

How Analyze a STL File

Stallion 3D accepts STL files saved in the ASCII format. Ideally, we recommend files that contain about 20,000 facets. Larger files will result in longer pre-processing times.

1. Click on the Design menu and choose Import/Edit STL File (this will invoke the Import/Edit File dialog box).

2. Click on the file menu in the dialog box and click the Open option. This will invoke the Open dialog box from which you can search for your file and enter it into the program. Choose aero1.stl (for examples) as the file.

3. The wireframe .stl file will appear in the dialog box. Use the View menu to see the other elevations.

Notice that the file might be sideways. In Stallion 3D, the up/down direction is the Z direction.

4. To change the file orientation, click on the Position/Orientation tab and enter 90 into the Y-Z Plane Rotation (Deg.) box. This will rotate the object 90 degrees to place it in the proper orientation when it appears in the Stallion 3D editor.

5. Click on the Size/Scale tab to set the units of the stl file. Use this window to scale the object and set dimensions.

6. Click OK to exit the Import or Edit STL dialog box.

7. The wireframe picture of the object is now available in the DesignEditor window. If there is an extra surface in the DesignEditor window, double-click on this surface to invoke the Edit Surface dialog box. Next, click the delete button to erase the surface.

8. Next, we setup the analysis by clicking on the CFD Solver and Setup CFD Solver option.

9. Under the Numerics tab, please do not change the default values.

To import and analyze your geometry please do the following:

10. Under the Flow Model tab, please use the Euler Equations option. This will compute the lift, moments and pressure drag.

11. Under the dimensions tab, please do not change the default values of the minimum x, y, and z values.

12. Under the boundary conditions tab, please click to set all the boundaries as radiation boundary conditions.

13. Click OK to exit.

14. Next, please set the air/water speed by clicking on the Flow Field menu and the Flow Parameters option. This will invoke the Flow Field Conditions dialog box.

15. Please change the angle of attack to your desired angle of attack

16. Please set your desired speed.

17. If the fluid is low-speed air or water, please click the Fluid Type Other option and set properties of: Air (incompressible) Sea Level if your airplane speed is below $M=0.2$ or one of the water options. This will help the program to converge faster.

18. The next step is to generate the grid and run the program. This is done by clicking on the CFD Solver menu and Generate Grid/Solve Flow option. This will start the grid generation process. The solver will also start after the grid is generated.

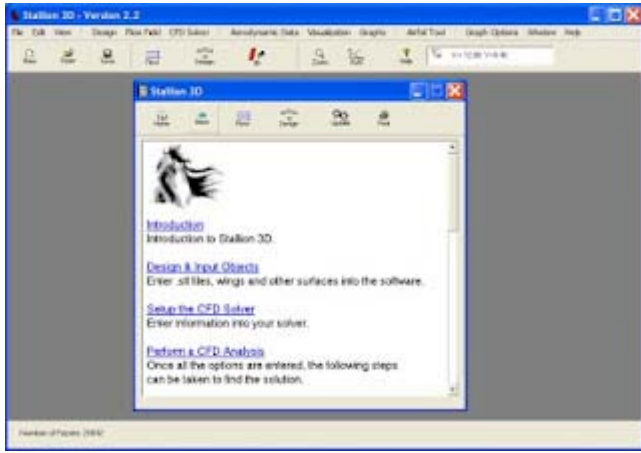
19. When the solver is running, you can click on the CFD Solver menu and the STOP solution option to pause the solver. This will allow you to view the solution and see how it is progressing. To view the data, please click on the Aerodynamics Data menu.

