Implementation in COMSOL

The transient Navier-Stoke equation will be solved in COMSOL. A text (.txt) file needs to be created that contains the velocity at the inlet of the common carotid (calculated as described earlier) as a function of time, so that it can be used as the boundary condition in COMSOL. The format of the file is:
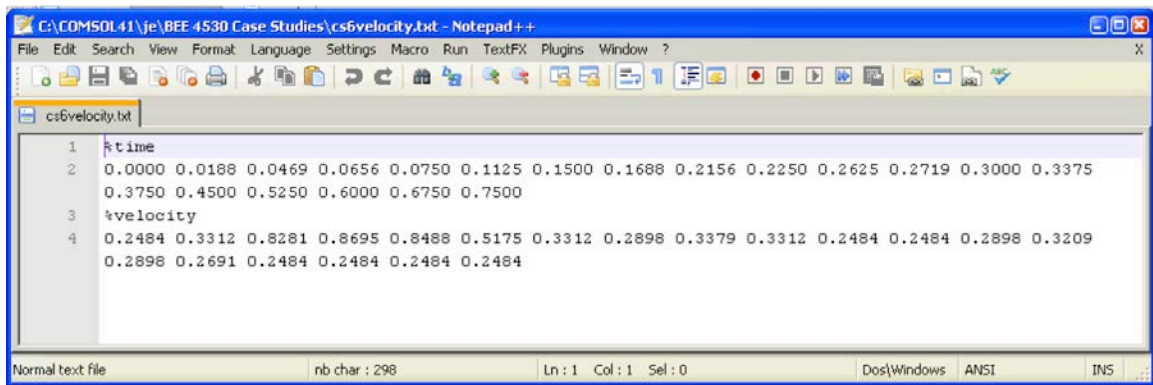
%time
Time values separated by spaces
%velocity
*Velocity values separated by spaces*

The file can be created in Notepad or Wordpad. It should be made sure that there are **no line breaks** between the data. Remember here that the velocities in the text file refer to the velocities at the center of the common carotid, vo. The text file should look like this:

Step 1: Problem Type Specification

(1) Start COMSOL Multiphysics 4.1.

(2) Select 3D under 'Select Space Dimension' tab.

(3) Click the blue 'Next' arrow next to the 'Select Space Dimension' title.

(4) Select Fluid Flow > Single-Phase Flow > Laminar Flow (spf).

(5) Click on the '+' button.

(6) Click the blue 'Next' arrow next to the 'Add Physics' title.

(7) Left click on Stationary.

(8) Click on the race flag icon next to the 'Select Study Type' title. Save often to prevent losing your work.
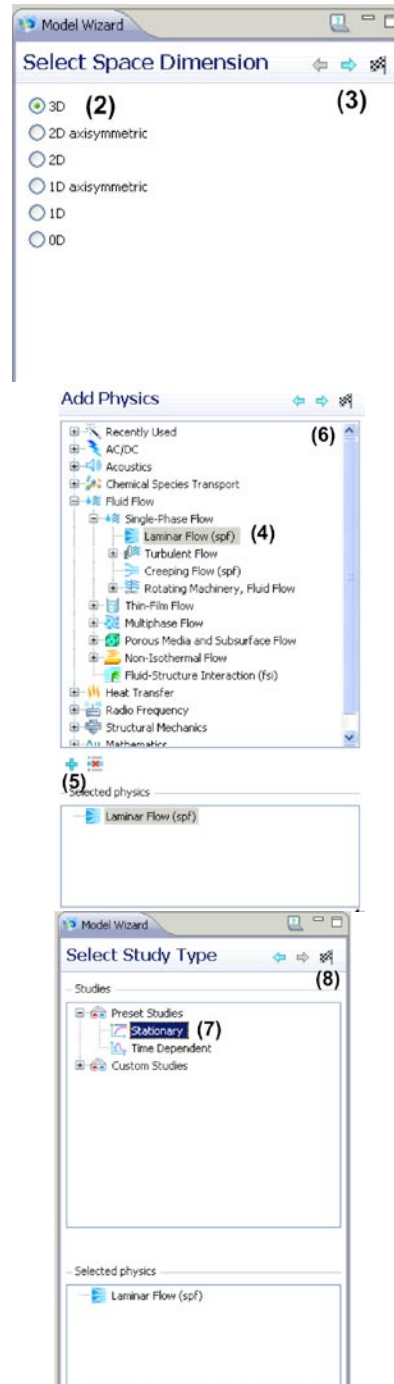
Step 2: Geometry Creation

