 0.001 Minimum Element
- No extrusion for inlets/outlets
- 1.2 for size function growth



The ACT toolkit will now proceed to create a finite volume mesh. This will take a few minutes. Once completed, you will see the surface of the volume mesh.

Note: To visualize the finite volume mesh once the toolkit has created it, go to the Display menu, and select “Grid...”. In the Display Grid window, highlight all Face Zones shown (including the zone named “interior-xxx”), and click Display. Then in the Clipping Planes panel on the top bar, check Insert Clipping Planes, select “Limit in Z” and drag the slider to examine the internal finite volume mesh structure at any desired location along the coronary artery.



Step 2B: Setting up the CFD Physics

You will note that the ACT work area has now changed to show the fluid properties and other settings. In this ACT step, we will set up the physics of the problem we are trying to solve. The first step is to set up the fluid properties.

Note: A fluid is characterized by two properties: density and viscosity.

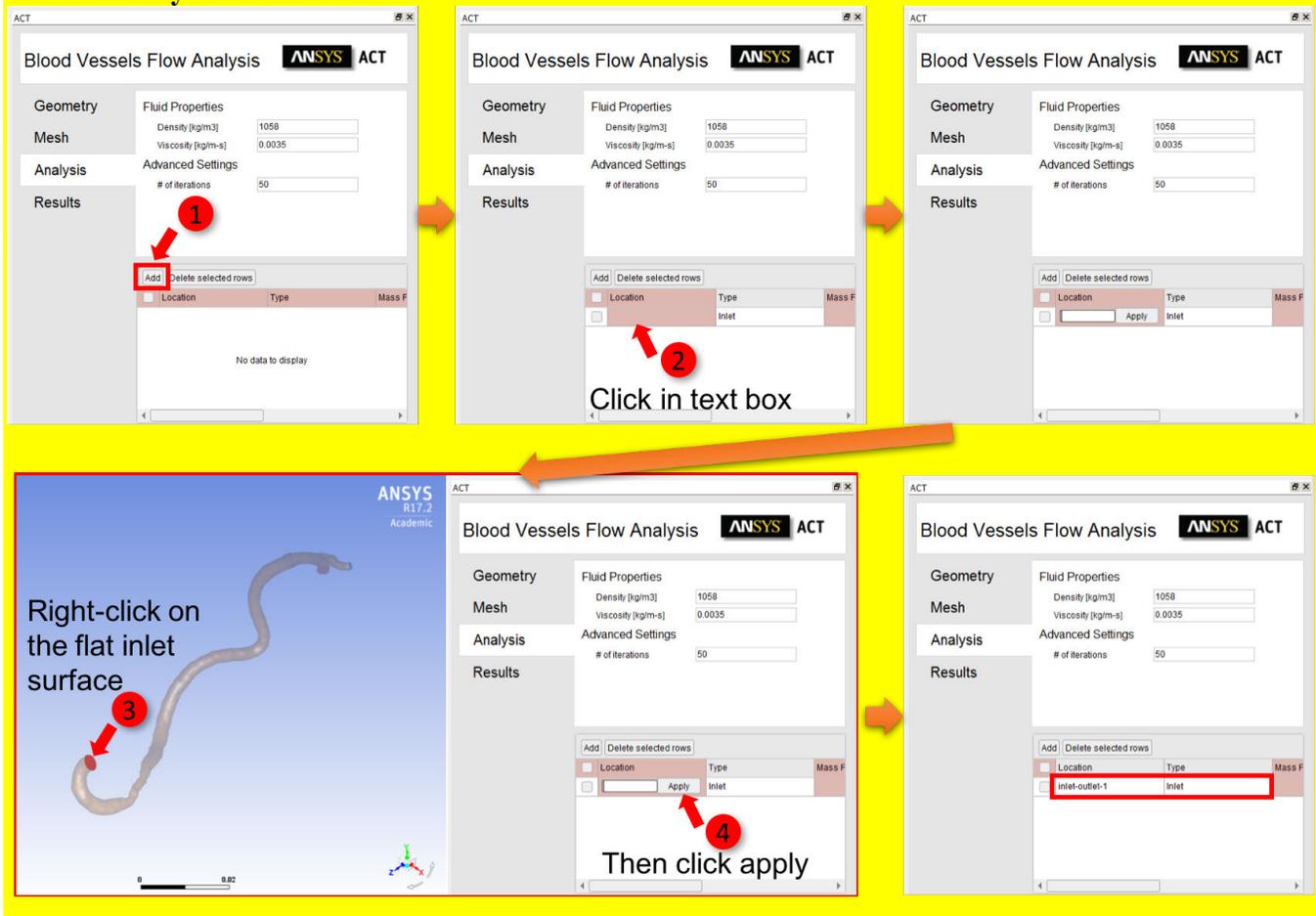
- Density (ρ) is the amount of mass per unit volume, measured in kg/m^3 . For blood, 1058 kg/m^3 is an appropriate density.
- Viscosity (μ) is the property of a fluid that offers resistance to flow or shear, measured in kg/m-sec or Pa-sec . Blood viscosity depends on a number of factors, including hematocrit and plasma proteins (8). Often, we assume that blood viscosity is constant at all shear rates (Newtonian fluid) and this exercise will make this assumption. A typical value used in the literature for constant viscosity is 0.0035 kg/m-s for blood with 45% HCT. For more information, including using a more appropriate non-Newtonian blood viscosity model, refer to the educational exhibit and (9).

In addition to setting the fluid properties, we also need to set the flow boundary conditions. We will do this graphically by creating 3 new boundary conditions (one for the inlet, and two for the two terminal branches in our 3D model), and right clicking on the corresponding surfaces in the 3D view of the mesh to identify each boundary.

