COMSOL: A Tutorial for the Befuddled

by

Loren B. Schreiber

Overview

COMSOL Multiphysics® is sophisticated computer program that can simulate diverse and coupled physical phenomena such as fluid flow, heat transfer, mass transfer, electromagnetic radiation, structural mechanics, and acoustics.

This tutorial provides a step-by-step guide to students who (1) are presently learning elementary fluid mechanics and (2) have had no prior experience using COMSOL.

The physical application in this tutorial is tank discharge. Connected to the bottom of the tank is an outlet pipe. This classic problem appears in popular textbooks. Students may thus compare the numerical solution generated by COMSOL with approximate analytical solutions.

Having completed this tutorial, students will be able to examine what-if scenarios in tank discharge. Further, students will be able to apply the discipline of the modeling process as they learn about additional physical phenomena. Students will also be able to start to play with the COMSOL package to explore its extensive features.

Screen shots with comments

Note

This tutorial was developed using COMSOL Multiphysics® version 4.3a.

Problem Statement
The system consists of a vertical cylindrical tank connected to a vertical cylindrical pipe. The connection is at the center of the bottom of the tank. Thus, the system is symmetrical about the centerline of the tank and pipe (or *axisymmetric* in COMSOL vocabulary). A cylindrical coordinate system, with r as the radial coordinate and z as the vertical coordinate, thus proves convenient.

A viscous liquid discharges from the system in laminar flow.

The problem is to predict the time for the liquid to discharge from an initial height in the tank to a specified lower height.

Problem Parameters

Symbol	Value	Description
Dp	0.00461 m	Diameter of the pipe
Dt	0.1466 m	Diameter of the tank
H	10 in	Height of the liquid in the tank
gaccel	9.80665 m/s^2	Gravitational acceleration
Lp	0.6096 m	Length of the pipe
mu	rho · nu	Viscosity of the liquid
nu	1.31 10^{-5} m^2/s	Kinematic viscosity of the liquid
Pbottom	0 Pa	Pressure at the bottom outlet of the pipe
Ptop	0 Pa	Pressure at the top inlet of the tank
rho	1041 kg/m^3	Density of the liquid
Rentr	0.0001 m	Radius of curvature at the pipe connection to the bottom of the tank
Rp	Dp / 2	Radius of the pipe
Rt	Dt / 2	Radius of the tank

Notes:
1. The value of H in the table is the initial height of liquid in the tank. The Parametric Sweep feature in COMSOL enables the user to specify additional values of H, at equal intervals. For this problem, we specify heights of 10 in, 9.5 in, 9 in, …, 1.5 in, 1 in.
2. In this problem, the pressures are gauge pressures, i.e. relative to atmospheric pressure.
3. COMSOL lacks the ability to input formatting features such as italic typeface, subscripts, and Greek letters. Accordingly, in this tutorial, symbols for physical quantities are written in ordinary roman fonts of the English alphabet.

Background

The document, cited below, provides background on the problem of tank discharge, including an extensive list of references:

> Schreiber, Loren B. *Expt 140 – Theoretical Background*. Unit Operations Laboratory, Department of Chemical and Biomedical Engineering, FAMU-FSU College of Engineering, Rev. 5a, October 28, 2012.

This document develops the macroscopic equations to describe tank discharge and outlines their solution. In addition, the document provides an example spreadsheet calculation for turbulent flow and an analytical solution for laminar flow.

A key word in the above paragraph is <u>macroscopic</u>. COMSOL solves the <u>microscopic</u> equations of fluid flow, i.e., partial differential equations.

The macroscopic solution predicts an average velocity in the tank and an average velocity in the pipe. COMSOL predicts the velocity (magnitude and direction) at any point within the tank and pipe. COMSOL also provides a convenient tool (Derived Values) that can integrate the point velocities across a surface to obtain a volumetric flow rate or an average velocity; the user may export these values to other software (such as Microsoft Excel) for subsequent calculations.

We obtain the equation for the Derived Values feature by integrating the velocity across the horizontal surface at the pipe outlet, recognizing the axial symmetry of this problem:

$Q = \iint \boldsymbol{v} \cdot d\boldsymbol{A} = \iint (-v_z) \cdot r \, d\Phi \, dr = 2\pi \int (-v_z) \cdot r \, dr$ →→ Or in COMSOL notation: ∫ (−2 * pi * r * w) dr

The Derived Values feature can also calculate the cross-sectional area at the pipe outlet:

$\text{Acs} = \iint d\boldsymbol{A} = \iint r \, d\Phi \, dr = 2\pi \int r \, dr$ →→ Or in COMSOL notation: ∫ (2 * pi * r) dr

And the average velocity across the pipe outlet:

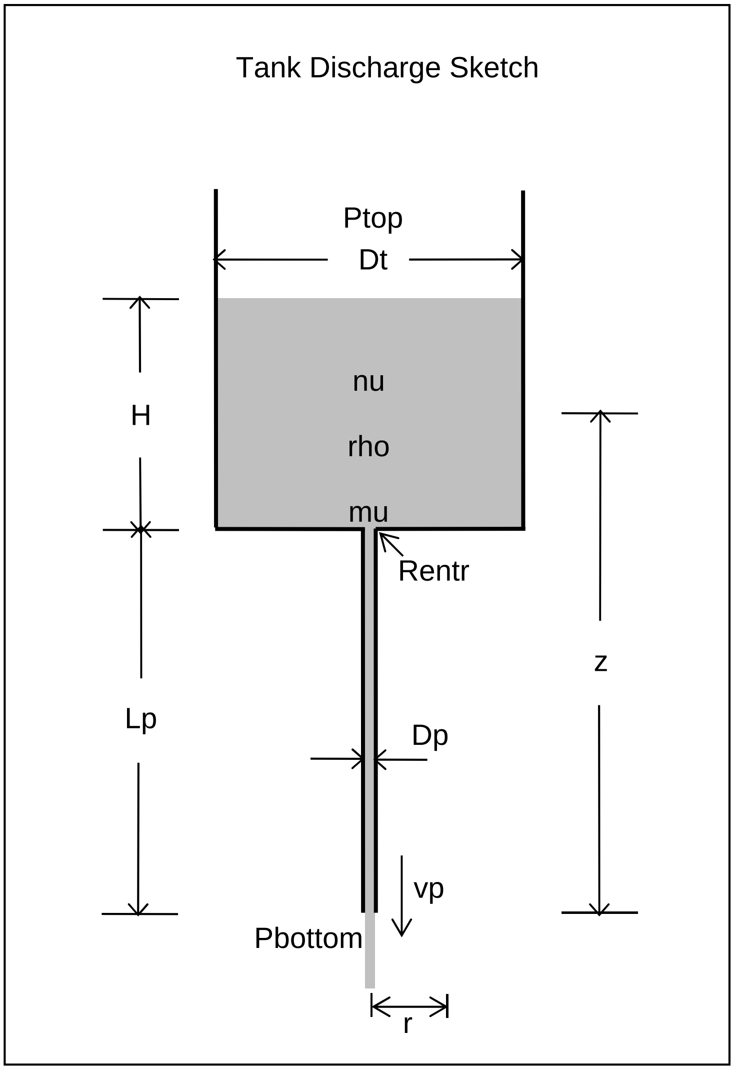
vp = Q / Acs = ∫ (−2 * pi * r * w)/(pi*Rp^2) dr

Although not used directly in this problem, COMSOL also predicts the pressure throughout the system.

There is an important practical difference in the macroscopic and microscopic approaches. The macroscopic solution requires estimates of parameters to account for the contraction from the tank into the pipe and for the development of the velocity profile in the pipe. Reliable estimates of these parameters may be unavailable. The microscopic solution in COMSOL has no such requirement.

This problem is dynamic. As the liquid level falls, the velocity decreases. COMSOL can handle this dynamic problem, but the computational time is excessive. Here we invoke the quasi-steady-state approximation. That is, we solve for the fluid velocity as if the system were steady state. This approximation is found to be sound in many practical applications of tank discharge.

After first solving for the velocity at many liquid heights, the user can then apply numerical integration to predict the discharge time. The document cited above explains this procedure and provides an example calculation, which is straightforward.

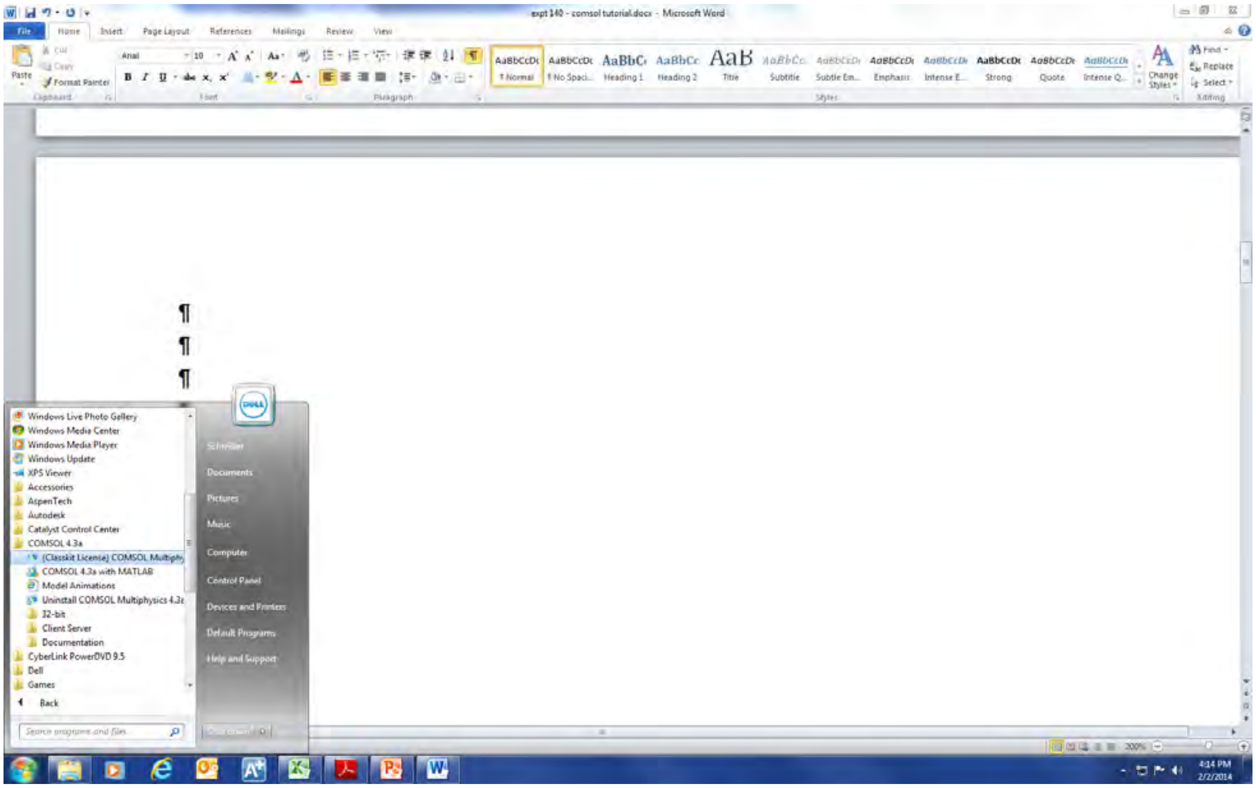


Acknowledgments
Dr. Bruce Locke and Dr. Ravi Chella conceived the idea to introduce COMSOL to our undergraduates, especially juniors taking their second semester of transport phenomena. They requested the Florida State University to fund a small project to develop COMSOL learning modules. Under their direction, in 2012 Mr. Chris Golding explored the application to several problems, including tank discharge.

In spring 2013 in Transport Phenomena Laboratory (ECH 3274L), team 2—Messrs. Joseph Duffy, Zlatko Sokolikj, and Andrew Suellentrop—applied COMSOL to predict discharge times for their Tank Discharge experiment under the guidance of Mr. Chris Golding. They also developed the first COMSOL tutorial (March 4, 2013) on tank discharge.

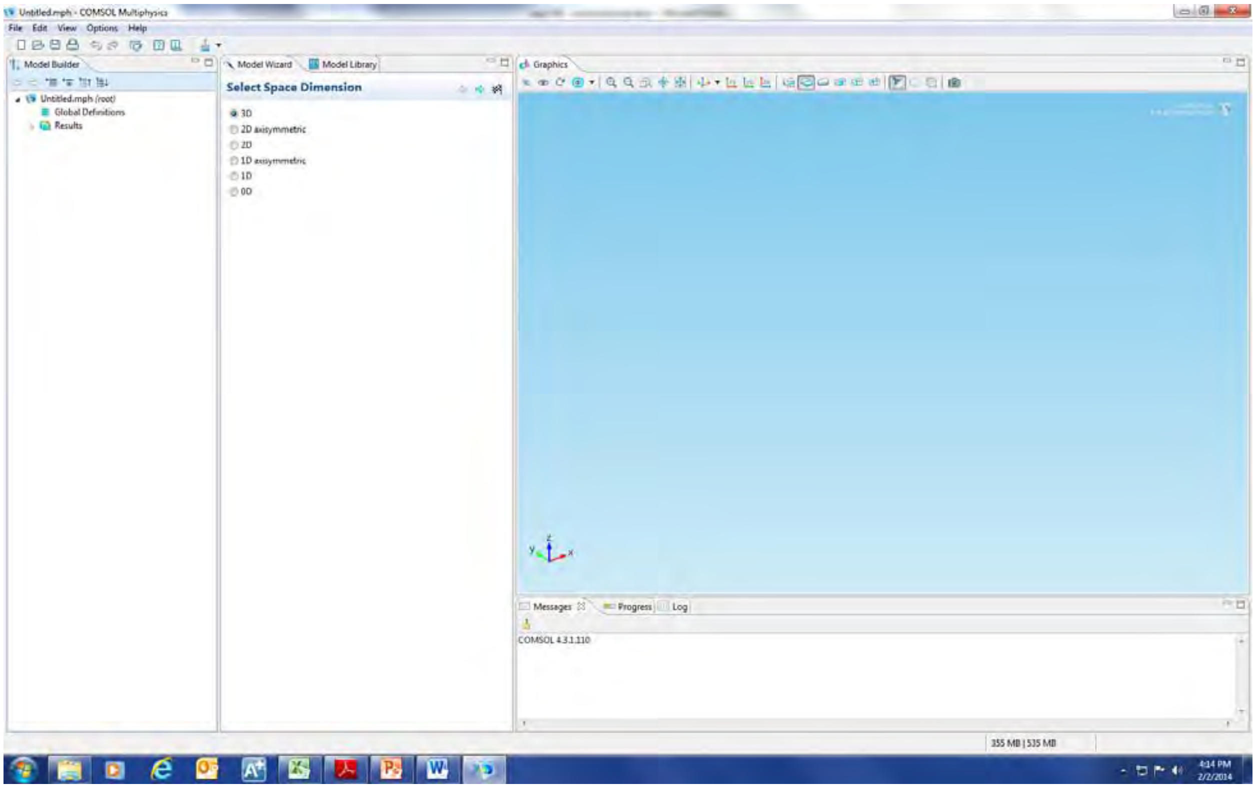
We are grateful to all the above persons for their solid contributions.

Click on the **Start** button in Windows.

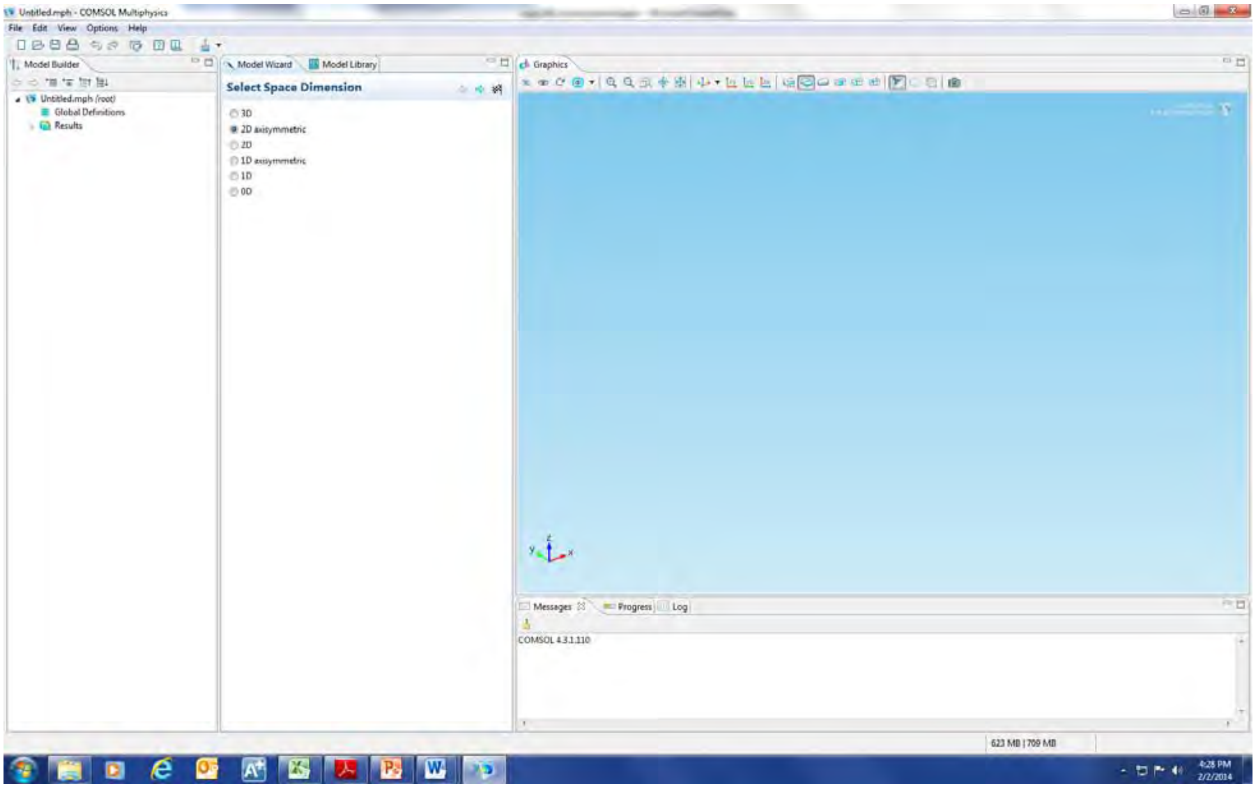


Click on COMSOL 4.3a > **(Classkit License) COMSOL Multiphysics**

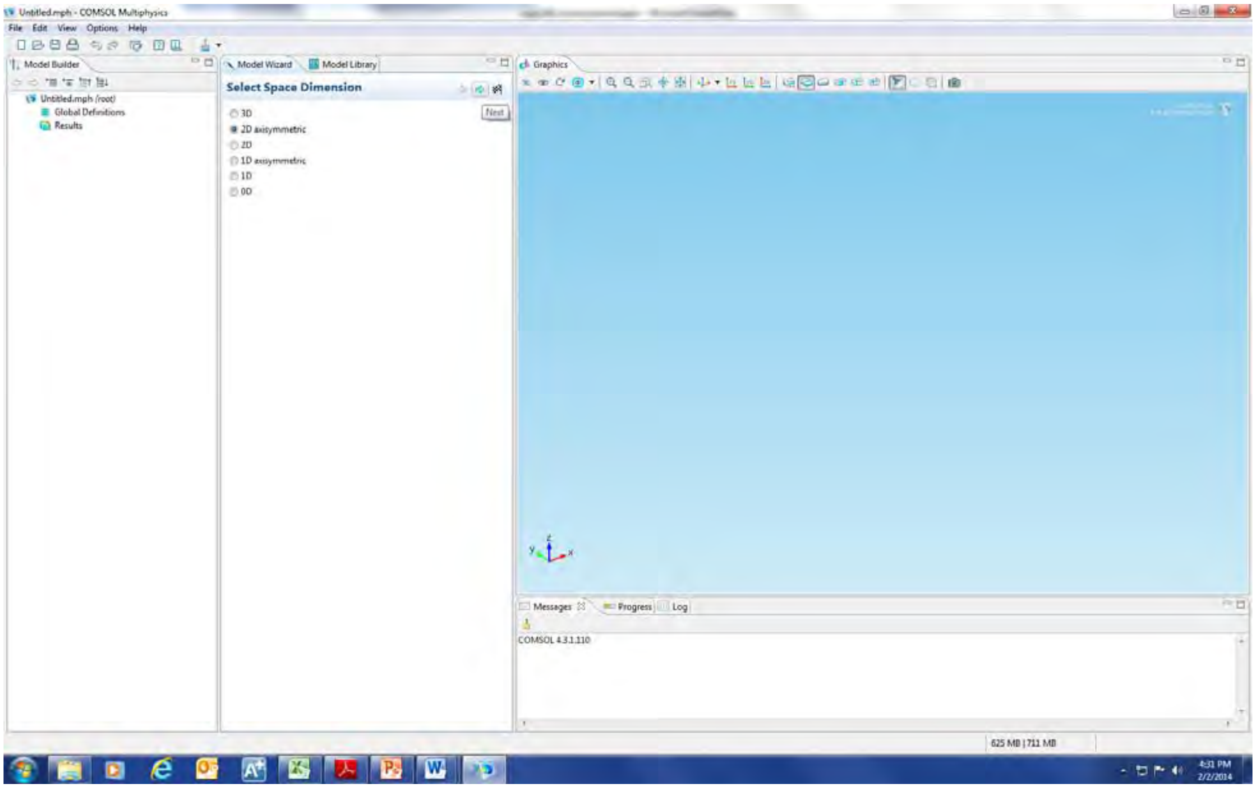
Opening screen shot

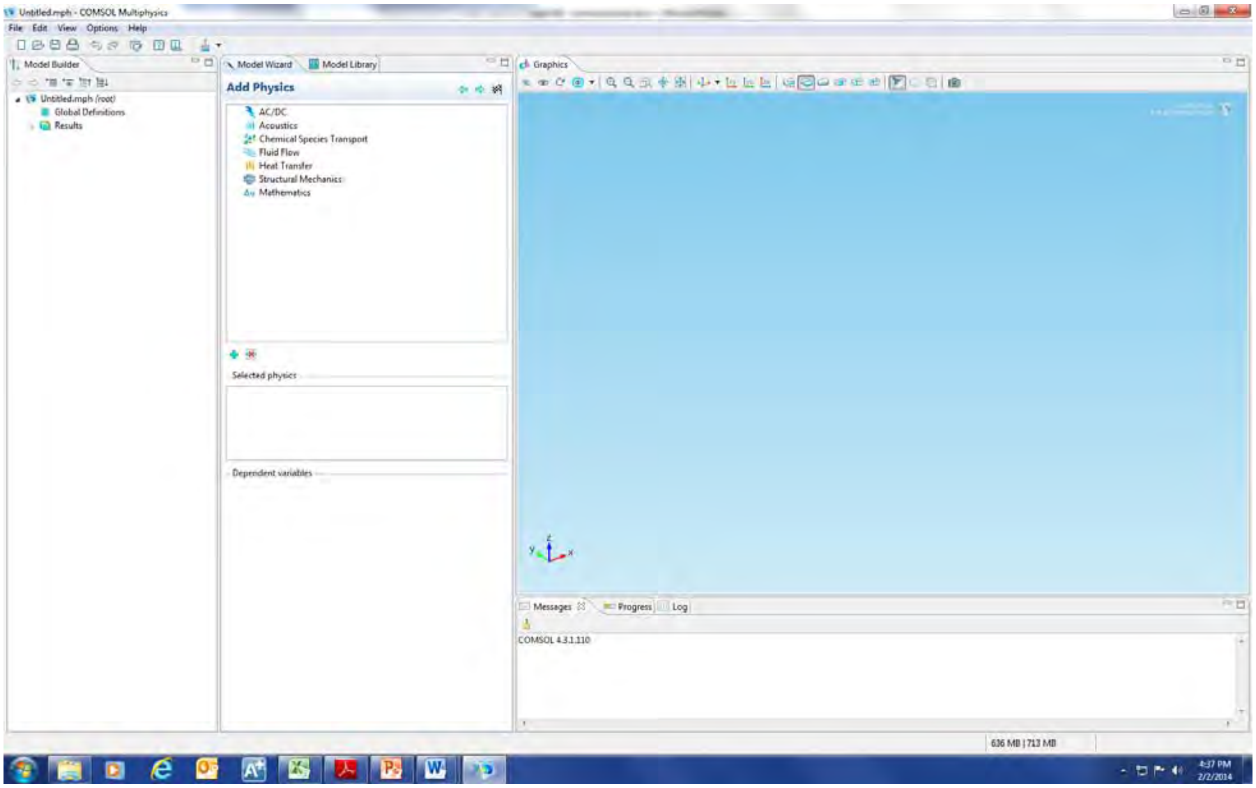


Select the radio button for **2D axisymmetric**.



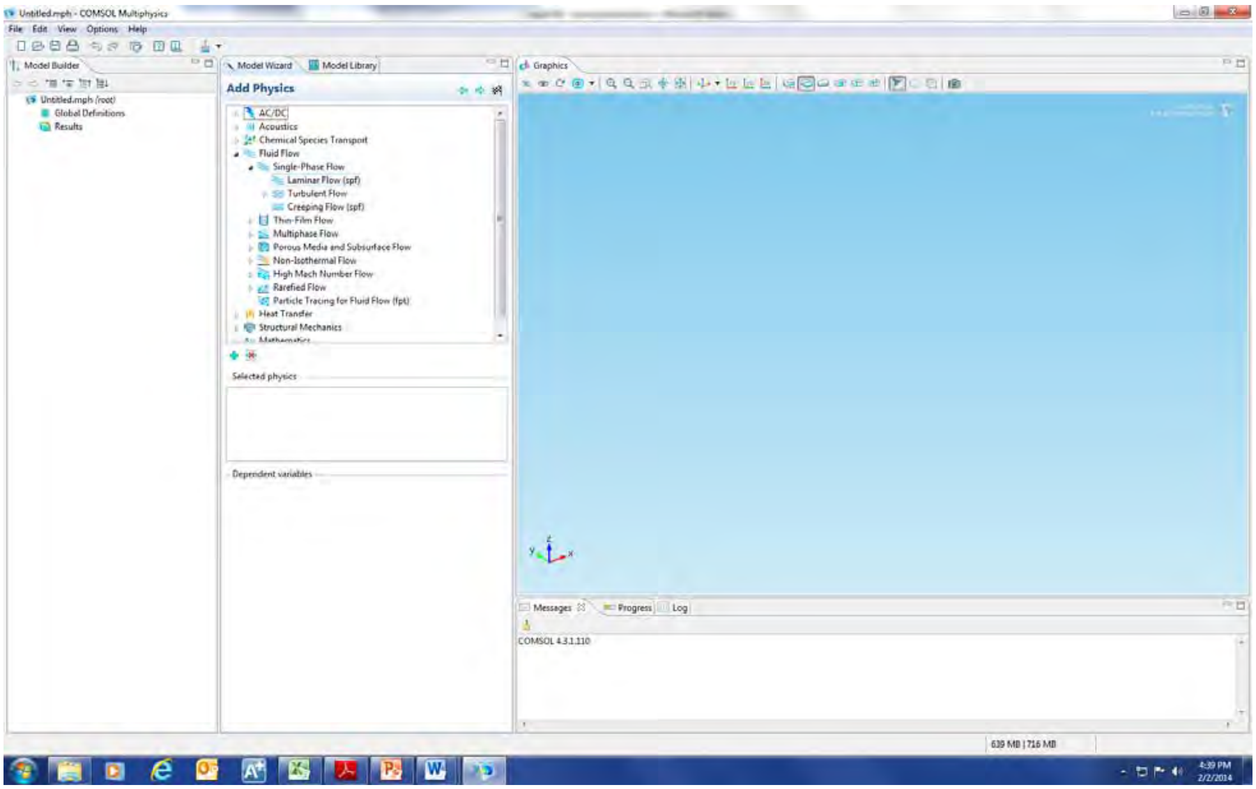
Click the Next button ➪ (located in the upper right corner of the Model Wizard window) to continue.


Untitled.mph - COMSOL Multiphysics
File Edit View Options Help
Model Builder
Untitled.mph (root)
Global Definitions
Results
Model Wizard
Model Library
Select Space Dimension
3D
2D axisymmetric
2D
1D axisymmetric
1D
0D
Next
Graphics
Messages
Progress
Log
COMSOL 4.3.1.110



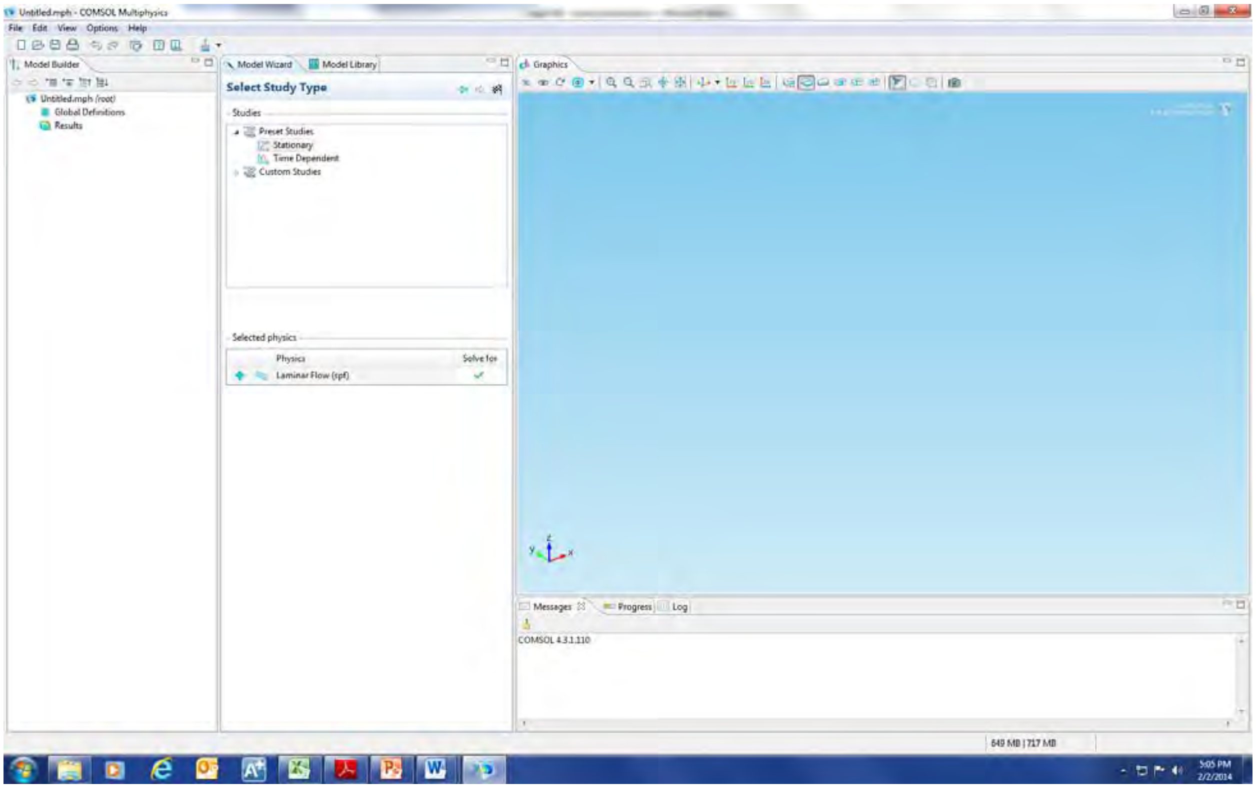
Expand the tree:

Model Wizard > Add Physics > Fluid Flow > Single-Phase Flow > Laminar Flow (spf)



Click on **Laminar Flow (spf)**.

