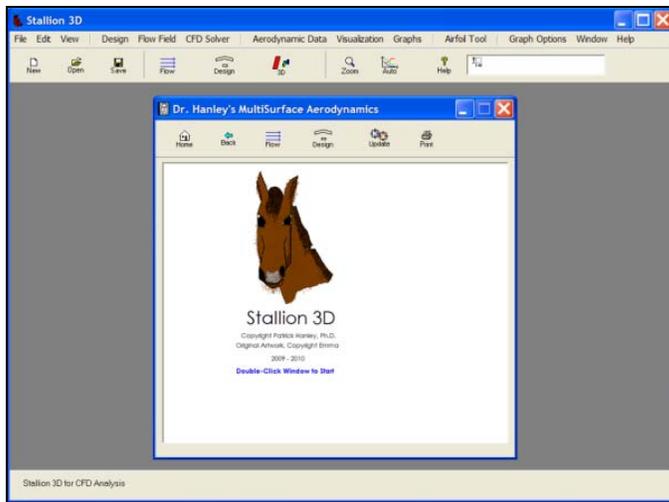


How to Enter and Analyze a Wing

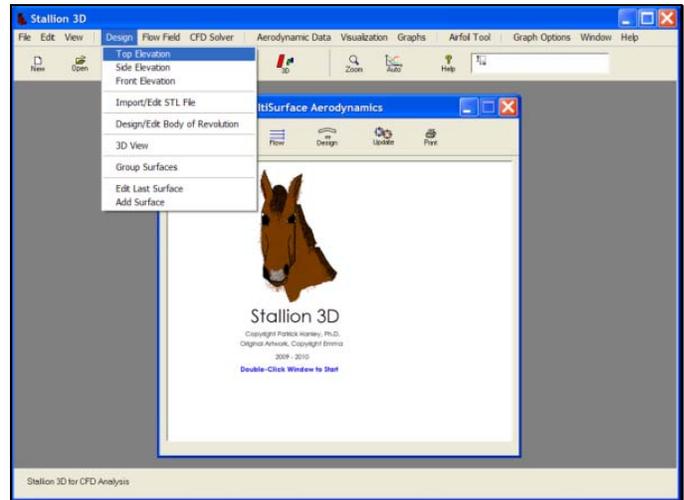
Entering the Wing

The Stallion 3-D built-in geometry creation tool can be used to model wings and bodies of revolution. In this example, a simple rectangular wing is modeled in the user interface. The wing span is three (3) meters and the chord is one meter in length.

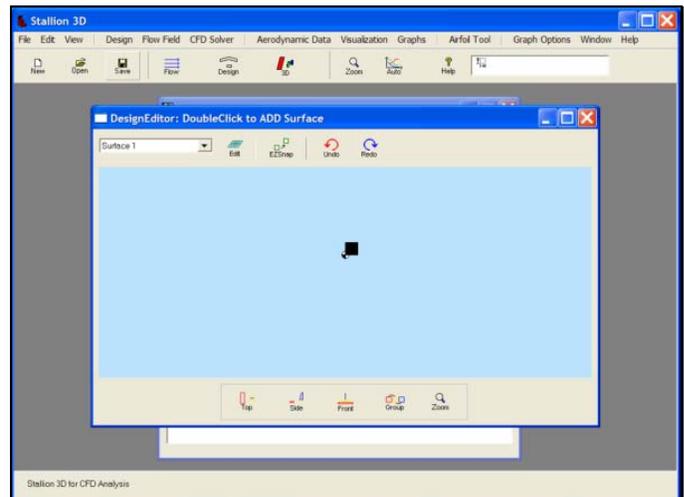
Please note that geometries can also be entered into Stallion 3D using .STL files created in a CAD software package.



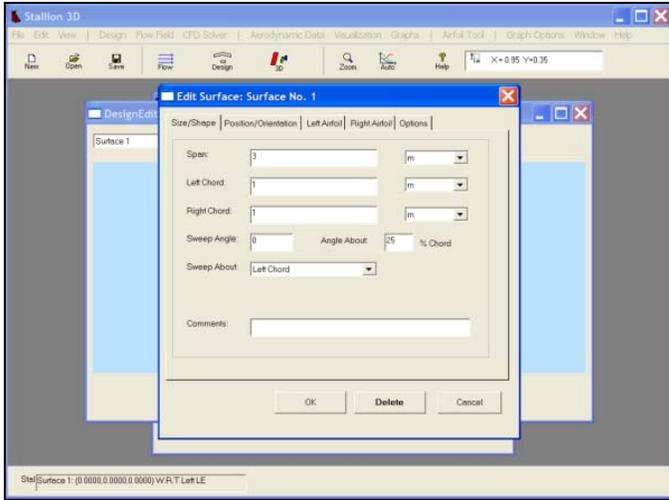
1. Start Stallion 3D and then click the Design menu followed by Top Elevation to invoke the DesignEditor screen showing top view of the default wing (square).