20. To resume the solution, please click on the CFD Solver menu and the Resume Solution option.

Car Analysis Example

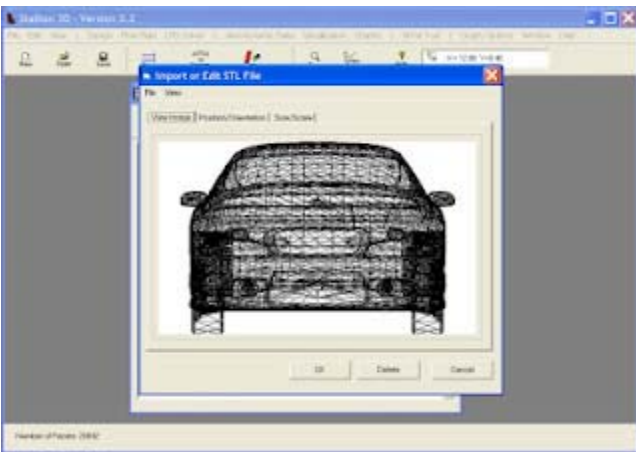
Often it is necessary to quickly determine the aerodynamics characteristics of a production vehicle or a new prototype before further testing or construction can move forward.

The following steps can be followed to determine the lift and drag on a road vehicle.



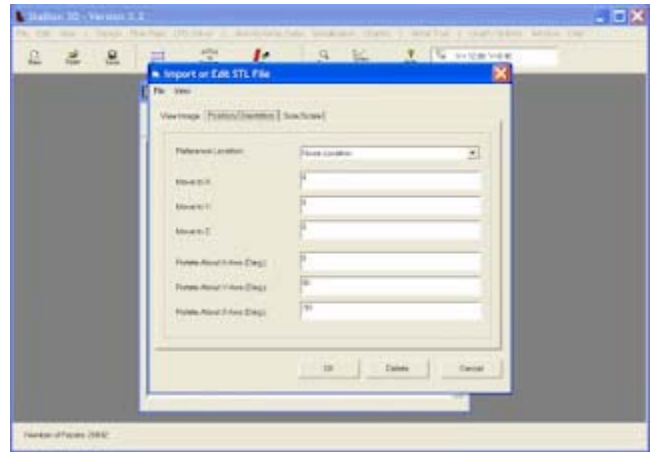
Stallion 3D Main Window

To start the analysis, click on the design menu and select the Import or Edit .STL file option. This will invoke the .stl file editor show below. The editor allows you to read in files in the ASCII format and then provide a quick preview of the input. Stallion 3D can analyze files that are water-tight. If your file has holes, you might be able to repair them with the free utility called [Netfabb-Studio Basic](#). Netfabb studio can also convert files from a variety of formats to the .stl format.



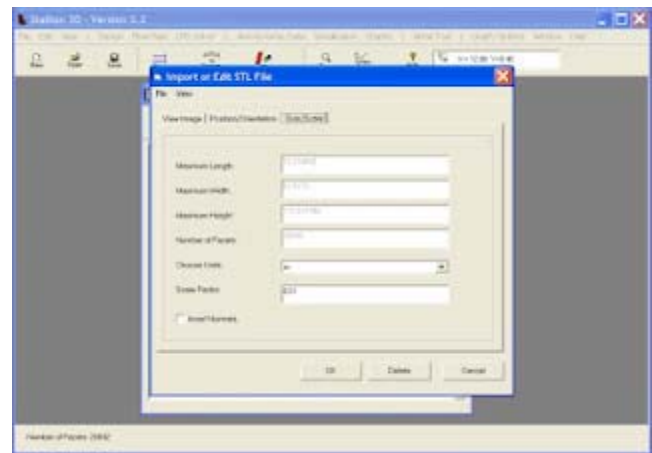
A car from Turbosquid.com as seen in Stallion 3D.

The .stl input box also allow you to rotate and translate the part. This allows you to point the leading edge of the model (longitudinal direction) along the x-axis and the lateral direction along the y-axis.