Then click the Next button ➡.



Note:

In the window for Model Wizard, a panel appears for Select Study Type.

Two sections appear: Studies and Selected physics.

The section for Selected physics lists Laminar Flow (spf).

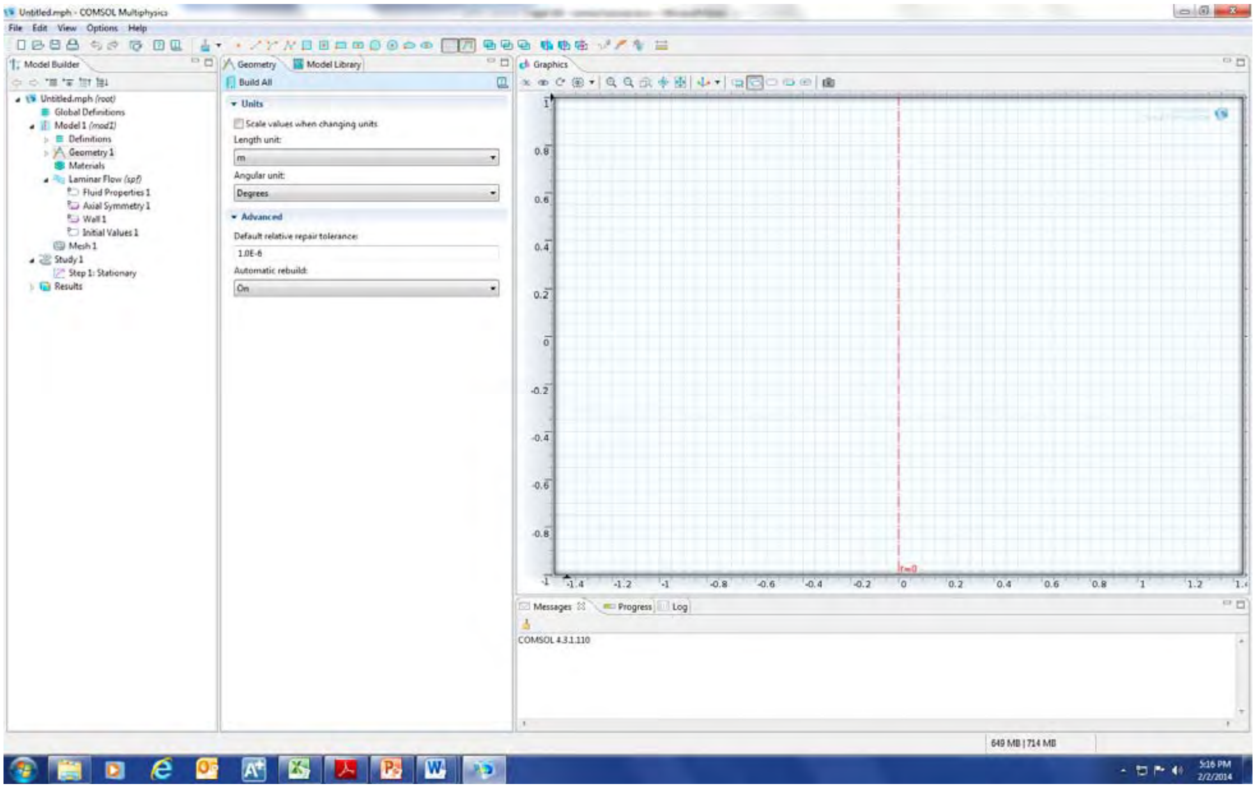
Expand the tree:

Model Wizard > Select Study Type > Preset Studies > Stationary

Click on **Stationary**.

The Next button is dim. That means we're finished with this phase of the setup.

Click on the Finish button  (located in the upper right corner of the Model Wizard window) to continue.



File name is presently Untitled.mph.

Let's rename the file and save it.

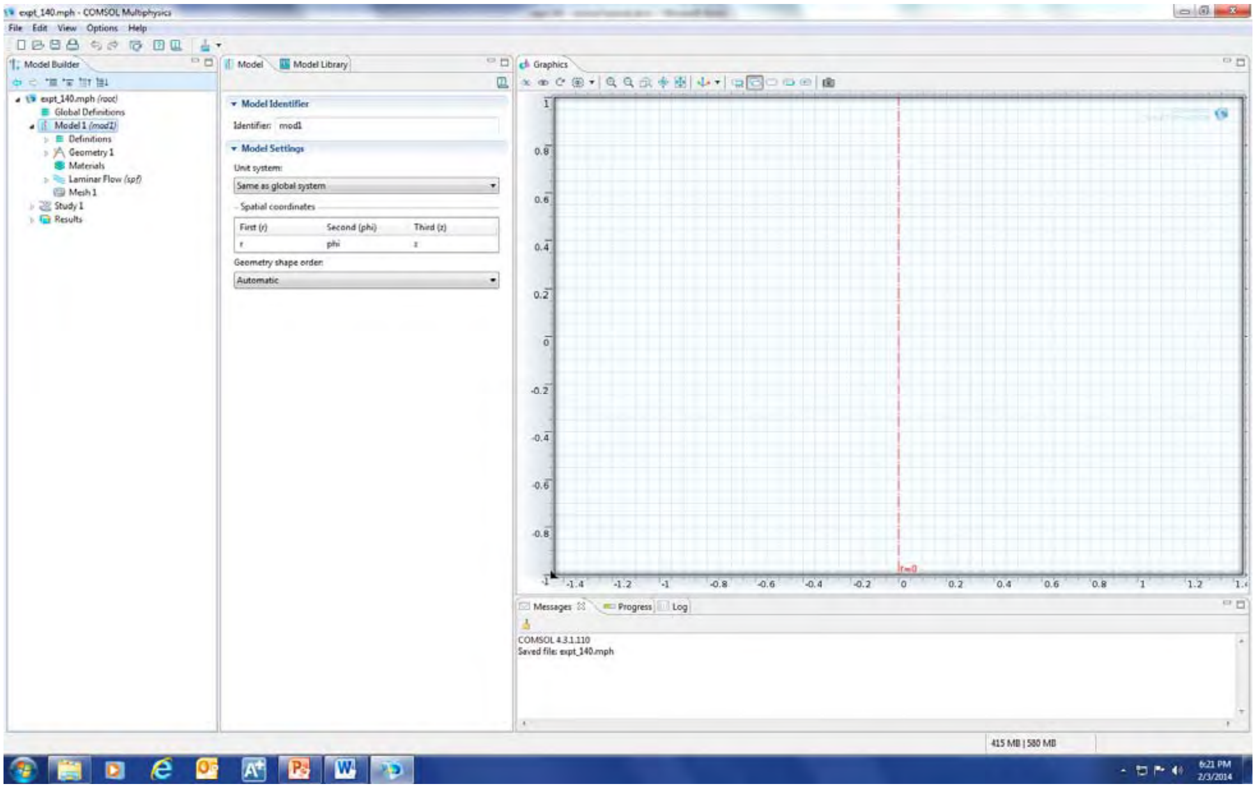
This can be done in the customary manner.

On the menu bar, click on **File**.

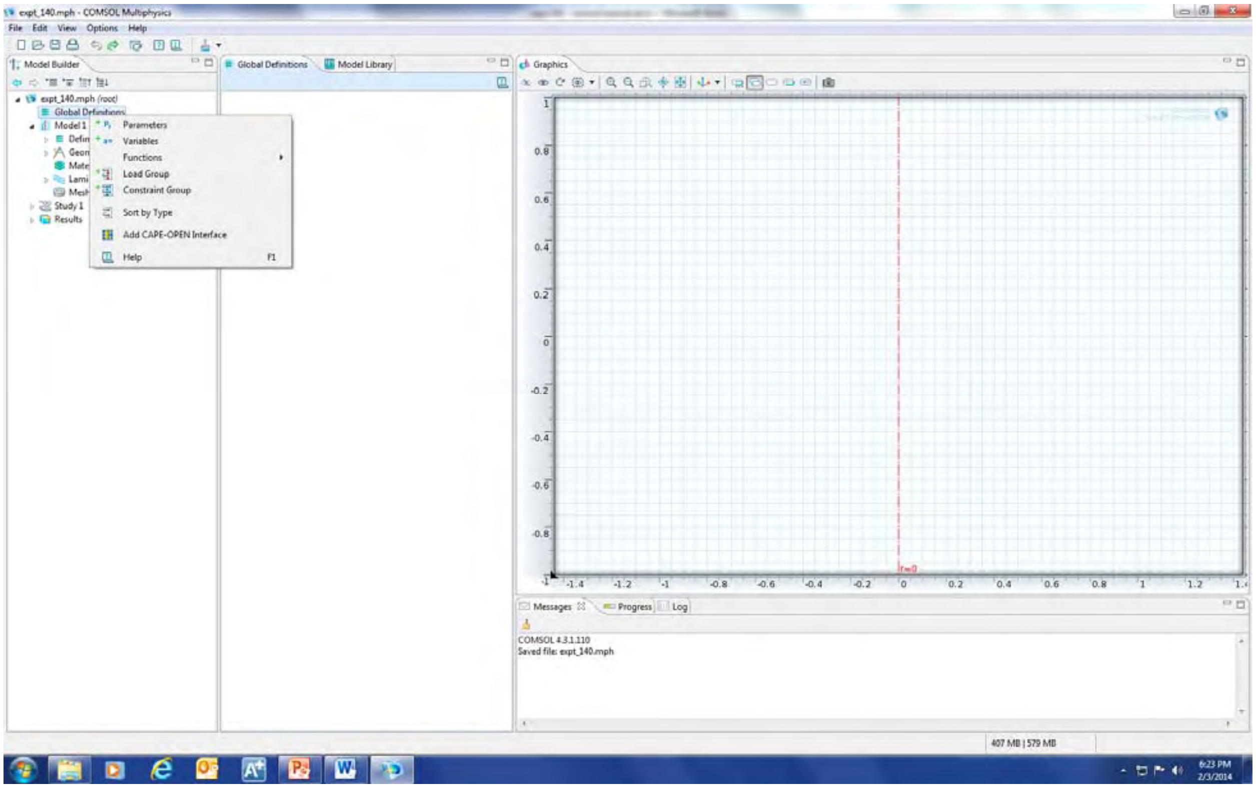
Then click on **Save As…**

Enter **expt_140** in the box for File name.

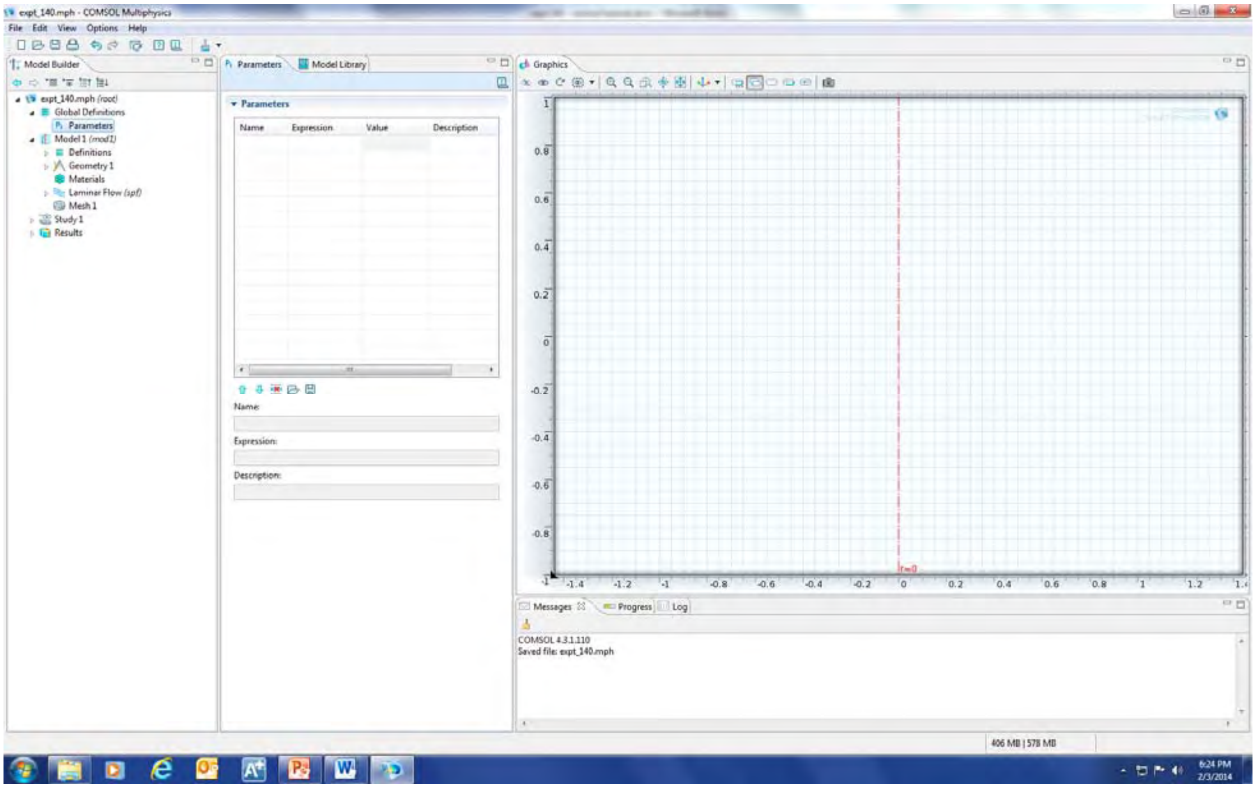
The new file name appears at the left side of the title bar at the top of the window.



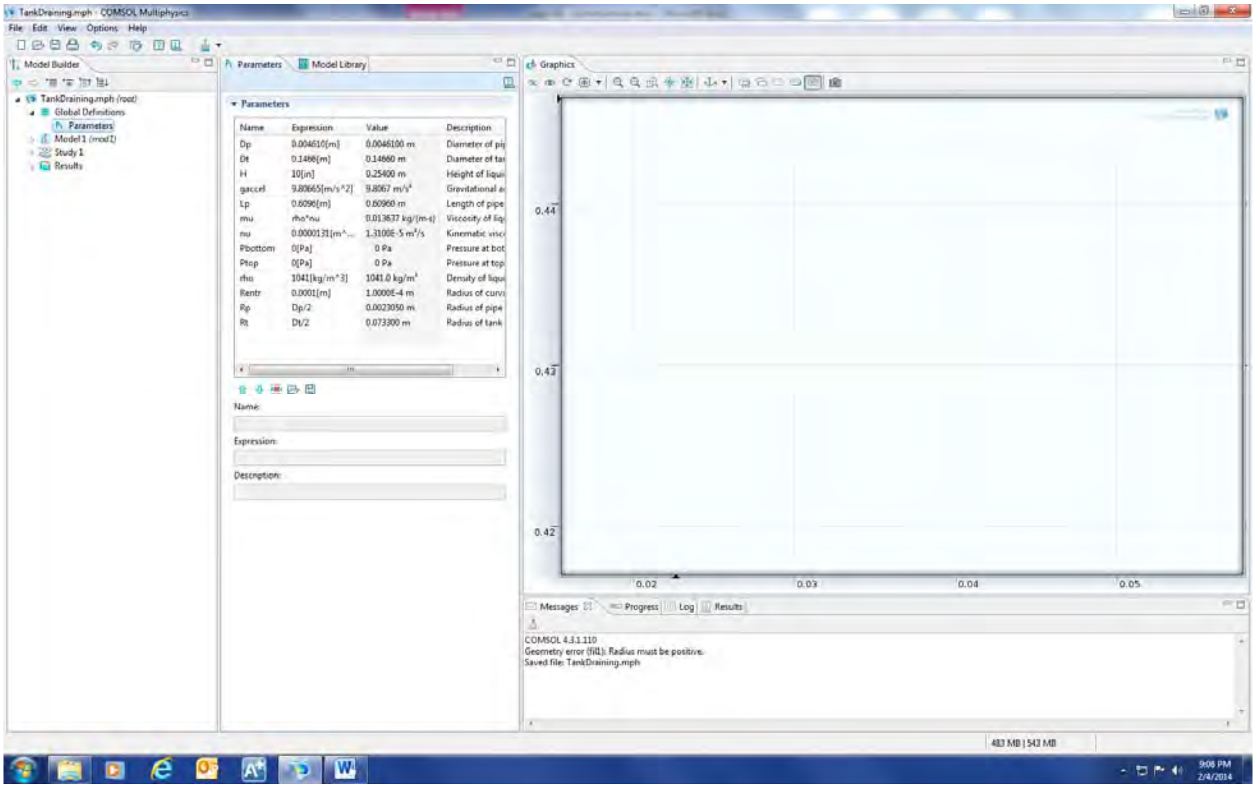
Right click on **Global Definitions** to display the pop-up menu.



Click on **Parameters**.

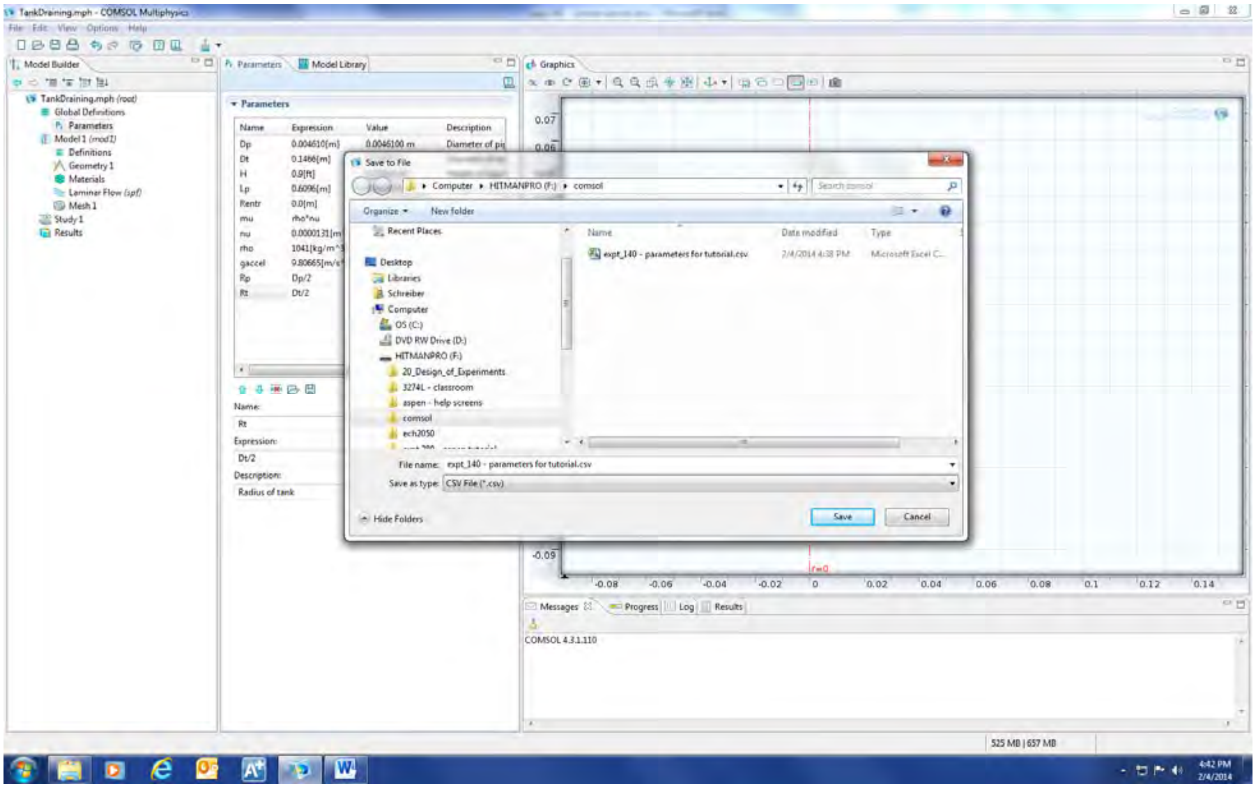


Enter the names of each parameter with the corresponding expression and description the Parameters window. COMSOL will automatically compute and display the value of the expression in the Value column.



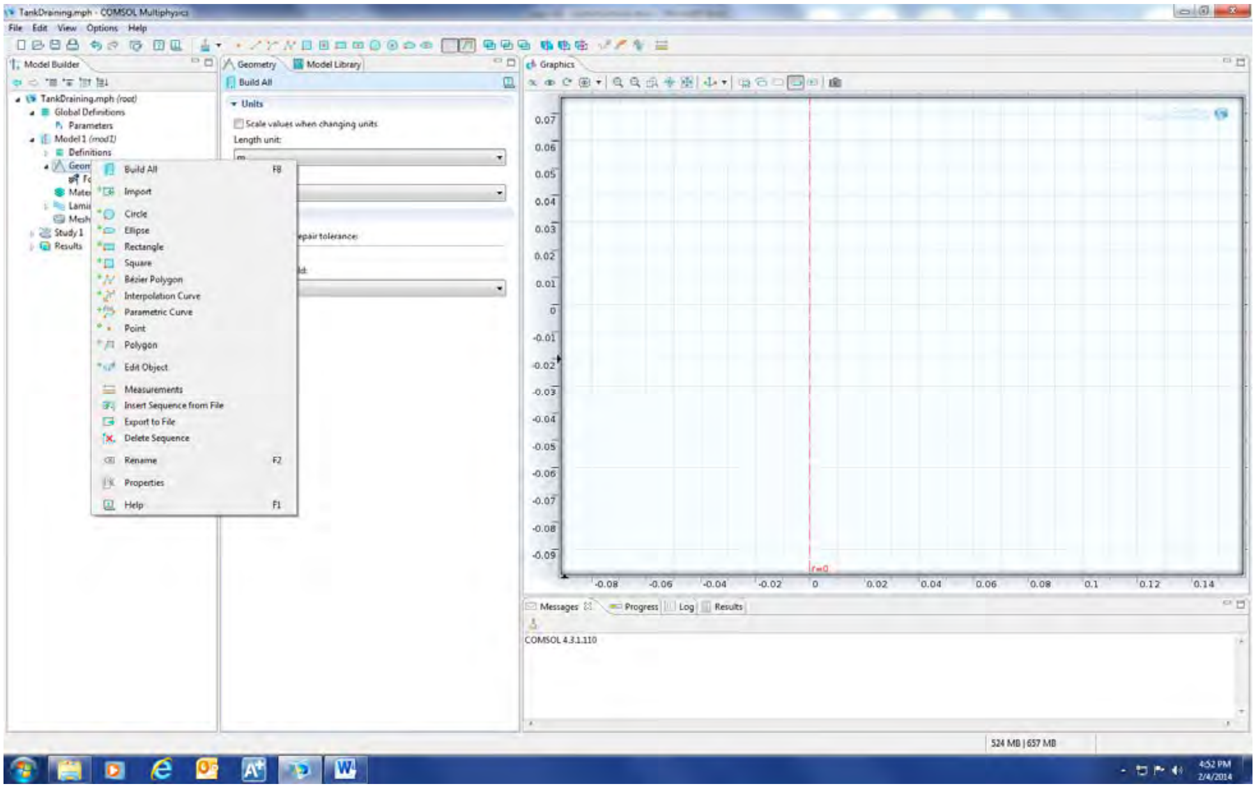
Beneath the Parameters window there is a blue up arrow and blue down arrow to adjust the location of a parameter row within the list. There is also a tool to delete an entry of a parameter.

Click on the save icon to save a copy of the parameter list for future reference.

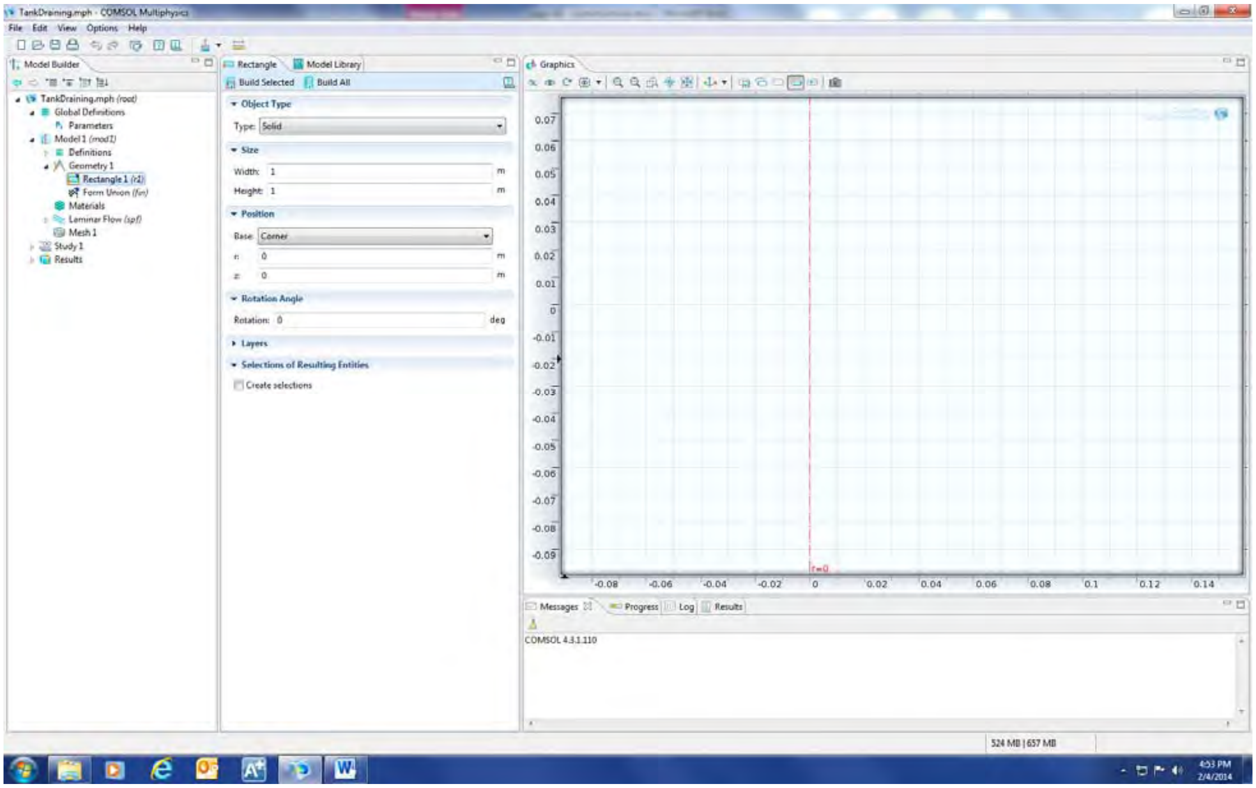


Click on the Save button.

Expand the branch for Model 1 > Geometry 1.
Right click on Geometry 1.



Click on **Rectangle**.



We shall use this rectangle to depict the tank.

In the Size section:

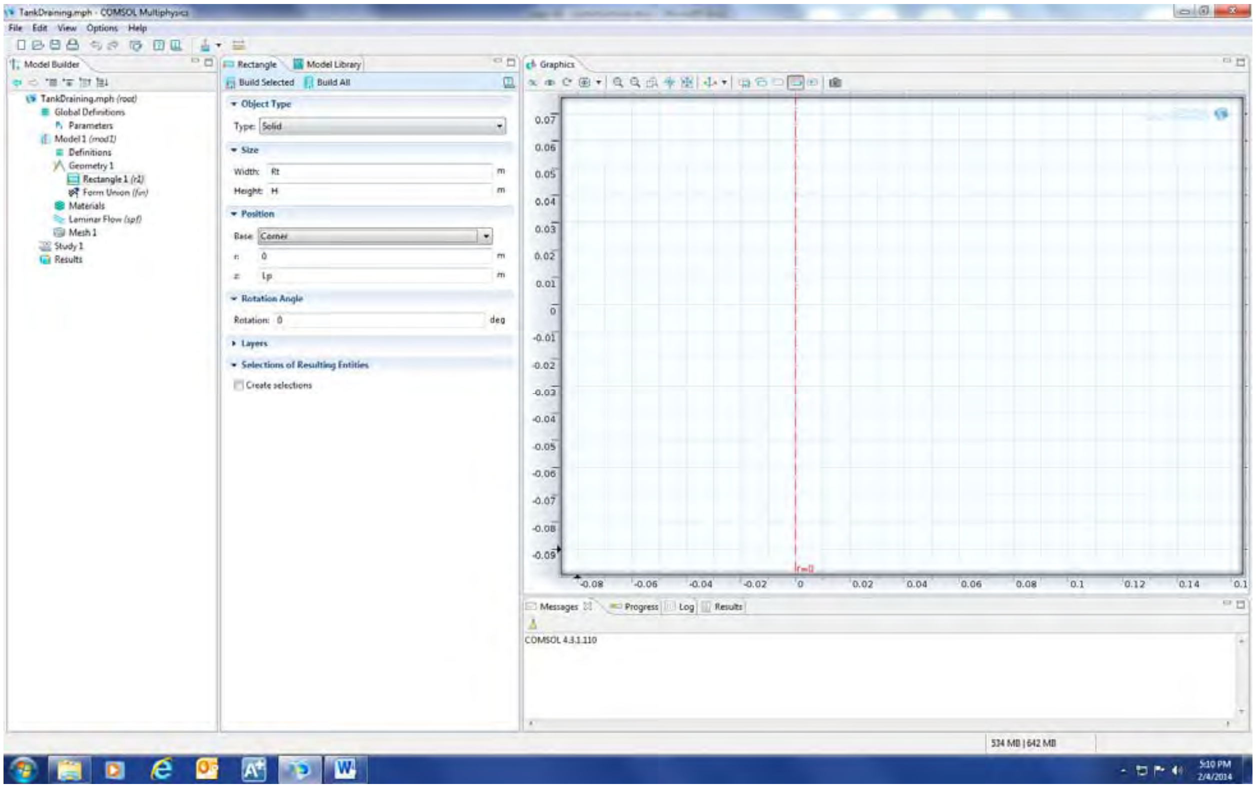
Enter **Rt** in the box for width.