(1) Right click on Geometry 1 and select Cylinder.
(2) Set the radius to 3.1 and height to 8.
(3) Under the Axis tab, make sure the Axis type is Cartesian. Set x, y, z to 0, 0, -1, respectively.
(4) Click on the Build All icon.
(5) Right click on Geometry 1 and select Work Plane.
(6) Right click on Geometry 1 > Work Plane 1 *(wp1)* > Geometry and select Circle.
(7) Set the radius to 3.1 and retain the default for other settings.
(8) Click on the Build All icon.
(9) Click on the Zoom Extents button to see the circle.
(10)   Right click on Work Plane *(wp1)* and select Revolve.
(11)   Under the Revolution Angles tab, set Start angle to 0 and End angle to -25.
(12)   Under the Point on the Revolution Axis tab, set x to -12.1 and y to 0.
(13)   Under the Direction of Revolution Axis tab, set x to 0 and y to 1.
(14)   Click on the Build All icon. The part of the common carotid with the bend has now been created and is shown in the 3D view.
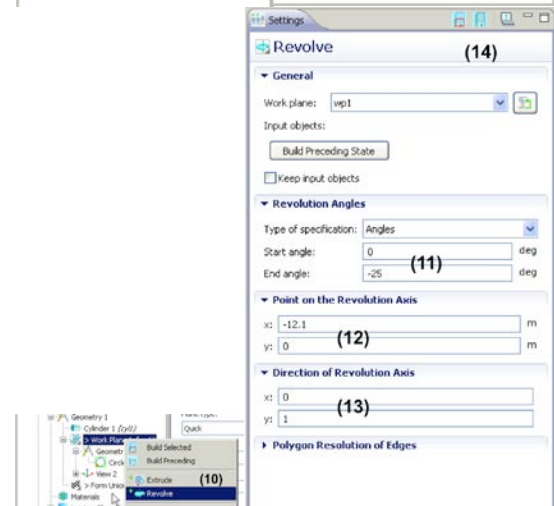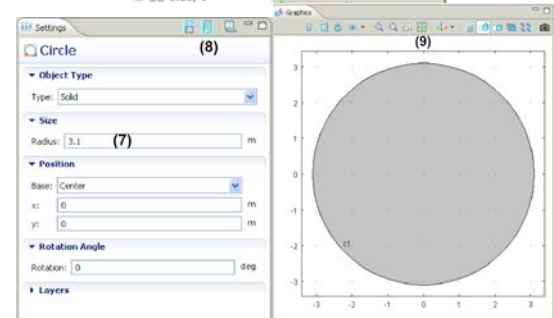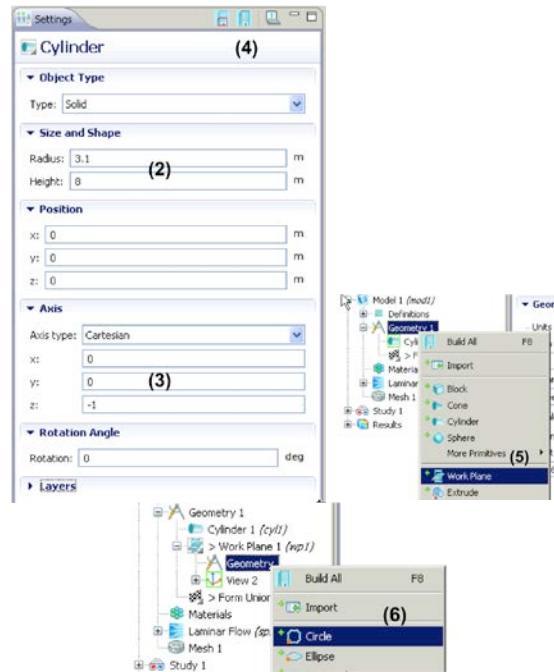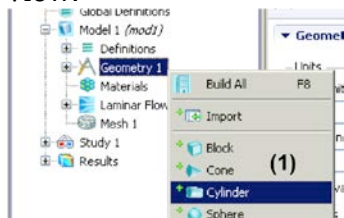
(15)   Right click on Geometry 1 and select Work Plane.

(16)   Under the Work Plane tab, open the drop-down menu for Plane type and select Face parallel.

(17)   Click on the top circular face of the bend (face 3) so that it becomes red.

(18)   Click on the '+' button.

(19)   Right click on Geometry 1 > Work Plane 2 *(wp2)* > Geometry and select Circle.

(20)   Set the radius to 3.1 and retain the default for other settings.

(21)   Click on the Build All icon.

(22)   Right click on Geometry 1 and select Extrude.

(23)   Under the General tab, make sure the wp2 is selected for Work plane.

(24)   Under the Distances from Work Plane tab, type in 4.5 for Distances (m).

(25)   Click on Build All icon.

(26)   Right click on Geometry 1 and select Cone.

(27)   Set Radius to 3.1, Height to 4.65, and Semi-angle to 11.

(28)   Under the Position tab, set **x** as 12.1*cos(25*pi/180)-4.5*cos(65*pi/180)-12.1, **y** as 0, and **z** as 12.1*sin(25*pi/180)+4.5*sin(65*pi/180)

(29)   Under the Axis tab, change the Axis type to Spherical and set **theta** to -25 and **phi** to 0.

(30)   Click on the Build All icon.
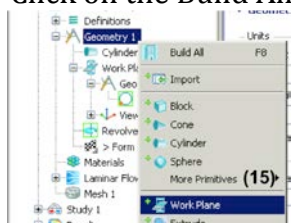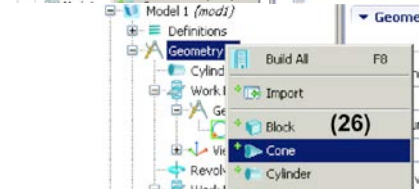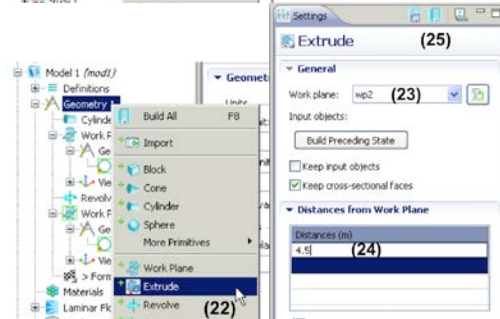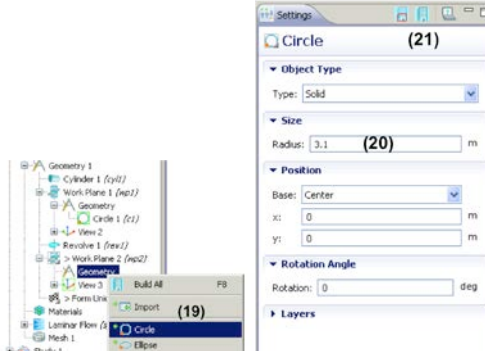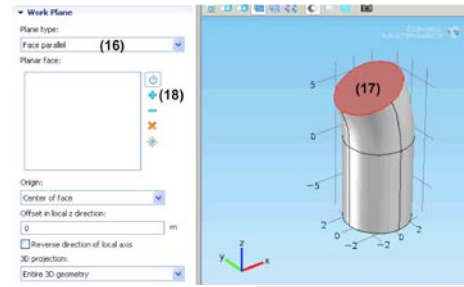
(31) Right click on Geometry 1 and select Work Plane.