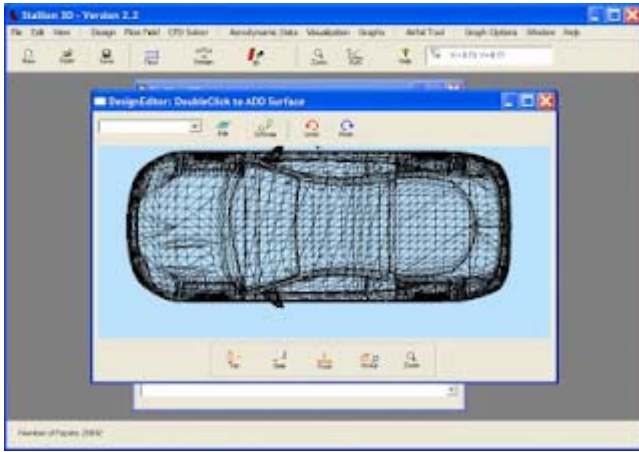
The .stl dialog box can be used to orient the model.

Before exiting the .stl import box, you can also explore the options to scale the model and set the correct dimensions. The default in Stallion 3D is meters, however, you can select inches, feet and cm.



Scale and dimension the solid model.

Once the part has been imported into Stallion 3D, it can be view in wire frame mode in the Design Editor. This screen can also be used to add more components (such as wings) to the model.



The design editor window can be used to add wings and other components to the model.

The next step is to analyze the model. Stallion 3D performs a 3-dimensional CFD analysis on an ordinary PC running MS Windows. The user can choose the model size and settings by clicking the CFD Solver menu followed by the Setup CFD option. In the first tab, the user can select the CFD problem size (number of nodes) and layout. The options are for a quick run (less accurate) or a moderately sized problem (more accurate). In this problem, I choose a Small Size (less than 240,000) nodes.



The user can choose the problem size. A quick run

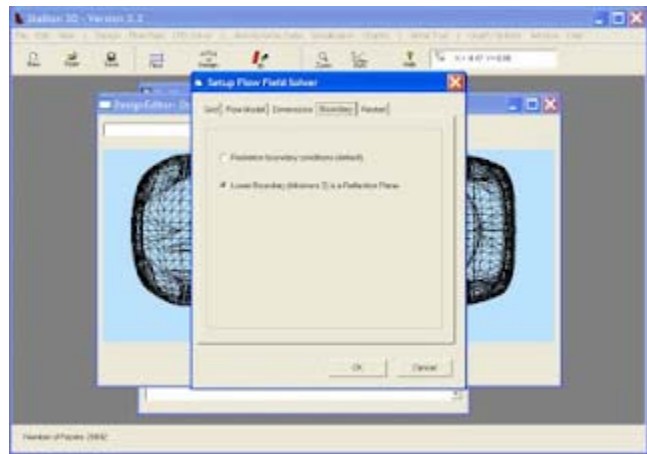
is helpful to test the setup.

The next option is to choose the flow domain dimensions and location. The flow domain is a big cube. I recommend that the length of the flow domain is about 10 times the maximum length of the part. This allows quicker convergence and better accuracy for the radiation boundary conditions.



Set the flow domain dimensions to simulate the physics and facilitate convergence.

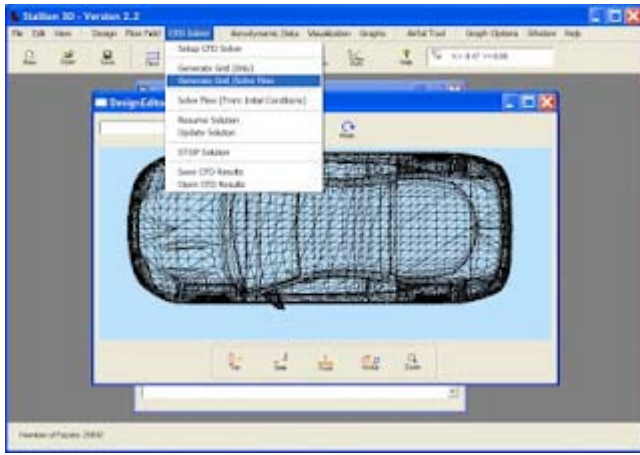
For the car problem, I select the lower z-dimension to coincide with the ground and then make the ground a reflection plane (instead of a radiating boundary). This will simulate the hard road.