Enter **H** in the box for height.

In the position section:

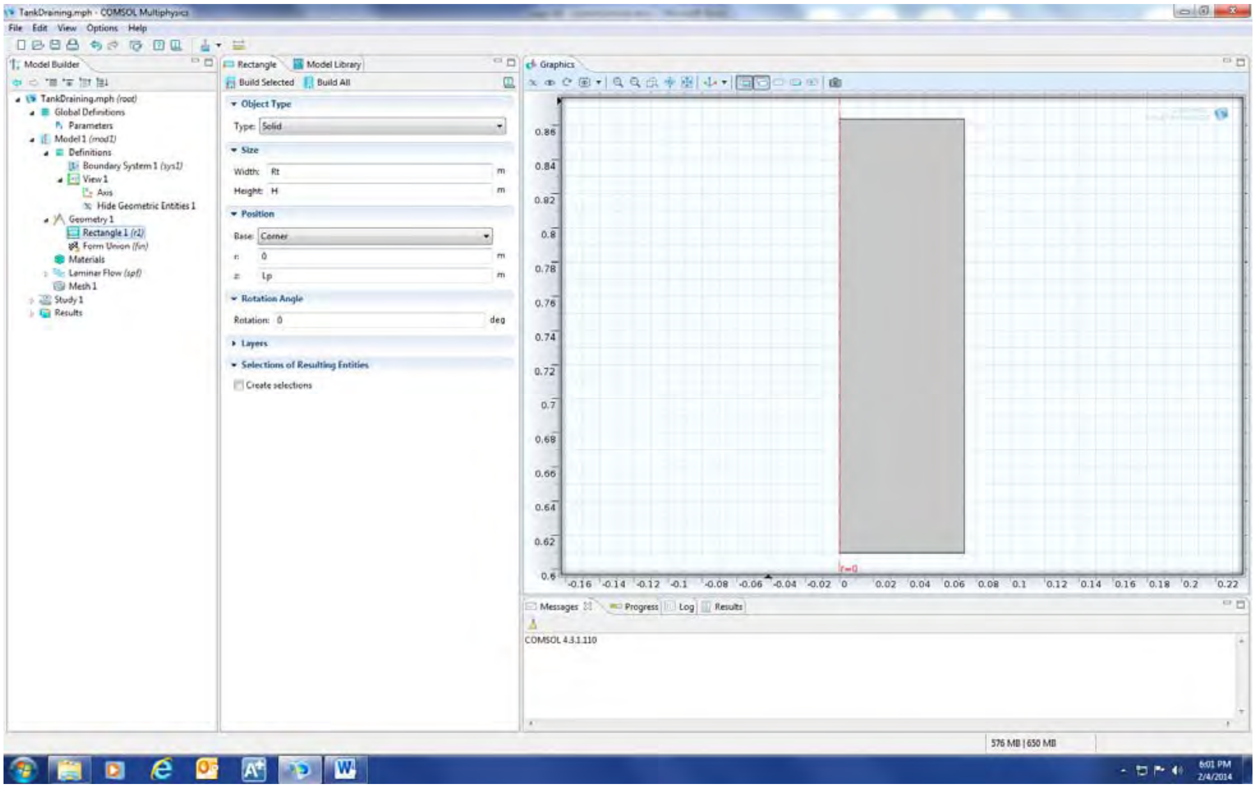
Enter **Lp** in the box for z.

Then click **Build Selected** near the top of the Rectangle panel.



Hover over each of the icons located at the top of the Graphics window. Gain familiarity with the names of the icons as the names appear.

Click on the icon for Zoom Extents.



The Graphics window now displays the rectangle.

The bottom left corner is located at r = 0 m and z = 0.610 m (Lp = 0.610 m).

The top right corner is located at r = 0.0733 m (Rt = Dt/2 = 0.1466 m/2 = 0.733 m) and z = 0.864 m (Lp + H = 0.610 m + 10 in × 0.0254 m/in = 0.864 m)

Next, we shall add a second rectangle to depict the pipe.

Try to do this yourself!

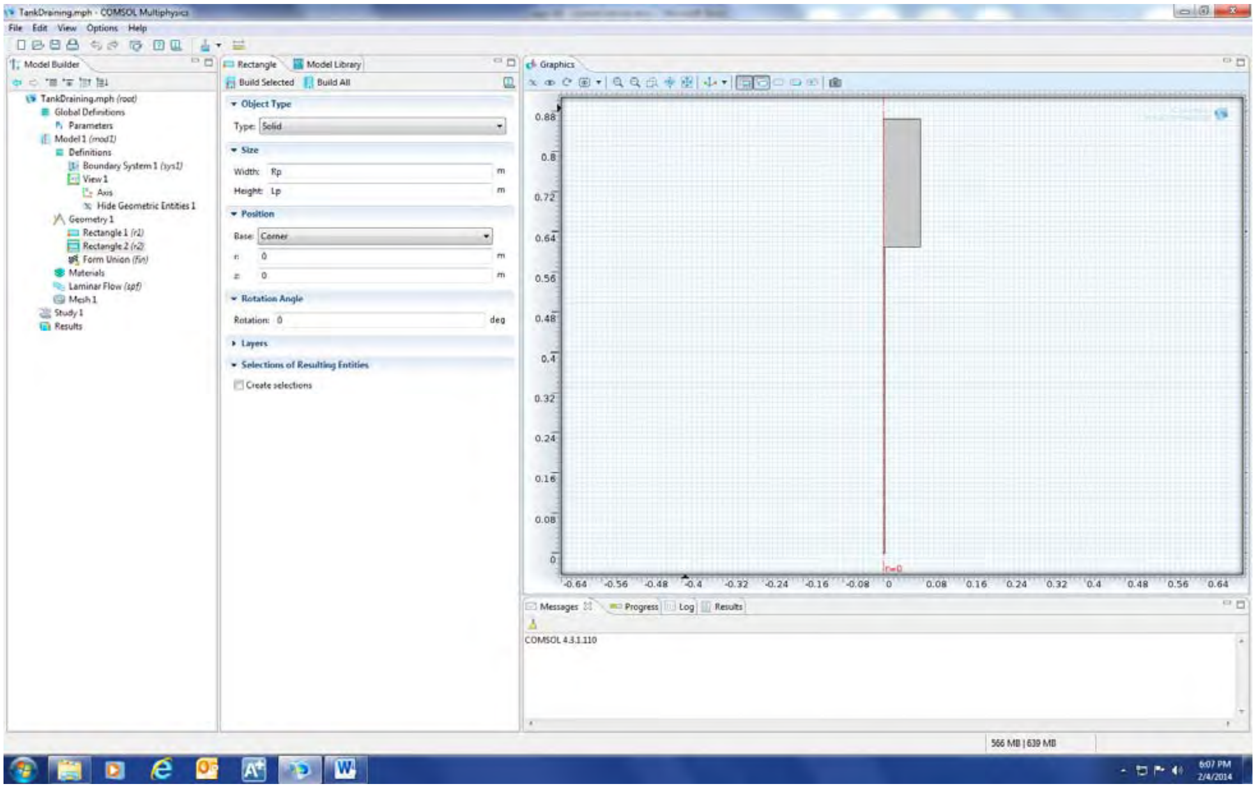
If you're having trouble, just go to the next page!

Right click again on Geometry 1.
Then click on rectangle.
We shall use this second rectangle to depict the pipe.

In the Size section:
Enter **Rp** in the box for width.
Enter **Lp** in the box for height.

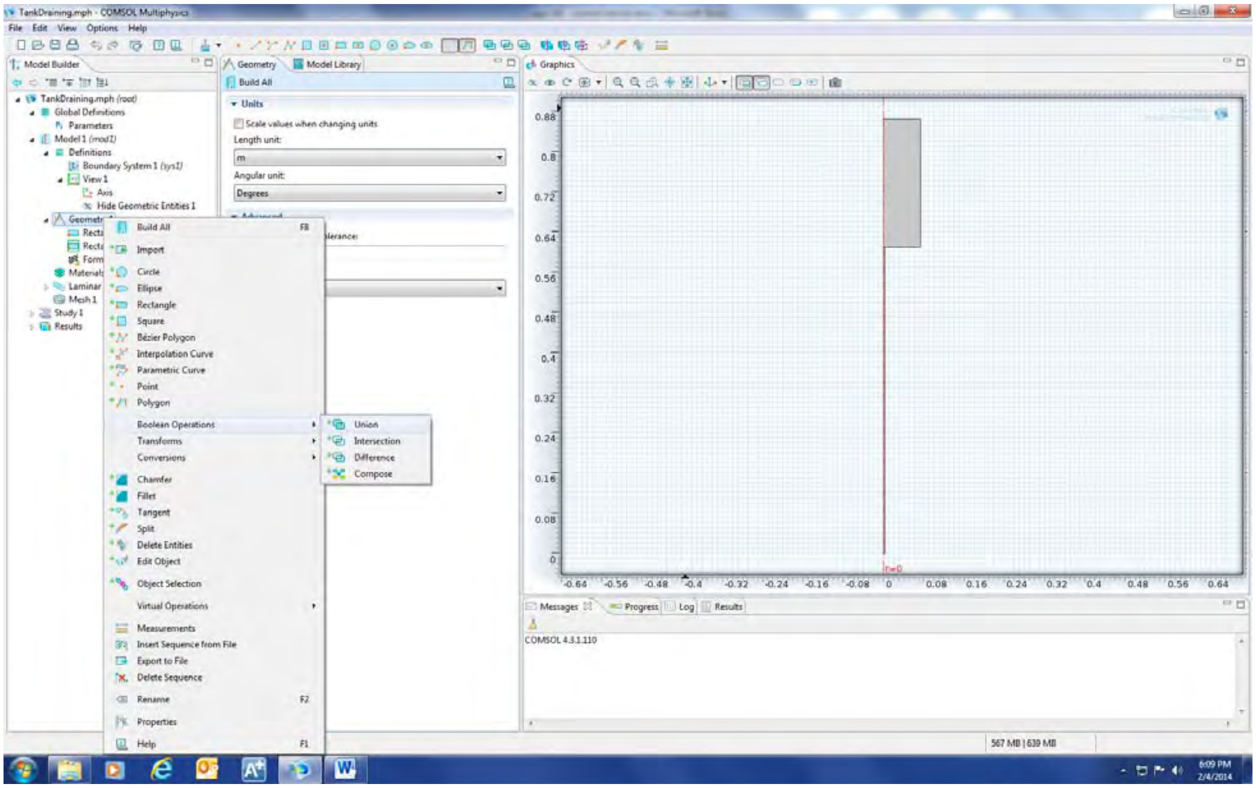
In the position section:
Enter **0** in the box for r and **0** in the box for z.

Click on Build Selected.
Then click on the icon for Zoom Extents.

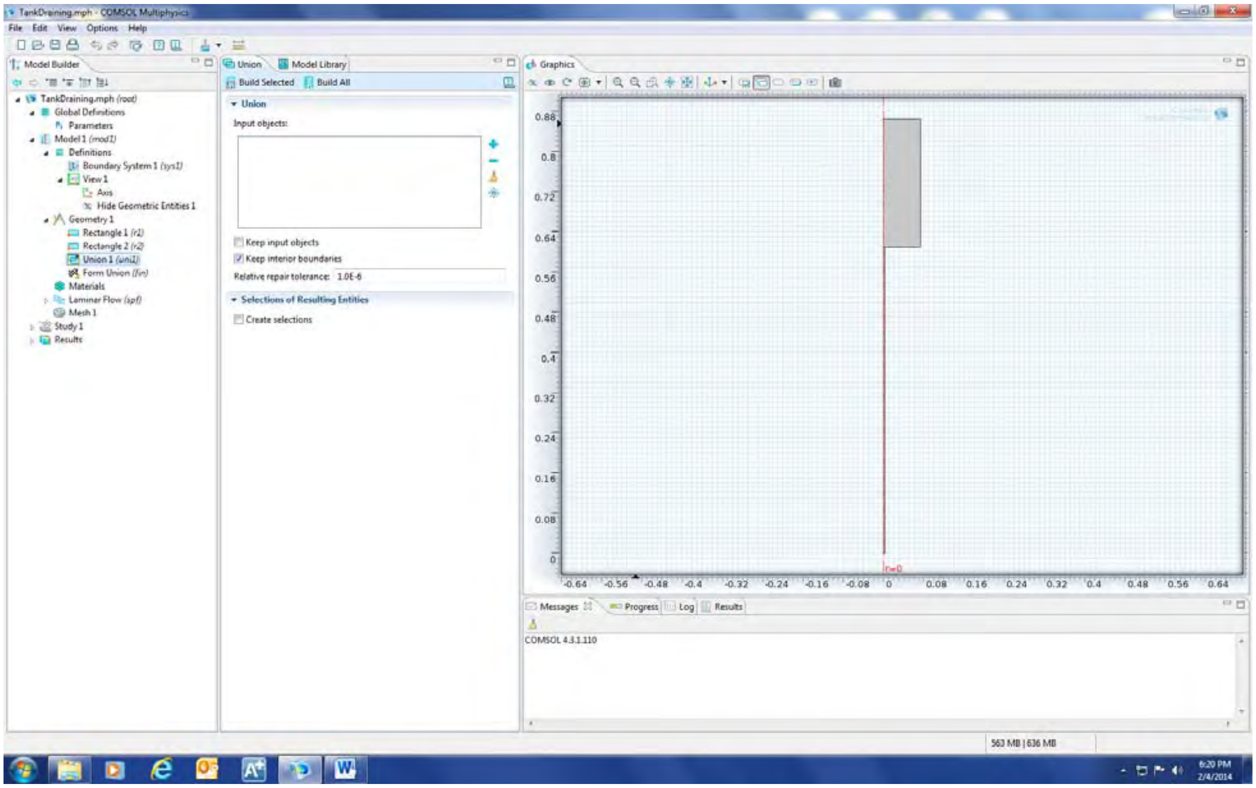


Right click on **Geometry 1**.

On the pop-up menu, click on **Boolean Operations**.



Click on **Union**.

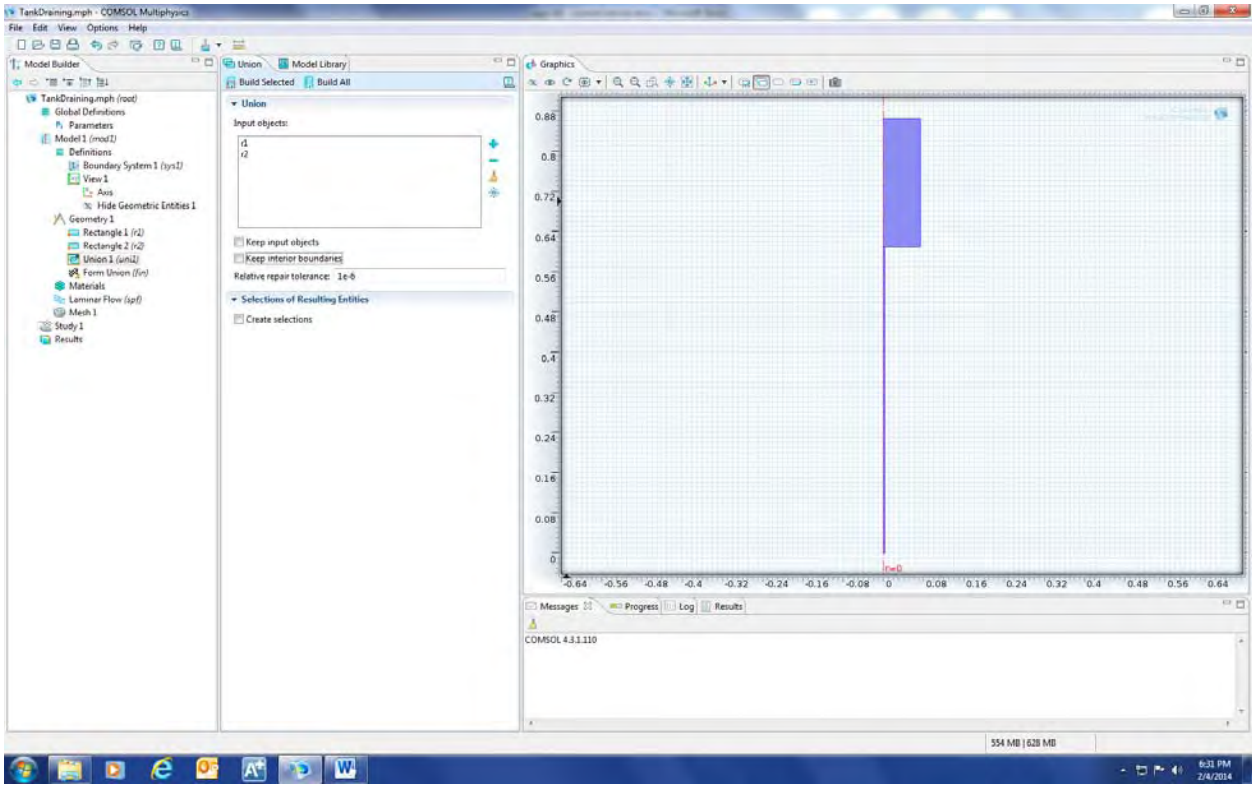


In the Graphics window, click on Rectangle 1.
Its color changes from gray to red.

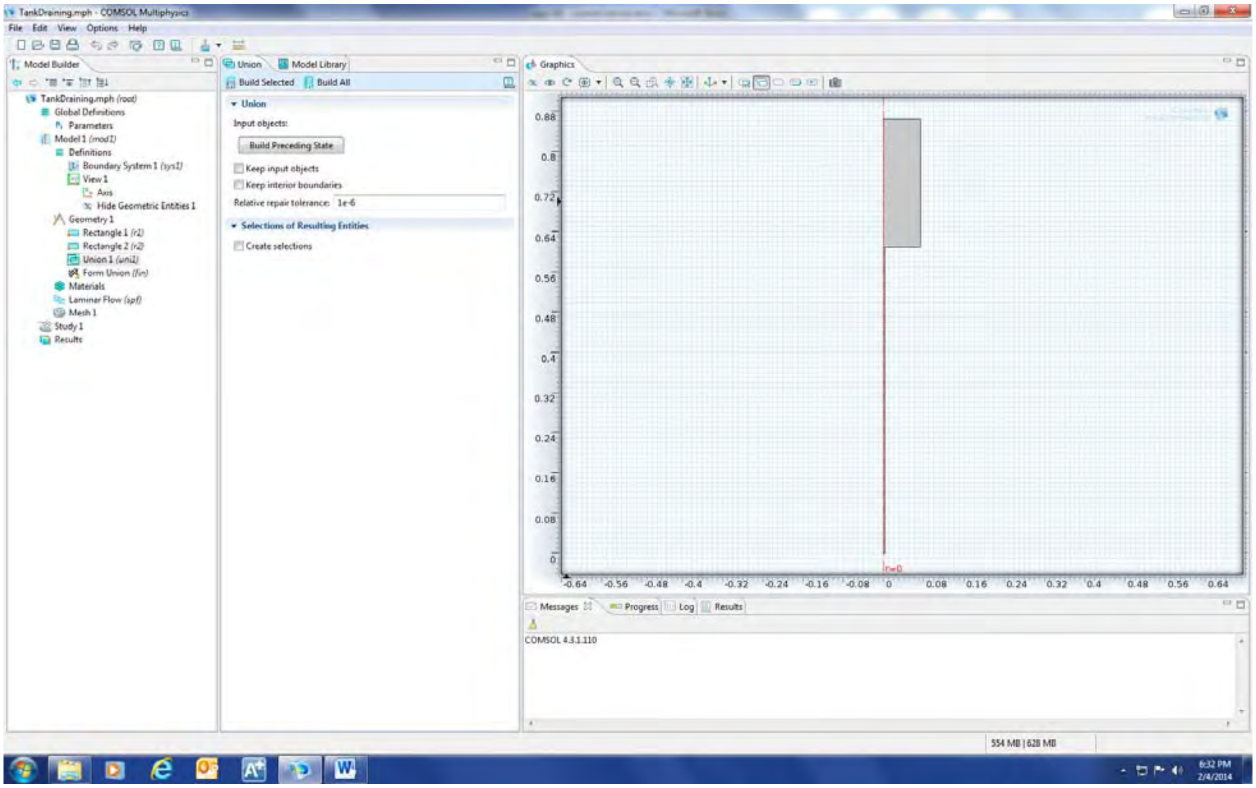
Then click on the tool adjacent to the box for Input objects.
r1 is added to the list of Input objects.
The color of Rectangle 1 in the Graphics window changes to violet.

In the same manner, and Rectangle 2 to the Input objects list.

Clear the checkbox for Keep interior boundaries.

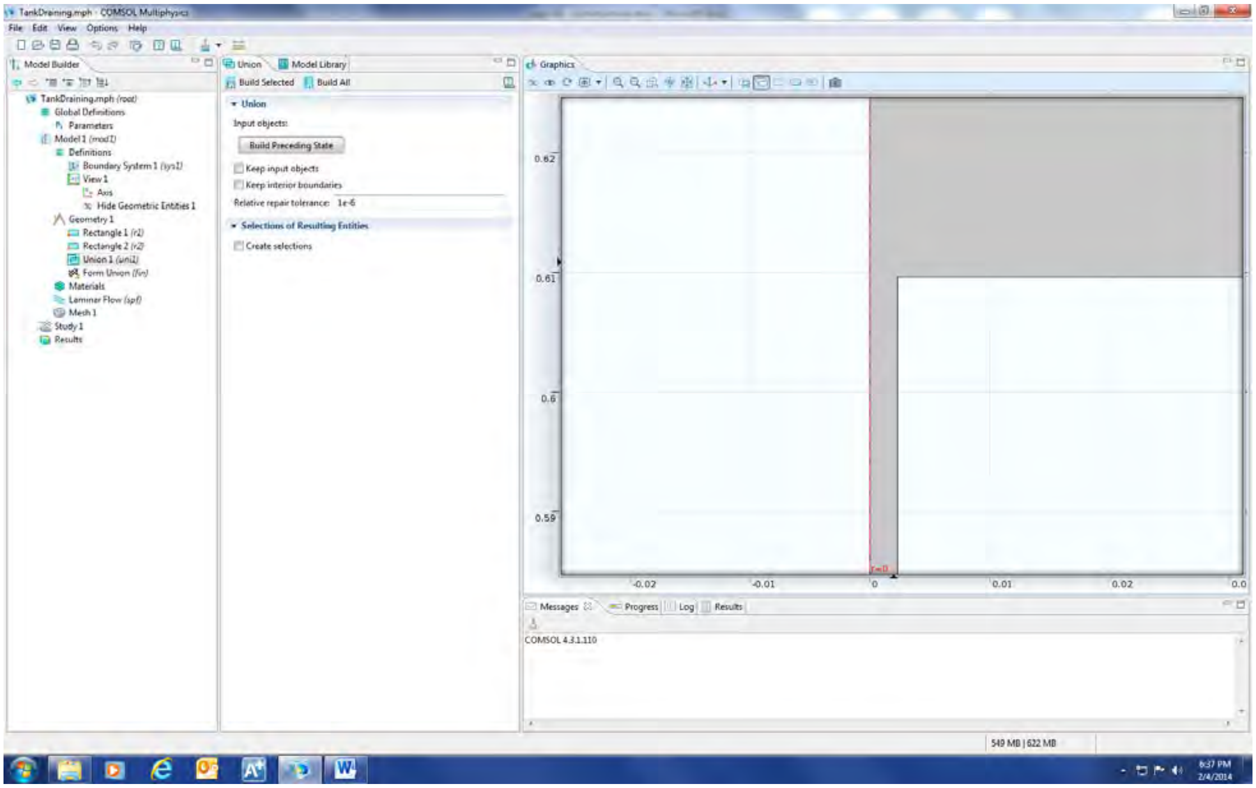


Click **Build Selected**.



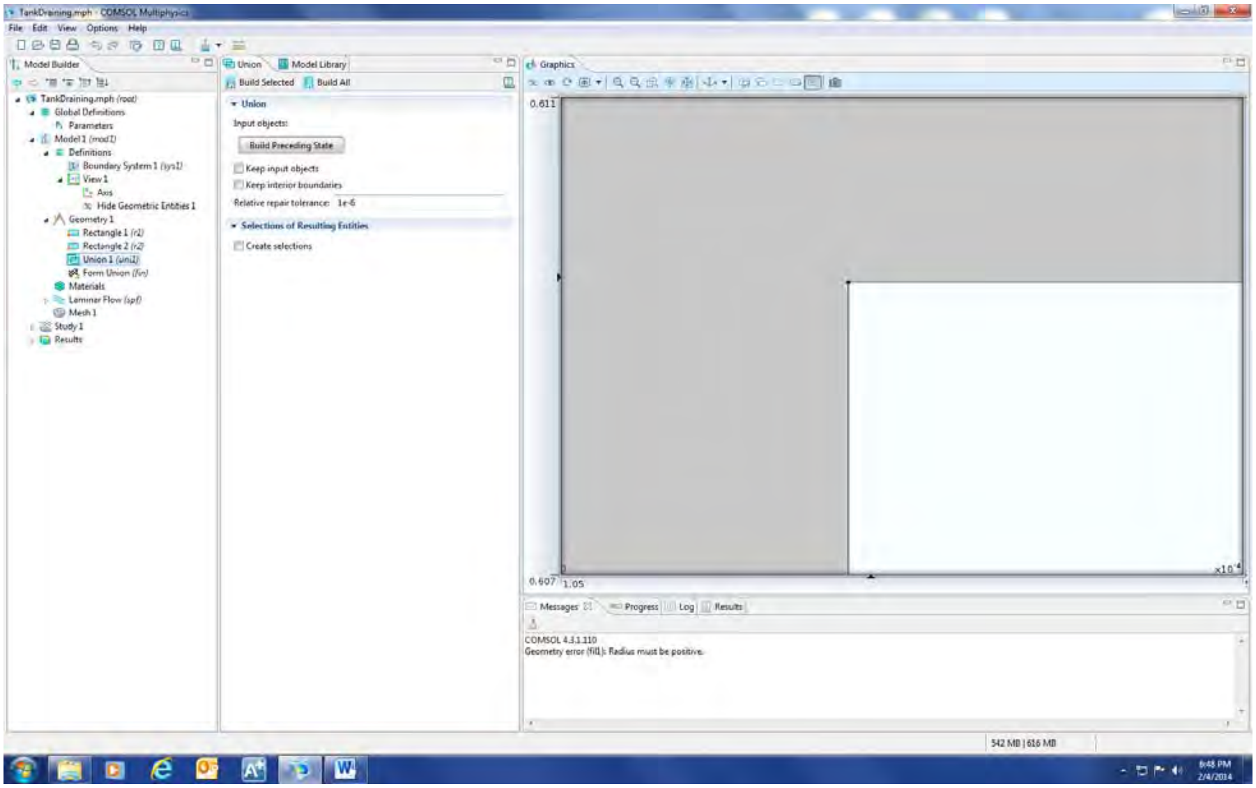
Click on the icon at the top of the Graphics window for Zoom Box.

Draw the box around the pipe entrance to the tank.



Notice that there is no boundary present in the connection between the pipe and tank. That is, the gray shading is continuous throughout the connection.

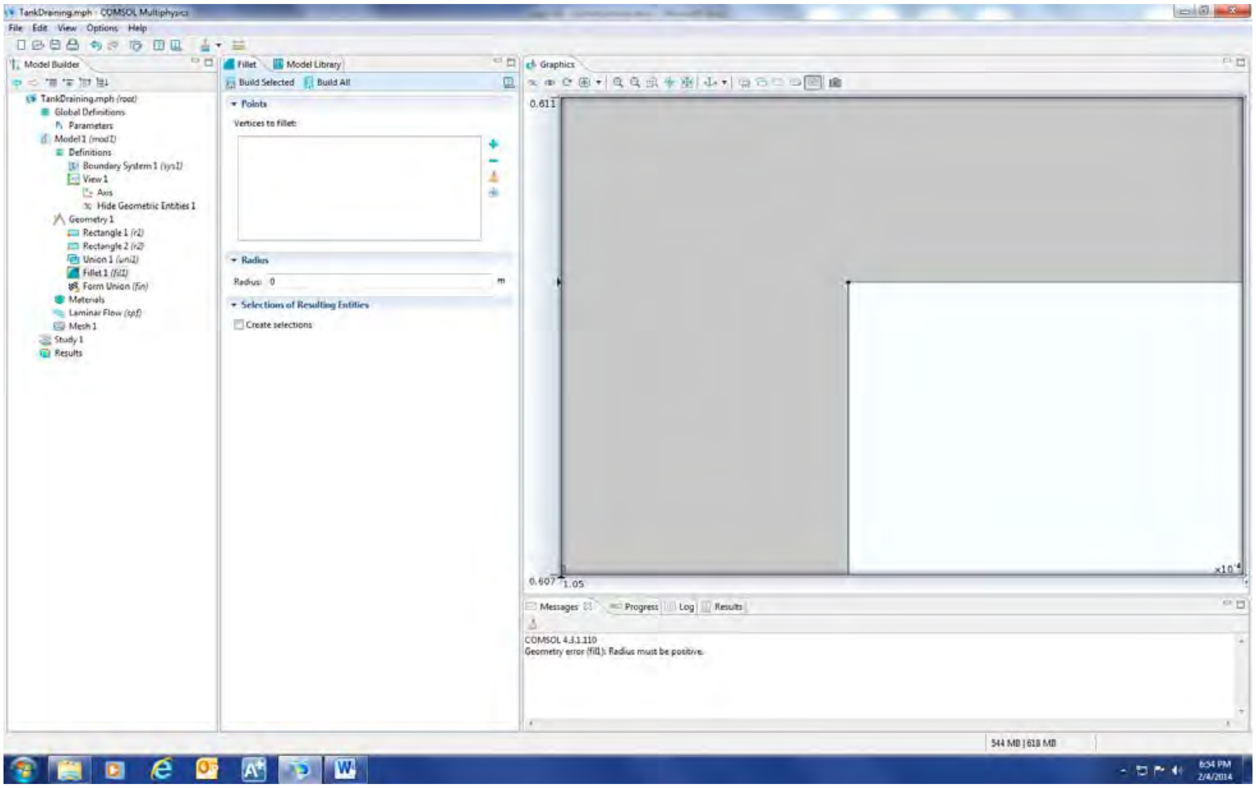
Use the **Zoom Box** to further magnify the corner of the connection between the pipe and the tank.