28. In the ACT work area, set Density to 1058, Viscosity to 0.0035, and # of Iterations to 50. Then,

1. Click the “Add” button to add a boundary condition surface. A new row will appear, designated as type Inlet.
2. Then, click the pink box that appeared in the “Location” column in order to enter the surface selection mode. A button named “Apply” will appear within the box in the column.
3. Move your mouse pointer to the flat inlet surface of the coronary tree, and *Right Click* it. The surface should become red in color.
4. While the surface is red (selected), in the Location column click the Apply button.

The ACT work area should now display a name such as “inlet-outlet-1” in the Location column of the boundary conditions list.

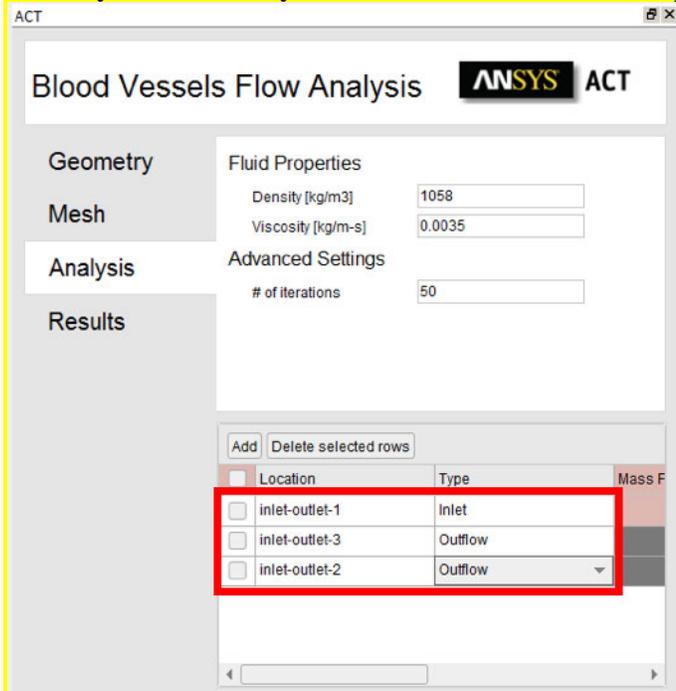


29. Repeat the process of adding boundary conditions for the two outlets, and select type “Outflow” for each of these two.

1. Click the “Add” button; in the new row that's appears, select “Outflow” from the Type drop-down menu.
2. Click the pink box in the “Location” column to enter surface selection mode.
3. Move your mouse pointer to the flat outlet surface of the coronary tree, and *Right Click* it.

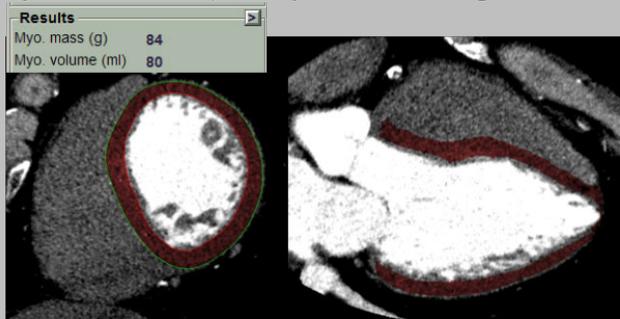
4. Click the Apply button in the Location column.

When you are done you should be able to see the populated list below:



Finally, we need to set up the physiologic values for the inlet flow and outflows that we wish to solve the fluid flow problem for.

Note: To determine these values, we will assume that myocardium requires, at rest, 0.8 ml/min/gram of tissue. Segmenting the left ventricular myocardial mass (which accounts for roughly 2/3rds of total myocardial mass) thus yields that the patient's myocardium demands 100.8 ml/min of blood.



Patient's Resting Flow requirements:

$$Q = 1.5 \times \text{Mass}_{LV} \times 0.8 = 100.8 \text{ ml/min}$$

Next, we need to determine the amount of flow required in hyperemia. For the purposes of this exercise, we will assume that the epicardial coronaries pose no resistance to flow. Then, given the knowledge that distal microvascular resistance drops to $\frac{1}{4}$ its resting state value after administrations of adenosine, we can determine the hyperemic coronary flow to be $Q_{\text{stress}} = 100.8 \times 4 = 403.2 \text{ ml/min} = 0.0071 \text{ kg/sec}$, where the last equality was obtained by using the density of blood, 1058 kg/m^3 . Note that the assumption of minimal epicardial coronary resistance can be relaxed, but the CFD below would need to be coupled to the distal microvascular resistance. This can be done explicitly by starting with an at rest CFD simulation and applying Kirchoff's circuit laws to obtain the distal resistance at rest. Then, the distal resistance can be fixed to $\frac{1}{4}$ the rest value obtained from this step, and using Kirchoff's laws, new

flow rates can be obtained for stress, and inserted again into the CFD. This would require multiple applications of the CFD and is not covered here.

The final step is to determine how much of that flow is delivered to the myocardium via the RCA, and subsequently how much of the flow in the RCA is delivered through the PDA versus the PLV (since our model did not include any other coronary branches). For this determination, we will apply Murray's law (10) which states that the flow through a vessel is proportional to its diameter to the third power. Two successive applications of this law, one to determine the relative RCA versus LM flow, and one to determine the relative PDA vs PLV flow, shown below, yield these values.