(32) Under the Work Plane tab, change the Plant type to Face parallel.

(33) Click on the top circular face of the cone so that it becomes red.

(34) Click on the '+' button.

(35) Right click on Geometry 1 > Work Plane 3 *(wp3)* > Geometry and select Circle.

(36) Set the Radius to 3.1-4.65*tan(11*pi/180) and retain the default settings for others.

(37) Click on the Build All icon.

(38) Right click on Geometry 1 and select Extrude.

(39) Under the General tab, make sure wp3 is selected for Work plane.

(40) Under the Distances from Work Plane tab, type in 15 for Distances (m).

(41) Click on Build All icon.
The last part of the internal carotid geometry has now been created.

(42) Right click on Geometry 1 and select Cylinder.

(43) Set the Radius to 1.8, Height to 6.

(44) Under the Position tab, set **x** to 1.25, **y** to 0, and **z** to 0.9.

(45) Under the Axis tab, change the Axis type to Spherical and set **theta** to 25 and **phi** to 0.

(46) Click on the Build All icon.

(47) This is what your geometry should look like by now.
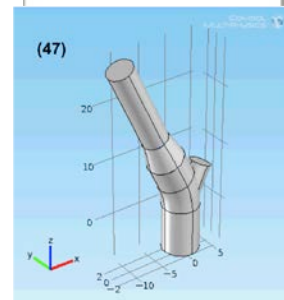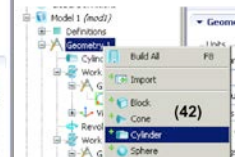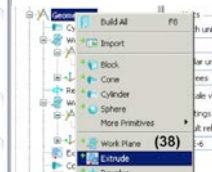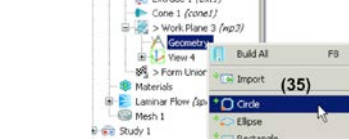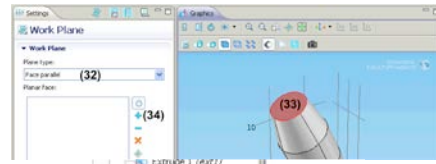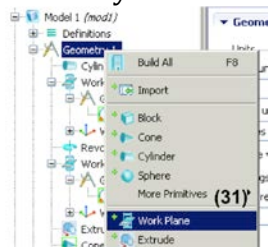
(48)   Right click on Geometry 1 and select Work Plane.

(49)   Under the Work Plane tab, change the Plant type to Face parallel.

(50)   Click on the top circular face of Cylinder 2 (the short cylinder you just drew on the external carotid artery side) so that it becomes red.

(51)   Click on the '+' button. After you click the '+' button, it should say cyl 2 > 4 in the Planar face box.

(52)   Right click on Geometry 1 > Work Plane 4 *(wp4)* > Geometry and select Circle.

(53)   Set the Radius to 1.8 and retain the default settings for others.

(54)   Click on the Build All icon.

(55)   Right click on Geometry 1 and select Extrude.

(56)   Under the General tab, make sure wp4 is selected for Work plane.

(57)   Under the Distances from Work Plane tab, type in 28 for Distances (m).

(58)   Click on Build All icon.
The final part of the external carotid geometry has now been created and can be seen in the 3D view.

