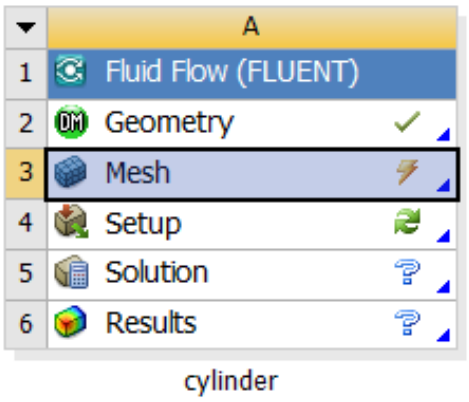
Steady Flow Past a Cylinder - Physics Setup

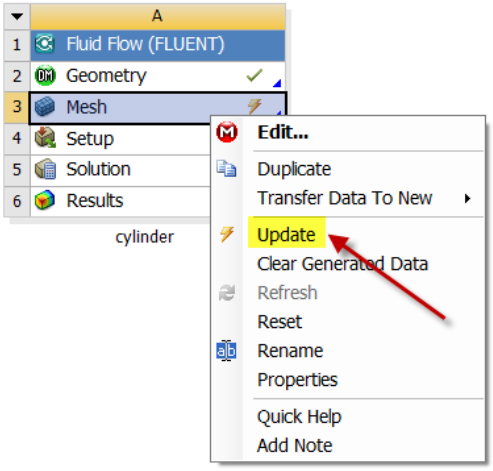
Authors: John Singleton and Rajesh Bhaskaran, Cornell University

Physics Setup

Your workbench project should look like this.



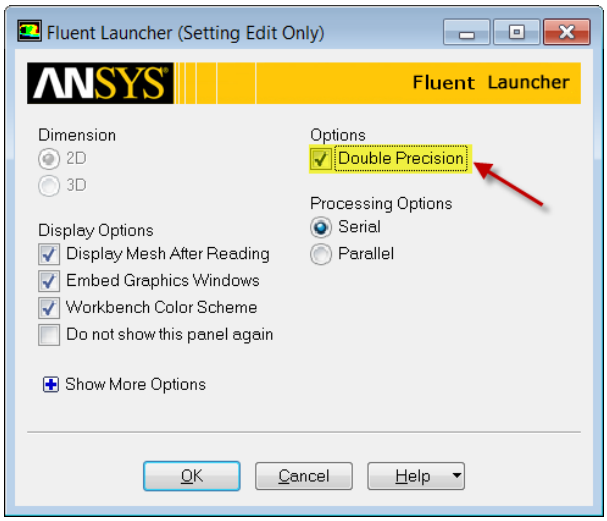
We are done with all the meshing steps but for some reason, a tick mark doesn't appear next to *Mesh* in the project page. To get the tick mark next to *mesh* , right-click on it and select *Update* as shown below.



Launch Fluent

(Double Click) Setup in the ***Workbench Project Page***.

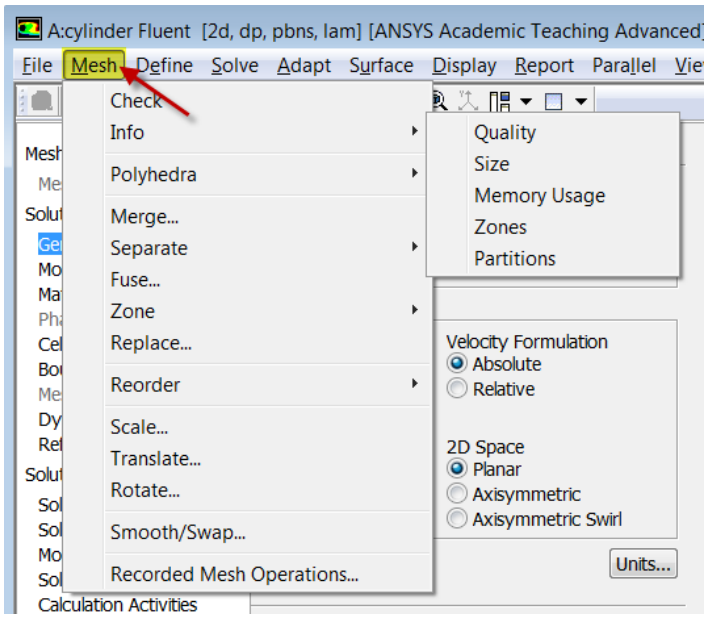
When the ***FLUENT Launcher*** appears change options to "Double Precision", and then click ***OK*** as shown below.The Double Precision option is used to select the double-precision solver. In the double-precision solver, each floating point number is represented using 64 bits in contrast to the single-precision solver which uses 32 bits. The extra bits increase not only the precision, but also the range of magnitudes that can be represented. The downside of using double precision is that it requires more memory.