Murray's Law: $Q_i \sim D_i^3$

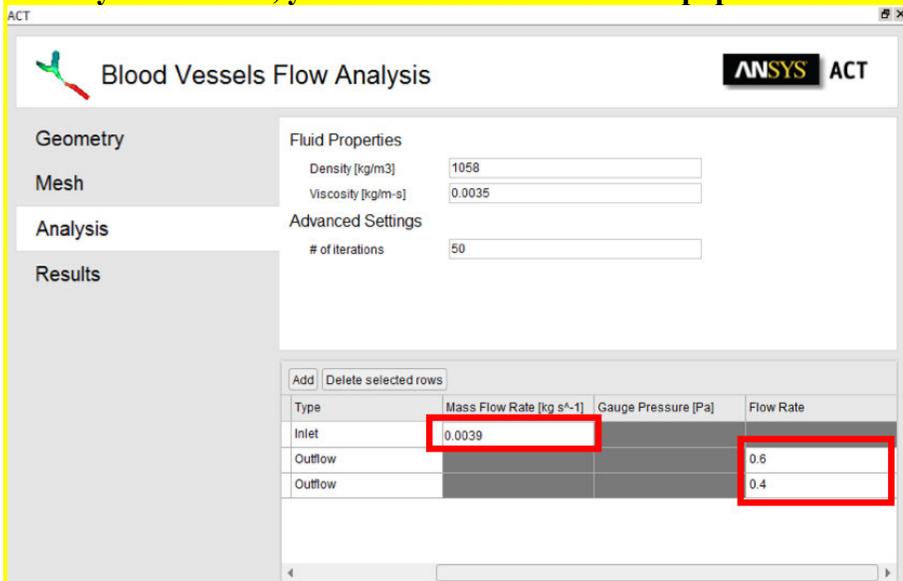
$Q_{\text{stress}} = Q_{\text{RCA}} + Q_{\text{LM}}$
 $Q_{\text{RCA}} \sim D_{\text{RCA}}^3 = 3.45^3 \Rightarrow Q_{\text{RCA}} \sim \frac{3.45^3}{3.45^3 + 3.23^3} \times Q_{\text{stress}} = 0.0039 \text{ kg/sec}$
 $Q_{\text{LM}} \sim D_{\text{LM}}^3 = 3.23^3 \Rightarrow Q_{\text{LM}} \sim \frac{3.23^3}{3.45^3 + 3.23^3} \times Q_{\text{stress}} = 0.0032 \text{ kg/sec}$

$Q_{\text{RCA}} = Q_{\text{PDA}} + Q_{\text{PLV}}$
 $Q_{\text{PDA}} \sim D_{\text{PDA}}^3 = 1.84^3 \Rightarrow Q_{\text{PDA}} \sim \frac{1.84^3}{1.84^3 + 2.10^3} = 40\% \times Q_{\text{RCA}}$
 $Q_{\text{PLV}} \sim D_{\text{PLV}}^3 = 2.10^3 \Rightarrow Q_{\text{PLV}} \sim \frac{2.10^3}{1.84^3 + 2.10^3} = 60\% \times Q_{\text{RCA}}$

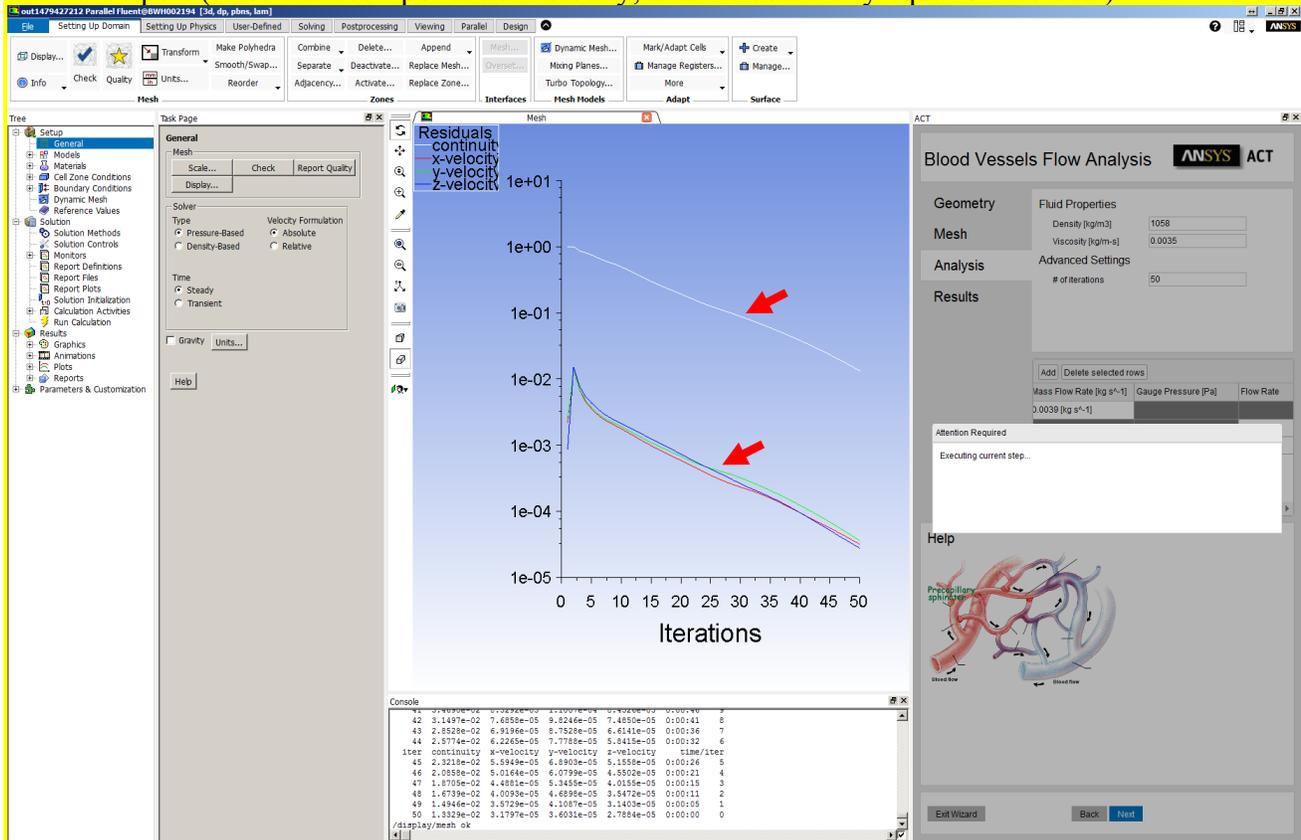
30. We are now ready to set the values for the flow boundary conditions. In the boundary surfaces listing, scroll to the right to reveal the text boxes in which to type the values we determined apply for this patient.