(59)   Right click on Geometry 1 and select Boolean Operations > Union.
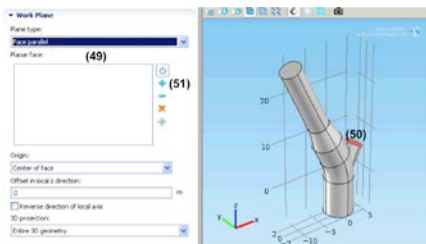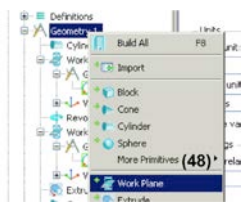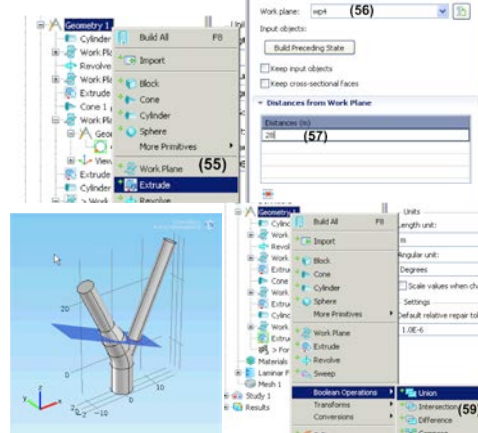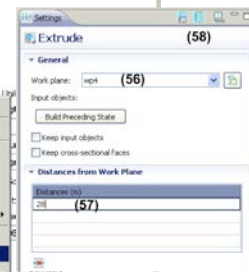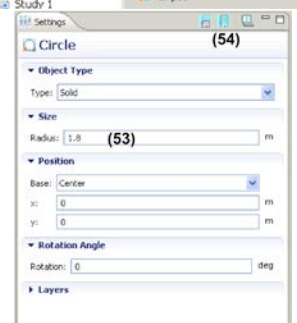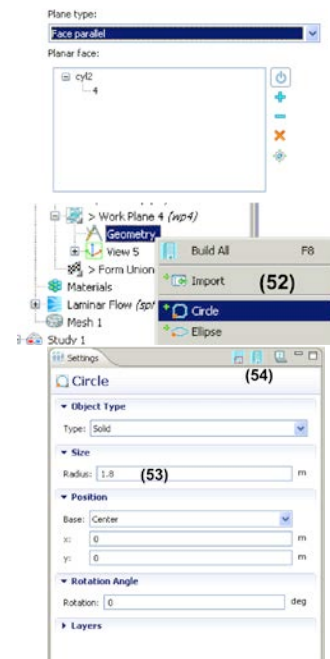
(60) Left click on the bended region of the internal carotid artery (rev1) so that it becomes red and then click on the '+' button. Refer to the picture if you are confused where this region is.

(61) Left click on the short cylinder region of the external carotid artery (cyl2; where the bifurcation occurs) and click on the '+' button. Again, refer to the picture if you are not sure where this region is.

(62) Uncheck Keep interior boundaries.

(63) Click on the Build All icon.

(64) Right click on Geometry 1 and select Work Plane.

(65) Select Quick for the Plane type (if not already selected) and zx-plane for Plane. Y-coordinate should be 0m (default value).

(66) Click on Build All icon.

(67) Right click on Work Plane 5 *(wp5)* > Geometry and select Rectangle.

(68) Set Width to 50, Height to 40. Under Position tab, set Base to Corner , x to -10, and y to -20.

(69) Click on Build All icon.

(70) Right click on Work Plane 5 *(wp5)* and select Extrude.

(71) Under the Distances from Work Plane tab, type in -4 for Distances (m).

(72) Click on Build All icon. A rectangular block is created to delete one half of the carotid geometry.
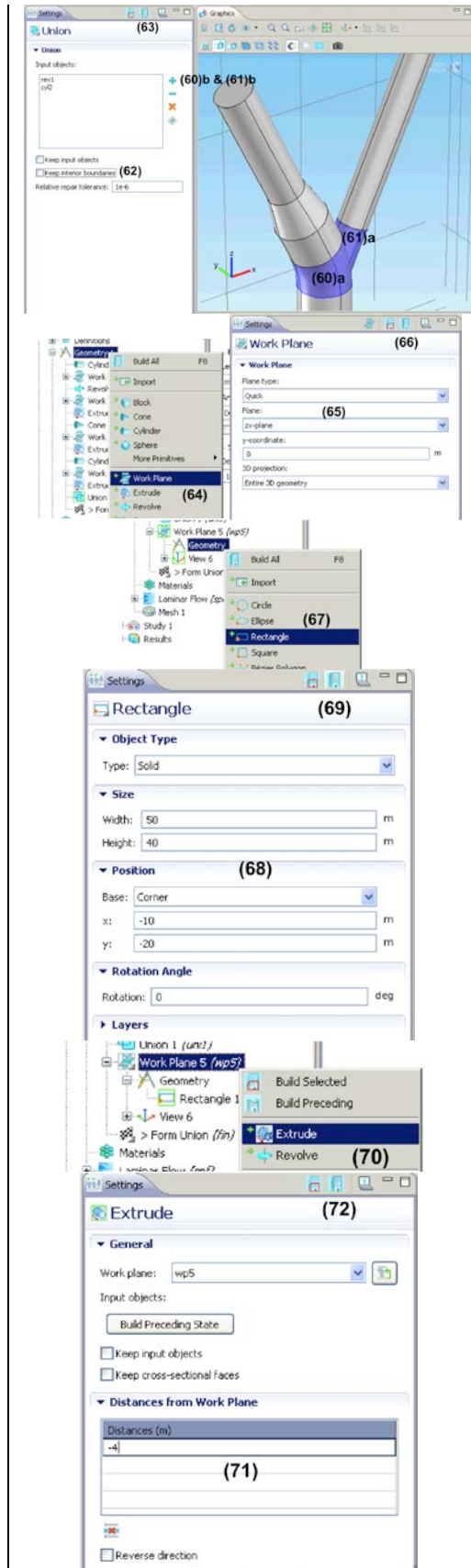
(73)   Right click on Geometry 1 and select Boolean Operations > Compose.

(74)   Turn the geometry around so you can see the rectangle and the carotid. You can do this by left clicking on geometry, holding the click, and then moving it around.

(75)   While pressing the shift button, click on each and every domain in the carotid geometry **and** the rectangle so they become red.

(76)   Click on the '+' button→cyl1, cone1, ext4, ext3, ext2, ext1, uni1 (total 7) now be added to the input objects list.