Twiddle your thumbs a bit while the FLUENT interface starts up. This is where we'll specify the governing equations and boundary conditions for our boundary-value problem. On the left-hand side of the FLUENT interface, we see various items listed under ***Problem Setup***. We will work from top to bottom of the ***Problem Setup*** items to setup the physics of our boundary-value problem. On the right hand side, we have the ***Graphics*** pane and, below that, the ***Command*** pane.

Check Mesh

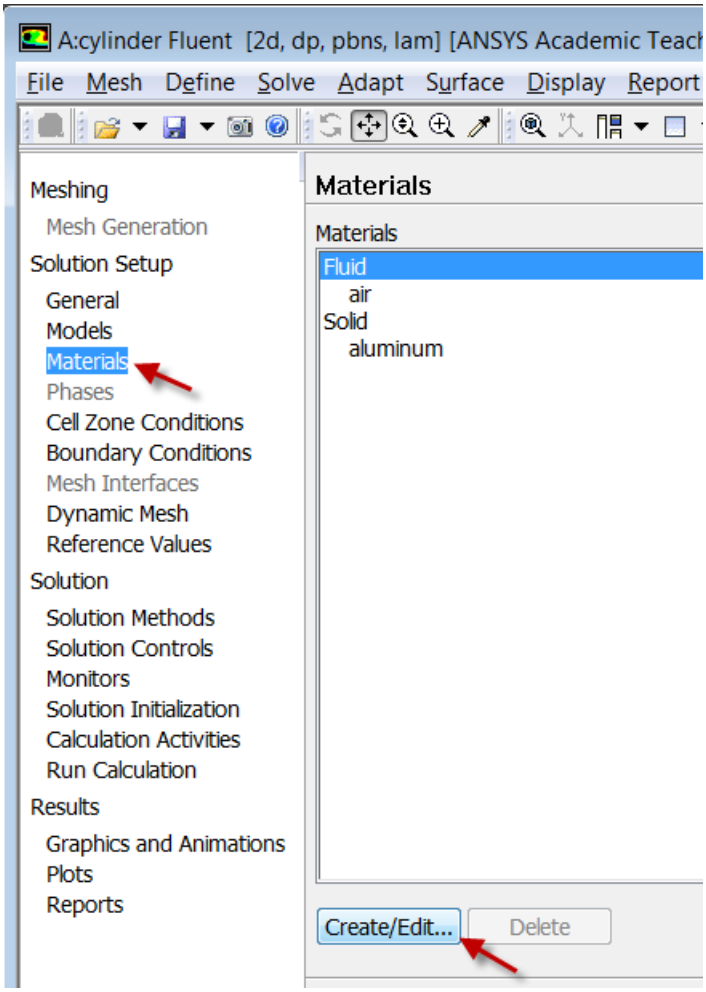
(Click) Info > Size
You should now have an output in the ***command*** pane stating that there are 18,432 cells.



(Click) Check > Perform Mesh Check
You should see no errors in the ***command*** pane.