1. For the Inlet, type “0.0039” in the pink box in the Mass Flow Rate. Do not enter units; units will appear after you enter the value, and will be in kg/sec as we determined in the note above.
2. For the larger Outlet of the two, type “0.6”, to indicate 60% of the flow as we determined.
3. For the smaller Outlet of the two, type “0.4”, to indicate 40% of the flow as we determined.

When you are done, you should be able to see the populated list below:



31. We will now run the CFD simulation. In the bottom of the ACT work area, click the “Next” button which should be highlighted blue. Wait for the operation to complete, in approximately 5 min. While the CFD is running, you will see the diminishing residuals of the solution being plotted in the main viewport (3 for each component of velocity, and the continuity equation residual).

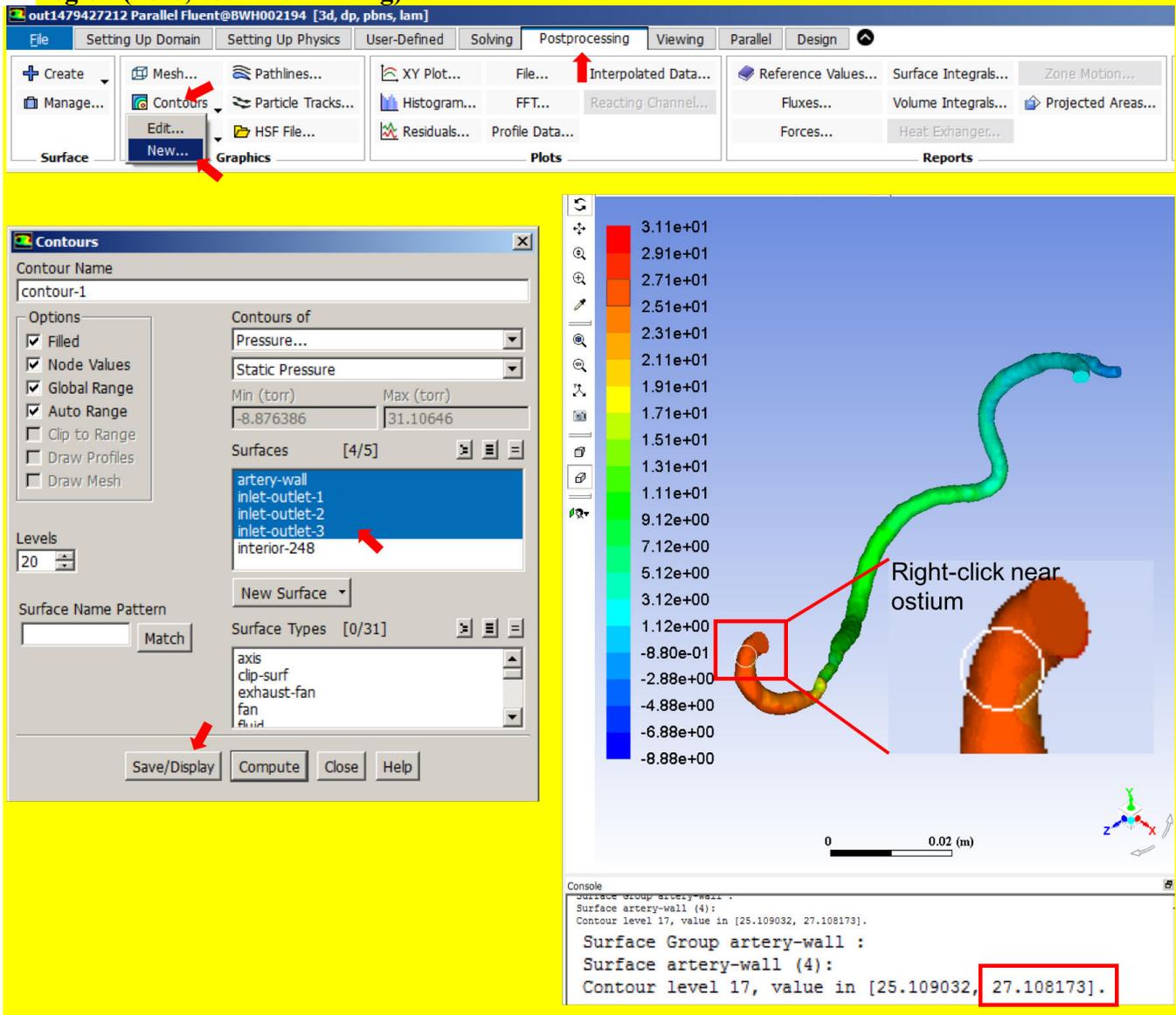


Step 2C: Interrogating the CFD Solution for Pressure and FFR

When the CFD solver has ran to completion (here, set to 50 iterations), we can interrogate solved variables such as pressure and velocity. We are interested in pressure and FFR. However, because we did not indicate in the CFD solver a known pressure at any location, such as the ostium, the resulting solution contains pressures relative to some arbitrary constant pressure. We will first need to set the mean aortic pressure as the constant (reference) pressure. We will do this by interrogating the pressure solved at the ostium, then creating a new variable that sets the ostial pressure to be the patient’s mean aortic pressure. Blood pressure measured just prior to the patient’s CTA was 158/82 (systolic/diastolic), and we estimate mean aortic pressure as $P_a = 1/3 \times \text{sBP} + 2/3 \times \text{dBP} = 107.3$. Once we have normalized the pressure at the coronary ostium to the patient’s mean aortic pressure, we will calculate FFR as the ratio of pressure throughout the coronary tree divided by the mean aortic pressure.