(77)   In the Set formula edit field, type in cyl1+cone1+uni1+ext1+ext2+ext3+ext3-ext4. Check Keep interior boundaries.
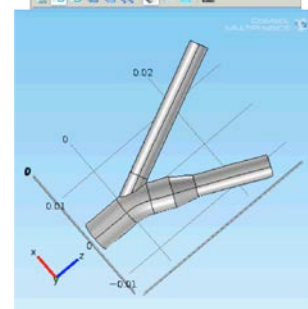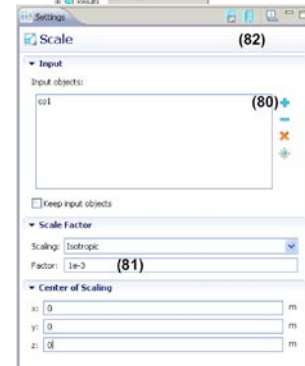
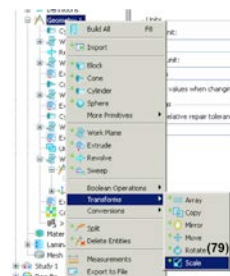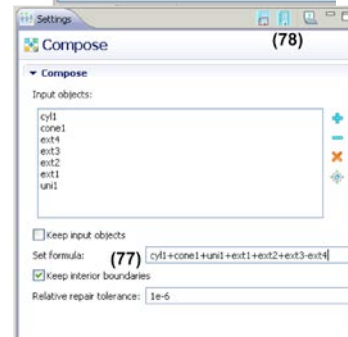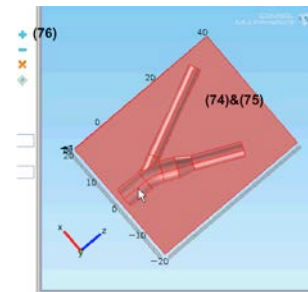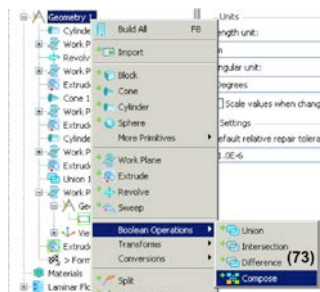(78)   Click on Build all icon.

(79)   Right click on Geometry 1 and select Transforms > Scale.

(80)   Click on the geometry to make it red and click on '+' button.

(81)   Under the Scale Factor tab, set Factor to 1e-3. We do this since we created the geometry in m but the actual dimensions are in mm.
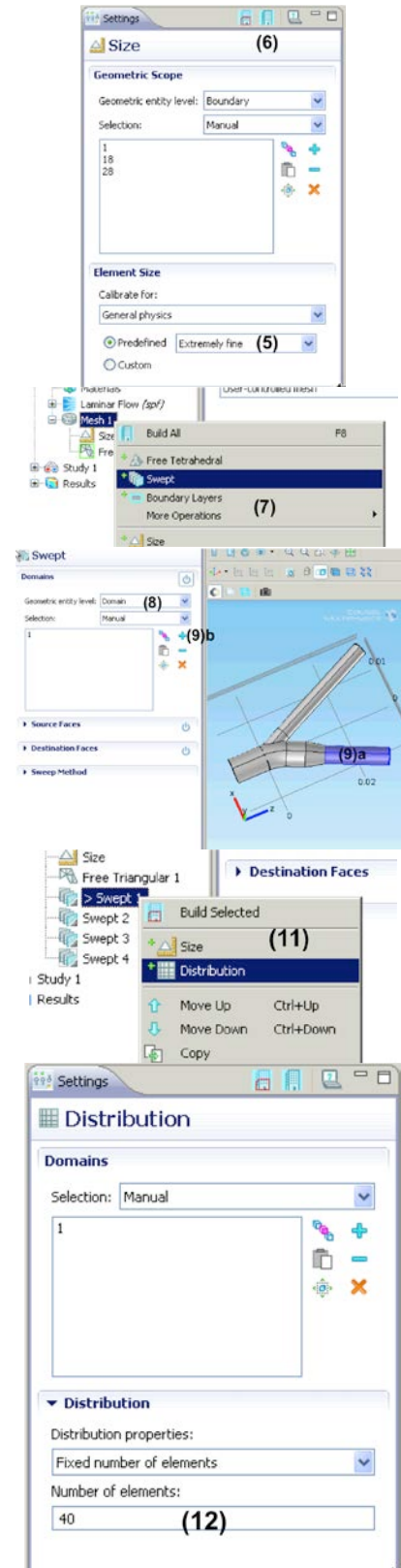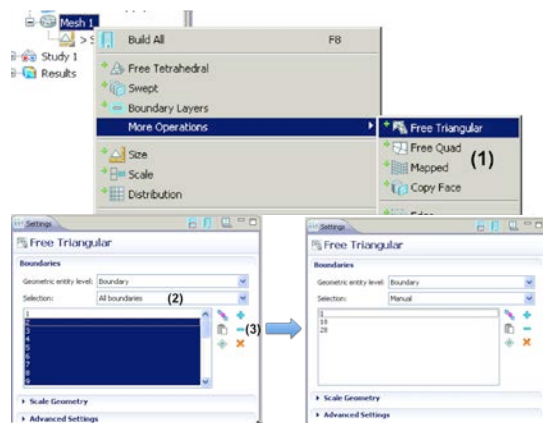
(82)   Click on Build All icon.