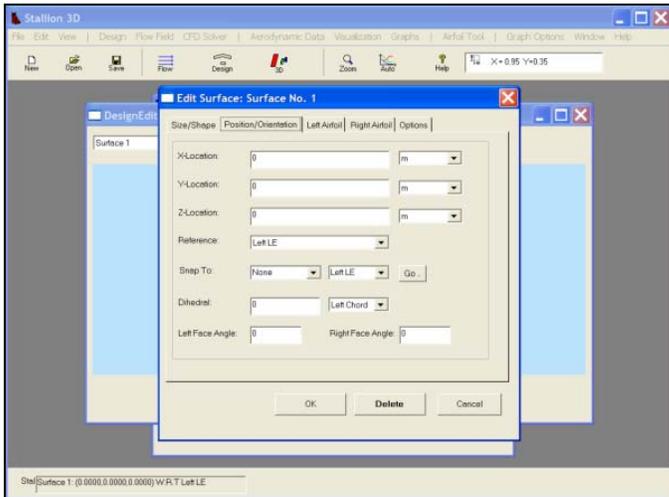
2. Click the Edit button on the DesignEditor menu to invoke the surface editor dialog box.



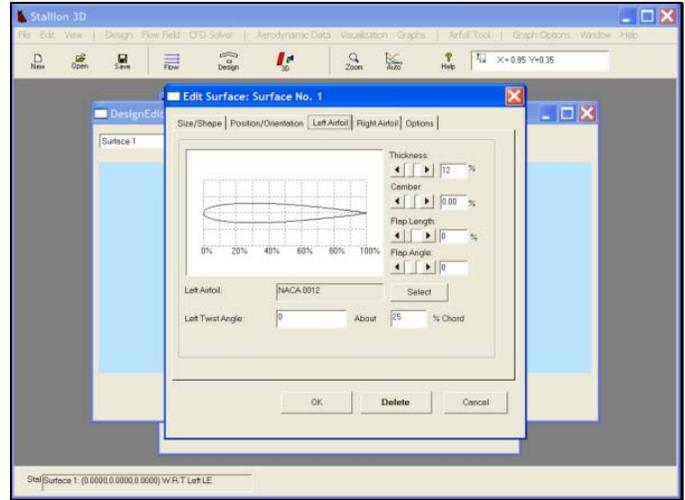
3. Under the Size/Shape tab, enter the size of the wing. Set Span to three meters and both the Left Chord and Right Chord width to one meter.



4. Under the Position/Orientation tab, set the location of the wing with respect to the left chord leading edge. The position should be $x=0$, $y=0$ and $z=0$. Set X-Location =0; Y-Location =0 and Z-Location=0.

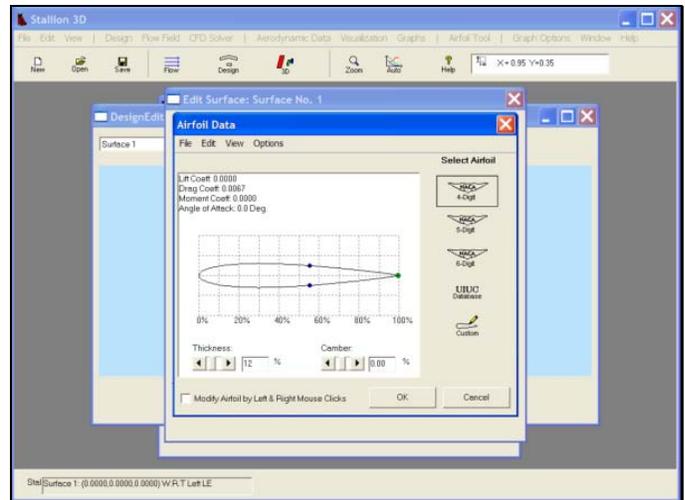


5. Click on the Left Airfoil tab and then click the Select button to enter the airfoil shape on the left side of the wing.