32. First, we will extract the relative pressure solved for at the ostium by the CFD simulation.

- In the menu bar, click on “Postprocessing”, click “Contours” () and select “New”.
- In the Contours window, select Contours of “Pressure...”, “Static Pressure, and select all surfaces except the “interior-xxx” one, and click Save/Display. Then dismiss the window by hitting close.
- The viewport should now show the pressure calculated by the CFD in the coronary artery tree in torr (unit synonymous to mmHg).
- Right-click on the artery near the ostium and the Console will print the pressure in that region (here, 25.1-27.1 mmHg).



The screenshot displays the ANSYS Fluent interface. The top menu bar is set to "Postprocessing". The "Contours" menu is open, and the "New..." option is selected. The "Contours" dialog box is visible, showing the following settings:

- Contour Name: contour-1
- Options: Filled, Node Values, Global Range, Auto Range, Clip to Range, Draw Profiles, Draw Mesh (all checked).
- Contours of: Pressure...
- Static Pressure
- Min (torr): -8.876386, Max (torr): 31.10646
- Surfaces: [4/5] (artery-wall, inlet-outlet-1, inlet-outlet-2, inlet-outlet-3, interior-248)
- Surface Name Pattern: Match
- Surface Types: [0/31] (axis, clip-surf, exhaust-fan, fan, fluid)
- Buttons: Save/Display, Compute, Close, Help

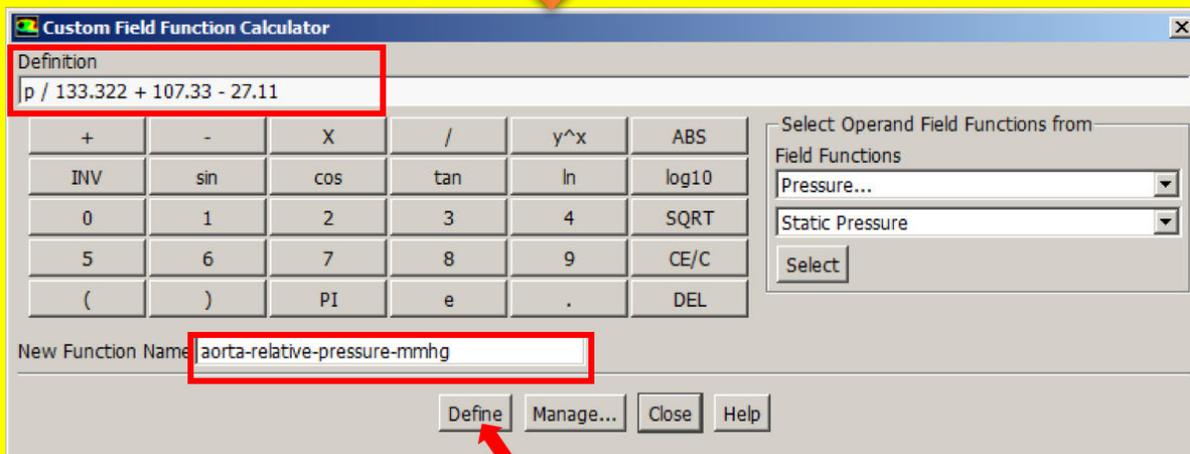
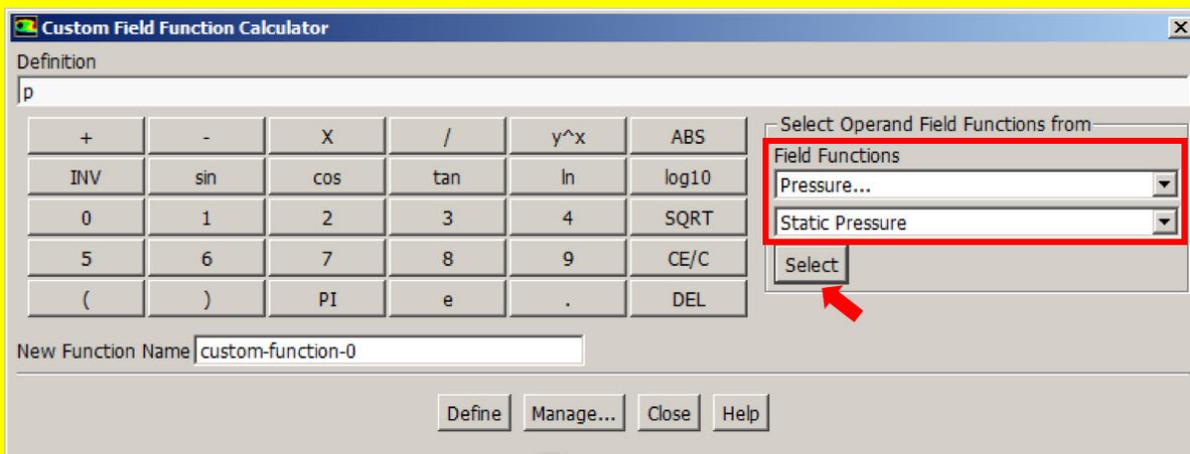
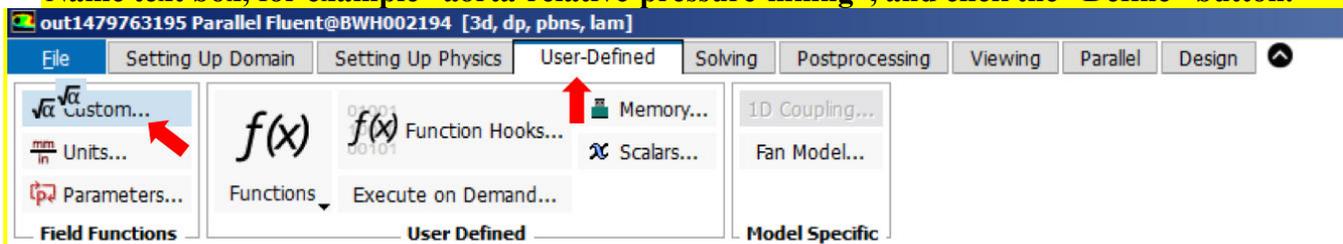
The 3D model shows a coronary artery tree with a color scale ranging from -8.88e+00 (blue) to 3.11e+01 (red). A red box highlights the ostium region, and a red arrow points to it with the text "Right-click near ostium".

