Instructions for MicroCFD ® 2D Virtual Wind Tunnel
Version 1.9 – Educational Edition © 2015 MicroCFD

NOTE: For ease of reference, please print out this document and follow its instructions step by step. Additional content, including video tutorials, is available online at www.microcfd.com/tutor.htm.

Optimizing the display:

1. Your display should be set to its maximum resolution and 32-bit color for optimum text and image rendering. Right click on your empty desktop and select Personalize and Display to verify highest resolution and color quality.
2. If your text appears small at maximum resolution, click Set custom text size (DPI) and increase your DPI setting from 96 to 120. The default *Microsoft Windows* setting is 96 DPI.
3. MicroCFD Virtual Wind Tunnel was designed for 120 DPI, and all its windows will be resized to 80% when run at 96 DPI. The 800*x*600 wind tunnel will only be displayed at 640*x*480 at 96 DPI, and the one-to-one mapping between flow cell and pixel will be lost.
4. A 96 DPI setting should only be used if the maximum screen resolution is less than 1280*x*960. This will ensure that the main window can be seen in its entirety on smaller displays.

Setting up a new test:

1. Select Clear from the File menu, if a previous test is loaded.
2. Set Tunnel Length, flow parameters, gas properties, simulation Time Step and Stop Time, or use the preset values. The Tunnel Length should be about four to five times the Model Length for slender shapes. For blunt shapes, the tunnel height should be about six times the model height.
3. Select Load Shape from the File menu to load a basic shape, an airfoil, or a custom designed shape. Shapes that extend beyond the tunnel dimensions are automatically scaled and centered upon loading, but can be resized and repositioned afterwards.
4. The Tunnel Coordinate System (TCS) is located in the lower left corner, with the x-axis pointing to the right and the y-axis pointing up. The height of the tunnel is 75% of its length. If the Tunnel Length is set to 8.00 (m), the height would be 6.00 (m), and any shape whose coordinates fall within $0 \le x \le 8$ and $0 \le y \le 6$ can be loaded directly, without scaling.
5. A shape that is not scaled and centered upon loading cannot later be modified. Proceed with Set Shape from the Edit menu, then load another shape, or continue with step 8.
6. For scaled and centered shapes, one can select Modify Shape from the Edit menu to move, resize, or rotate the currently loaded shape. For large step increments use the Page Up / Page Down keys, to maximize or minimize a modifier use the Home / End keys.
7. The position of a scaled shape is now measured from the tunnel center to the center of the shape. The shape center point is the arithmetic mean of its maximum and minimum points in *x* and *y*. When finished, click Done and select Set Shape from the Edit menu.
8. Place a blue seed pixel within the interior of your set shape through “point and click”. Select Fill Shape from the Edit menu to fill the interior of the shape. Repeat if not completely filled.
9. To set up a multi-element configuration repeat steps 3 through 8. Although one can build a composite shape through merging of individual shapes, shape overlap should be avoided.
10. Select Create Boundary from the Edit menu. For multi-element configurations, all shapes have to be loaded, set, and filled before the boundaries can be created.
11. To enhance the boundary layer effect, select Surface Model from the Flow menu and check Rough. Otherwise leave the surface boundary Smooth.
12. For 2D planar flow verify that Axisymmetry from the Flow menu is <u>not</u> selected. By checking Ground Effect, the tunnel bottom will act as a ground plane, which also disables axisymmetry.
13. Select Axisymmetry only for bodies of revolution. When checked, the tunnel bottom is the symmetry axis for the revolved shape. Axisymmetry selection disables ground effect.
14. For a vertically repeating 2D geometry, such as an airfoil cascade, select Cascade Loop to connect the upper and lower tunnel boundaries. Assure consistent spacing across boundaries.
15. Select Stability from the Run menu and set it to Medium or High if the shape is blunt or the Flow Mach Number is below 0.75. Increased Stability requires longer computation times.
16. Select Save Setup from the File menu to save all your preliminary work.