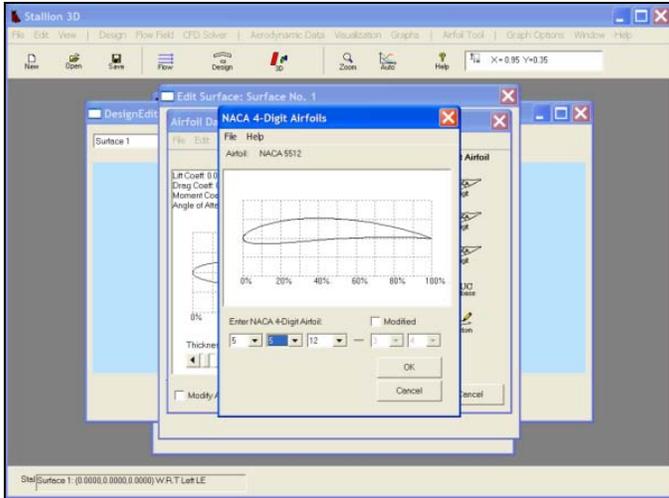


6. On the AirfoilData dialog box, click the NACA 4-Digit airfoil button to invoke the NACA 4-Digit Airfoils dialog box.

Note, there are other options available to enter airfoil shapes. For example, the dialog box below can also be used to enter custom airfoils (either coordinates or .DXF files), NACA 5 & 6 digit airfoils and airfoils from the UIUC airfoil database.

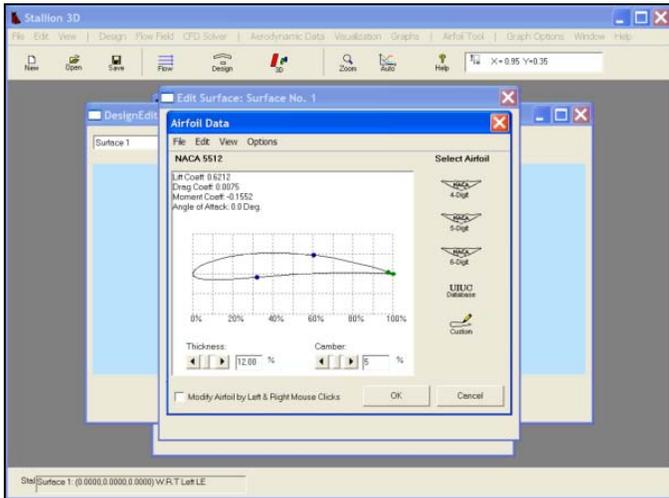


7. Use the NACA 4-digit dialog box to enter each number for the NACA 4-digit airfoil as shown below. In this case, the airfoil shape is the NACA 5512 airfoil.

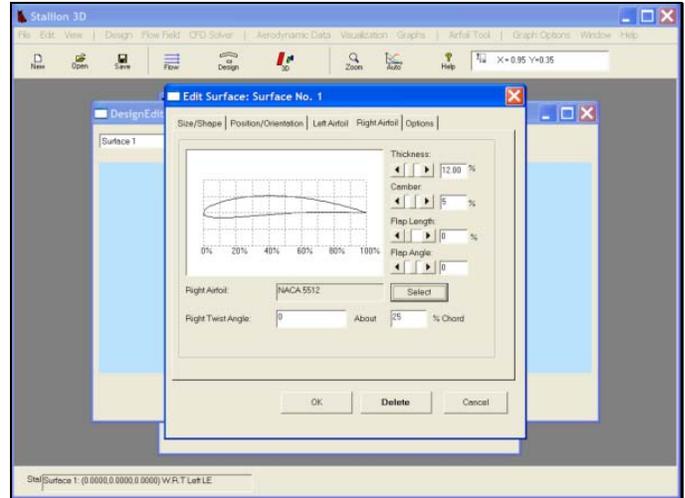


8. Click the OK button to accept the airfoil data.

9. The airfoil now appears in the Airfoil Data dialog box. Click the OK button to accept the new airfoil.

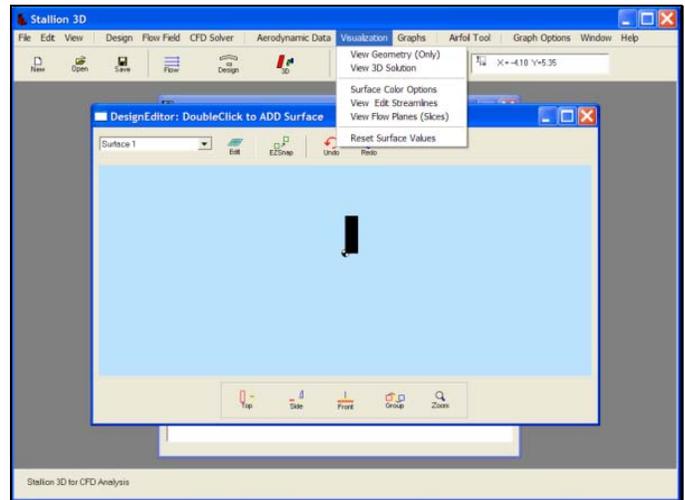


10. Click on the Right Airfoil tab and then accept the airfoil that is already in the Airfoil Data dialog box. This airfoil (the NACA 5512) was entered in the previous steps.