The console output shows the following text:

```
Surface Group artery-wall :  
Surface artery-wall (4):  
Contour level 17, value in [25.109032, 27.108173].  
Surface Group artery-wall :  
Surface artery-wall (4):  
Contour level 17, value in [25.109032, 27.108173].
```

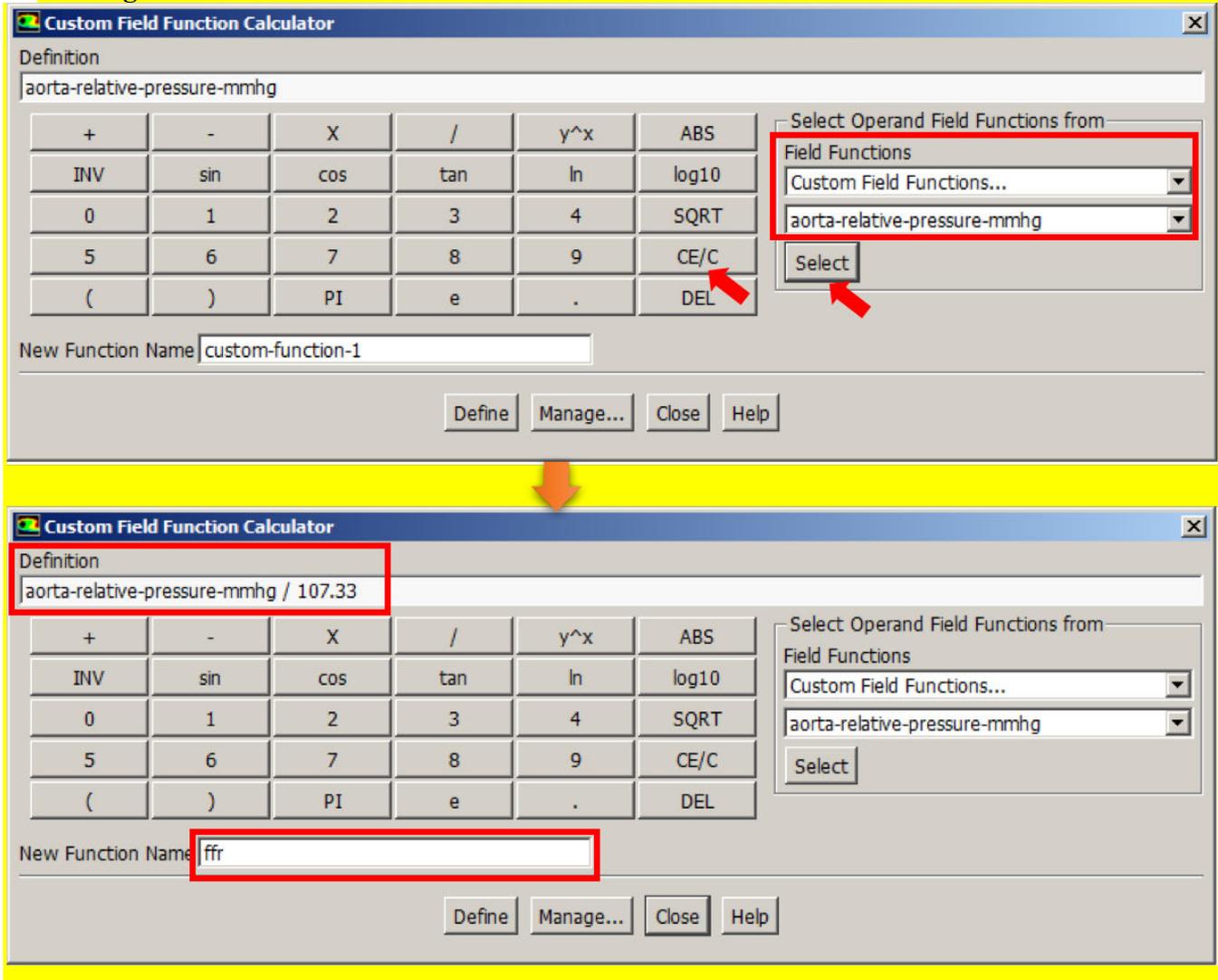
33. Next, we will create custom functions to report pressure and FFR.

- In the menu bar, click on “User Defined”, and on the top left corner, click the “Custom” button (\sqrt{x}).
- In the Custom Function Calculator window that appears, under “Field Functions” select “Pressure...”, “Static Pressure” and click select. In the calculator Definition text box you should now see the letter “p” appear.
- Because the pressure “p” is internally stored in pascal, we first divide “p” with 133.322 to yield the pressure in mmHg. Use the mouse to click each character i.e., “/”, “1”, “3”, “3”, “.” etc.
- Continue entering the remaining equation shown below, by adding the patient’s mean aortic pressure of 107.33, and subtracting the arbitrary pressure the CFD deduced at the ostium (27.11 mmHg as we found in the previous step).
- When the equation is completed, type in a name for the new function in the New Function Name text box, for example “aorta-relative-pressure-mmhg”, and click the “Define” button.