Choosing a proper Time Step and Stop Time:

1. The actual integration time step is computed internally based on the CFL condition of the flow and the selected Stability in the Run menu. Only the simulation time displayed in the pop-up clock advances in actual integration time steps.
2. The Time Step selected by the user should be a large multiple of the actual integration time step. It specifies the time elapsed between graphical updates of the flow color maps.
3. A lower-bound estimate for a Time Step dt is one-tenth the ratio of Tunnel Length L and flow speed V, $dt \sim 0.1\ L / V$, or the time it takes for the air to advance 10% through the wind tunnel.
4. The flow speed V is given by $V = M \cdot a$, where M is the flow Mach number, and a is the free stream speed of sound. For air at room temperature, $k = 1.4$, $R = 287$J/kg·K, and $T = 288$K, the free-stream speed of sound has the well-known value of $a = \text{SQR}(k \cdot R \cdot T) = 340$m/s.
5. For $M = 0.8$ and $L = 8.0$m, $V = 272$m/s, and $dt \sim 0.0029$s = 2.9ms, thus any Time Step between three and six milliseconds would be appropriate to show the flow development.
6. In the transonic regime ($M = 0.8$ to 1.2), the flow reaches steady state after the tunnel has purged itself three times. A sufficient Stop Time would be $T \sim 3 \cdot L / V$, thus $T \sim 0.090$s in this case.
7. For subsonic ($M < 0.8$) and supersonic flow ($M > 1.2$), steady state is reached after the tunnel has purged itself twice. Overall computation times decrease with increasing Mach number.

Running a new or saved test:

1. Before running a test, select Automatically… from the Run menu and specify which tasks, if any, you want to be performed every time the color maps are updated (each Time Step):
   • Check Save Slides if you only want to save the color maps and flow parameters.
   • Check Save Frames if you want to save all flow data (equivalent to Save Test).
   • Check Print Frames, if you want all output sent to a printer in Landscape Orientation.
2. Select the Processor which best fits your hardware and computational needs. GPU processing is only available if a CUDA enabled graphics card from NVIDIA ® is installed in your computer.
3. To start a newly set up test, click Start Test from the Run menu. When processing on the CPU, a pop-up clock at the top of the screen displays the current simulation Run Time and CPU Time.
4. To start a previously saved test, select Load Test from the File menu and select a test to be loaded (ensure that the filename carries the ".tst" extension if typed in manually).
5. If the test was completed (Run Time = Stop Time), increase the simulation Stop Time and, if desired, modify the Time Step. Select Start Test.
6. While a test is running, both the main window and the pop-up clock, if shown, can be minimized to perform other tasks. The pop-up clock is not shown when processing on the GPU.
7. A test in progress can be stopped by selecting Stop Test from the Run menu. In order to continue, select Start Test from the Run menu again. The color map is updated every Time Step.
8. The pop-up clock and main window may appear to freeze occasionally, depending on other operating system tasks. Upon completion of the test, the main window will return to normal.
9. When the computation is finished, select Save Test from the File menu to save the result. If steady state has not yet been reached (color maps are still changing), go back to step 5.

NOTE: Screen Savers use significant CPU / GPU time while the program is computing and should be disabled! Select an automatic shut-off time for your display (15-30 minutes) or turn it off manually.

Interpreting the results:

1. After a test is completed or has been manually stopped, the color map can be changed to a different flow property. Select Property from the View menu and choose Mach, Density, Pressure, or Temperature from the submenu. Streamlines can also be plotted.
2. All color maps show a non-dimensional scale. The absolute values of local pressure, density, and temperature have been divided by their free stream values. To obtain the full range of absolute values, in their respective units, select Statistics from the View menu.
3. Also available in the Statistics window is data on lift, drag, and pitching moment. For 2D flow, all forces and moments, whether dimensional or non-dimensional, are per unit depth. For example, lift and drag are given in units of force per unit depth (N/m).

4. All 2D aerodynamic coefficients are based on the total length of the model. For a model length L and a free stream dynamic pressure $q = ½\ Rh \cdot V^2$, lift, drag, and pitching moment are non-dimensionalized as follows: $Clift = Lift / (q \cdot L)$, $Cdrag = Drag / (q \cdot L)$, $Cpitch = Pitch / (q \cdot L^2)$.
5. The pitching moment is computed with respect to the Aerodynamic Coordinate System (ACS) located at the tunnel center. A positive pitching moment acts counterclockwise. The line of action of the resultant aerodynamic force is given by the equation, $x \cdot Lift - y \cdot Drag = Pitch$, and can be plotted by selecting Force Line from the View menu.
6. For axisymmetric flow, lift and pitching moment are identically zero, the TCS and ACS coincide, and the Force Line runs along the symmetry axis. The drag is 3D and is non-dimensionalized by the free stream dynamic pressure, $q = ½\ Rh \cdot V^2$, and by the frontal area of the model, $A = pi \cdot R^2$, where R is the largest model radius, measured from the symmetry axis.