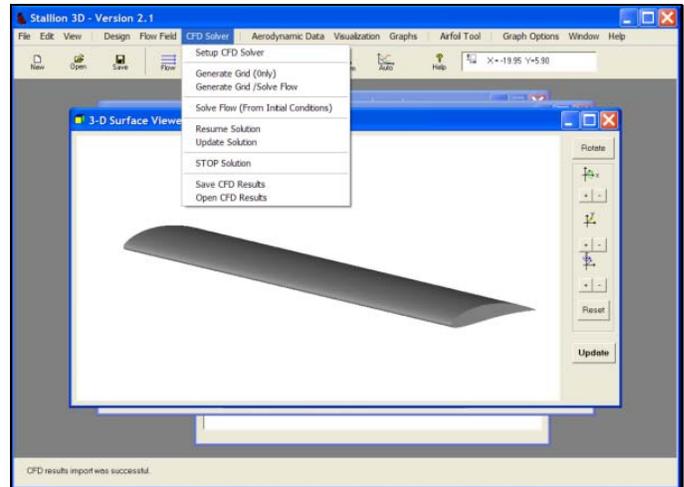
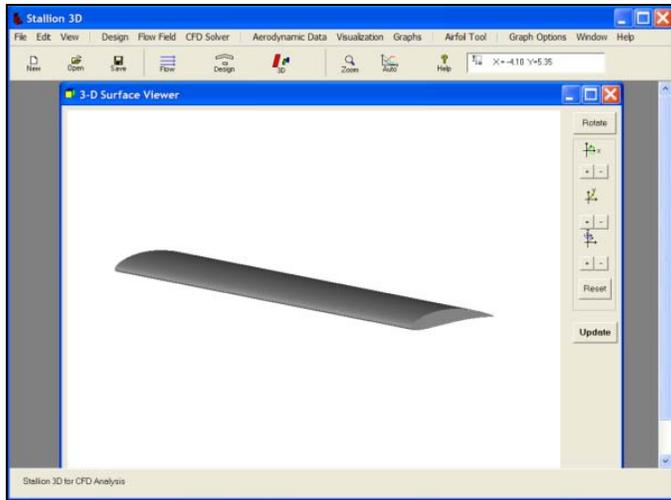
The airfoil now appears in the Right Airfoil tab window.

11. Click OK to exit the Edit Surface dialog box and return to the Design Editor window.



The Design Editor window now shows the wing element in the plan view.

Click on the Visualization menu and then click View Geometry (Only) to view the wing.



The Aerodynamics Analysis

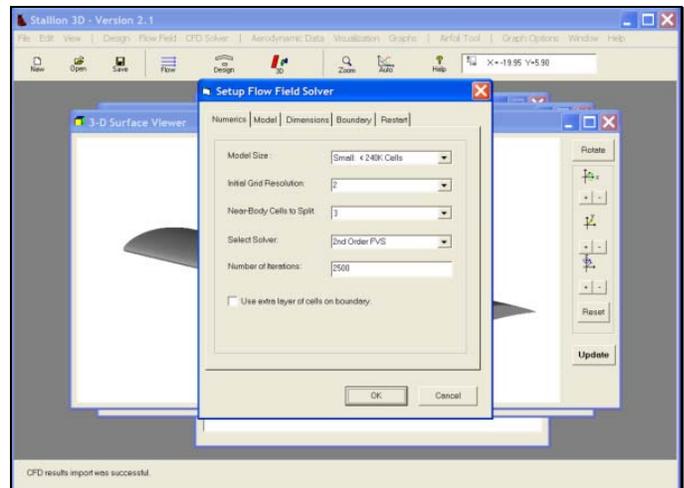
Stallion 3D can perform the analysis at various levels of accuracy. More accurate solutions require more available memory and longer computer times. It is recommended that the initial analysis use a moderate sized model to evaluate the problem before performing a final analysis with increased accuracy.

The following steps are used to select the model size, accuracy, physical model and boundary conditions.