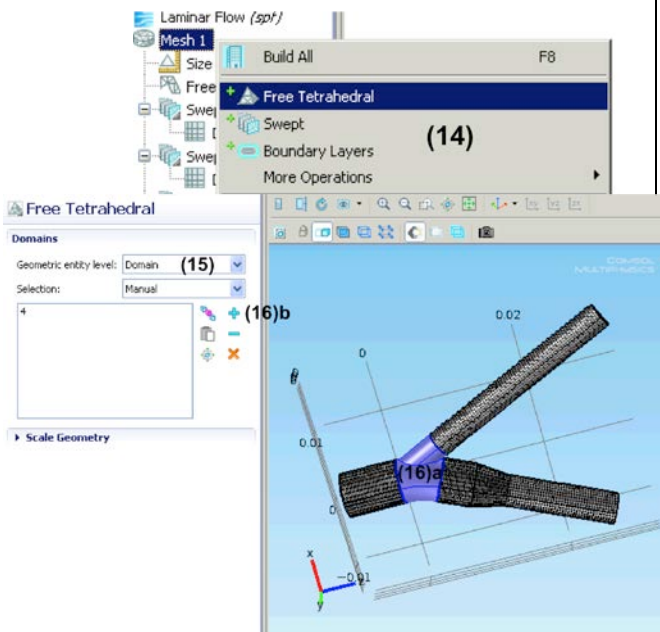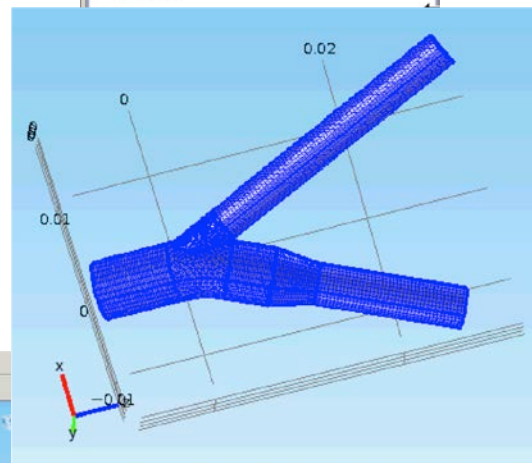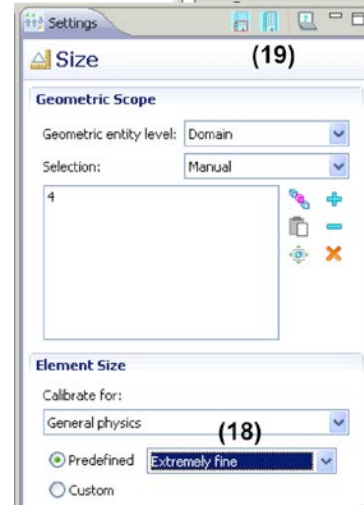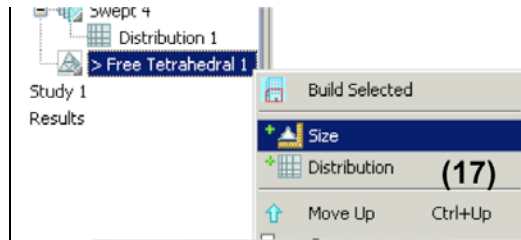(83)   Click on Zoom Extents to see the finished geometry.

Step 3: Meshing

(1) Right click on Mesh 1 and select More Operations > Free Triangular.

(2) Open the drop-down menu for Selection and select All boundaries.

(3) Remove all boundaries except 1, 18, 28 by clicking on the unwanted boundaries in the list and then pressing the '-' button. The final list should only contain 1, 18, 28.

(4) Right click on Free Triangular 1 and select Size.

(5) Under the Element Size tab, select Extremely fine for Predefined.

(6) Click on Build all icon.

(7) Right click on Mesh 1 and select Swept. Repeat 3 more times to end up with four Swepts in total (Swept 1-4).

(8) Click on Swept 1. Change the Geometric entity level to Domain.

(9) Click on subdomain 1 and press the '+' button.

(10) Repeat (8)-(9) for Swept 2-4. For Swept 2, add subdomains 2&3. For Swept 3, add subdomain 5. For Swept 4, add subdomain 6.

(11) Right click on Swept 1 and select Distribution.

(12) Set Number of elements to 40.

Build Selected

Size

Distribution **(17)**

Move Up   Ctrl+Up

Settings

Size   **(19)**

Geometric Scope

Geometric entity level: Domain

Selection: Manual

4

Element Size

Calibrate for:

General physics   **(18)**

⦿ Predefined   Extremely fine

○ Custom

(13)   Repeat (11)-(12) for Swept 2-4. For Swept 2, the number of elements is 20. For Swept 3, the number of elements is 30. For Swept 4, the number of elements is 70. Click on Build All icon when you are done with all Swept 1-4.

(14)   Right click on Mesh 1 and select Free Tetrahedral.

(15)   Change the Geometric entity level to Domain.

(16)   Click on subdomain 4 and press the '+' button.

(17)   Right click on Free Tetrahedral 1 and select Size.

(18)   Under the Element Size tab, select Extremely fine for Predefined.

(19)   Click on the Build All icon.

Laminar Flow *(spf)*

Mesh 1

Size

Free

Swep

Build All   F8

Free Tetrahedral

Swept

Boundary Layers   **(14)**

More Operations

Free Tetrahedral

Domains

Geometric entity level: Domain   **(15)**

Selection: Manual

4   **(16)b**

▸ Scale Geometry

(16)a

Step 4: Governing Equations, source terns, IC, BC
Subdomain settings

    (1) Click on Laminar Flow (spf) > Fluid Properties 1.

    (2) Select User Defined for density (ρ) and dynamic viscosity (μ).