Notice that there is a sharp corner at the connection of the pipe to the tank.

Right click on Geometry 1.

In the pop-up menu, click on **Fillet**.

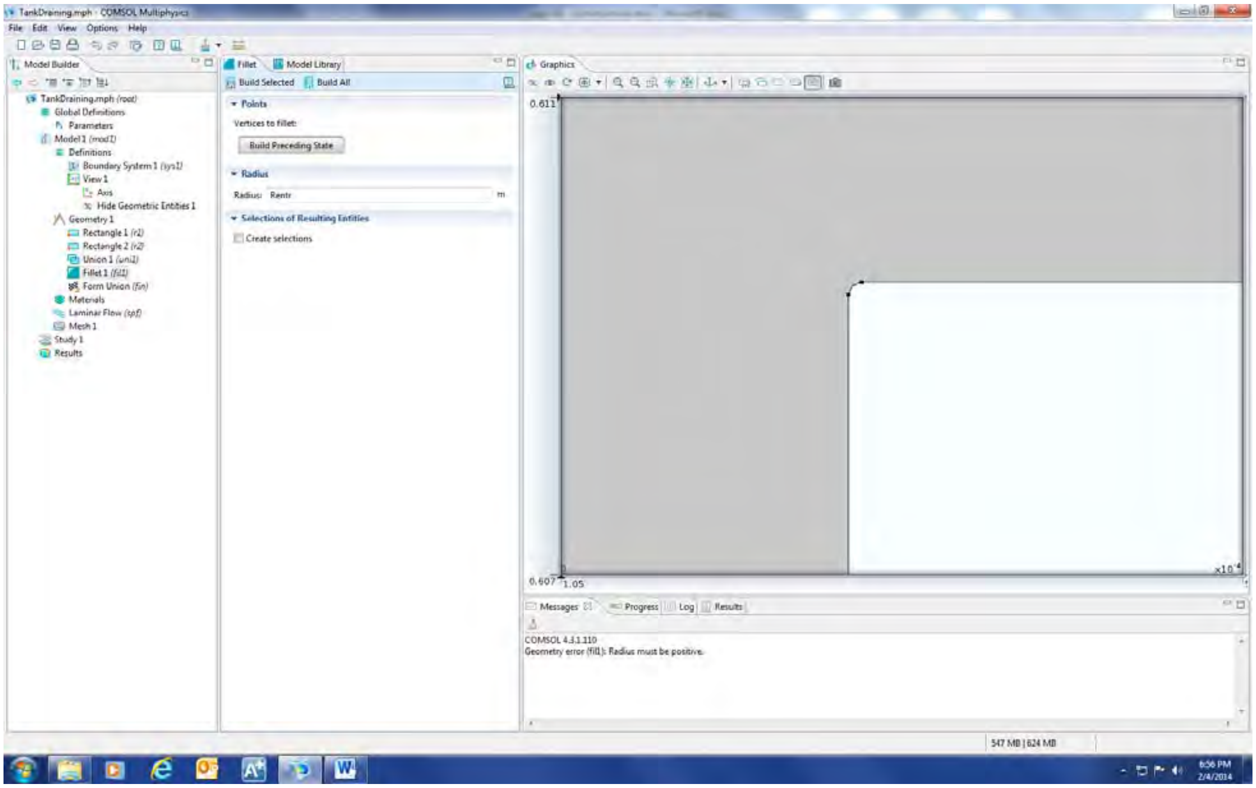


In the Graphics window, click on the vertex at the corner of the connecting of the pipe to the tank. The color of the vertex changes to red.

Then click on the + icon adjacent to the box for Vertices to fillet.
The color of the vertex changes to blue.

In the box for Radius, enter **Rentr**.

Click on Build Selected.

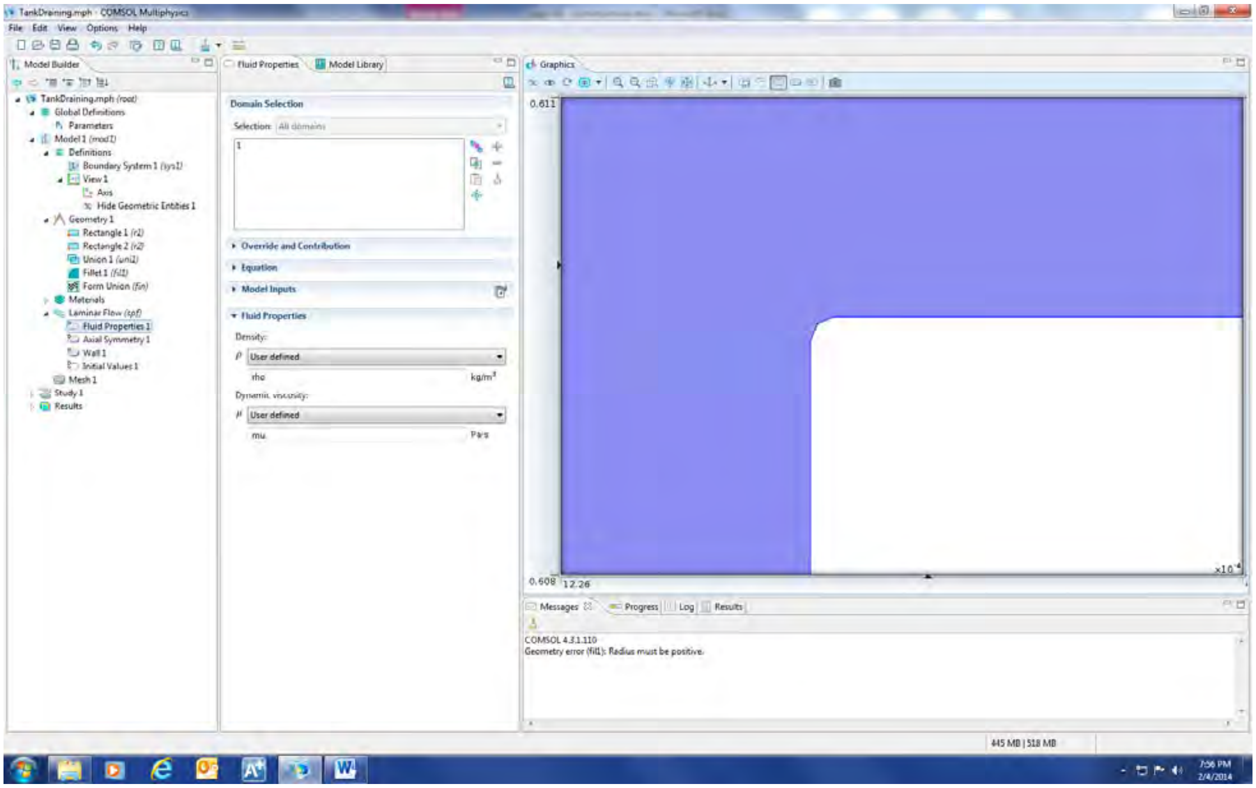


The corner connection is now rounded, as specified by Rentr.

Under Geometry 1, click on **Form Union**.
In the Finalize window, click on **Build All**.

The model geometry is now complete.

We proceed to specify the branch for Laminar Flow *(spf)*.
Right click on **Fluid Properties 1**.
In the box for density, enter **rho**.
In the box for viscosity, enter **mu**.

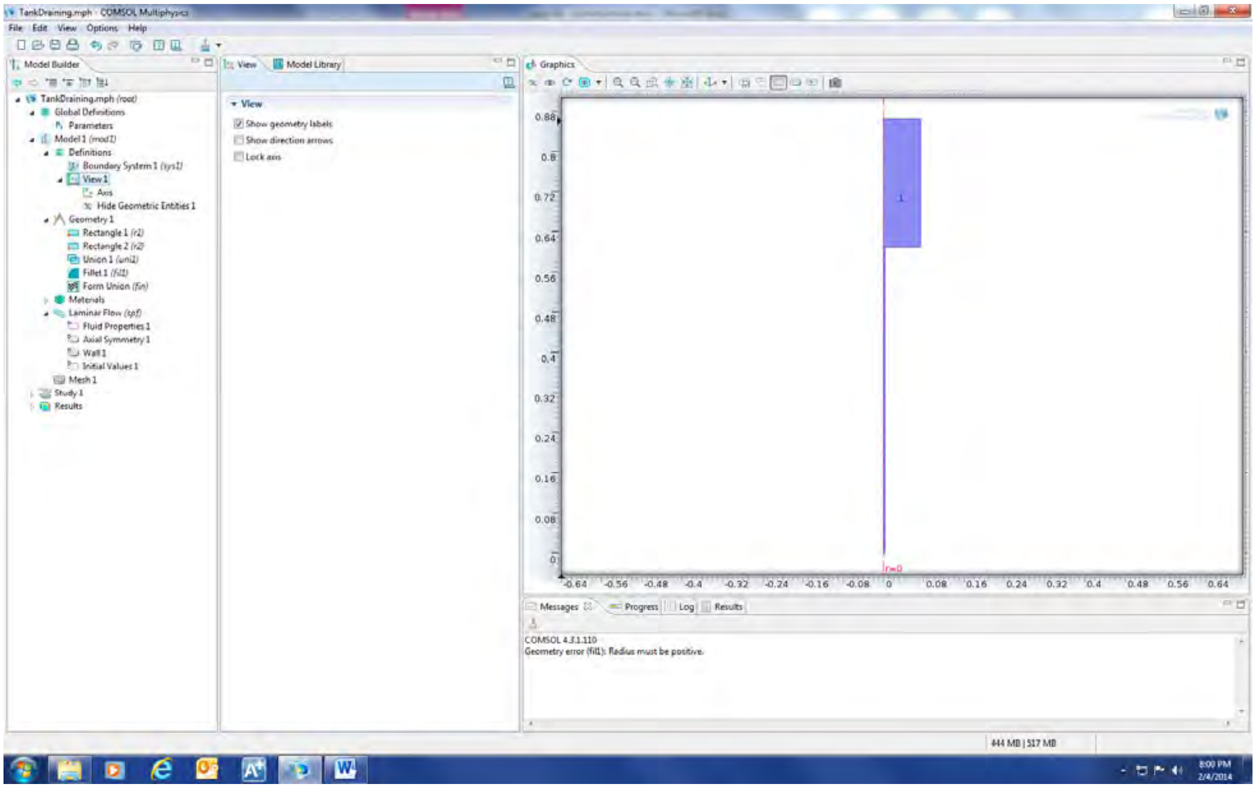


Before examining the boundary conditions, we enhance the view in the Graphics window.

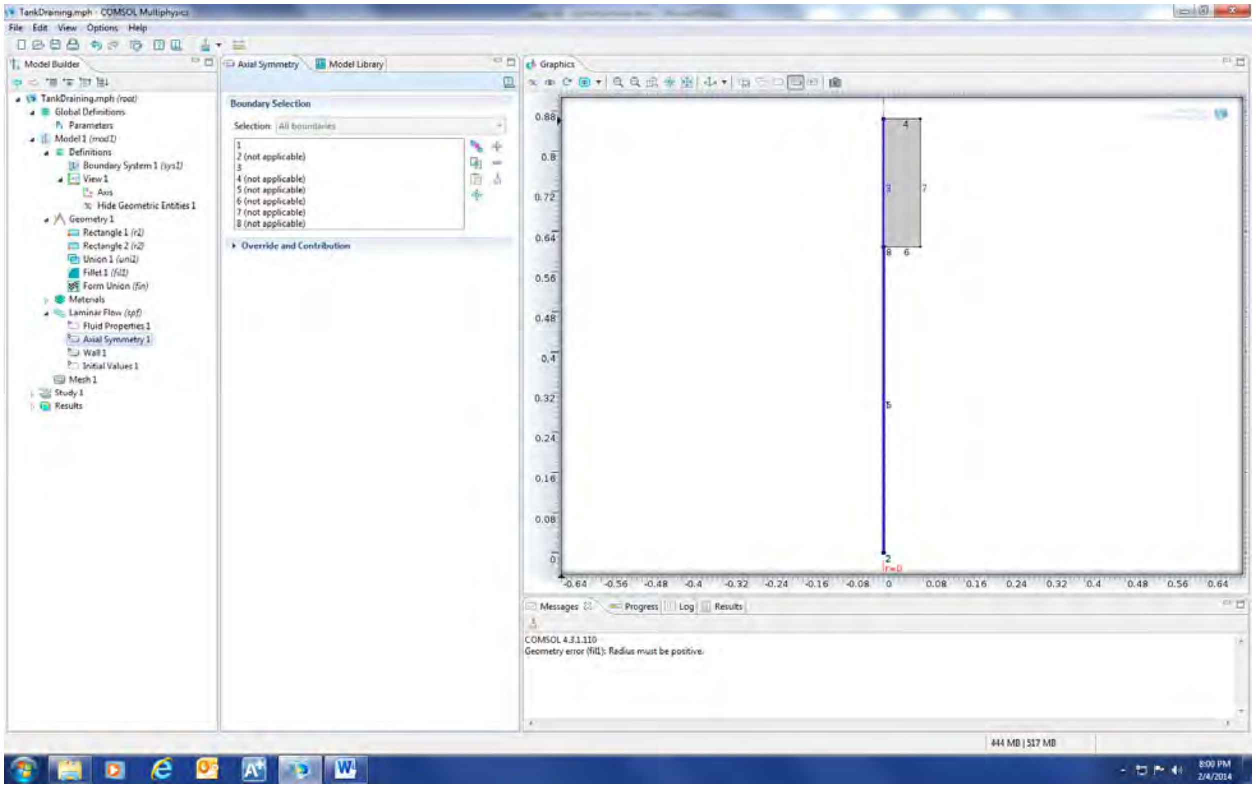
Click on Zoom Extents.

Then go to Model 1 > Definitions > View 1.

Mark the checkbox for Show geometry labels.



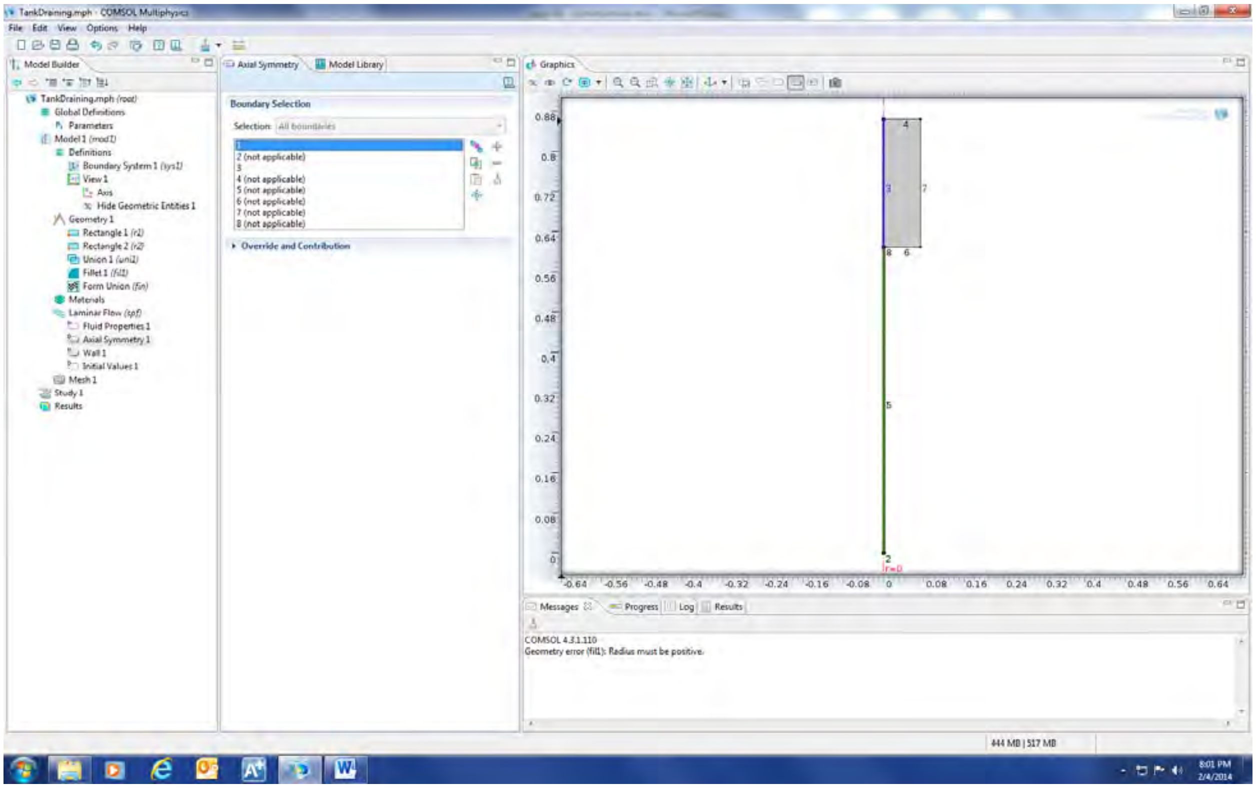
Go to Laminar Flow > Axial Symmetry 1.



The blue color highlights the axis of symmetry, which is the centerline of the pipe and the tank.

Using the Zoom Box, one can confirm that the numeral 5 is covering the numeral 1.
Moreover, numeral 1 refers to the centerline of the pipe, whereas numeral 5 refers to the wall of the pipe.

Click on **1** in the Selection list in the Boundary Selection window.

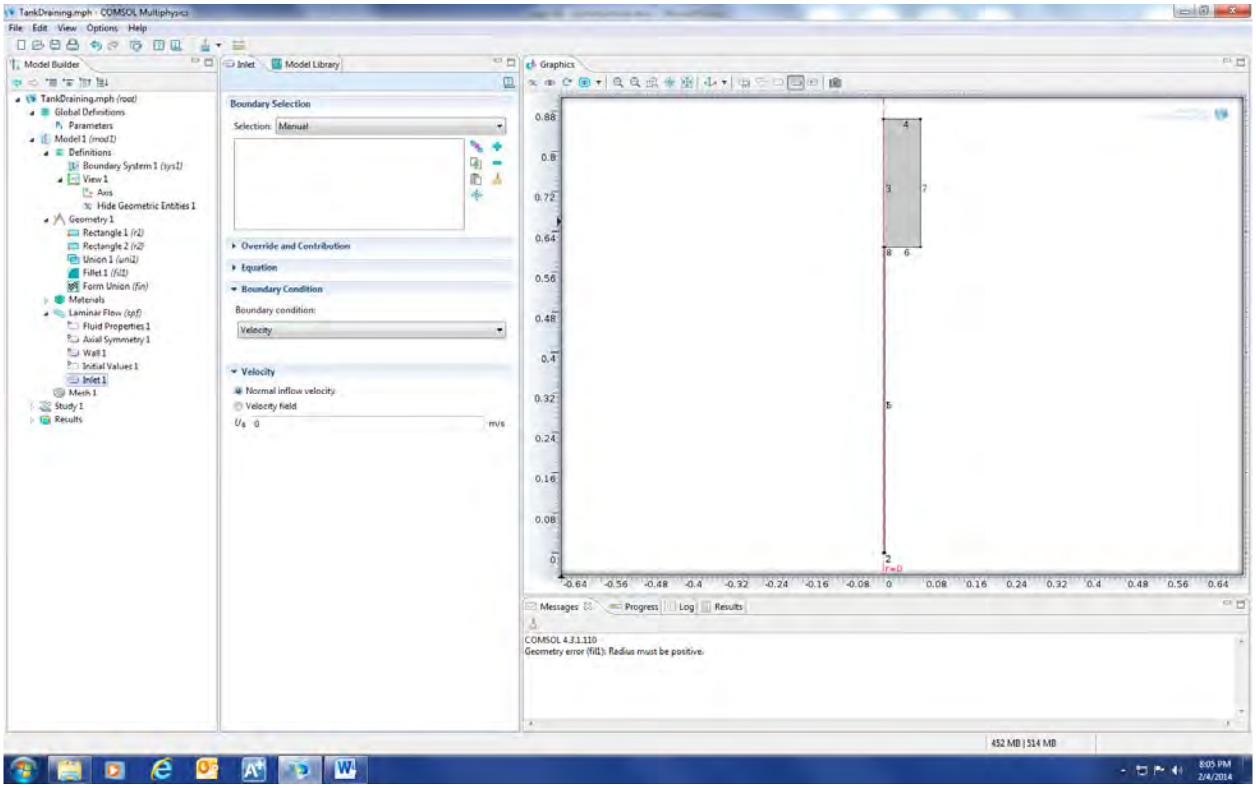


The green line is on the centerline of the pipe, as can be verified using the Zoom Box tool or the Zoom In tool.

Click on the other numerals in the list in the Selection box. The centerline of the tank appears in green (as in GO at a traffic light), but all others appear in red (as in STOP at a traffic light).

Right click on Laminar Flow *(spf)*.

In the pop-up menu, click on Inlet.

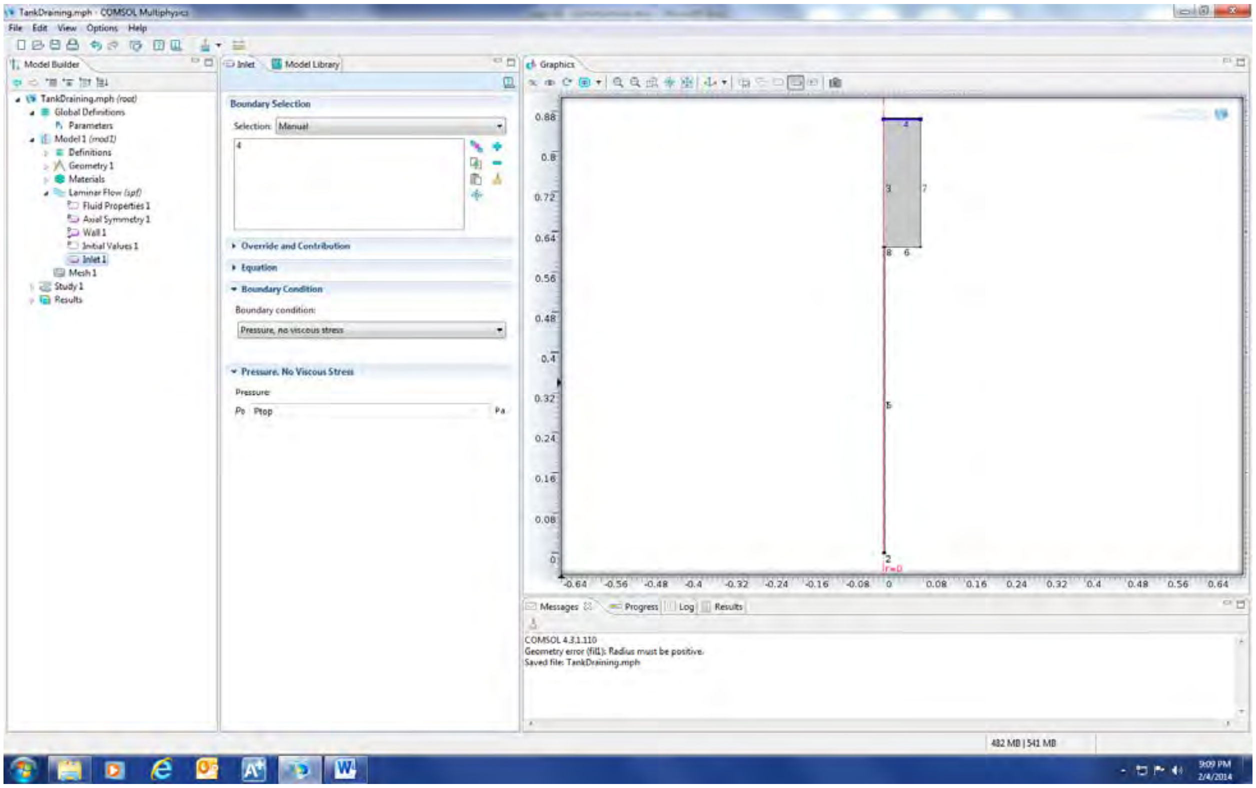


In the Graphics window, click on the **top boundary**, labeled 4.
The color of the boundary becomes red.

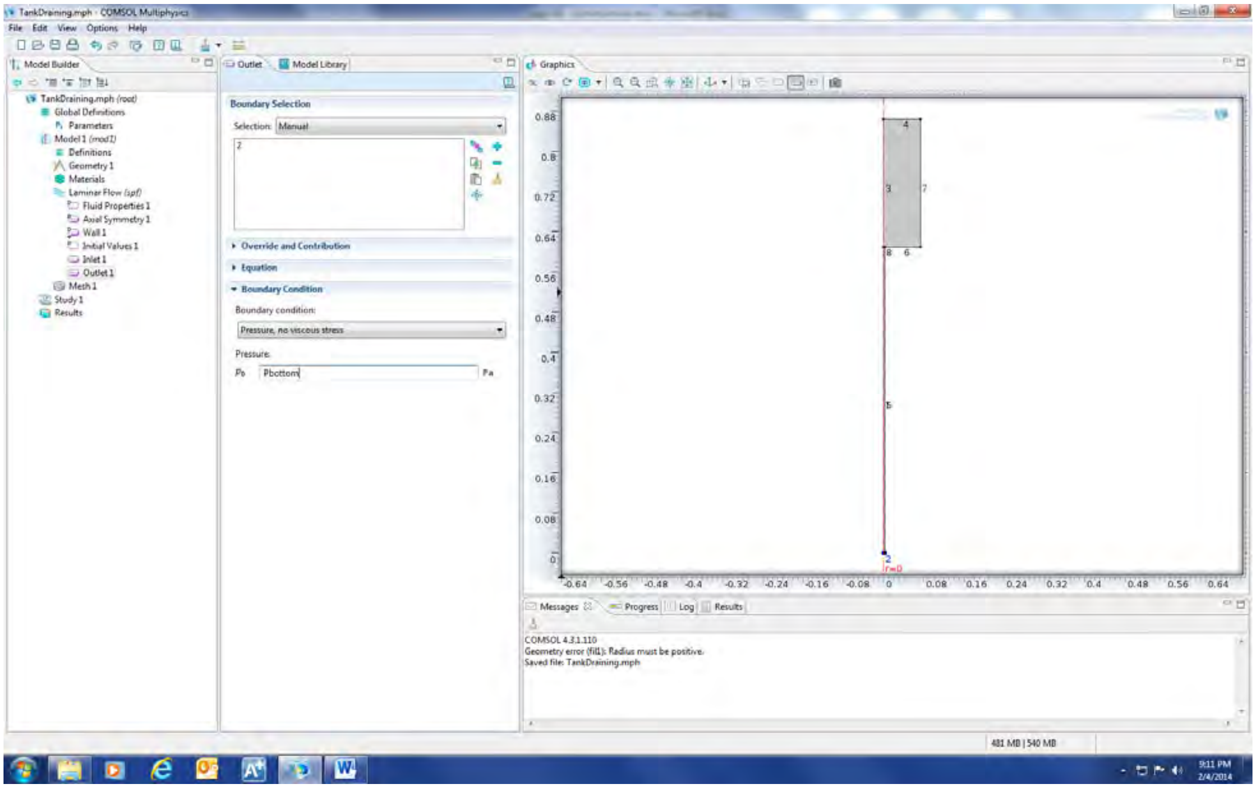
Click the + icon adjacent the box for Selection.
The numeral 4 is added to the Selection box.
The color of the boundary becomes blue.

Under boundary condition, select **Pressure, no viscous stress**.

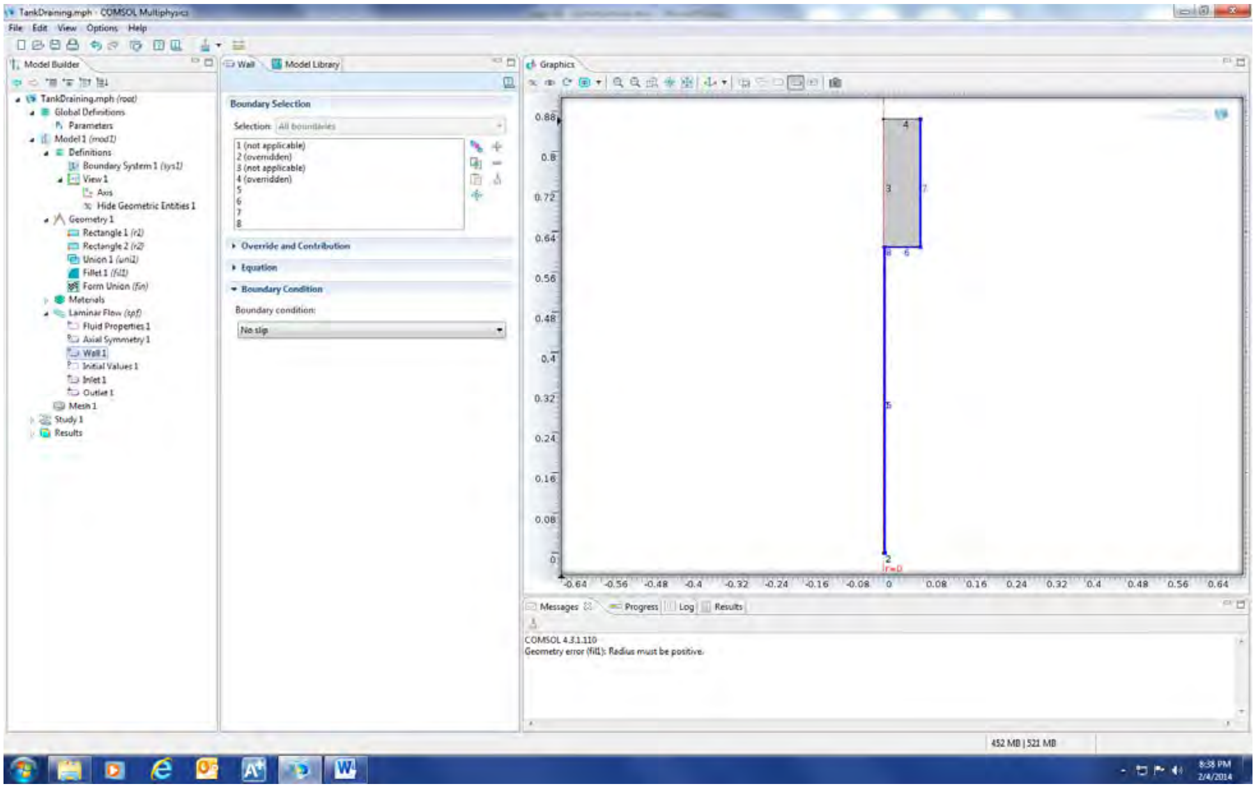
In the box for p_0, enter **Ptop**.



In a similar manner, add an outlet to the model for the bottom boundary labeled 2.



Click on Wall 1.