Specify Material Properties

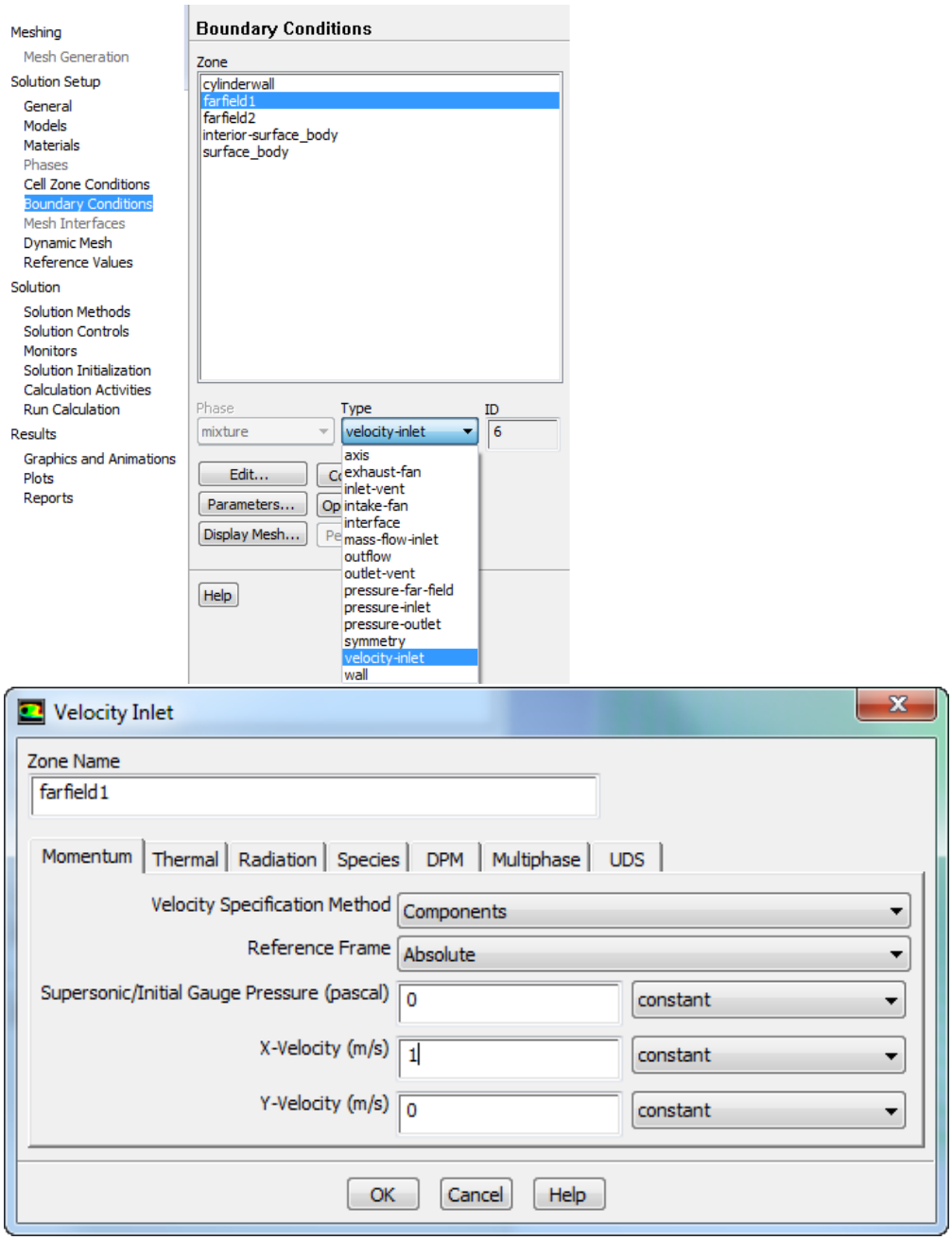
Solution Setup > Materials > Fluid > Create/Edit....
Then set the ***Density*** to 1 kg/m^3 and set ***Viscosity*** to 0.05 kg/m*s. Click ***Change/Create*** then click ***Close***.



Boundary Conditions

FarField1

Solution Setup > Boundary Conditions > farfield1.
Set ***Type*** to ***velocity-inlet***. Click ***Edit...***. Set ***Velocity Specification Method*** to ***Components***, set ***X-Velocity*** to 1 m/s, and set ***Y-Velocity*** to 0 m/s.



FarField2

Solution Setup > Boundary Conditions > farfield2..
Set ***Type*** to ***pressure-outlet***.

Note that pressures in the FLUENT interface are specified in terms of *gauge* values where: *gauge pressure = absolute pressure - reference pressure*. The reference pressure is 1 atm by default. So in our case the gauge pressure at the outlet is 0 atm which is also the default. The following video from the laminar pipe flow module in our free online course explains the advantages of working in terms of gauge pressures. The video discusses this in the context of laminar pipe flow but the same ideas apply for our cylinder flow too.

Cylinder Wall

Solution Setup > Boundary Conditions > cylinderwall.
Set ***Type*** to ***wall***.This is the default.

Reference Values

Solution Setup > Reference Values.
Set the ***Density*** to 1 kg/m^3. The other default values will work for the purposes of this simulation.

Save Project

Go to Step 5: Numerical Solution

Go to all FLUENT Learning Modules