    (3) Enter 1000 (kg/m^3) for ρ and 0.0035 (Pa·s) for μ.

Boundary settings

    (4) Right click on Laminar Flow (spf) and select Inlet.

    (5) Click on boundary 17 (the semi-circle on the bottom of the geometry) and press '+' button.

    (6) Click the bubble for Velocity field.

    (7) Set $\mathbf{u_o(x)}$ to 0, $\mathbf{u_o(y)}$ to 0, and $\mathbf{u_o(z)}$ to vel(0)*(1-(x^2+y^2)/0.0031^2)

    (8) Right click on Laminar Flow (spf) and select Symmetry.

    (9) Press shift and click on boundaries 3,6,9,13,16,25 (the flat sides of the geometry).

    (10)   Press the '+' button.

    (11)   Right click on Laminar Flow (spf) and select Outlet.

    (12)   Press shift and click on boundaries 1 and 28 (the top two semi-circles).

    (13)   Press the '+' button.
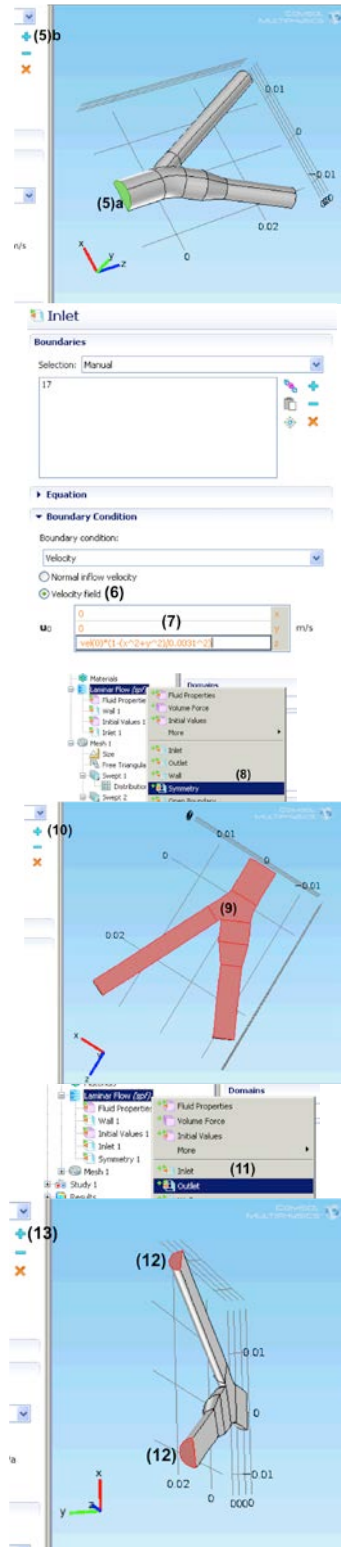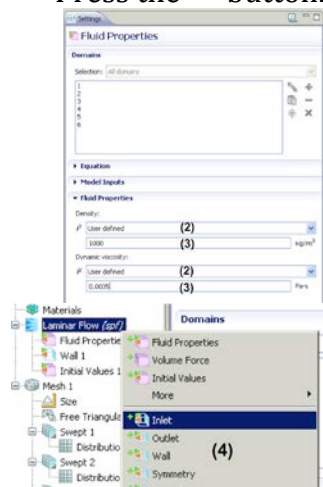
Step 5: Define function

(1) Right click on Global Definitions and select Functions > Interpolation.
(2) Change the Data source to File.
(3) Click on Browse and select the text file, vel.txt (or your filename) that was created earlier.
(4) Click on Import.
(5) Change the function name to vel
(6) Click on Plot icon to see the velocity as a function of time.

Step 6: Solver settings and solution

(1) Right click on Study 1 and select Compute.

This solution for Study 1, which was for Stationary, will be used as the initial condition values for Study 2, which is for Time-dependent.

(2) Under the **Model Builder** tab, right click on the first node (the one that has the filename of your mph file (e.g. casestudy6.mph *(root)*)

(3) Select Add Study.

(4) Select Time Dependent.

(5) Click on the race flag icon.

(6) Under the Study Settings tab, enter range(0,0.001,0.75) for Times edit field.

(7) Right click on Study 2 > Solver Configurations and select Show Default Solver.

(8) Click on Study 2 > Solver Configurations > Solver 2 > Dependent Variables 1.

(9) Under the 'Initial Values of Variables Solved For' tab, select Solution for Method and Solver 1 for Solution.