The remaining boundary (labeled 5, 6, 7, 8) is displayed in blue color and is listed in the boundary Selection box.

At the bottom, we notice that the boundary condition is set at the default No slip.

The next step is to add gravitation force to the model.

Right click on Laminar Flow *(spf)*.

Then click on Volume Force in the pop-up menu.

In the Graphics window, click on the **domain** (that is, the region of the tank and the pipe), labeled 1.

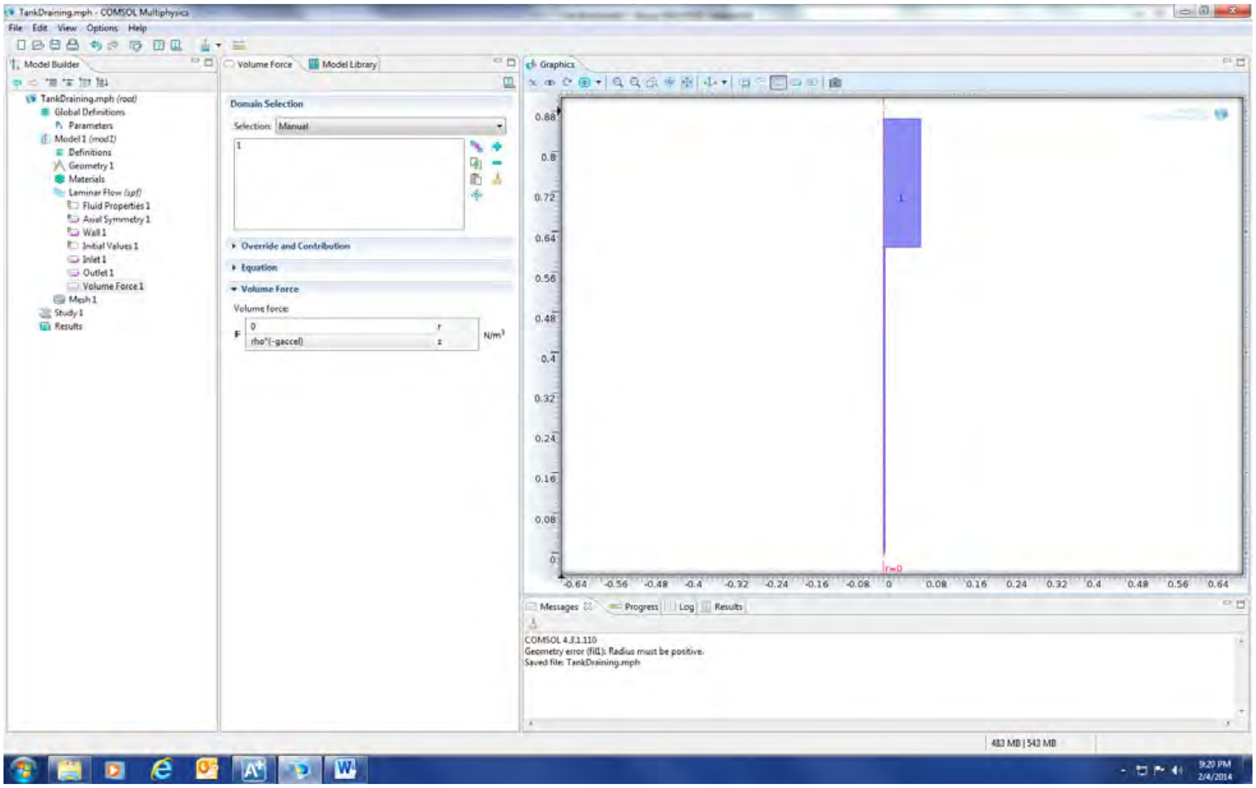
The color of the domain becomes orange-red.

Click the icon adjacent the box for Selection.

The numeral 1 is added to the Selection box.

The color of the domain becomes violet.

Under box for Volume force, enter **rho*(–gaccel)** for the force in the z direction on the volume in the tank and pipe.



Click on **Initial Values 1** under Laminar Flow *(spf)*.

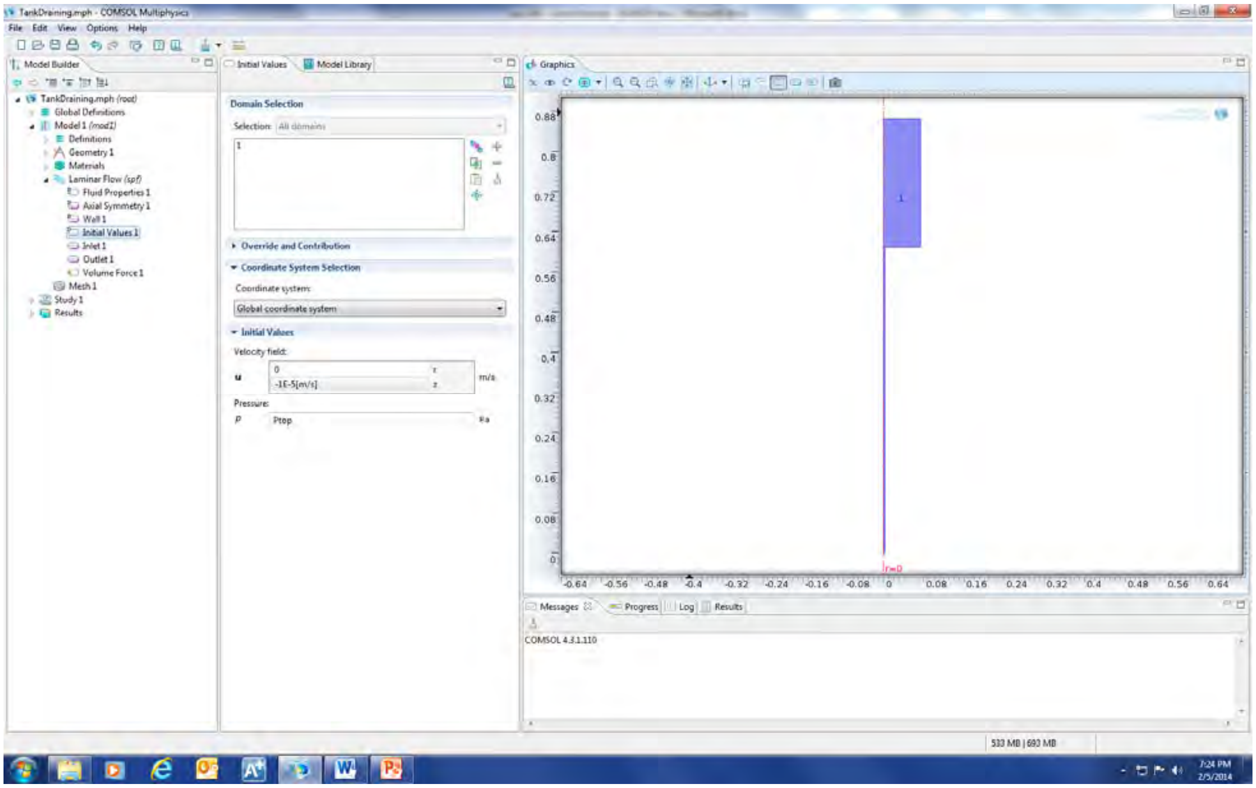
In the Graphics window, click on the tank or pipe to select it.

Then on the ✚ icon next to the Domain Selection box to add domain 1 to it.

In the box for section for velocity field, enter **0** m/s for the initial velocity in the r direction.

Enter **−1E-5** m/s for the initial velocity in the z direction.

In the box for pressure, enter **Ptop**.



Click on Mesh 1.

Select **Physics-controlled mesh** for Sequence type.

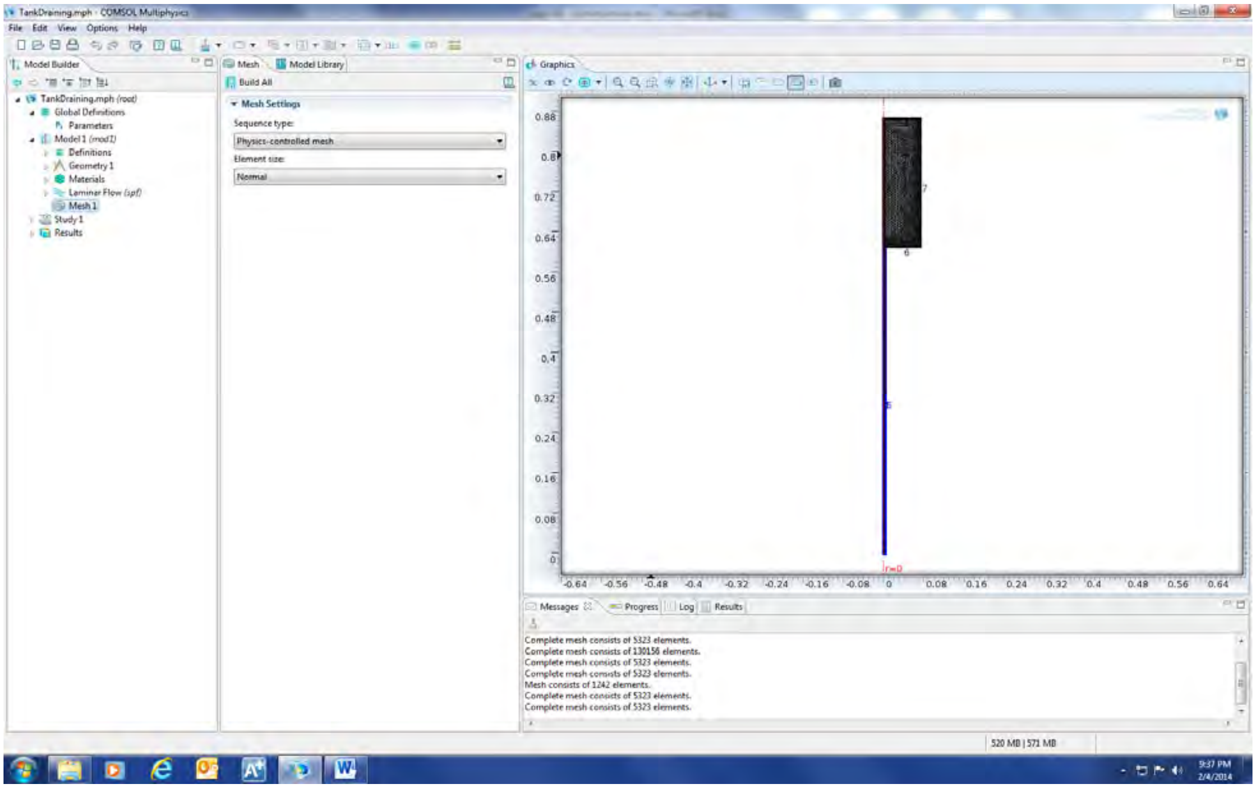
Select **Normal** for Element size.

COMSOL offers nine element sizes for the mesh:

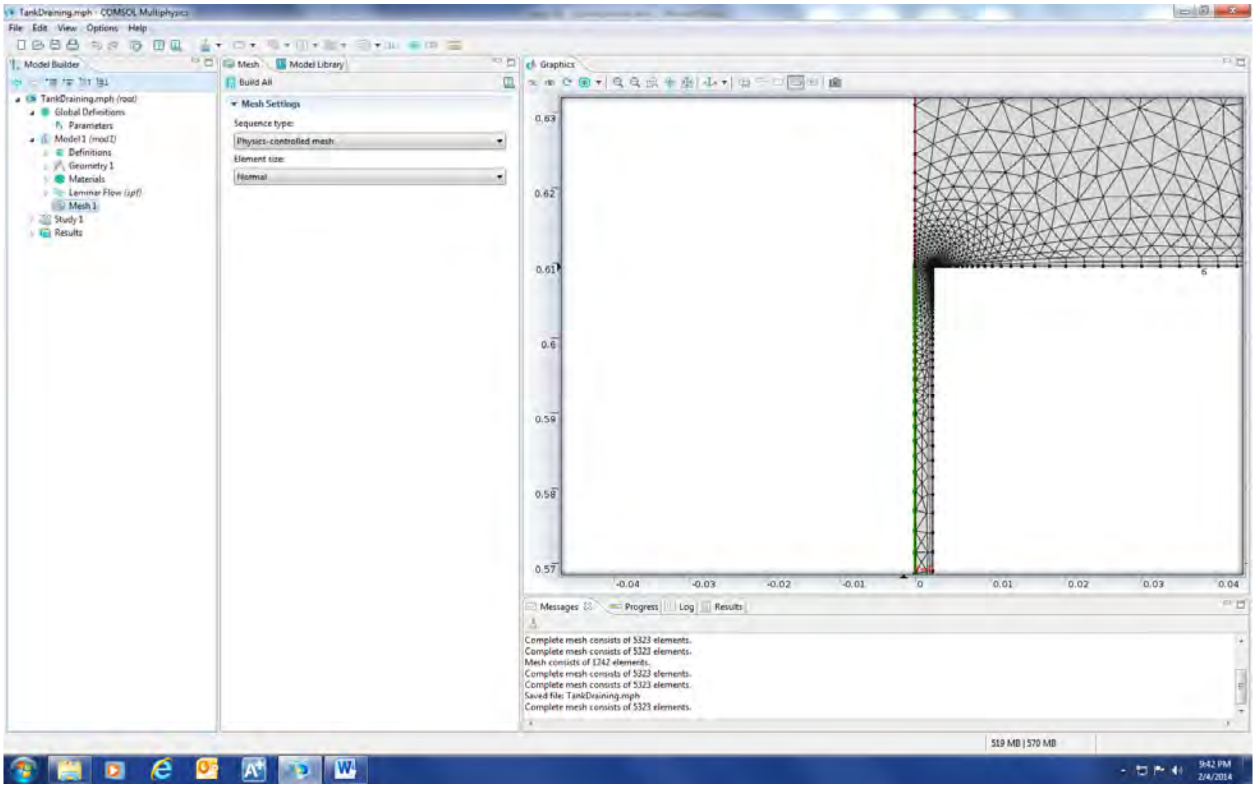
Extremely fine – Extra fine – Finer – Fine – Normal – Coarse – Coarser – Extra coarse – Extremely coarse

A reducing the element size will tend to lead to a more accurate solution, but also tend to increase the time for the computation.

Click on Build All.



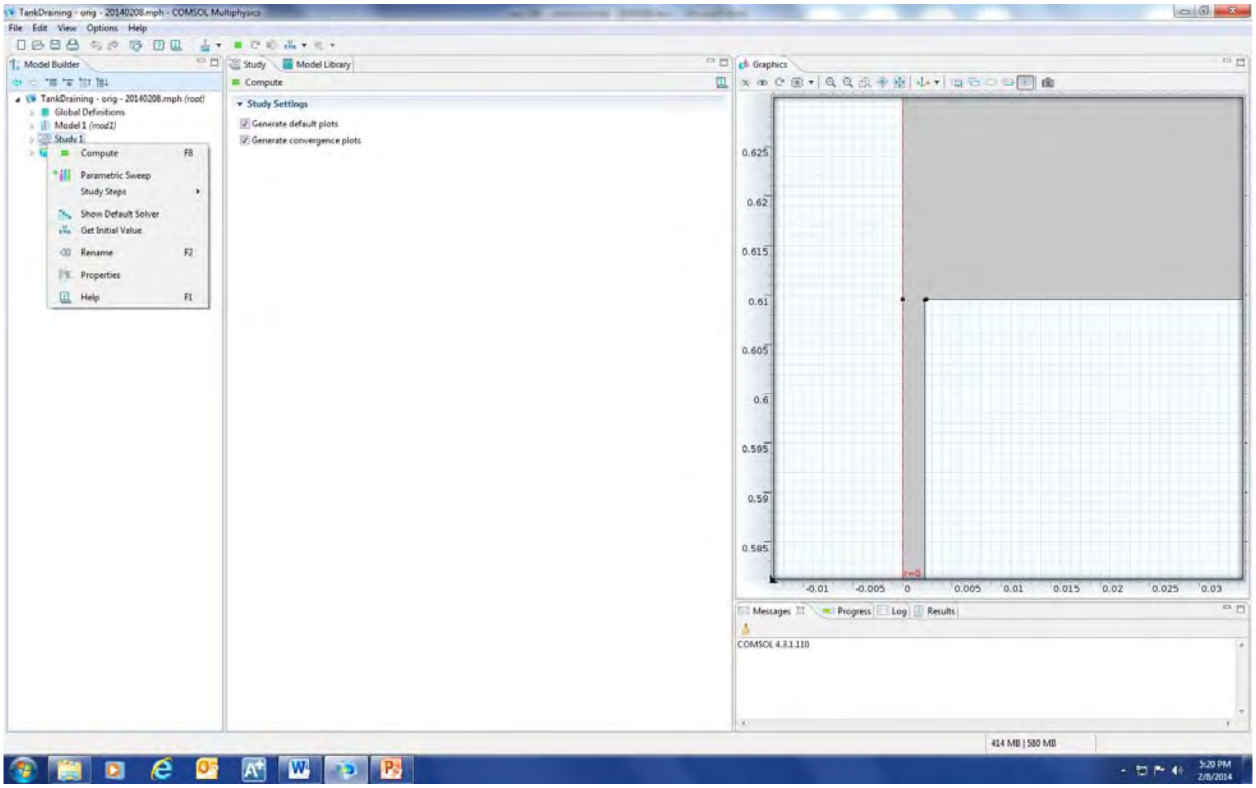
One can use the Zoom Box tool to examine the region near the pipe entrance to the tank.



That completes building the model.

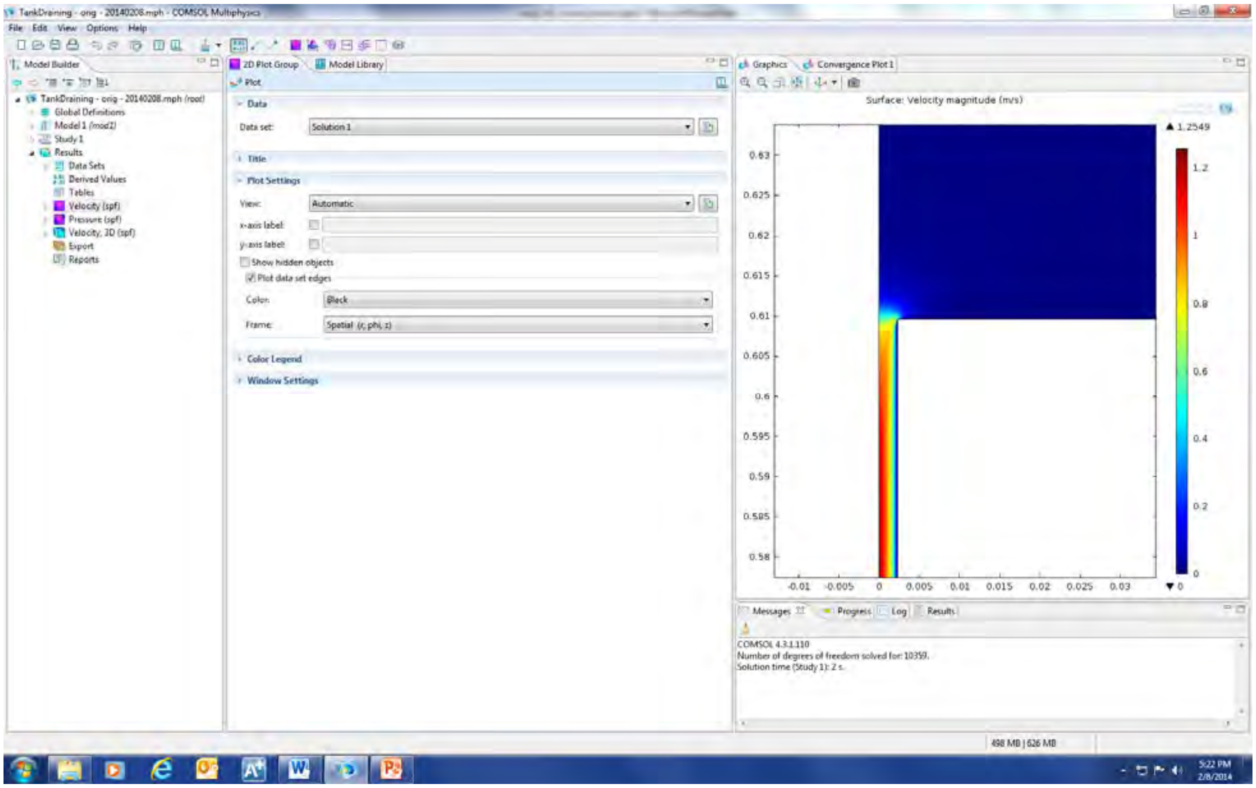
The next step is to set up to perform simulations.

Right click on Study 1.



Click on **Compute**.

And presto!



The x-axis is the radial position r.

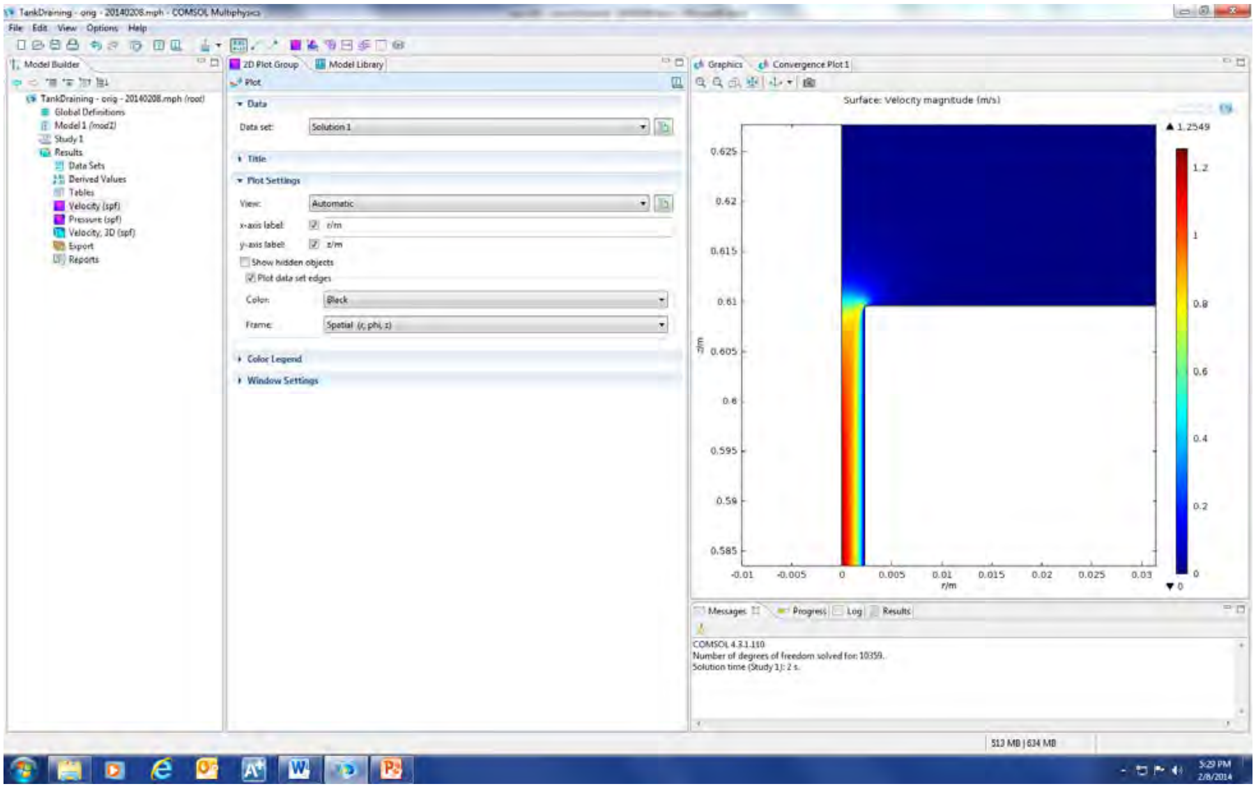
The y-axis is the vertical position z.

Click on **Velocity (spf)**.

In the Plot Settings section:

Mark the checkbox for the x-axis and enter r/m.

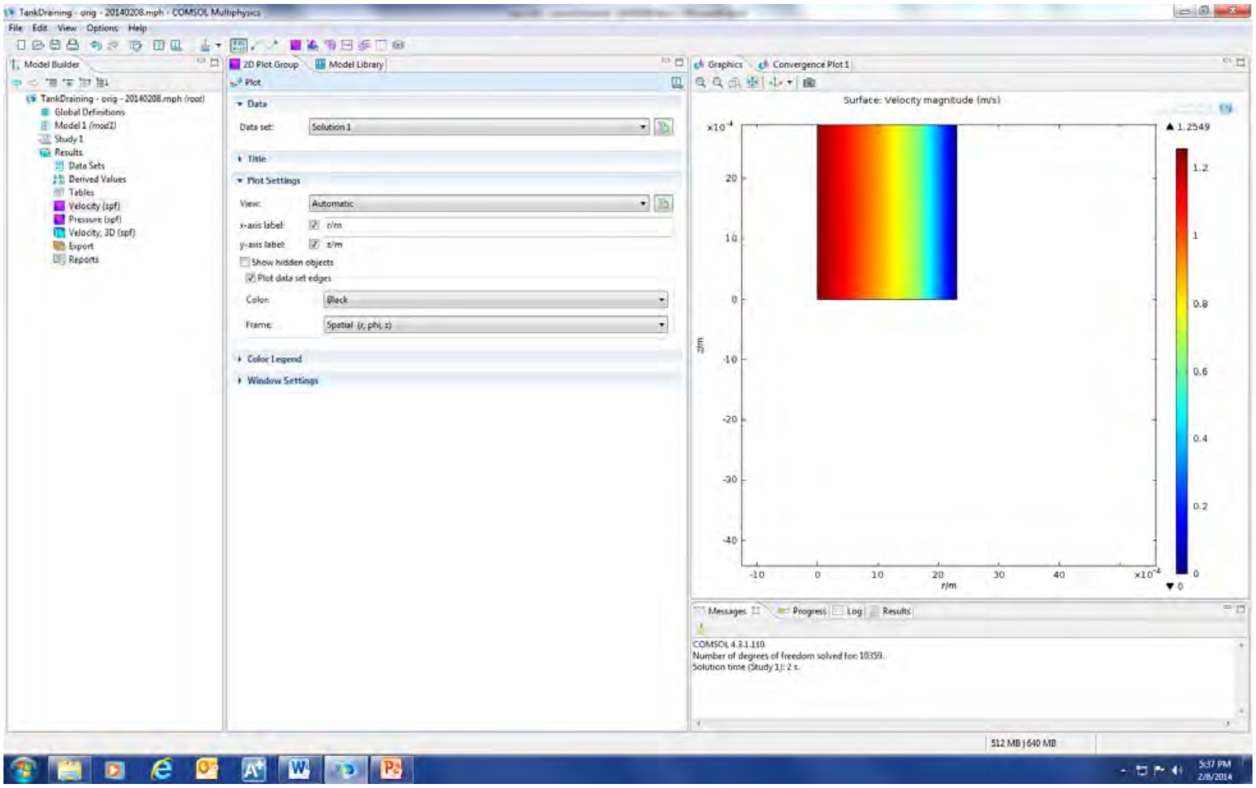
Mark the checkbox for the y-axis and enter z/m.



The color scheme indicates the magnitude of the velocity in the z-direction in units of m/s. The color legend is the vertical bar on the right side of the graphics window.

Above the color legend is the value 1.2549 (in m/s); it is the largest velocity in the system. Beneath the color legend is the value 0 (in m/s); it is the smallest velocity in the system.

Using tools Zoom Extents and Zoom Box, we can display the pipe outlet.

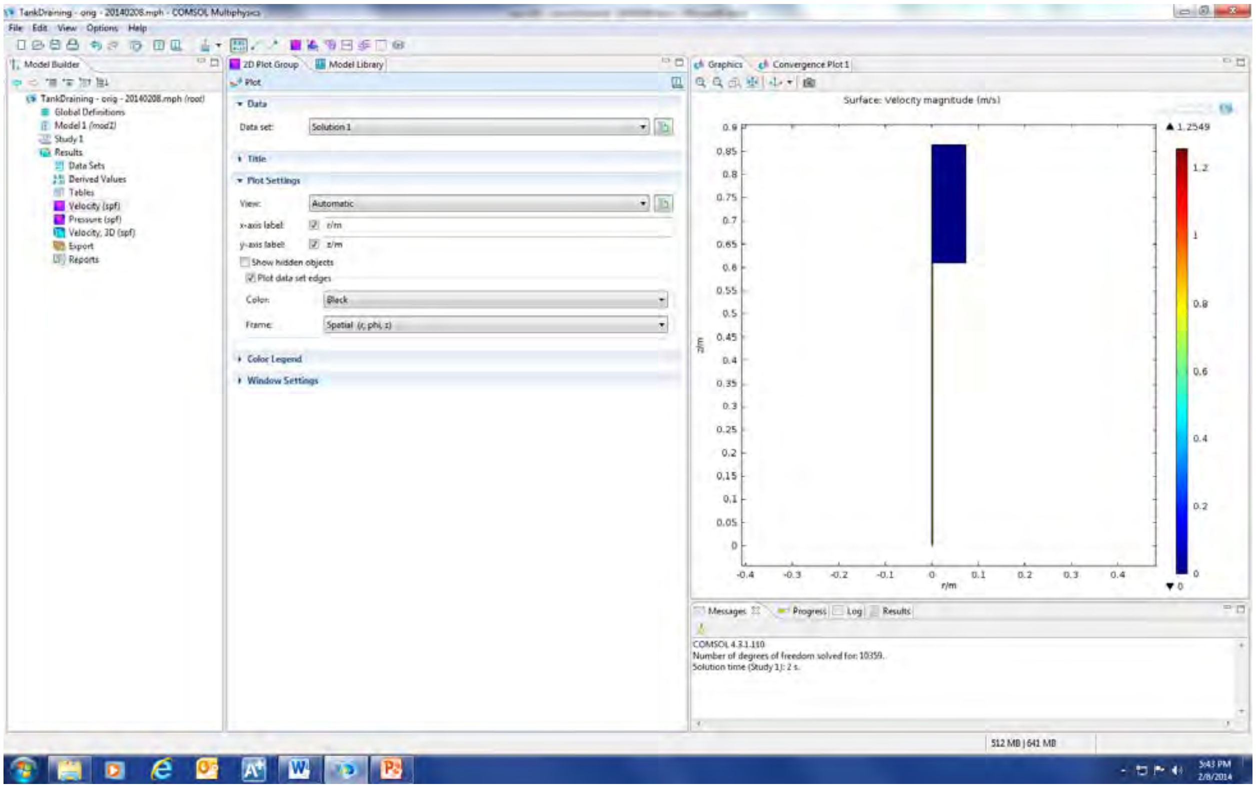


Notice that the axes are now magnified by a factor of 10,000.