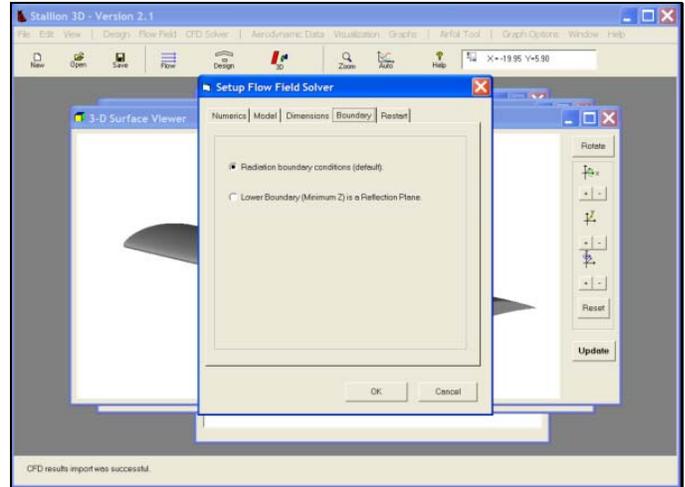
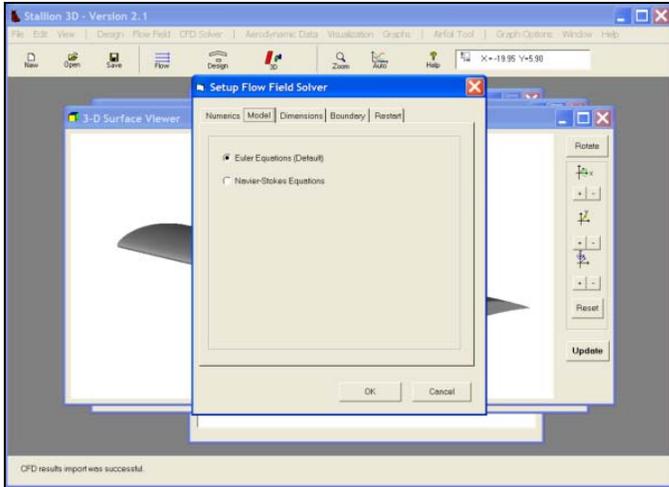
1. Click on the CFD Solver and then choose Setup CFD. This will invoke the Setup Flow Field Solver dialog box.

2. Under the Numerics tab, set the Small Model Size parameter to generate less than 240,000 cells. For this particular example, this setting generated about 200,000 computational cells.

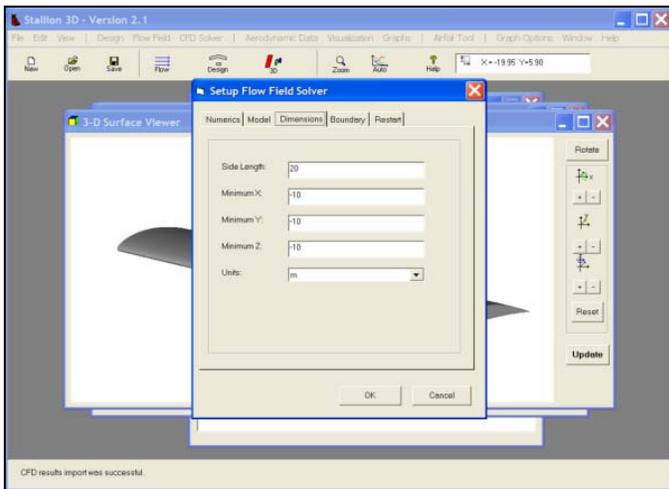
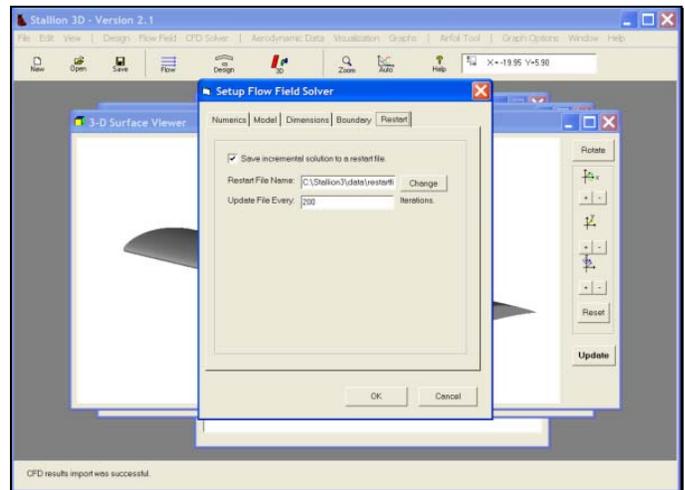
Set the Initial Grid Resolution to 2; the Near Body Cells parameter to 3; Select Solver to 2nd Order FVS. Set the Number of Iterations to 2500.



3. Under the Model Tab, select Euler Equations. The 3-dimensional compressible Euler equations are the default set of equations solve by Stallion 3D.