(10) Under the 'Values of Variables Not Solved For' tab, select Solution for Method and Solver 1 for Solution.
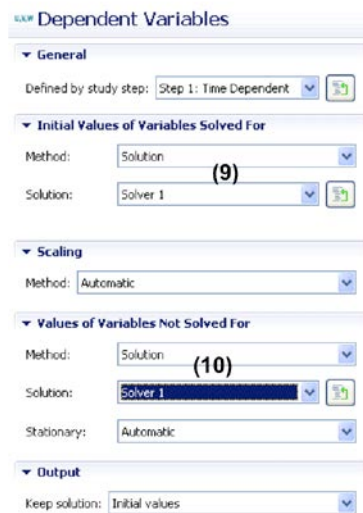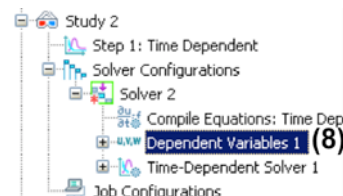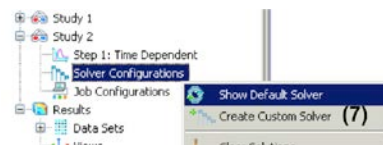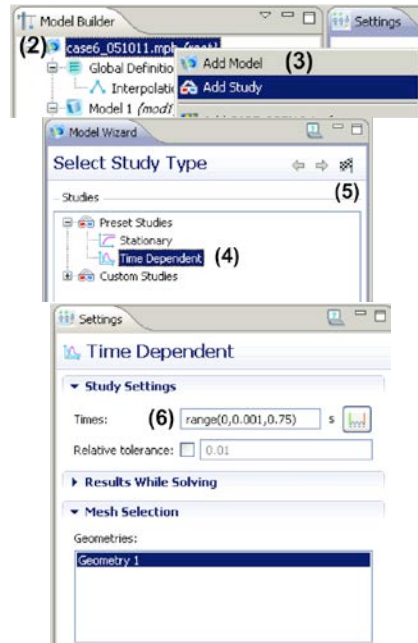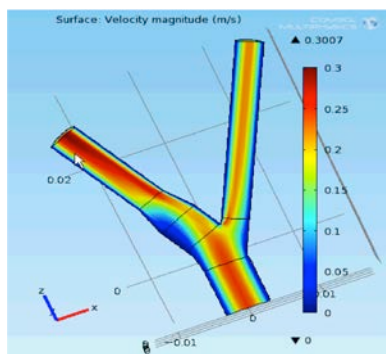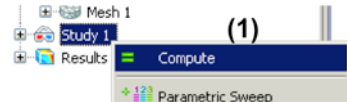
(11)   Right click on Study 2 > Solver Configurations > Solver 2 > Time-Dependent Solver 1 > Direct and select Enable.

(12)   Under the General tab, select PARDISO for Solver.

(13)   Click on Study 2 > Solver Configurations > Solver 2 > Time-Dependent Solver 1.

(14)   Under the Time Stepping tab, select Strict for Steps taken by solver.

(15)   Click on Model 1 (mod1) > Laminar Flow (spf) > Inlet 1.

(16)   Under the Boundary Condition tab, change the expression for $u_0(z)$ from vel(0)*(1-(x^2+y^2)/0.0031^2) to vel(t)*(1-(x^2+y^2)/0.0031^2)

(17)   Right click on Study 2 and select Compute.

Note that this problem is very large and needs considerable memory (more than 4Gb) to solve it. It will take around 1.5-2 hours to solve the problem.
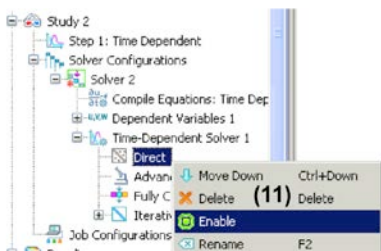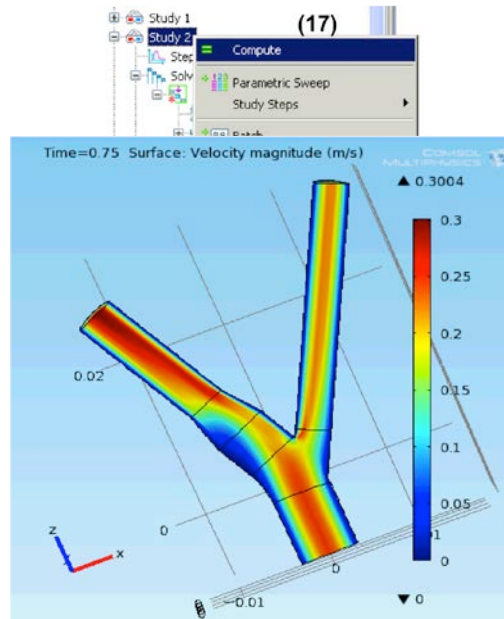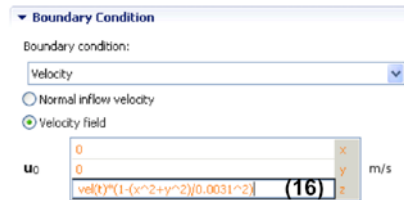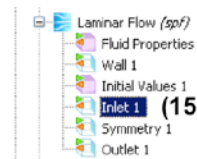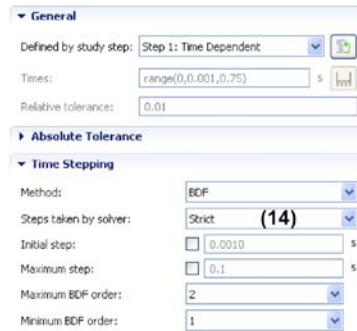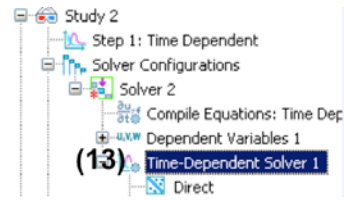
Step 7: Postprocessing

We will plot the velocity distribution in the domain at 0.75s (end time) at 15 different slices along the z-axis.

(1) Click on Results > 3D Plot Group 4.
(2) Make sure that under the Data tab, Solution 2 is selected for Data set and 0.75 is selected for Time.
(3) Click on Results > 3D Plot Group 4 > Slice 1.
(4) Under the Plane Data tab, select xy-planes for Plane and Number of planes for Entry method.
(5) Enter 15 in the Planes edit field.
(6) Click on the Plot icon.