The darkest shade of red is at the centerline, as expected from the symmetry.

The darkest shade of blue is at the wall, as expected from the boundary condition for no slip.

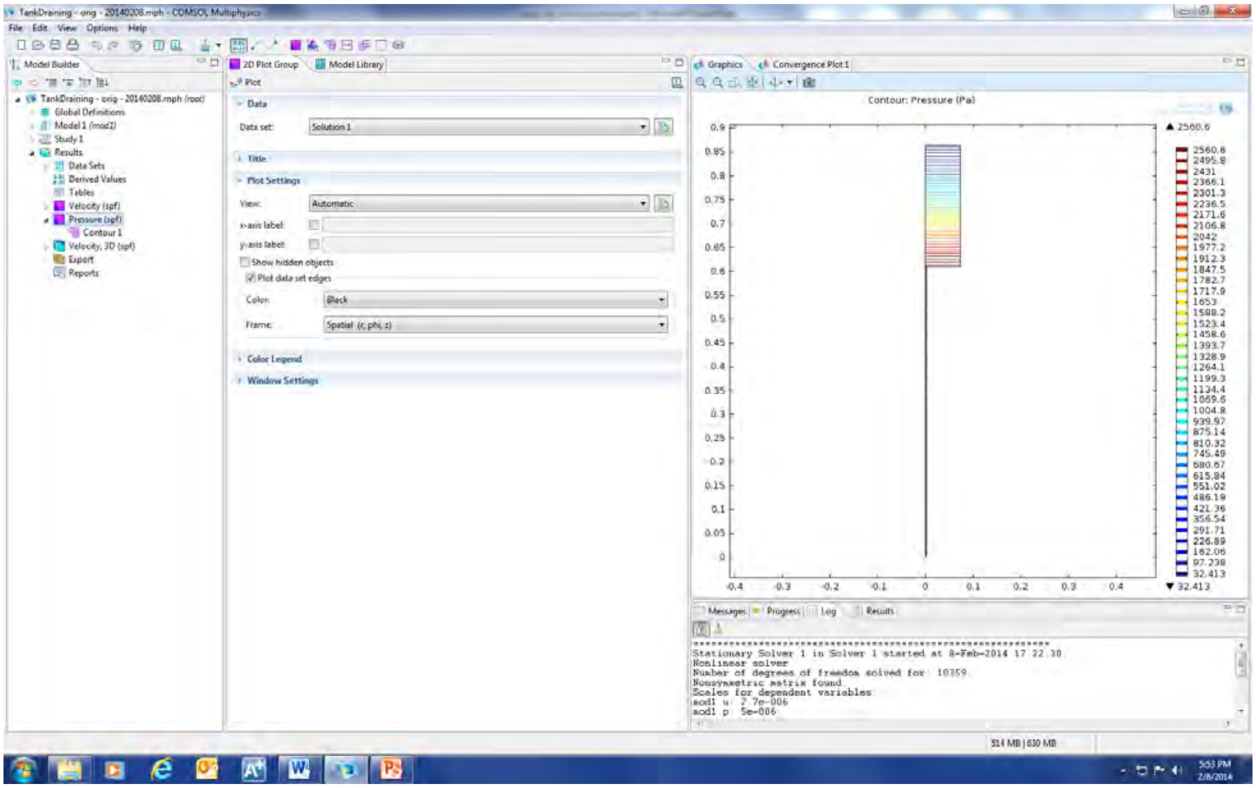
Using the Zoom Extents tool, we can display the entire system.



The tank is dark blue, indicating velocities well below 0.1 m/s.

Beneath the Graphics window, the Messages tab informs us:
Number of degrees of freedom solved for: 10359.
Solution time (Study 1) was only 2 s.

Click on **Pressure (spf)**.

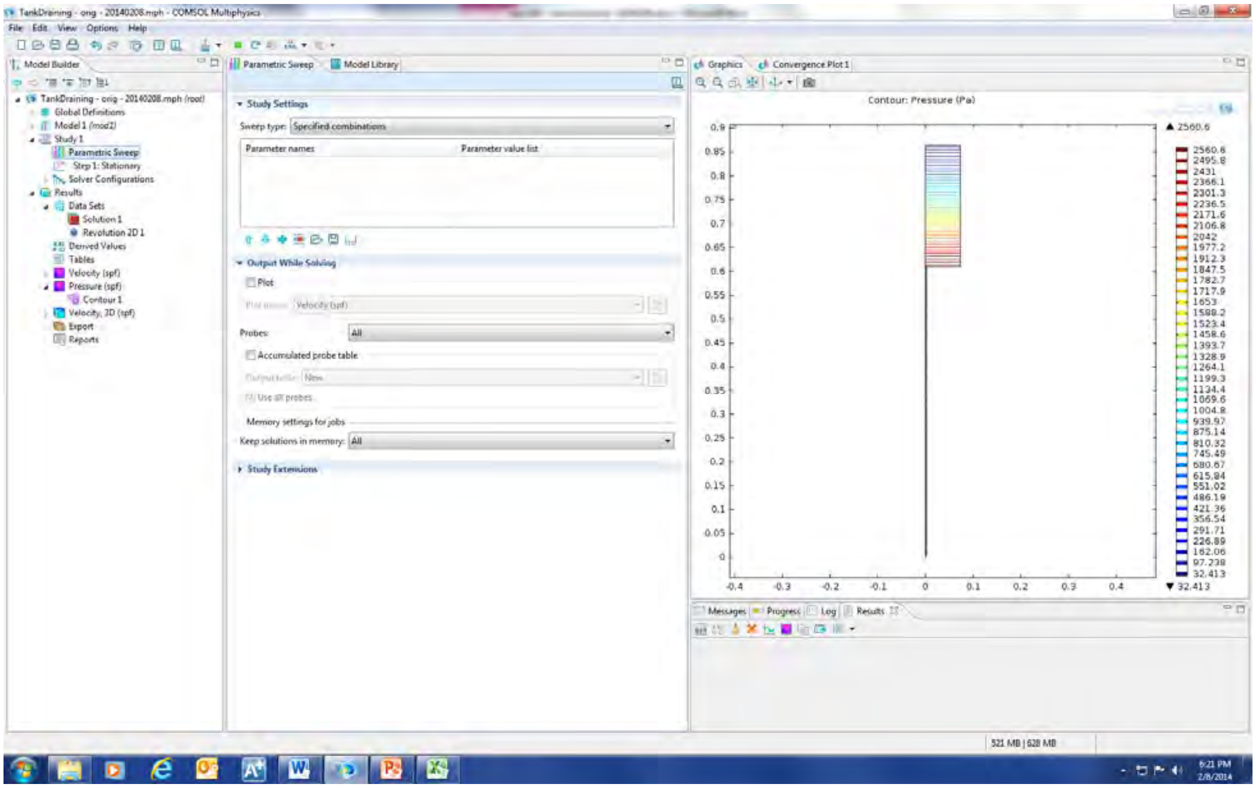


Pressure varies over the range of 32.413 Pag to 2560.6 Pag.

Thus, no vacuum is present anywhere in the system.

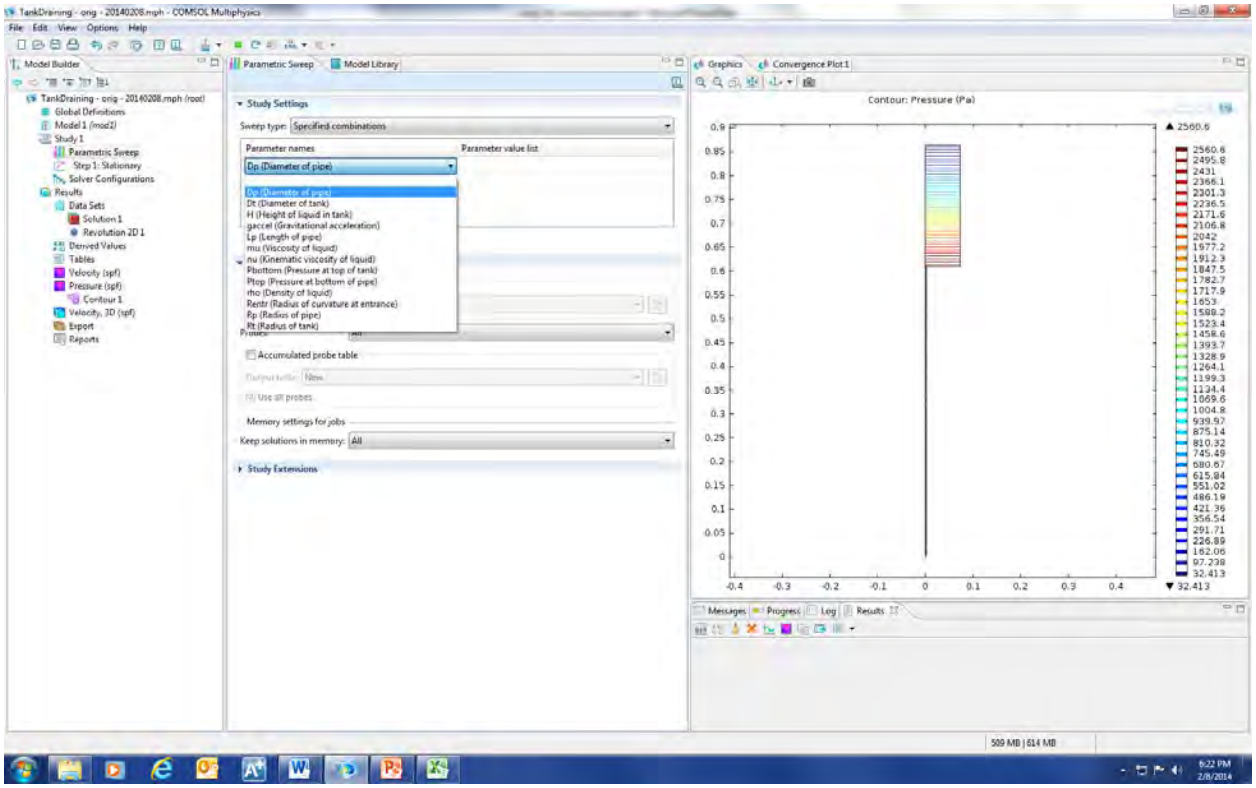
We now set up to evaluate the flow at other values of the liquid height in the tank.

Right click on Study 1 and then on the pop-up menu click on Parametric Sweep.



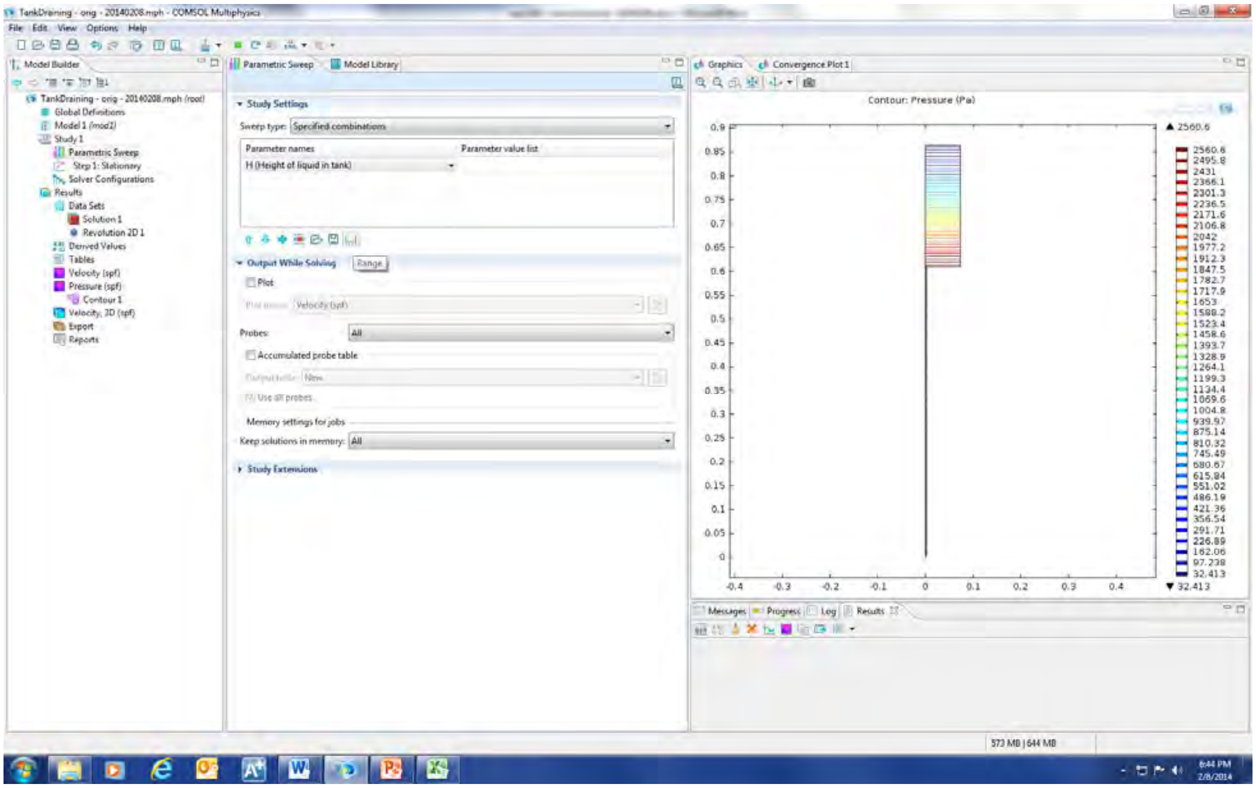
Click the **Add** button (blue plus sign ✚ **)** located beneath the table.

Expand the dropdown menu.



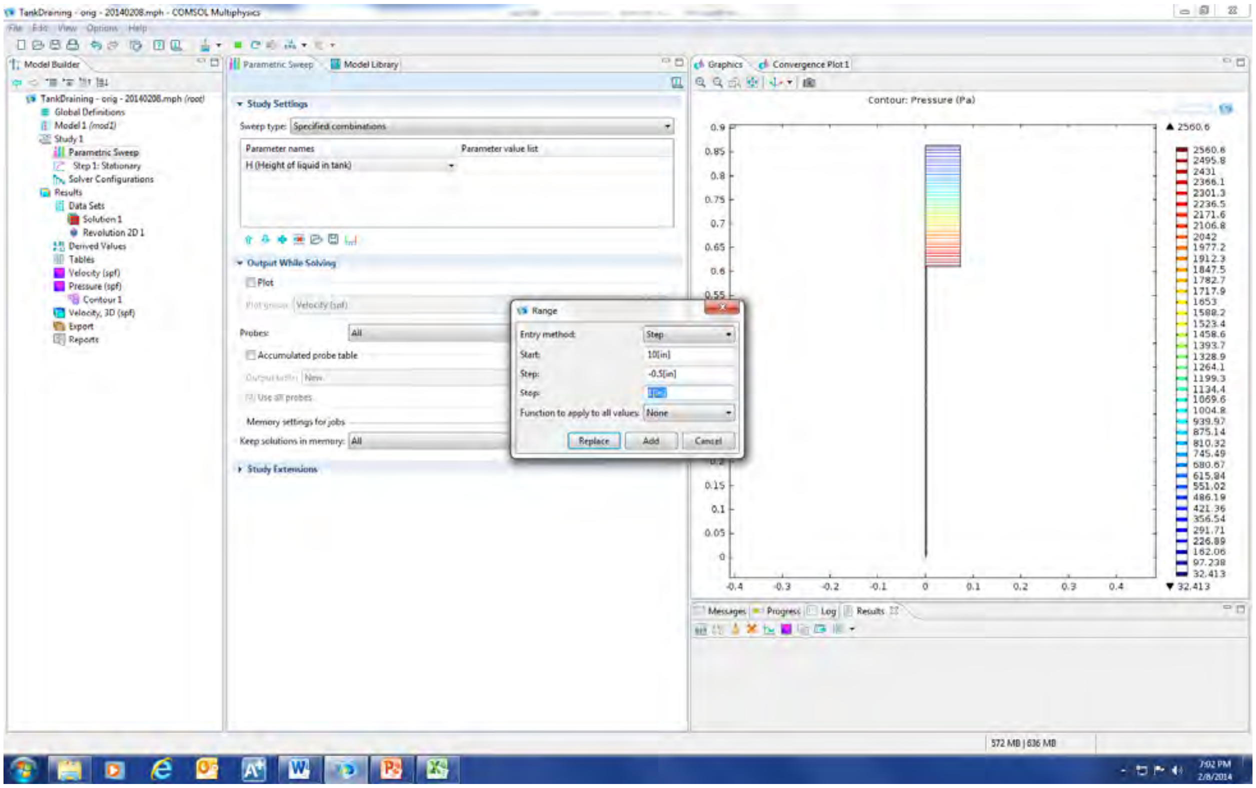
Click on H (Height of liquid in tank).

Hover over the buttons beneath the table to show their names and find the Range button.



Click on the **Range** button.

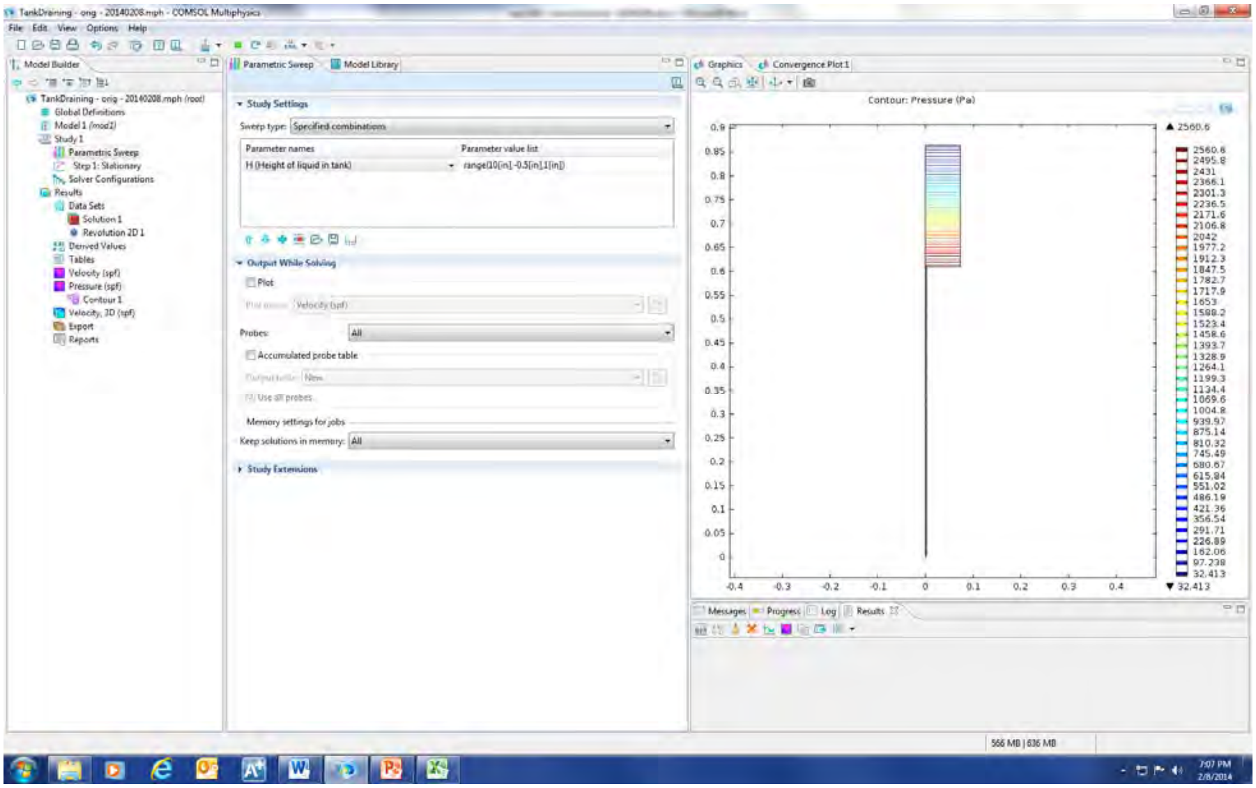
Then enter the values into the table.



These values must have the same dimensions as the parameter (in this instance, length).

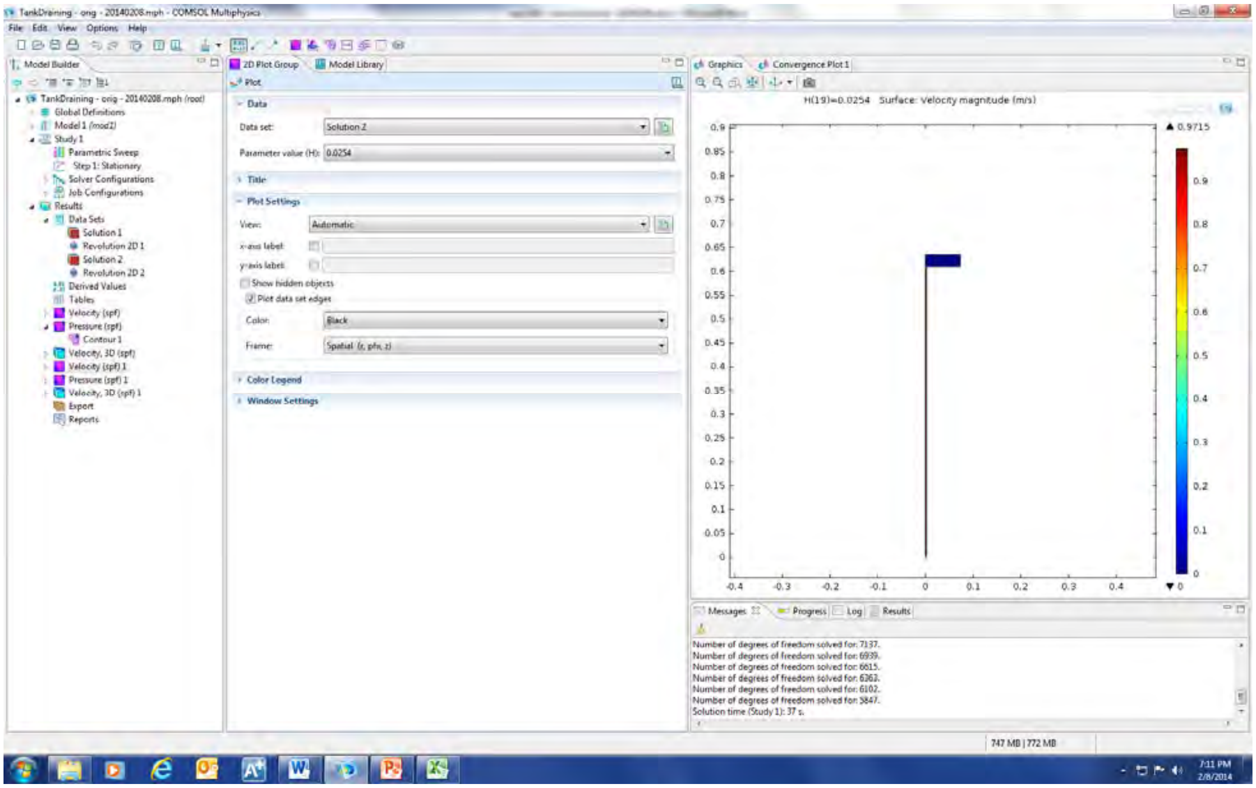
In addition, the step size is negative because the starting value of the parameter is larger than the stopping value.

Click the **Add** button on the entry form.



Right click on **Study 1**.

Click on **Compute**.

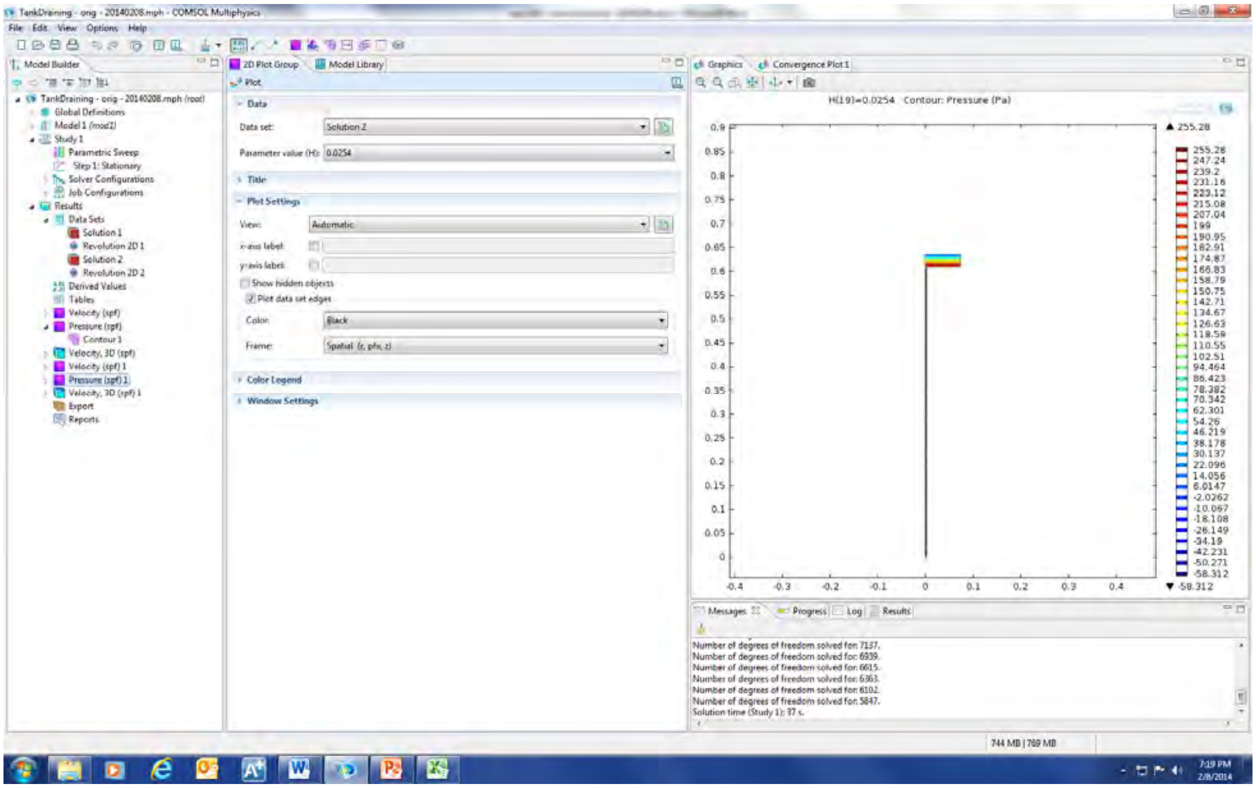


The Plot panel in the center of the screen indicates:

Parameter value (H): 0.0254 (the unit is m, so this is H = 1 in, the last value of the parametric sweep.)

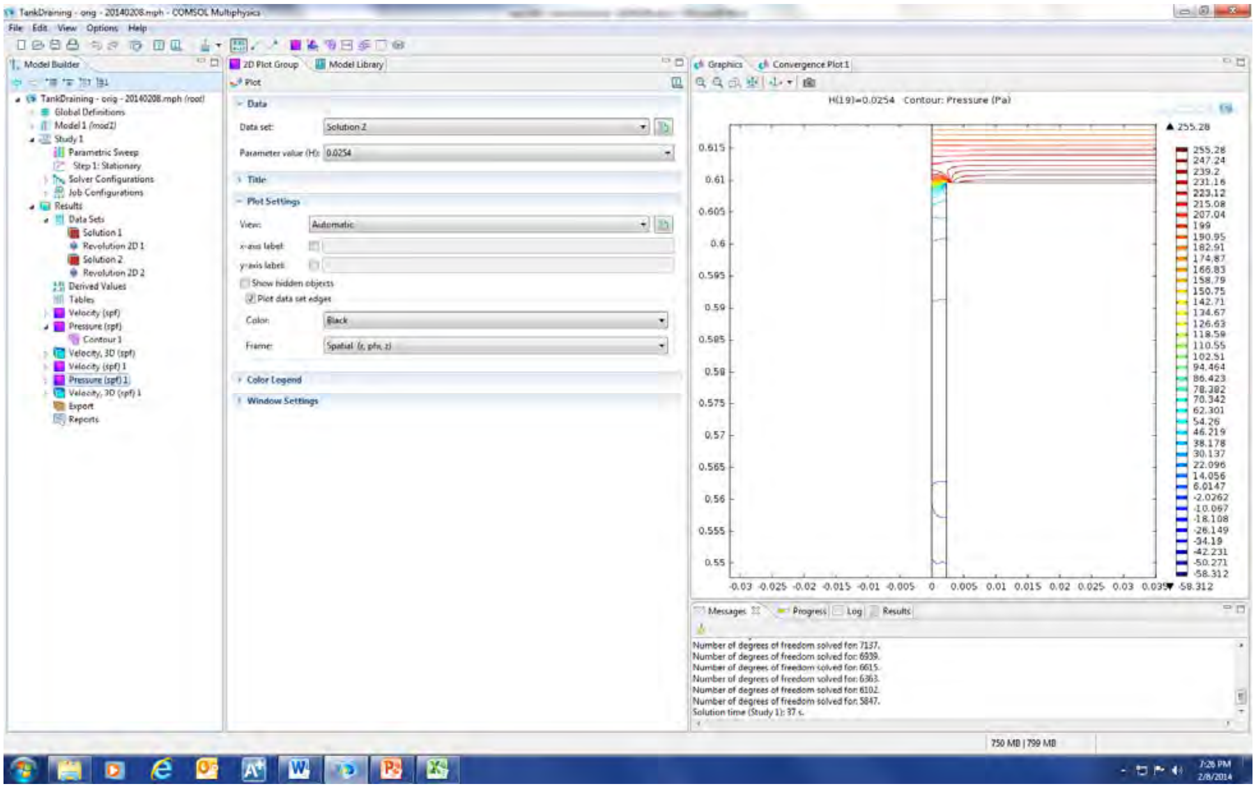
The highest velocity is 0.9715 m/s at H = 0.0254 m, whereas it was 1.2549 m/s at H = 0.254 m.