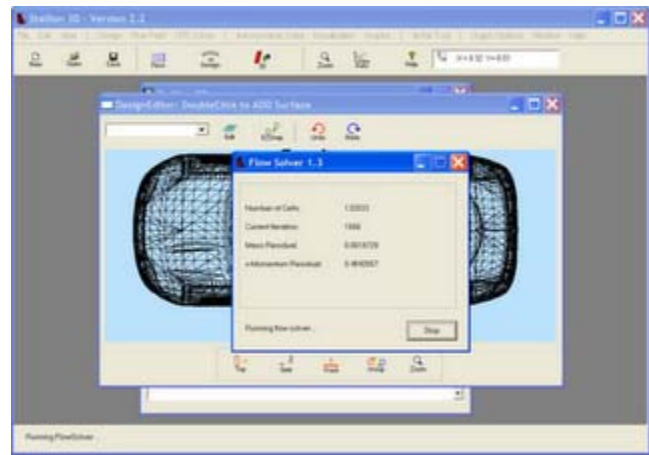
Set the lower boundary as reflective (instead of radiative) to simulate the road.

The next step in the solution is to set the flow speed and reference areas and lengths for the lift, drag and moment computations.

Finally, simply click the CFD Solver menu and select Generate Grid/Solve flow. The program will automatically generate the grid and launch the flow solver. For this specific problem, the grid generation took about 20 - 30 minutes on my laptop. The algorithm generated 132,833 nodes.

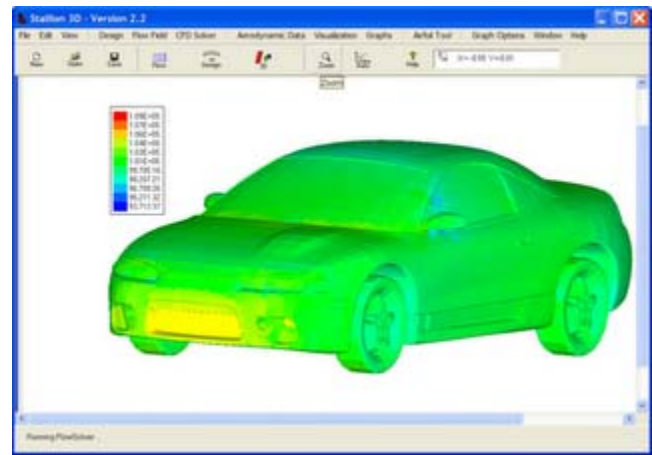


Once the grid was generated, the flow solver started to solve the 3-dimensional compressible Euler equations (with the low speed flow option). The solver computes the lift, moments and pressure drag for the 3-dimensional configuration with good accuracy. Stallion 3D can also solve the 3-D Navier-Stokes equations for the same configuration.



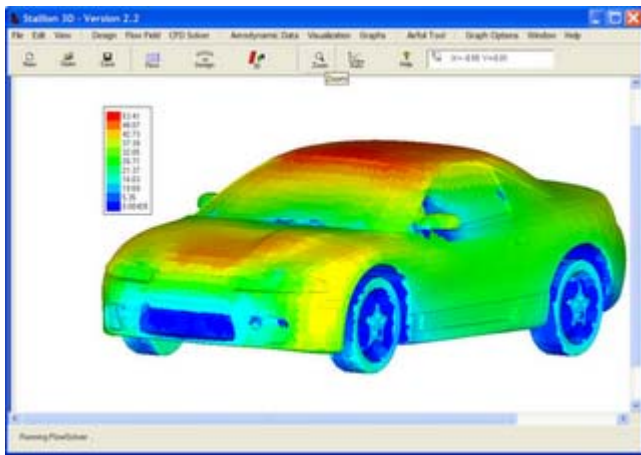
The flow solver updates 132,833 nodes every iteration using a 4-stage Runge-Kutta method.

The solution converges in less than 2 hours for this problem on my laptop computer. The window below shows the pressure computed on the surface of the car. This is obtained by clicking on the Visualization menu and selecting the View Solution option.