34. Next, we will create the custom functions to report FFR.

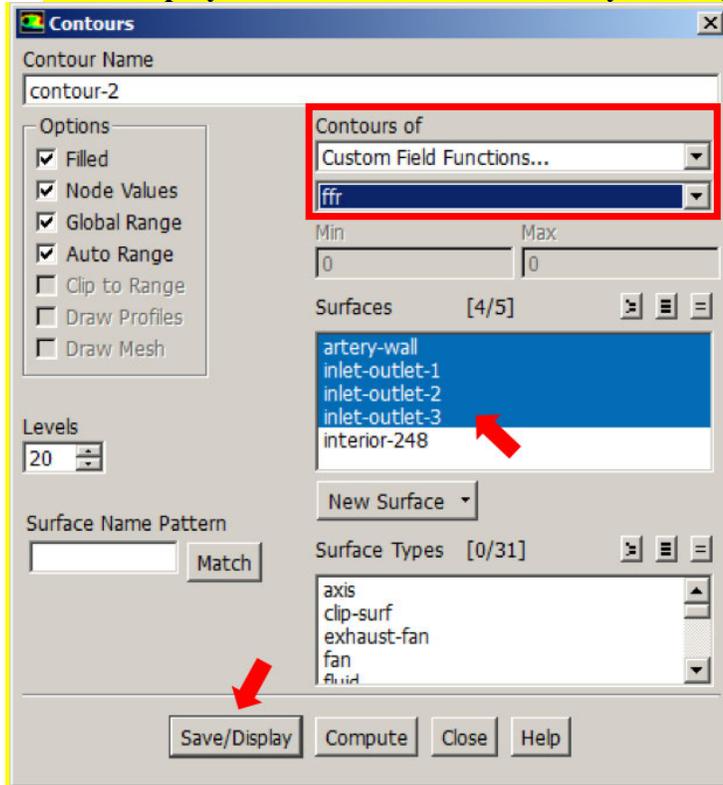
- In the Custom Function Calculator window, click the “CE/C” button to clear the previous function.
- Under “Field Functions” select “Custom Field Functions...”, and select your function, “aorta-relative-pressure-mmhg”, then click select. In the calculator Definition text box you should now see “aorta-relative-pressure-mmhg”.
- Enter the remainder of the equation, namely dividing by the mean aortic pressure (107.33)
- Name the new function “ffr”, and click Define. You can now dismiss the window by clicking Close.



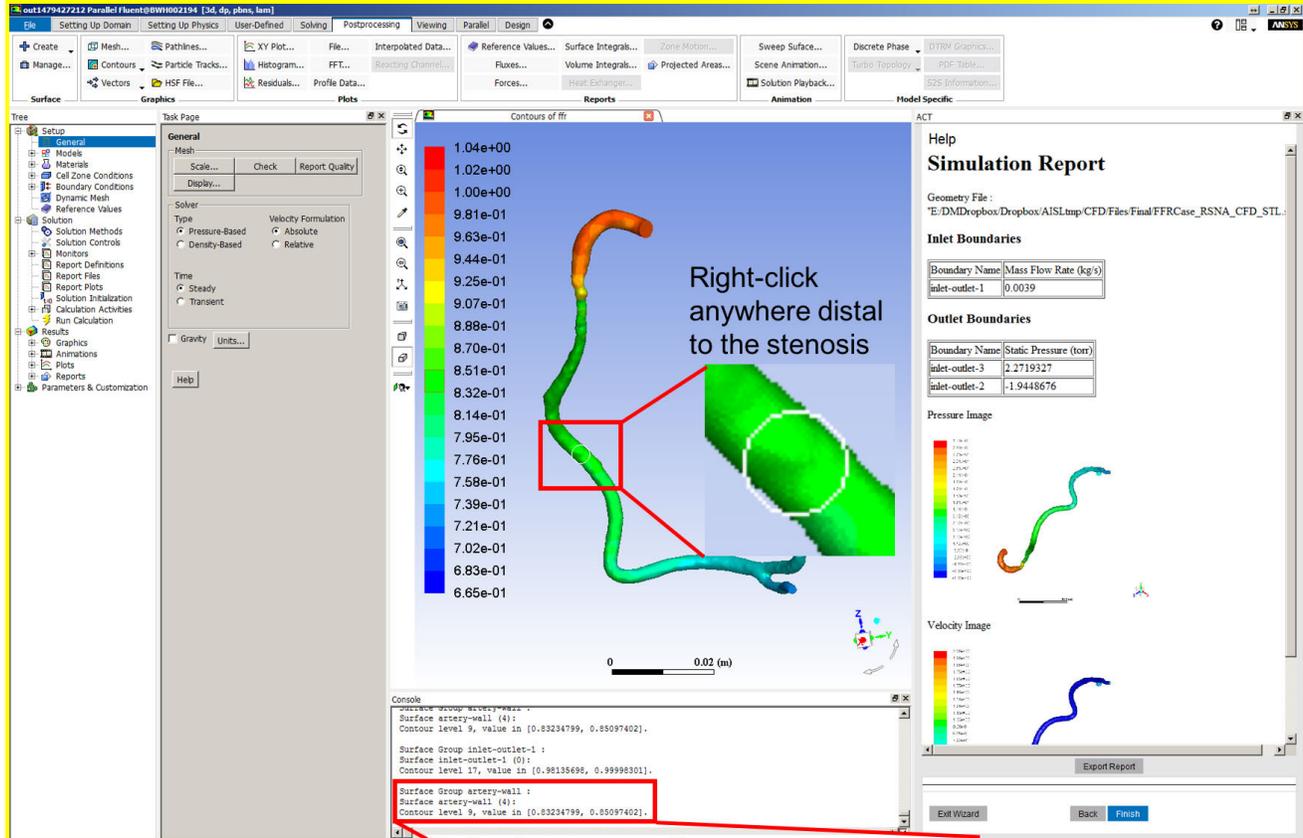
We are now done with our CT-FFR simulation! The last step is to plot the computed CT-FFR in the viewport, and interrogate it at a location distal to the stenosis!

