WIND TUNNEL 2D - V3.0
PROGRAMMED BY: AARON K. KNOLL
DATE: September 2010
==================================================

DESCRIPTION:

This program simulates a two dimensional wind tunnel for low Reynolds' number flows. The code uses the Semi-Lagrangian technique for solving the viscous incompressible Navier Stokes equations.

==================================================

GETTING STARTED:

The best way to learn is by trying. This simple tutorial is all you need to know to get started.

Step 1: Define your geometry.

This step is aimed at simplicity rather than geometric accuracy. All you need to do is create a JPEG picture of the object or domain you wish to analyze. All solid surfaces should be black and the fluid domain should be white. That's it! Check "wing.jpg" for an example.

Step 2: Run the program.

Now run the program "windtunnel2d.exe" (or windtunnel2d on a UNIX machine). Enter the following input when prompted:

>>Input JPEG image: wing.jpg
This sets the geometry file you wish to use.

>>Reynold's No.: 100
This tells the program what inlet velocity to use. The Reynolds' number is a non-dimensional measure of the flow speed given by V*L/Nu, where V is the velocity, L is the length reference (channel height in this program), and Nu is the kinematic viscosity. Check the following website for further details regarding the Reynold's number:

http://en.wikipedia.org/wiki/Reynolds_number

>>Kinematic viscosity [m^2/s]: 1.46e-5
This sets the kinematic viscosity of the fluid. Here we are using the viscosity of air at standard temperature and pressure. Here are some common kinematic viscosities:

Air = 1.46e-5
Water = 1.0e-6
Oil = 9e-4

>>Density [kg/m^3]: 1.184
This sets the density of the fluid. Here we are using the density of air. Here are some common material densities:

Air = 1.184
Water = 997
Oil = 920

>>Wind tunnel height [m]: 1
Here we set the wind tunnel height to 1 meter.

>>Wind tunnel length [m]: 2
Here we set the wind tunnel length to 2 meters.

>>Mesh spacing (0=coarse, 1=standard, 2=fine): 0
This is an especially important parameter that defines the mesh density used in the calculation. The finer the mesh spacing the more accurate the solution. However, a fine mesh will take much longer to calculate. Here is a rough estimate. On a decently fast computer, a coarse mesh will take 5 to 10 minutes, a standard mesh takes 1/2 hour to an hour, and a fine mesh will take a couple hours. So, try your calculation on a coarse mesh first. Then

move to the standard mesh or fine mesh when you are ready
for the final accurate solution.

>>Maximum number of iterations: 10000
This sets an upper limit on the number of iterations. It
should be used when you feel you are waiting too long for
a solution, and want to see how things are progressing.
For a complete simulation, set this parameter to a vary
large value so that the simulation completes before
reaching the maximum count. Try 100000 in most cases
to make sure.

Alright, that's it! Now the simulation will run to completion
and you can view your first results.

Step 3: Analyze the results.

The program will automatically output the lift and drag of
the object when the simulation completes. Also, new JPEG
pictures will be produced:

pressure.jpg: A picture of the pressure field.

velocity.jpg: A picture of the velocity field and streamlines.

U.jpg: A picture of the x-component of velocity.

V.jpg: A picture of the y-component of velocity.

Also, for detailed analysis, a text file containing the x-
component of velocity (U.dat), y-component of velocity
(V.dat), and pressure terms (P.dat) will be output.

==================================================

CONTACT: Aaron K. Knoll
e-mail: a.knoll@surrey.ac.uk