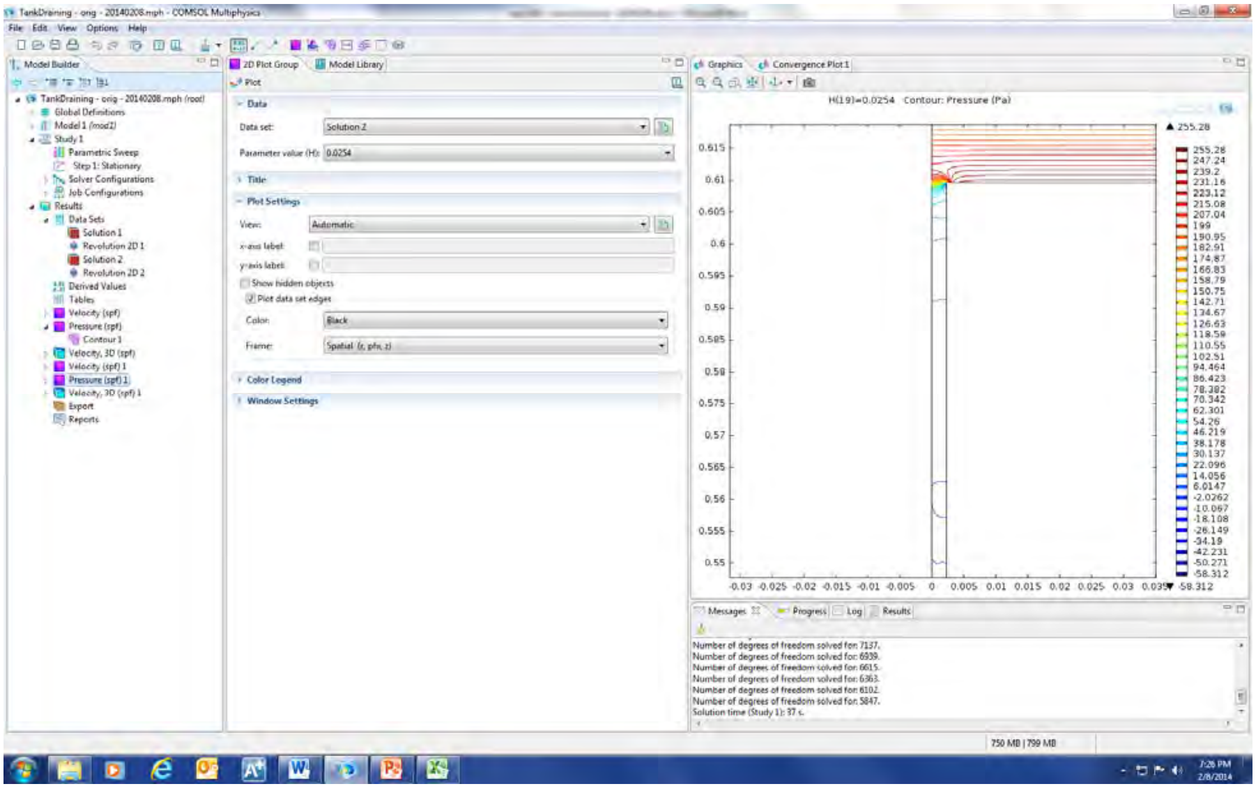
Click on **Pressure (spf) 1**.



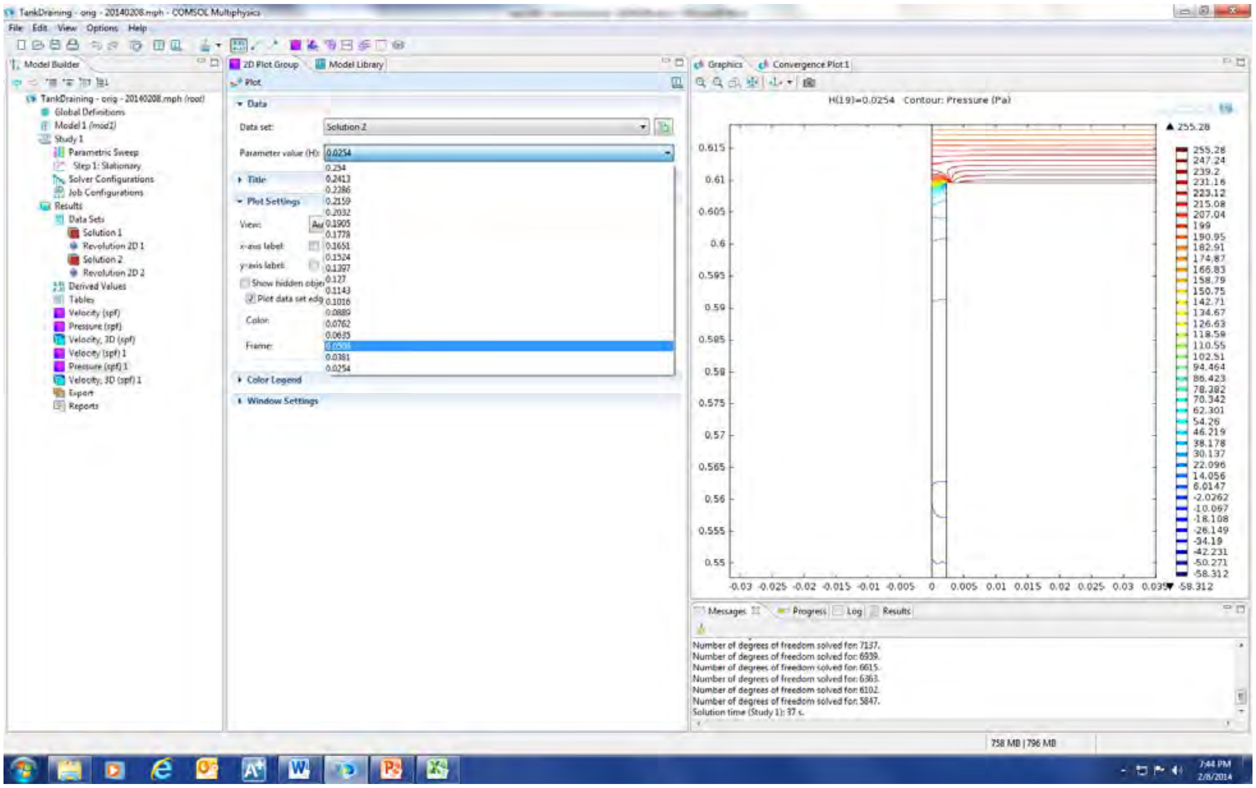
The lowest pressure is no −58.312 Pag, i.e. negative! A vacuum!

We can look at the pipe entrance to see the rapid decrease in pressure.

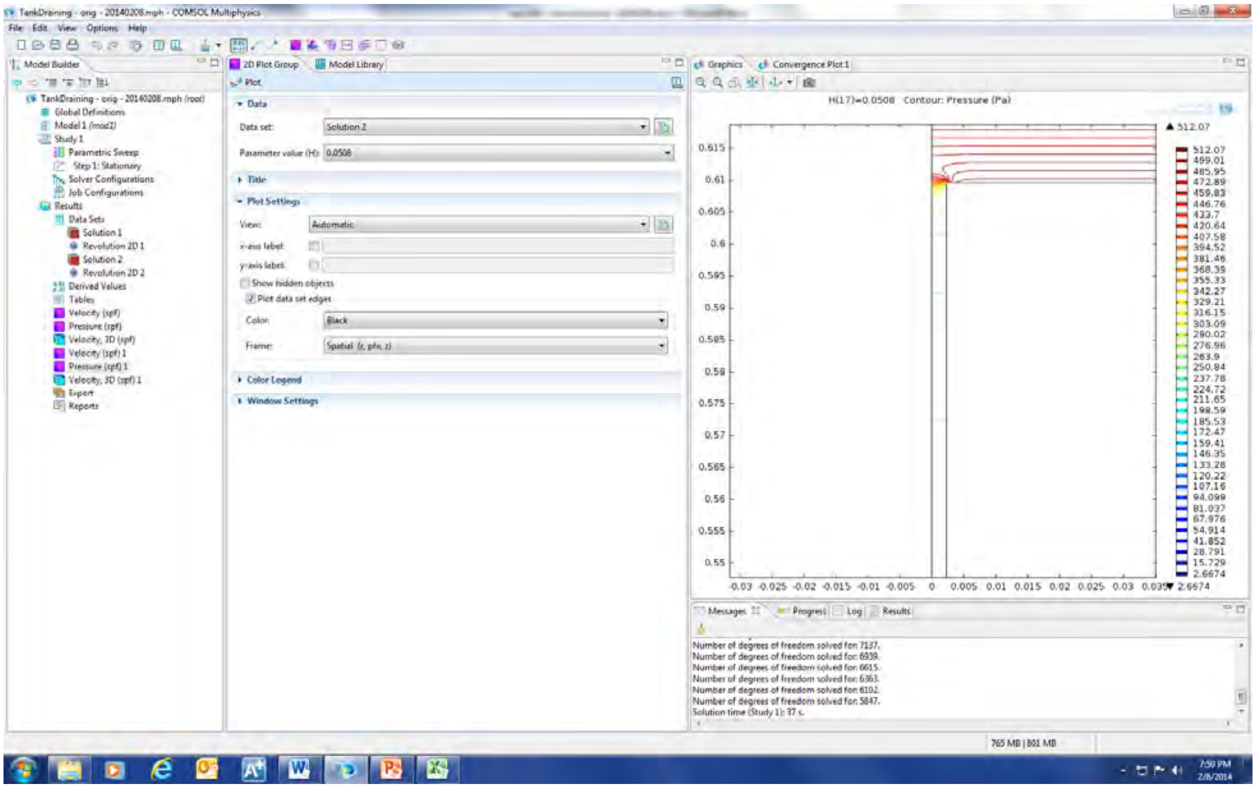

H(19)=0.0254 Contour: Pressure (Pa)
Number of degrees of freedom solved for: 7137.
Number of degrees of freedom solved for: 6939.
Number of degrees of freedom solved for: 6615.
Number of degrees of freedom solved for: 6363.
Number of degrees of freedom solved for: 6102.
Number of degrees of freedom solved for: 5847.
Solution time (Study 1): 37 s.



Click on the box for Parameter value (H) to select another height for examination.



Then click the **Plot** button at the top of the window.

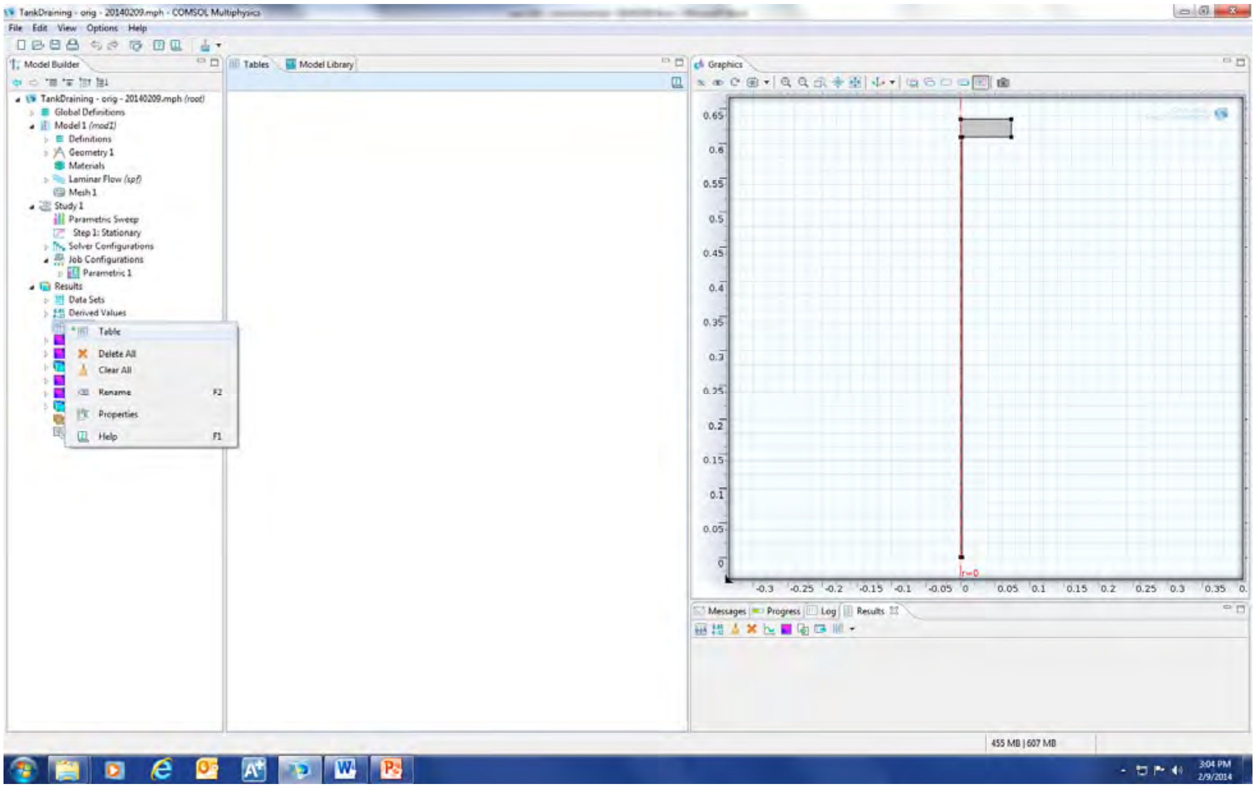


At a liquid height of H = 0.0508 m (2 in), all pressures are positive.

We now set up to grab some key values from the simulation.

First, let's set up a location to place the key values.

Right click on **Tables**.

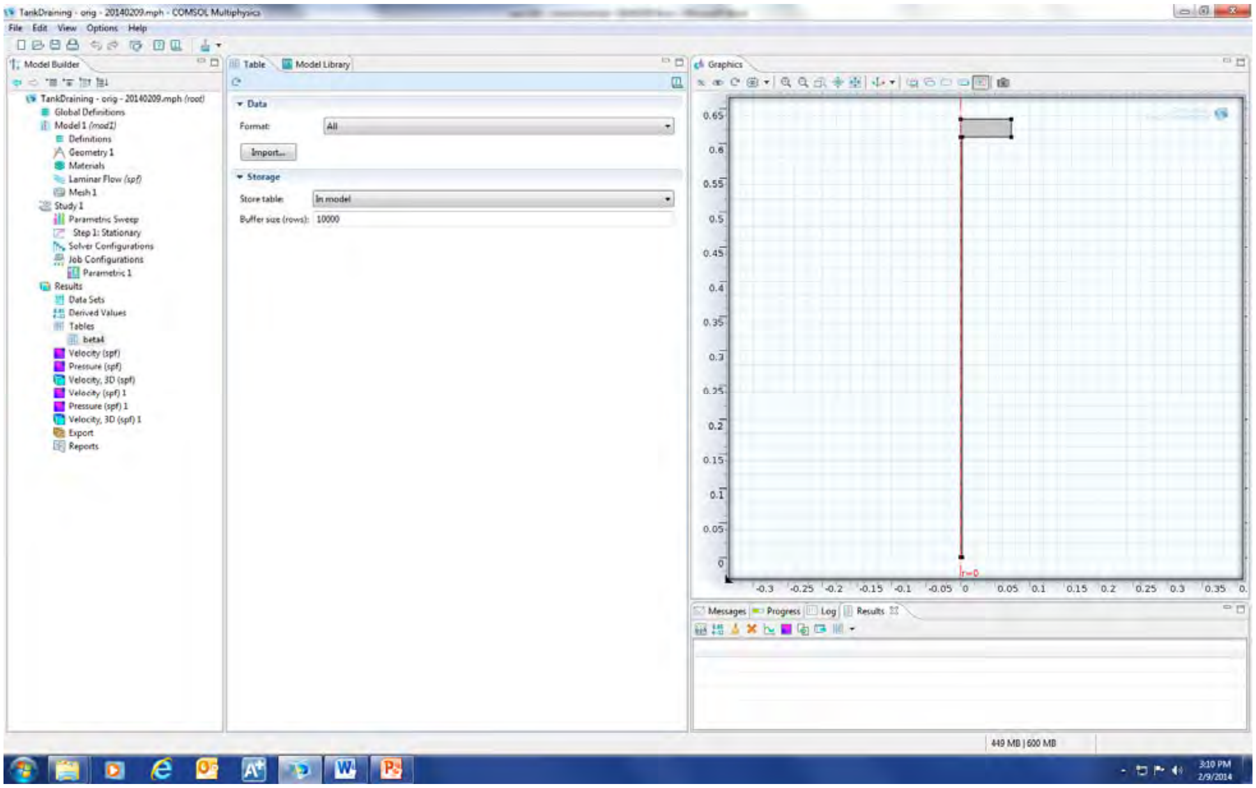


Click on **Table** on the pop-up menu to create a new table.

The new table has a default name of Table 1.

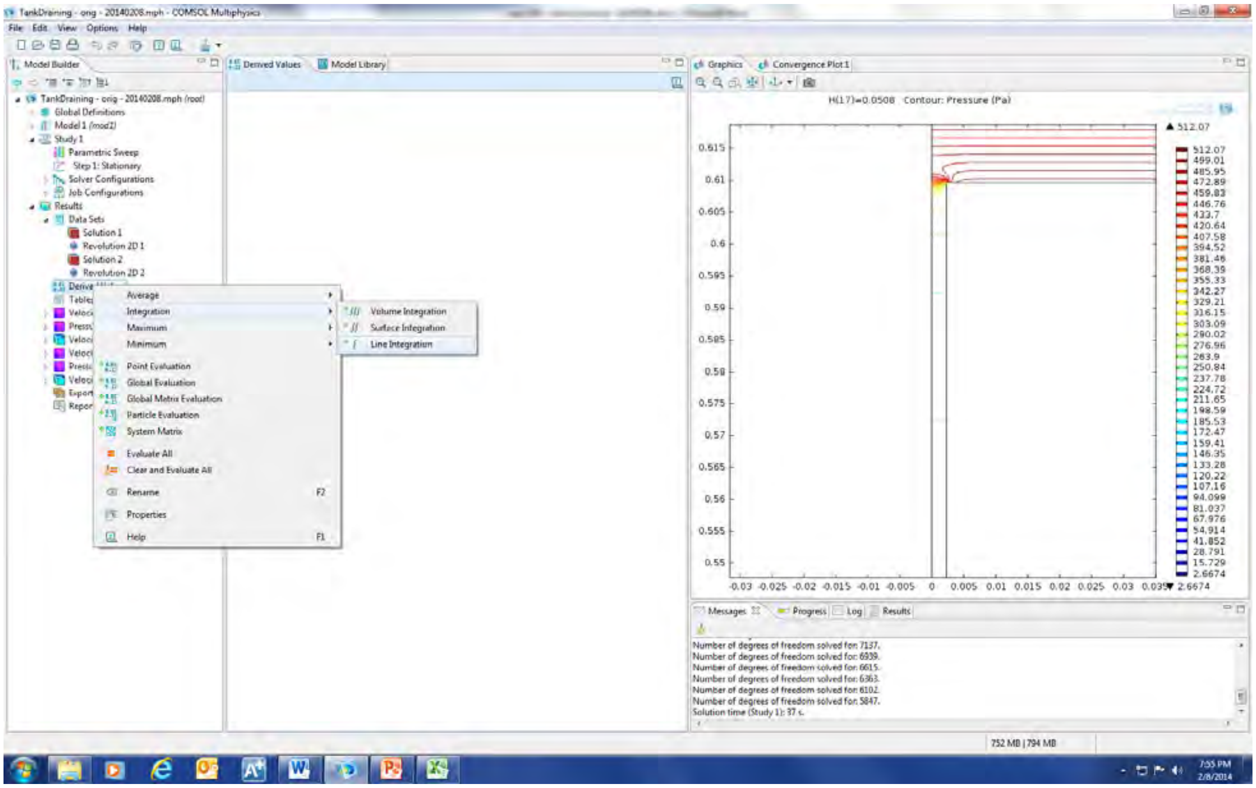
Right click on **Table 1** and rename it **beta4.**

This nomenclature aids bookkeeping because it is consistent with our lab manual.



The first value to add is the volumetric flow rate, Qdot/(m^3/s).

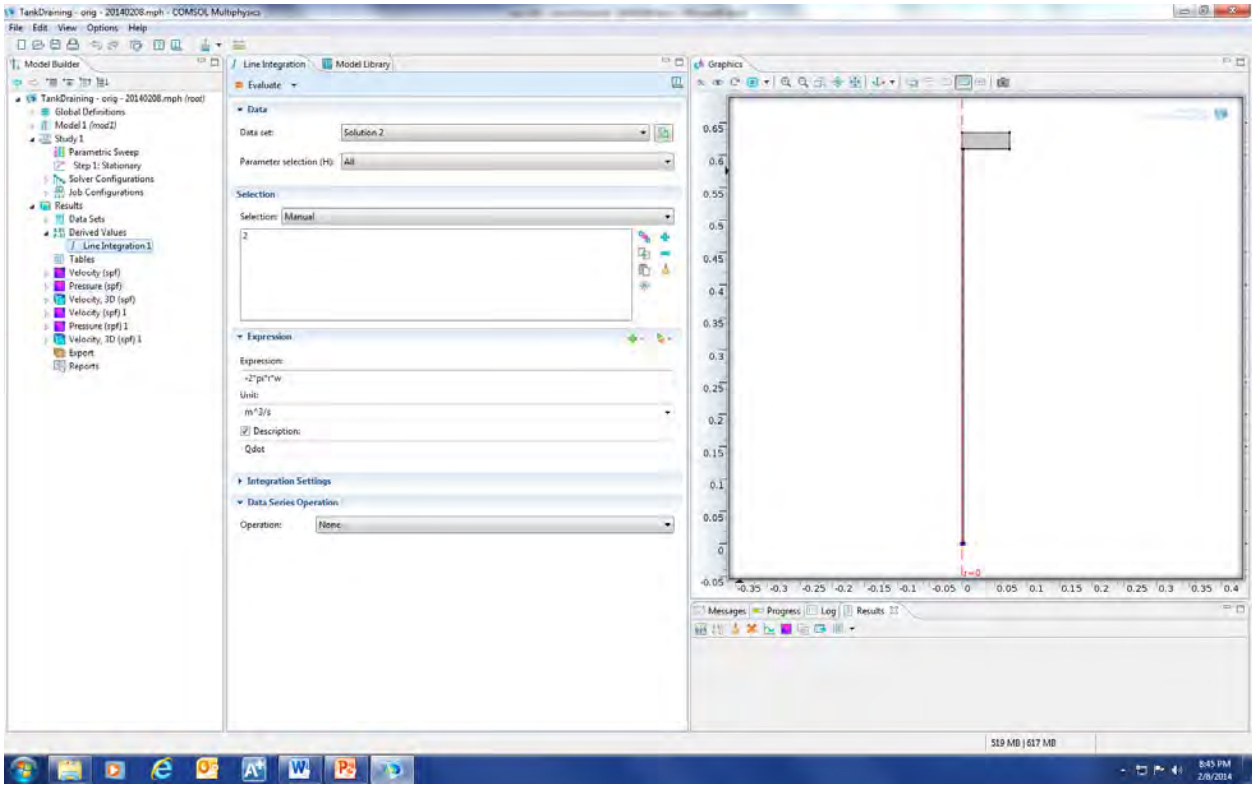
Right click on **Derived values** and then on **Integration**.



Select line integration.

Click on the bottom outlet of the pipe in the Graphics window to select the outlet boundary.
Click on the Add button (the blue plus sign ✚) to add the outlet boundary to the selection box.

Then enter the information in the boxes as indicated on the COMSOL image below.



For Data set, we entered **Solution 2**.

Why? Solution 1 was for the initial simulation at H = 0.254 m (10 in) only.

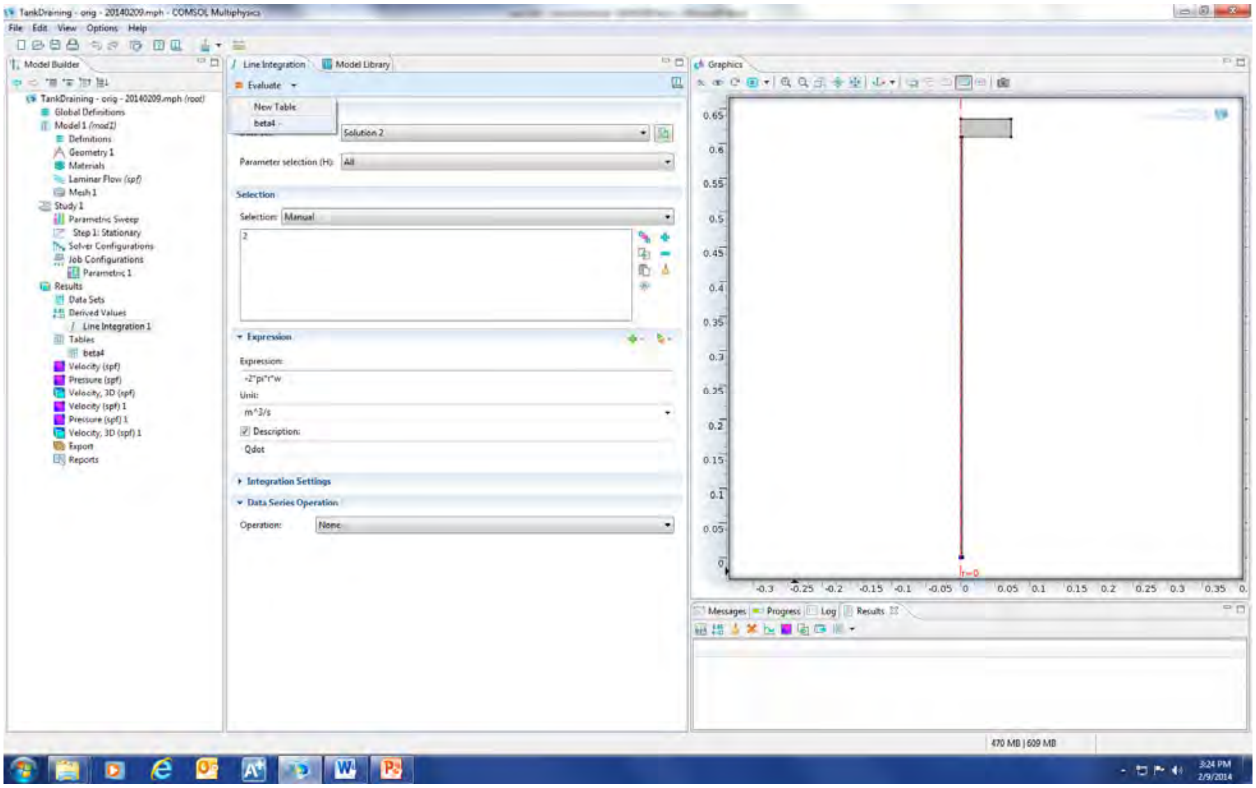
Solution 2 contains the results for the Parametric sweep.

For Parameter selection (H), enter **All**. COMSOL will then find the derived values at all heights in the parametric sweep.

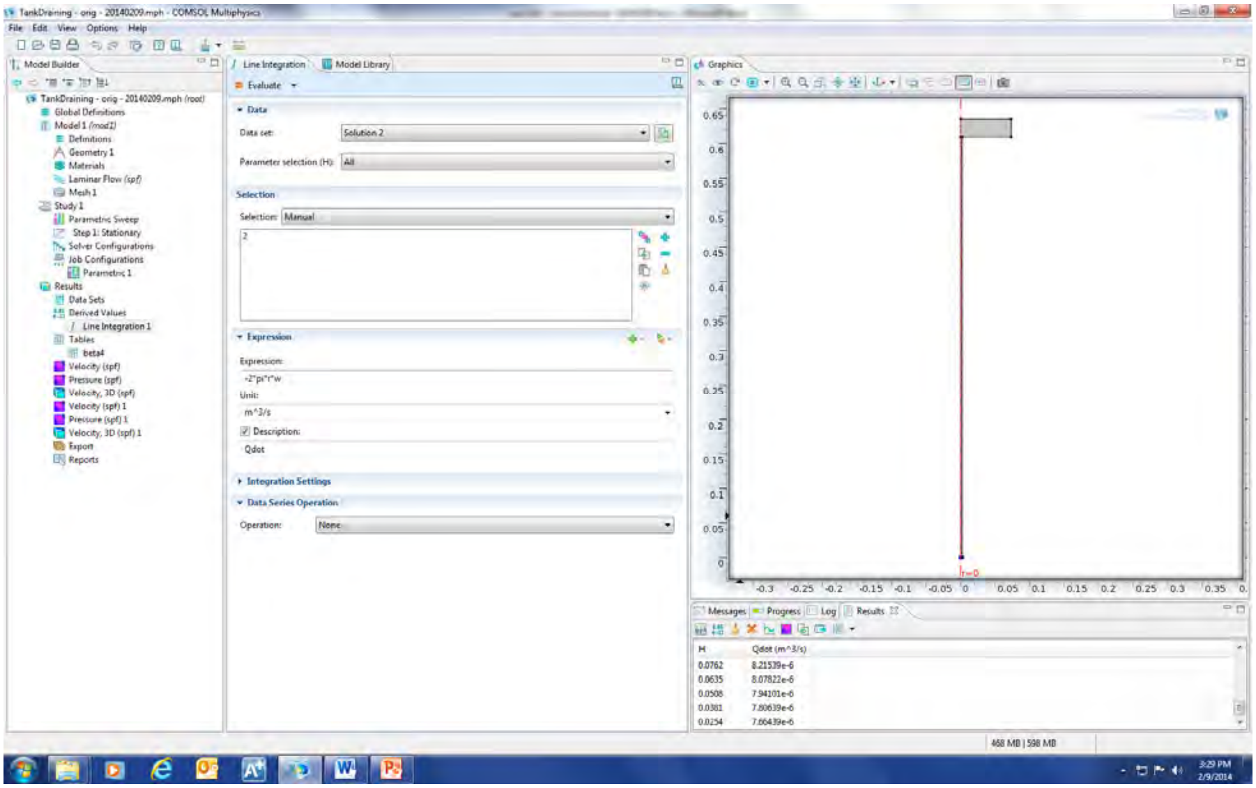
Next step:

Do NOT click on the Evaluate button.

Instead, click the on the **arrow** to the right of the Evaluate button at the top of the Line integration window.