Creating a custom shape file:

1. Shapes are saved in ANSI text format as a sequence of points connected in a closed loop. The following will describe how to create your own shape file using a simple text editor such as *Microsoft Notepad*. (A word processor in text mode is even more suitable, since it can also display hidden characters such as tabs and return keys).
2. After starting *Microsoft Notepad*, Select Open… from the File menu.
3. Change the current folder to …\MicroCFD\Shapes\Basic\…
4. Select Files of type: All Files (*.*) in the drop-down box.
5. Select the file Square.shp, and click Open.
6. You should see the following on your *Notepad*:
   • The first row simply contains the string "shp", which is the shape file identifier.
   • The second row should read "4", which is the number of points to follow, starting at 0.
   • The next five rows contain the data points, with *x* and *y* values separated by tabs.
   • Points 0 through 3 describe a square; point 4 is identical to point 0.
7. To define your own shape, simply modify the second row (*N* number of points), followed by the *x* and *y* coordinates of points 0 through *N*, with 0 and *N* being identical (closing the loop).
8. When done, select Save As… from the File menu and change the current folder back to …\MicroCFD\Shapes\Basic\…
9. Enter the file name Custom.shp. Select Save as type: Text Documents (*.txt) with Encoding: ANSI and Click Save. Older versions of *Notepad* have no encoding option but save in ANSI.
10. For multi-element configurations, separate shape files need to be created, which have to be loaded one at a time during setup. Try the nanoCAD plug-in for easy shape file creation.

NOTE: A decimal point must be represented by a period and not a comma. For example, one-tenth should be written as 0.1 and not 0,1. Otherwise your shape file may not load properly.

Defining custom colors:

1. With your display set at 24-bit color or higher, each RGB (Red, Green, Blue) component is represented by exactly one byte, which ensures proper rendering of the MicroCFD default colors. At lower settings, some of the default colors can only be presented through *dithering*, a process of mixing pixels of different colors from a limited (16-bit) color palette.
2. The color plots in MicroCFD Virtual Wind Tunnel will not display properly, if dithered colors are used. Either use a 24-bit color setting in your display properties, or change the colors to match the system palette. Another reason to change colors is to make them more distinguishable when printed. Sometimes two colors can be clearly differentiated on the display, yet they may look almost identical on paper.
3. To change colors, select Custom… from the Color menu and adjust each color separately by modifying its RGB components. When done, select Save Colors from the File menu to save your custom colors. Select Close from the File menu for the new colors to take effect. Closing the color window in its upper right corner will leave the current colors unchanged.
4. Although colors can be changed at any time, any previously saved slides remain fixed in their color composition and when reloaded will always display their original colors.
5. MicroCFD 2D Virtual Wind Tunnel will always start up with its default colors, and custom colors that were saved have to be reloaded each time the application is run.

Creating flow animations:

1. The development or periodicity of a flow can be viewed by turning a time sequence of slide images into an animated GIF file (*.gif) with third party software. The *GIF Movie Gear* from www.gamani.com is an excellent tool and can be used free on a trial basis.
2. Smooth animations contain a minimum of 100 slide images per tunnel purge time, thus the Time Step *dt*, based on Tunnel Length *L* and flow speed *V*, should be $dt \leq 0.01\ L / V$.
3. Prior to starting the test, select Automatically… from the Run menu and check Save Slides, which will save the color maps of Mach, Pressure, Density, and Temperature as bitmap files (*.bmp) into four separate subfolders within the application Slides folder (…\MicroCFD\Slides\...).
4. When the test is completed, open each of the subfolders with *GIF Movie Gear*, or similar software, and select all the bitmap files. Click OK and the GIF animation will be created.
5. The animated GIF file will be comparable in size to a single BMP file, even for 100 slides or more, due to the digital compression employed in the GIF file creation.

NOTE: If MicroCFD is installed in the Program Files folder, *Windows* may redirect any files that the application saves into its subfolders to a different location, which makes it difficult to retrieve such files with other applications, including *Windows Explorer*.