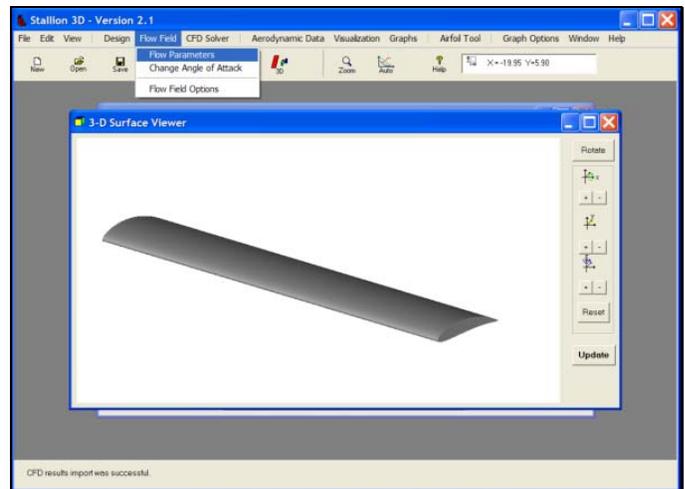


4. Under, the Dimensions tab, the boundaries of the flow field are established for the computational domain. Set Side Length parameter to 20. Set Minimum X to -10. Set Minimum Y to -10. Set Minimum Z to -10. Set Units to m (for meters). These settings enclose the wing with a computational domain that is a cube that has sides of 20 meters in length and the minimum corner is located at (-10, -10, -10).



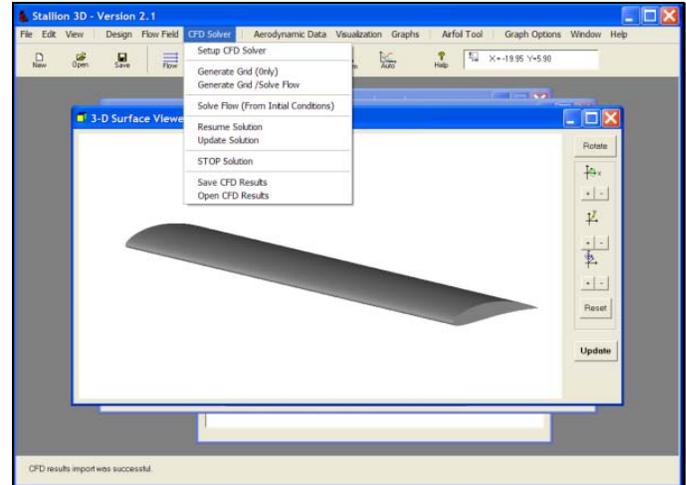
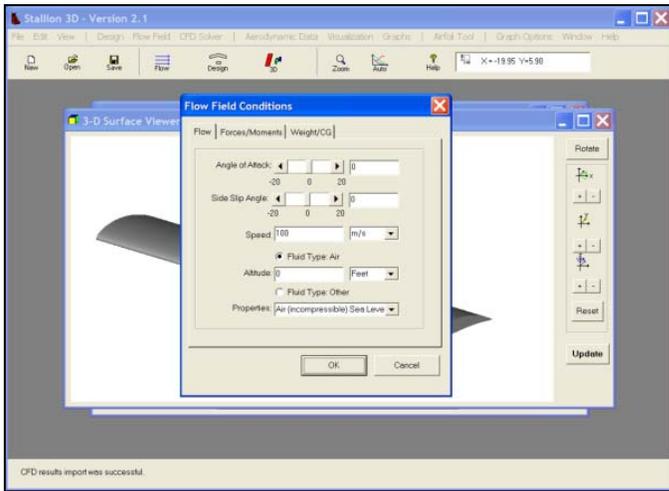
Setting the Flow Field

Click on the Flow Field menu and then select Flow Parameters.



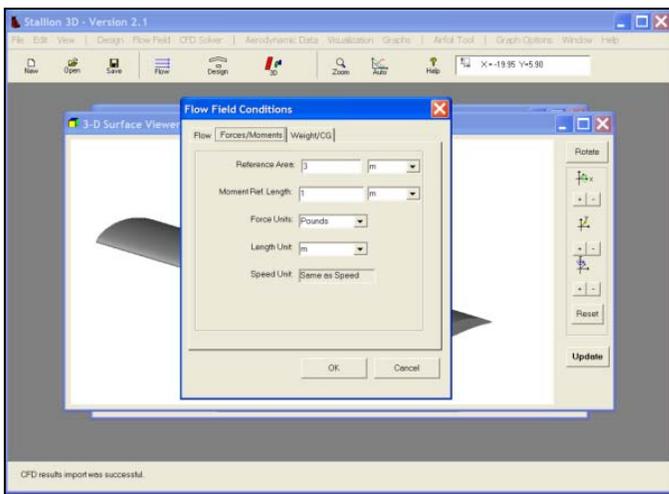
5. Under the boundary tab, click Radiation Boundary Condition to set the radiation boundaries on all six boundaries. This will simulate a wing in the free air.

Enter the angle, speed, and fluid type. For water, select Fluid Type: Other and then choose Sea Water 60 degrees F.



The Flow Solver will start and update Stallion 3D every 50 iterations.

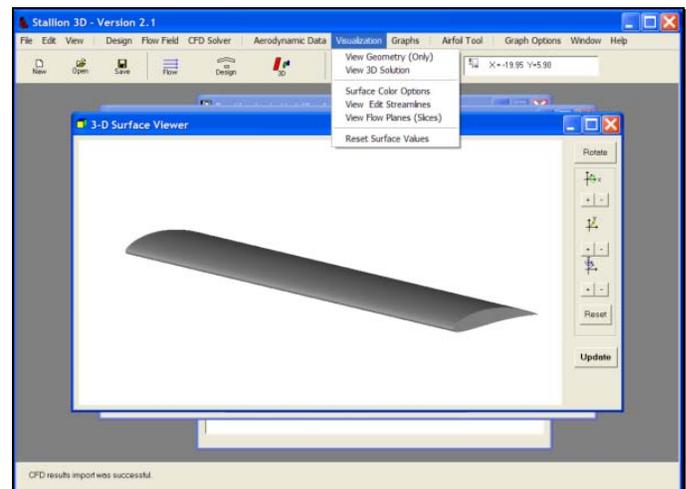
Under the Forces/Moment tab, enter the reference area of 3 square meters and the reference chord length of one meter.



Viewing the Results

After 50 iterations, the intermediate results will be ready for viewing. Please note that the solution will not be accurate until after a substantial amount of iterations (about 2000). However, it is still instructive to view the solution as the solver progresses.

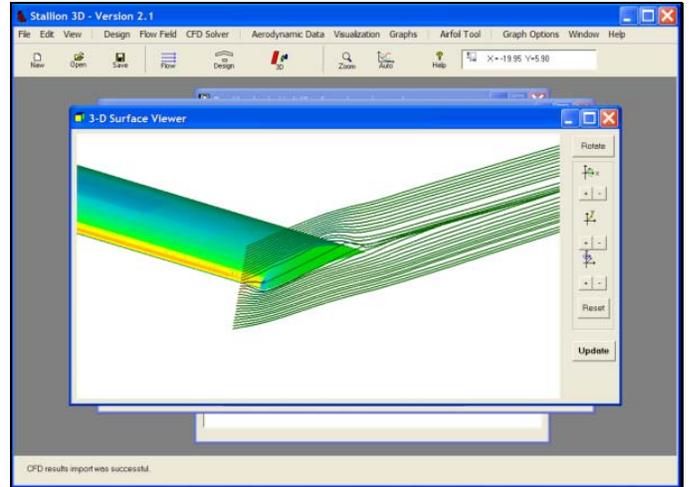
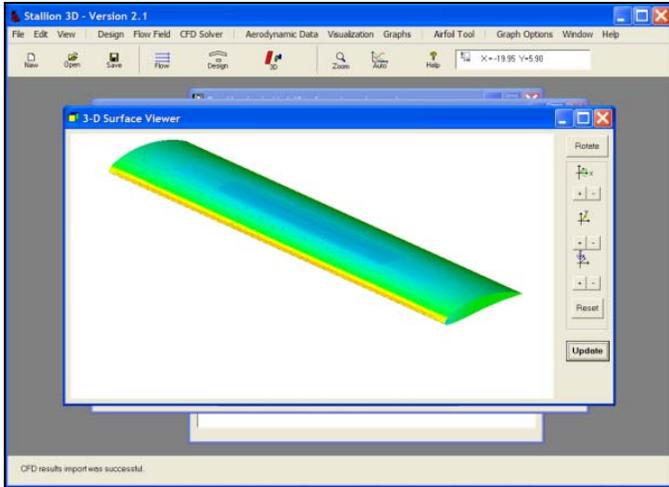
To view the solution, click on the Visualization menu and then choose View 3D solution. The 3D viewer will then appear as shown below.



The initial window shows the geometry colored according to the pressure values (red is high and blue is low).

Running the Analysis

To start the analysis, click on the CFD Solver tab and then choose Generate Grid/Solve flow to generate the grid and run the analysis.

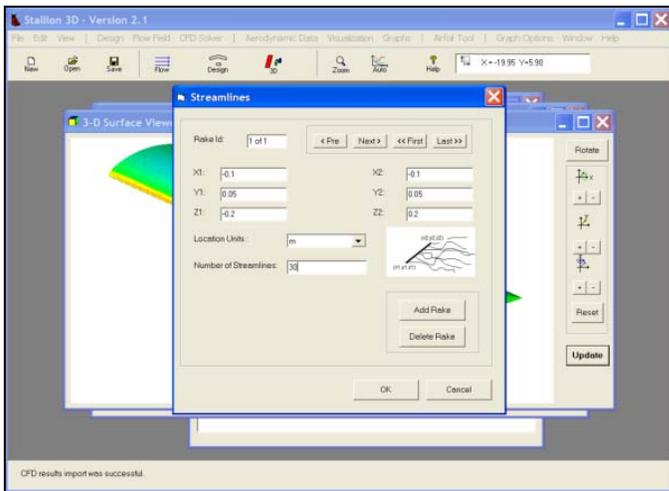


To change the streamlines, click on the Visualization menu and then choose View Edit Streamlines.

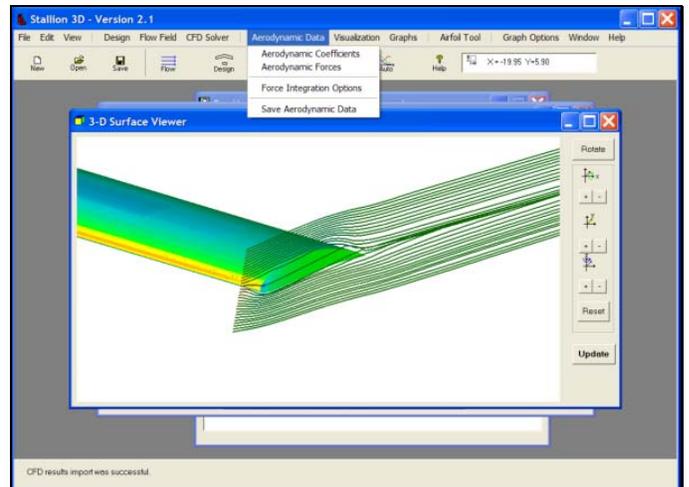
Computed Results

For this example, the fluid is air and the flow velocity is set to 100 m/s. The analysis is stopped after 2500 iterations.

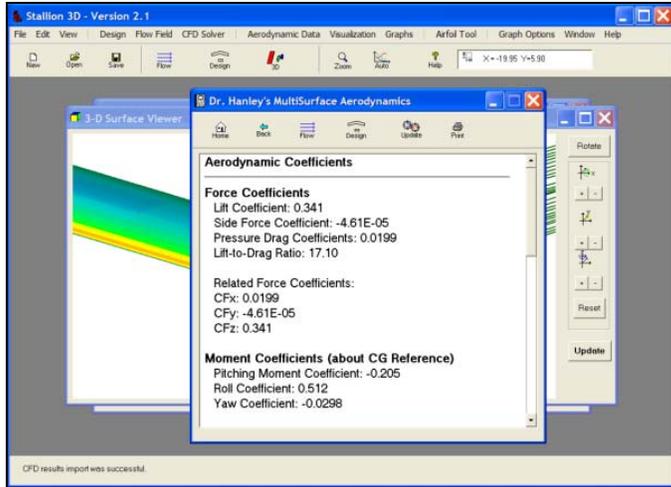
To view the results, click on the Aerodynamics Data menu and then click on the Aerodynamics Coefficients option as shown below.



Change the location of the endpoints to change the starting location of the streamlines as shown in the following screens.



The program will display the lift, drag and moment coefficients based on the user defined reference area and reference length.



Angle of Attack = 10 Degrees

	Stallion 3D	MSA
Cl	0.859	0.888
Cd	0.104	0.0891
L/D	8.260	9.966
Cm	-0.352	-0.355

The results of Stallion 3D and MultiSurface Aerodynamics are in good agreement. More accurate results can be obtained by choosing more nodes in the Stallion 3D simulations. This will improve the results for low angles of attack where the pressure drag is small.

The computations are repeated for angles attack of zero, five, and ten degrees. The results are compared to the results of MultiSurface Aerodynamics (MSA) and tabulated below. MultiSurface Aerodynamics is based on a vortex lattice method. It computes induced (vortex) drag in addition to profile (pressure & skin friction) drag. The pressure drag computed by Stallion 3D does not compute the skin friction component of drag.

Angle of Attack = 0 Degrees

	Stallion 3D	MSA
Cl	0.341	0.338
Cd	0.0199	0.0166
L/D	17.14	20.35
Cm	-0.205	-0.224

Angle of Attack = 5 Degrees

	Stallion 3D	MSA
Cl	0.608	0.616
Cd	0.0456	0.0444
L/D	13.33	13.87
Cm	-0.286	-0.290