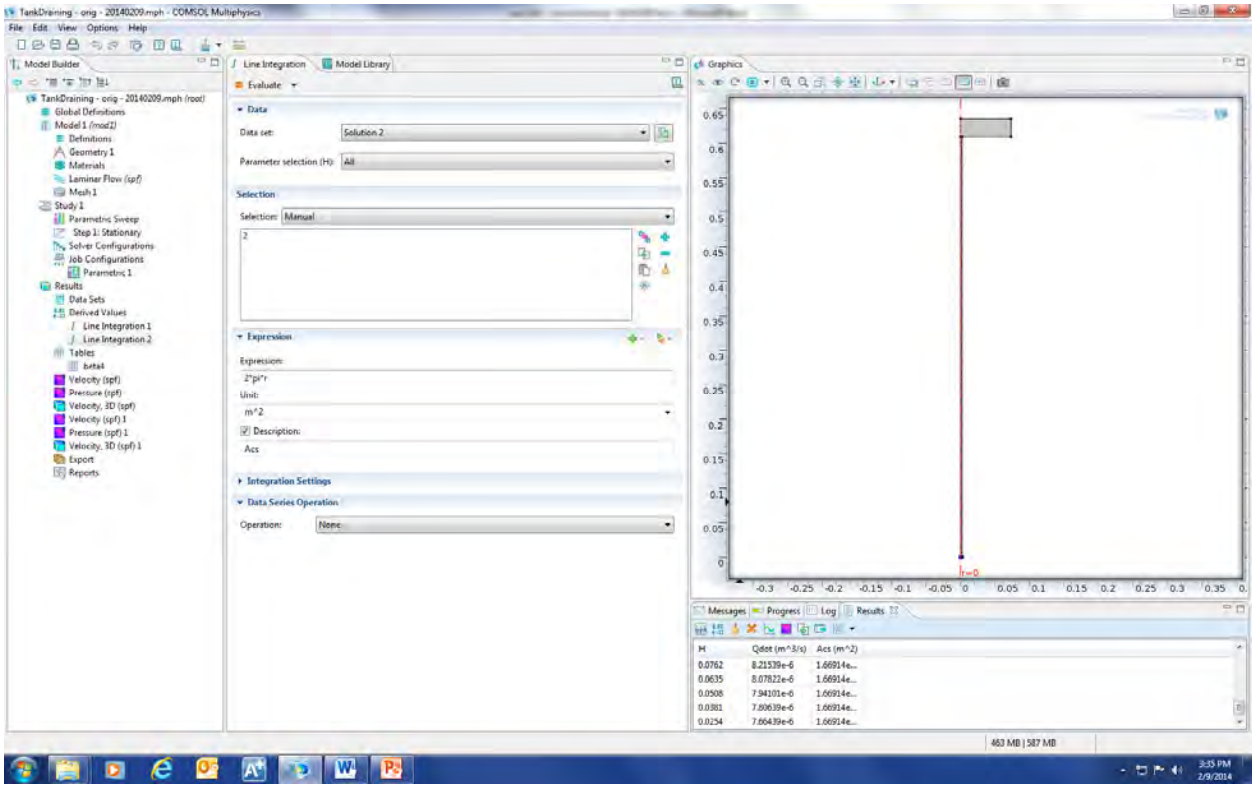


Click on **beta 4** in the dropdown list under the Evaluate button.



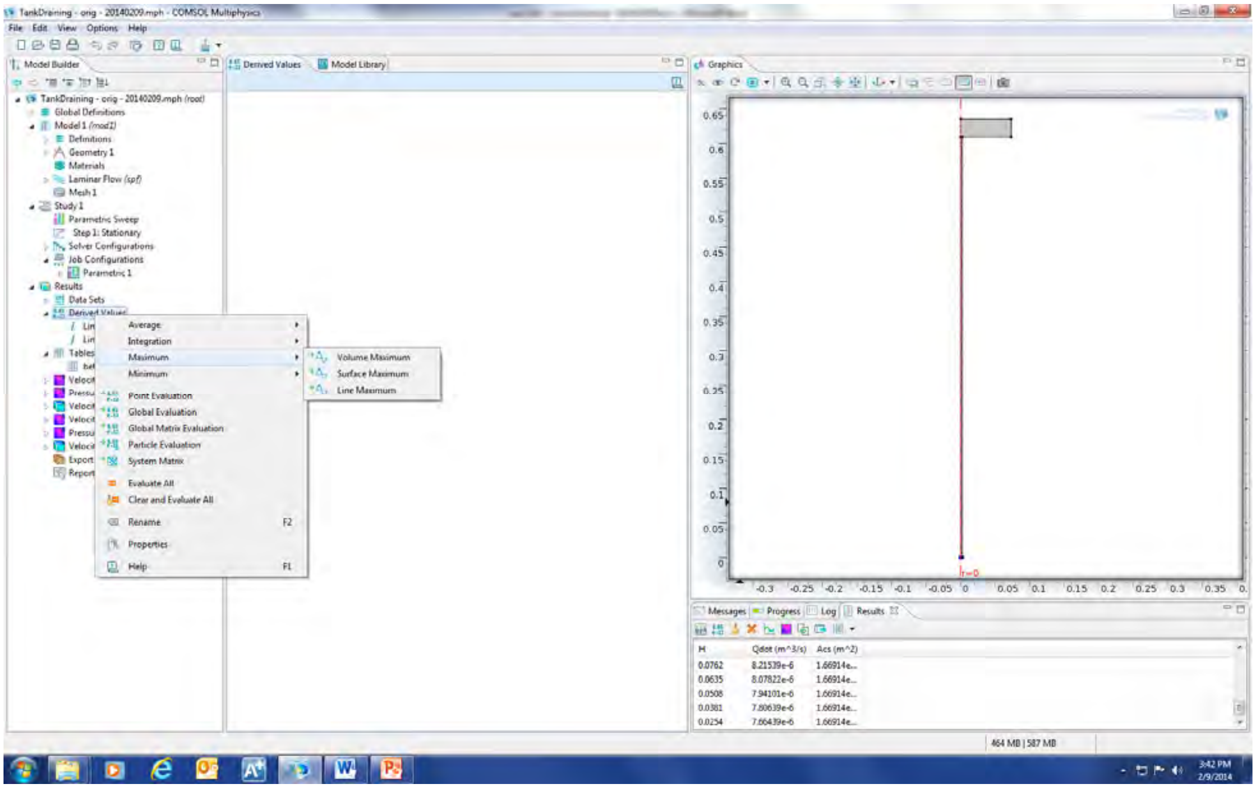
Right click on **Derived values** and then proceed in a similar manner to enter information to compute the cross-sectional area of the pipe outlet.

Add the values for the cross-sectional area to the table beta4 using the dropdown menu adjacent to the Evaluate button.



We next wish to find the maximum value of the magnitude of the velocity in the z-direction at the pipe outlet.

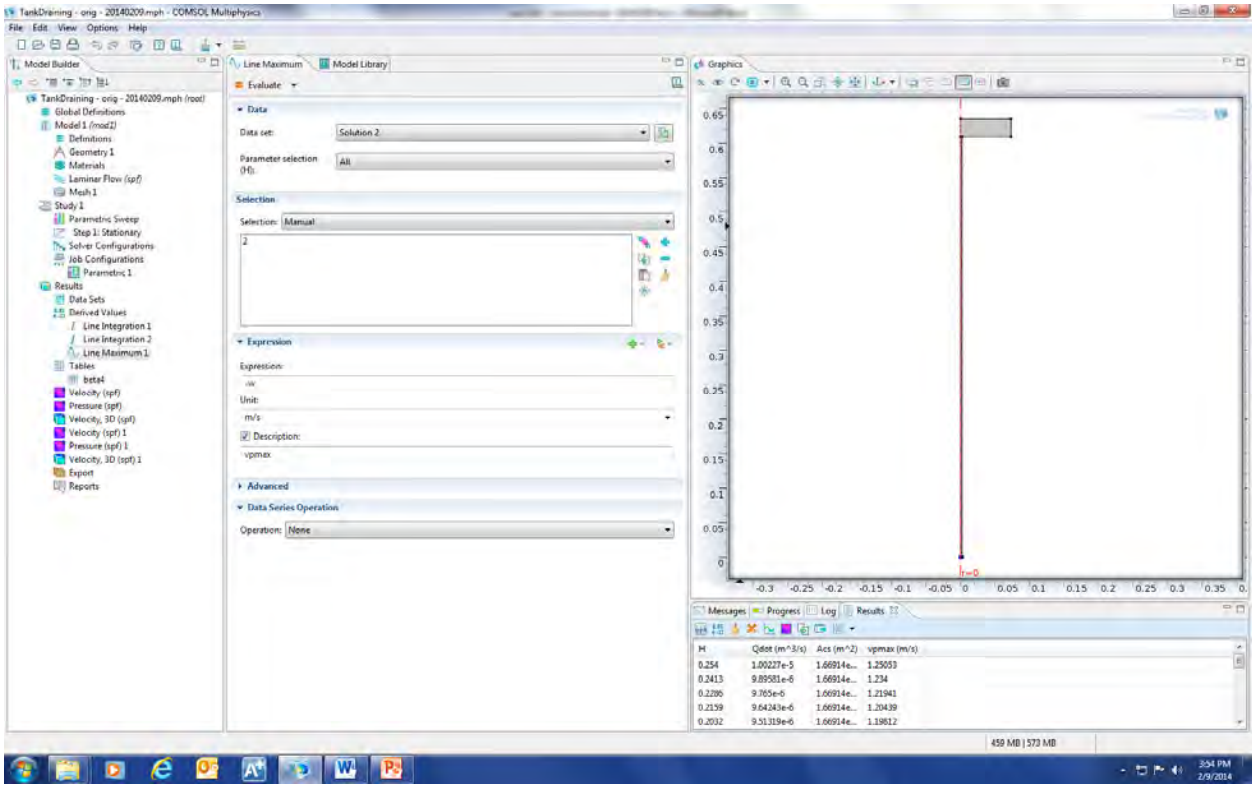
Right click on **Derived Values**, but this time in the pop-up menu expand **Maximum**.



Click on Line Maximum in the pop-up menu.

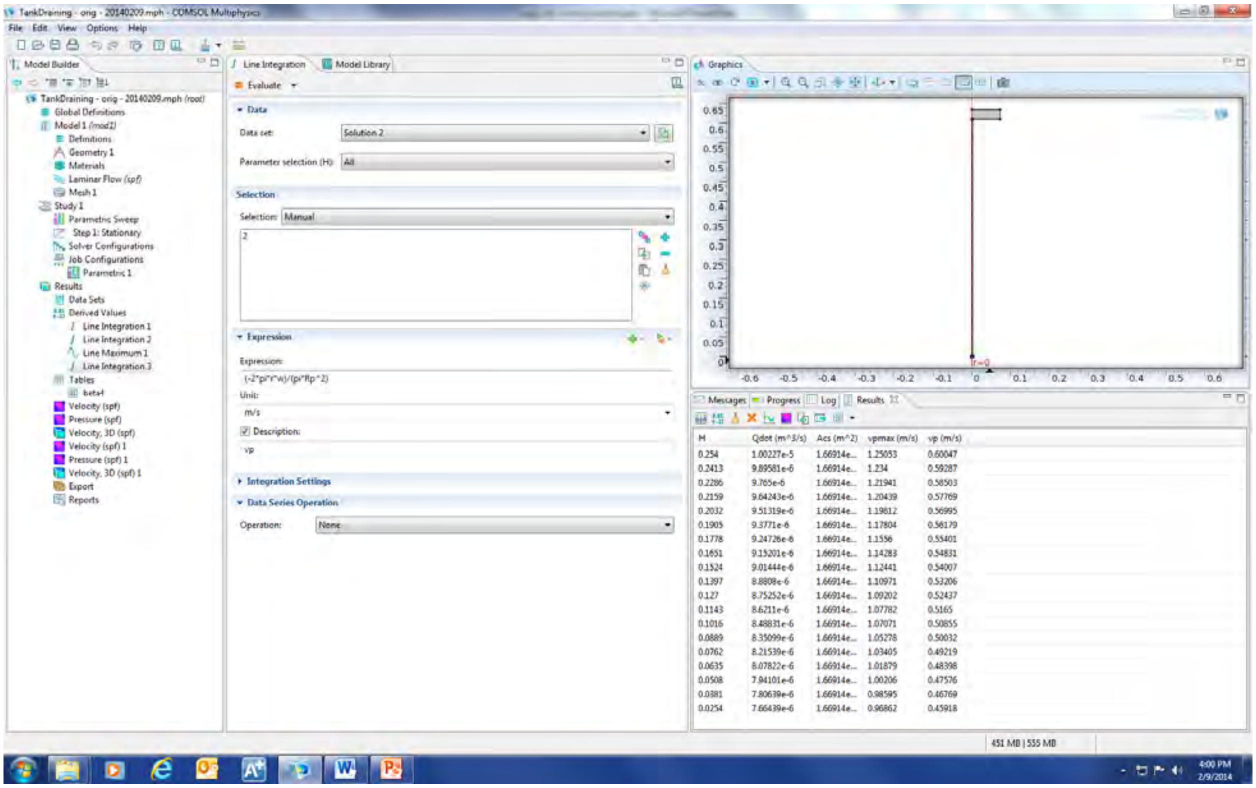
Enter the appropriate information on the form for Line Maximum.

Add the values for the maximum magnitude of the velocity to table beta4 using the dropdown menu adjacent to the Evaluate button.



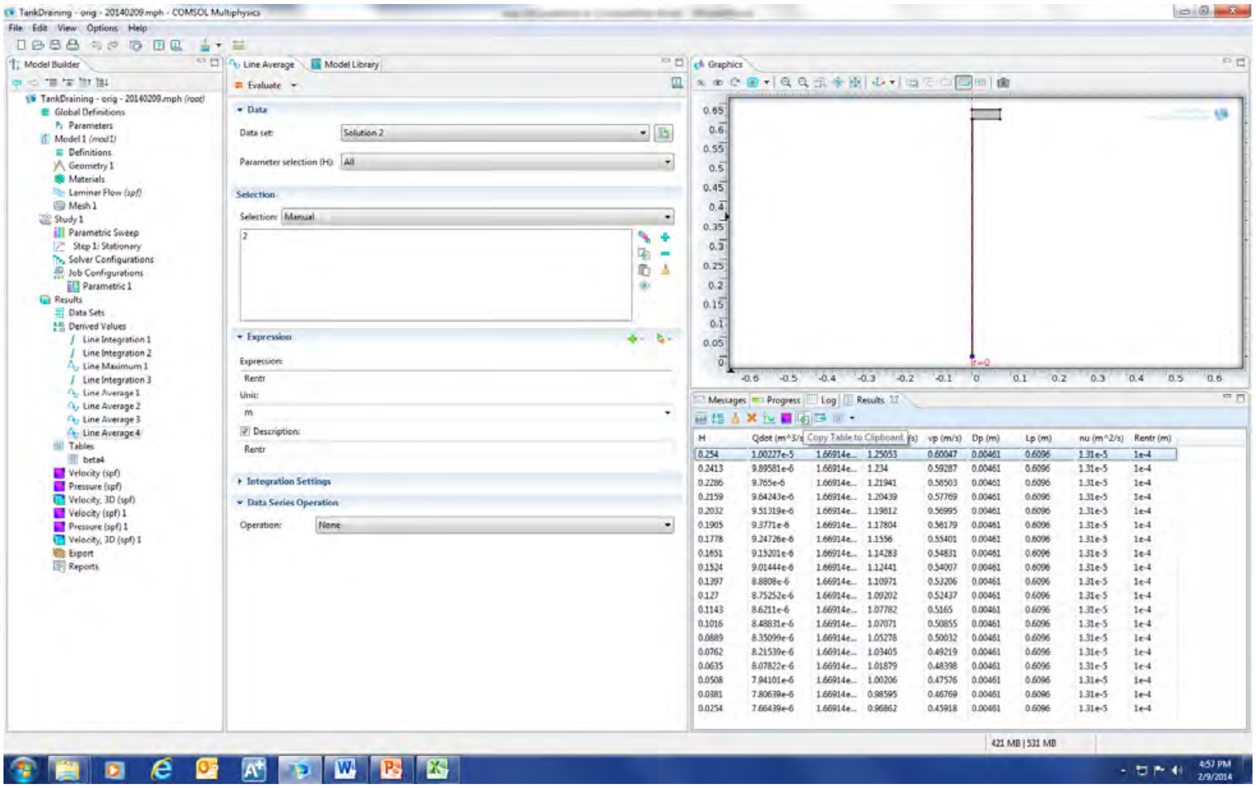
Right click on **Derived values** and then proceed in a similar manner to enter information to compute the magnitude of the average velocity in the z-direction in the pipe outlet.

Add the values for the velocity to the table beta4 using the dropdown menu adjacent to the Evaluate button.



Another feature of Derived Values is Line Average.

One can use it to add parameter values such as Dp, Lp, nu, and Rentr, which then serves as a convenient record of the conditions for the simulation.



Above the table, there is a useful button for Copy Table to Clipboard.

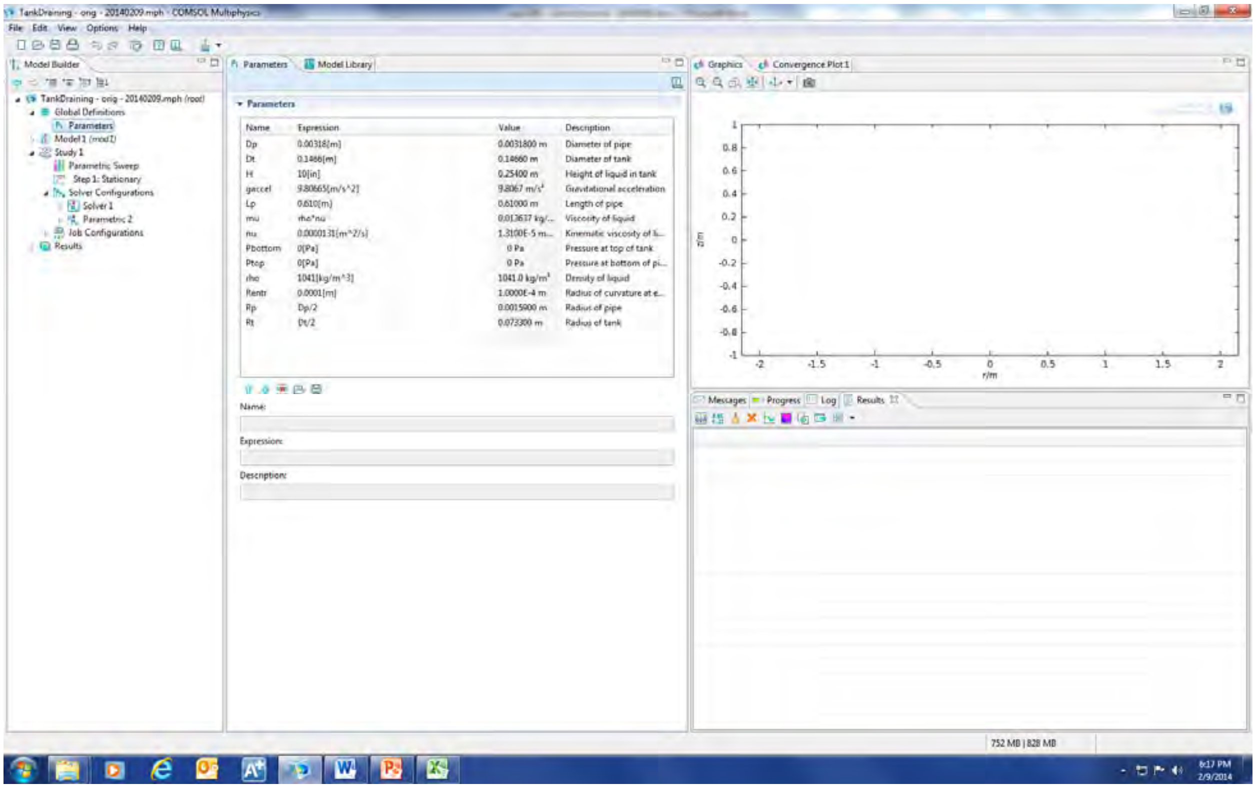
Thus, one can copy the data and paste it into an Excel worksheet for further manipulation.

This concludes the simulation of pipe beta4.

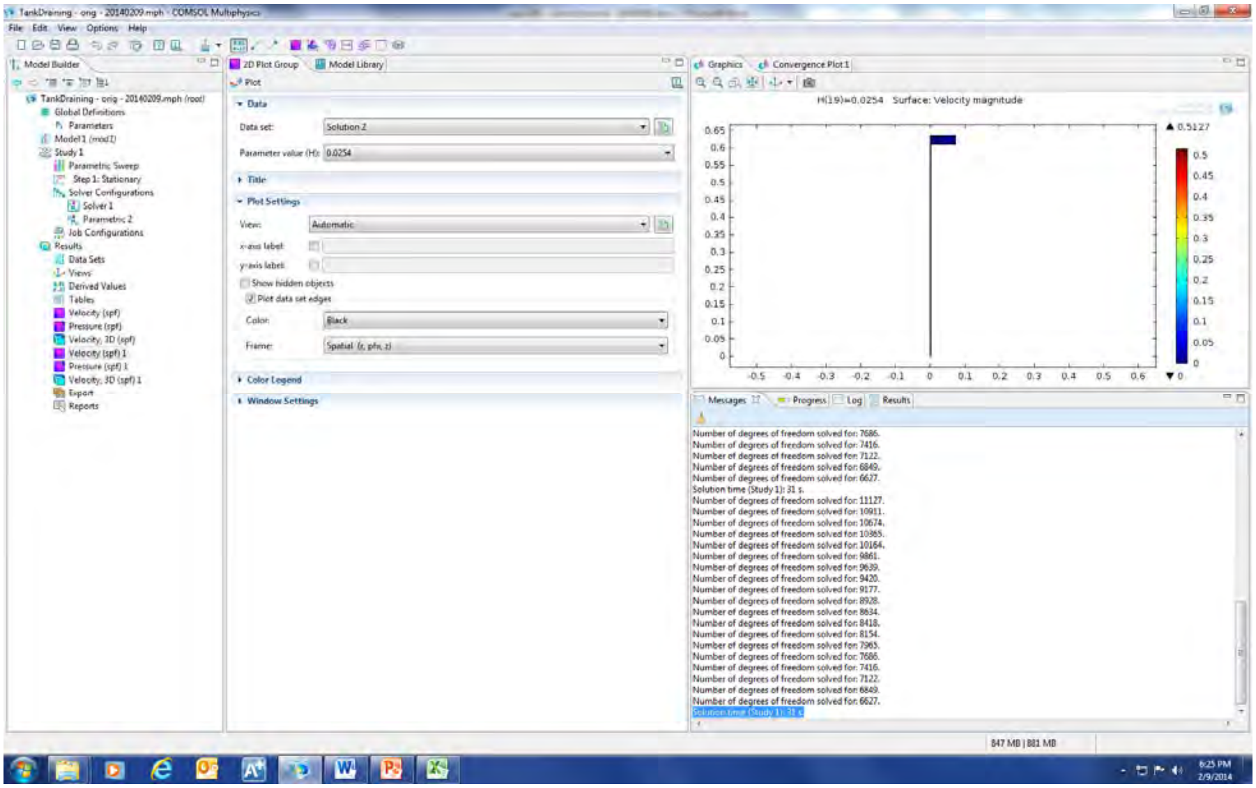
With the setup of the model, computation, and data manipulation, we now proceed to examine the impact of changing parameters.

For example, consider pipe gamma 4, which has Dp = 0.00318 m and Lp = 0.610 m.

Click on Global Definitions > Parameters. Then enter the values for these parameters.



Right click on **Study 1** and then click on **Compute**.



The calculation completes.

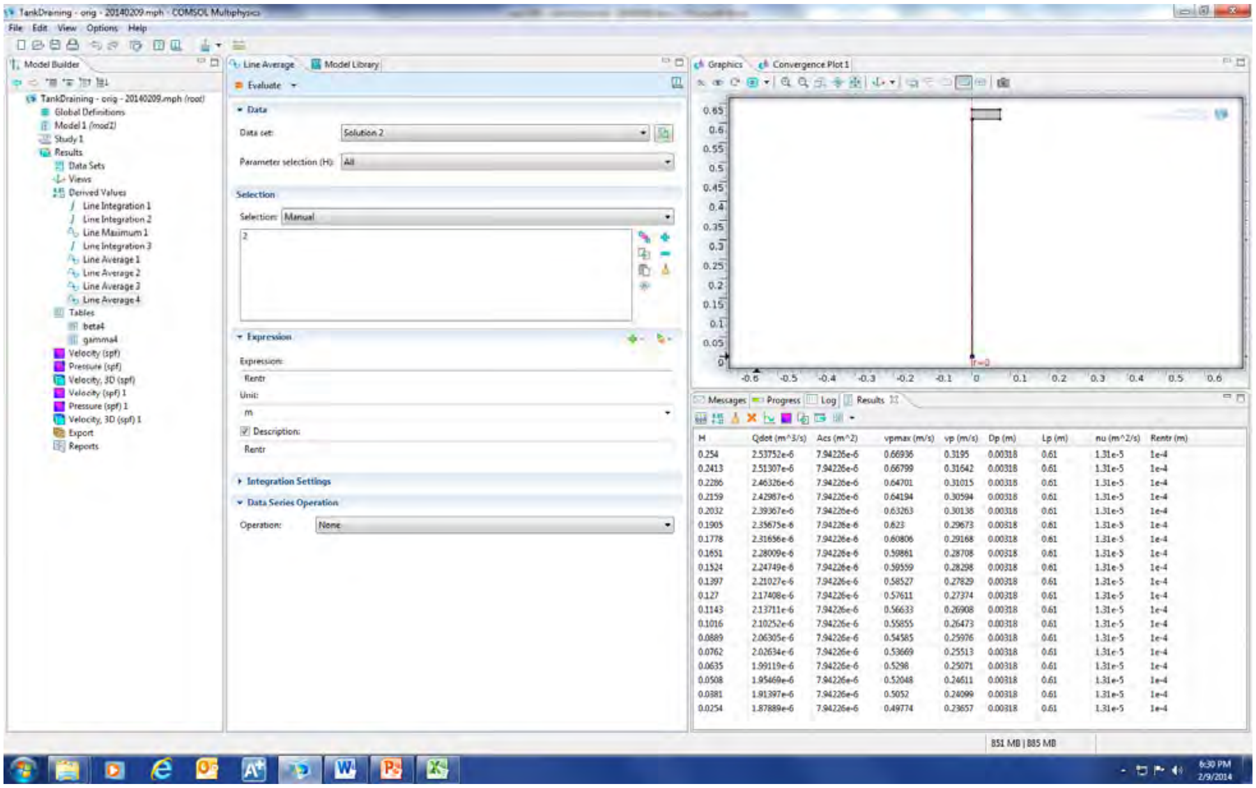
Inspection shows that COMSOL has updated the data in Solution 2.

Right click on Tables. Add table and rename it as gamma4.

Then click on the first derived value (for line integration 1).

Add the results for the first derived value (**volumetric flow rate**) to table **gamma4** using the dropdown menu adjacent to the Evaluate button.

Similarly, add the results for the other derived values to table gamma4.


TankDraining - orig - 20140209.mph - COMSOL Multiphysics
File Edit View Options Help
Model Builder
TankDraining - orig - 20140209.mph (root)
Global Definitions
Model 1 (mod1)
Study 1
Results
Data Sets
Views
Derived Values
Line Integration 1
Line Integration 2
Line Maximum 1
Line Integration 3
Line Average 1
Line Average 2
Line Average 3
Line Average 4
Tables
beta4
gamma4
Velocity (spf)
Pressure (spf)
Velocity, 3D (spf)
Velocity (spf) 1
Pressure (spf) 1
Velocity, 3D (spf) 1
Export
Reports
Line Average
Model Library
Evaluate
Data
Data set:
Solution 2
Parameter selection (H):
All
Selection
Selection:
Manual
2
Expression
Expression:
Rentr
Unit:
m
Description:
Rentr
Integration Settings
Data Series Operation
Operation:
None
Graphics
Convergence Plot 1
Messages
Progress
Log
Results
H Qdot (m^3/s) Acs (m^2) vpmax (m/s) vp (m/s) Dp (m) Lp (m) nu (m^2/s) Rentr (m)
0.254 2.53752e-6 7.94226e-6 0.66936 0.3195 0.00318 0.61 1.31e-5 1e-4
0.2413 2.51307e-6 7.94226e-6 0.66799 0.31642 0.00318 0.61 1.31e-5 1e-4
0.2286 2.46326e-6 7.94226e-6 0.64701 0.31015 0.00318 0.61 1.31e-5 1e-4
0.2159 2.42987e-6 7.94226e-6 0.64194 0.30594 0.00318 0.61 1.31e-5 1e-4
0.2032 2.39367e-6 7.94226e-6 0.63263 0.30138 0.00318 0.61 1.31e-5 1e-4
0.1905 2.35675e-6 7.94226e-6 0.623 0.29673 0.00318 0.61 1.31e-5 1e-4
0.1778 2.31656e-6 7.94226e-6 0.60806 0.29168 0.00318 0.61 1.31e-5 1e-4
0.1651 2.28009e-6 7.94226e-6 0.59861 0.28708 0.00318 0.61 1.31e-5 1e-4
0.1524 2.24749e-6 7.94226e-6 0.59559 0.28298 0.00318 0.61 1.31e-5 1e-4
0.1397 2.21027e-6 7.94226e-6 0.58527 0.27829 0.00318 0.61 1.31e-5 1e-4
0.127 2.17408e-6 7.94226e-6 0.57611 0.27374 0.00318 0.61 1.31e-5 1e-4
0.1143 2.13711e-6 7.94226e-6 0.56633 0.26908 0.00318 0.61 1.31e-5 1e-4
0.1016 2.10252e-6 7.94226e-6 0.55855 0.26473 0.00318 0.61 1.31e-5 1e-4
0.0889 2.06305e-6 7.94226e-6 0.54585 0.25976 0.00318 0.61 1.31e-5 1e-4
0.0762 2.02634e-6 7.94226e-6 0.53669 0.25513 0.00318 0.61 1.31e-5 1e-4
0.0635 1.99119e-6 7.94226e-6 0.5298 0.25071 0.00318 0.61 1.31e-5 1e-4
0.0508 1.95469e-6 7.94226e-6 0.52048 0.24611 0.00318 0.61 1.31e-5 1e-4
0.0381 1.91397e-6 7.94226e-6 0.5052 0.24099 0.00318 0.61 1.31e-5 1e-4
0.0254 1.87889e-6 7.94226e-6 0.49774 0.23657 0.00318 0.61 1.31e-5 1e-4
851 MB | 885 MB

Closing thoughts

COMSOL is a tool for conducting simulated experiments.

But is any of this valid? Are the results to believed?

We need to be cautious.

First, as beginning transport students, we may have misapplied principles of transport phenomena.

We thus need to learn and explore.

For example, maybe the boundary conditions are suboptimal to model the system.

Second, as novice COMSOL users, we may have misunderstood and bungled COMSOL operations.

We thus need to devise cross checks in our work.

For example, we chose an element size of normal for the mesh. But maybe this is inadequate.

Or, are the COMSOL results consistent with analytical results.

Finally, from time to time, maybe we should compare the COMSOL predictions to real results from real experiments!