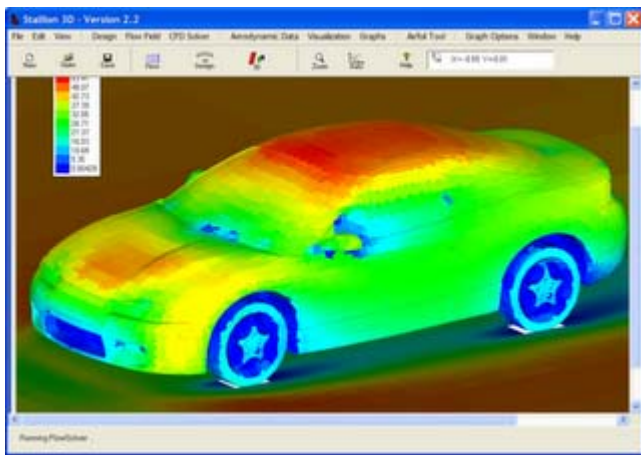
Pressure on the surface of the car.

The speed near the surface of the car is a useful design parameter for drag reduction and flow visualization. Stallion 3D allows the user to see the pressure, Mach number, velocity and temperature on the surface of the part.

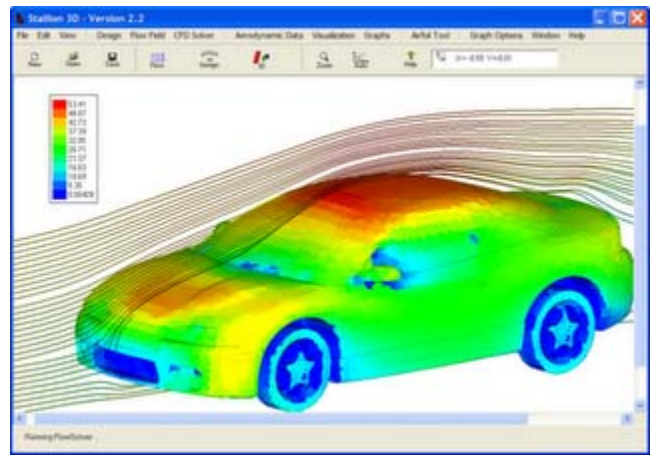


Velocity on the surface of the car.

Another technique to assess the quality of the solution is to look at the streamlines and flow at the ground plane. The following graphs show these two images.

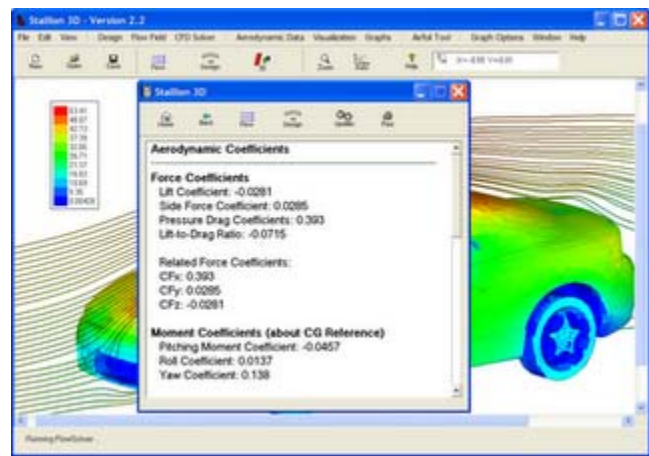


Velocity at the ground plane.



Flow streamlines.

Stallion 3D computes the overall lift, drag and moments acting on the vehicle. These numbers can be used as design parameters to improve and determine the performance of the vehicle. The moments can also be used to determine stability with subsequent computations.



Computed lift, drag and moment coefficients based on vehicle frontal area.

Note, the .stl file must be in very good shape for good computation of the forces and moments. It must not have multiple facets (in the same location) and inverted facets. If this is the case, the user can approximate the forces on the Cartesian-front (an approximate representative of the car). This method, however, is not valid for downforce calculation. A good .stl file is required for downforce calculation when the lower boundary is used as a solid wall.