35. First, we will plot the FFR function we created, similar to step 32 above; go to the “Postprocessing” menu tab, click “Contours” (📐) and select “New”.

- In the Contours window, select Contours of “Custom Field Functions...”, and select the “ffr” function you created.
- In the Surfaces area, select all surfaces except the “interior-xxx” one, and click Save/Display. Then dismiss the window by clicking “Close”.



36. You can now right-click anywhere on the coronary artery tree to interrogate the CT-FFR value!



Surface Group artery-wall :
 Surface artery-wall (4):
 Contour level 9, value in [0.83234799, 0.85097402].

Literature Cited

1. McQueen DM, Peskin CS, Yellin EL. Fluid dynamics of the mitral valve: physiological aspects of a mathematical model. *Am J Physiol.* 1982;242(6):H1095-110.
2. Davies PF. Hemodynamic shear stress and the endothelium in cardiovascular pathophysiology. *Nat Clin Pract Cardiovasc Med.* 2009;6(1):16-26.
3. Morris PD, Narracott A, von Tengg-Kobligk H, et al. Computational fluid dynamics modelling in cardiovascular medicine. *Heart.* 2016;102(1):18-28.
4. Koo BK, Erglis A, Doh JH, et al. Diagnosis of ischemia-causing coronary stenoses by noninvasive fractional flow reserve computed from coronary computed tomographic angiograms. Results from the prospective multicenter DISCOVER-FLOW (Diagnosis of Ischemia-Causing Stenoses Obtained Via Noninvasive Fractional Flow Reserve) study. *J Am Coll Cardiol.* 2011;58(19):1989-97.
5. Pijls NH, De Bruyne B, Peels K, et al. Measurement of fractional flow reserve to assess the functional severity of coronary-artery stenoses. *N Engl J Med.* 1996;334(26):1703-8.
6. Kakouros N, Rybicki FJ, Mitsouras D, Miller JM. Coronary pressure-derived fractional flow reserve in the assessment of coronary artery stenoses. *Eur Radiol.* 2013;23(4):958-67.
7. Tonino PA, De Bruyne B, Pijls NH, et al. Fractional flow reserve versus angiography for guiding percutaneous coronary intervention. *N Engl J Med.* 2009;360(3):213-24.
8. Hussain MA, Kar S, Puniyani RR. Relationship between power law coefficients and major blood constituents affecting the whole blood viscosity. *Journal of Biosciences.* 1999;24(3):329-37.
9. Sochi T. Non-Newtonian Rheology in Blood Circulation. *Physics.* 2013.
10. Murray CD. The Physiological Principle of Minimum Work Applied to the Angle of Branching of Arteries. *The Journal of general physiology.* 1926;9(6):835-41.

Dear participant, thank you for your time in attending this course! We hope it has been an interesting source of information, and, with your support in reviewing it, we hope to deliver a better version next year!

Until then, we would greatly appreciate it if you would help us to gather some information regarding your particular interests in CFD. This will help us to prepare more targeted course sessions next year, but it will also help us potentially build a web-based resource of CFD information for radiology users. Our initial goal is to make the software used in this course available for download as a demo, and to potentially share tutorials with more advanced information, such as how to perform pulsatile flow simulations or use non-Newtonian fluid models. To this end we would like to ask that you fill in the questionnaire on the next page, and return it to the course faculty. If you are able to fill it out at a later time, feel free to email a scanned copy or photograph to dmitsouras@alum.mit.edu.

Thank you and see you next year!

RSNA 2016

Introduction to Computational Fluid Dynamics from Medical Images: A Step-by-Step Demonstration (Hands-on)

Participant Questionnaire

Please return to course faculty or email scan/photo to dmitsouras@alum.mit.edu

Do you wish to be contacted toward potentially creating a CFD group through RSNA? Yes / No

If yes, your email:

	Circle One		Notes (e.g., specific anatomy/pathology)
Do you currently perform CFD in your research or practice?	Yes	No	
If no, would you like to?	Yes	No	
Of the following, which do you do or would like to do:			
CT-FFR (Coronary)	Yes	No	
ESS (Coronary)	Yes	No	
Axial/Radial Stresses (Coronary)	Yes	No	
Other Pressure (e.g., PAD)	Yes	No	
Other ESS (e.g., AAA)	Yes	No	
Other Stresses	Yes	No	
Aorta	Yes	No	
Congenital Heart	Yes	No	
Cerebrovascular	Yes	No	
Peripheral	Yes	No	
What would facilitate broader use of CFD in your research?	Rate 1 to 5, with 1 least important and 5 most important to you		
1. Regarding Tools:	More Training / Guides	Better Ease-of-Use	Used By My Peers
Segmentation Software			
CFD Software			
	Check the column that most describes your opinion		
2. Regarding Data Quality:	Definite Benefit	Marginal Benefit	No Benefit
Better Image Resolution			
Better CFD Material Properties			
Better CFD Boundary Conditions			
	Rate 1 to 5, with 1 least important and 5 most important to you		
3. Regarding Clinical Translation:	Access to Metrics	Peer-Reviewed Guidance Documents	
Validation of CFD Results			
Clinical Utility/Patient Risk			

Other comments/suggestions: