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.

Stallion 3	D								
File Edit Vier	v Design	Flow Field	CFD Solver	Aerodynamic Da	ata Visualizati	on Graphs	Airfol Too	of Graph Options	Window Help
D New Of	en Save	Flow	Design	1	Zoon	Auto	Pep 1		
				57.6	1				
	Desig	nEditor: I	DoubleClick (to ADD Surface	•				
					0 0				
	[Surface]		East	EZSnap	Undo Redo				
			Q.	- <u>4</u>	1 Front	Group	Q. Zoon		
				1		14	<u> </u>	_	
Stallion 3D for	CFD Analysis								

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.

Edit Sur	
DesignEdit	face: Surface No. 1
Surface 1	Position/Orientation Left Airfoil Plight Airfoil Options
Spen:	3 m 💌
Let Char	d 1 m
Right Ch	ord 1
Sweep A	Mg/e 0 Angle About 25 to Chard
Sween A	hout la choud
Commen	
	OK Delete Cancel

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.

New Open Save Flow Design 30 Zoon Auto Help 14 X*0.95 1*0.35
Edit Surface: Surface No. 1 Ser/Bage Poston/Coleman Ser/Bage Poston/Coleman Watcosin Paternoce Latt Actol Paternoce Latt Actol Paternoce Latt Actol Diaded: 0 Latt Actol Diaded: 0 Latt Face Angle: 0K Delete

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.

1	-	Edit Surface: S	urface No. 1			X	
	DesignEdit	Size/Shape Position	n/Orientation	Niffoil RightAirloi	I Options nickness:	-s	
		0% 20%	40% 60% I	80% 100% p	lap Length Dap Angle Select	- N	
		Left Twist Angle	Ø	About 7	5 % Cho	rd	
			OK	Del	ete	Cancel	

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.

New Open	Sava Fi	low Design "30 Zoon Auto"	Hep	
	DesignEdit	Edit Surface: Surface No. 1		
	Surface 1	File Edit View Options		
			Select Airfoil	
		Lift Coeff: 0.0000 Drag Coeff: 0.0067 Moment Coeff: 0.0000	40gr	
		Angle of Attack: 0.0 Deg.	THE A	
			SOM BOOM	
			UIUC Database	
		0% 20% 40% 60% 80% 100%	Custon	
		Camber:		
		Modify Airfoil by Left & Right Mouse Clicks OK	Cancel	

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.

	đ	Edit Surf	ace: Surface No. 1	×	L
	esignEdit	Airfoil Da	NACA 4-Digit Airfolls		
Surte	nce 1	File Edit	File Help		
	_		Antor NACA 5512	Airfoil	
		Drag Coett (4	
		Angle of Atte		a7	
				and a second sec	
				a a	
			0% 20% 40% 60% 80% 100	N	
			RANN HANKET CHORE O LEVYONG WEDE DON	base	
		0%	Enter NACA 4-Digit Airtoit	2	
		Thicknes	5 • 5 • 12 • - 1 • 4 •	1	
		112	OK		
		- Madha	Cancel	Descrit	
		- Mudiy			

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

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

Image: Construction Image: Construction	Stallion 3D No Edit View Design Flow I	ield CPD Solver Aerodynamic Data: Vacalization Graphs	Arfoli Tool Graph Options Window Help
Angle of Antock 0.0 Dirg Angle of Antock	Cot Vew Design Rev 1	And COD Solve Anodynamic Data Vavakation (Graph Comp. Col. 2008) Comp. Col. 2008 Comp. Col. 2008 Col. 2008	Artis Tool Crack Option: Window High

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.

No Edit View Design Plow	Teld CHD Scher Aerodynamic Data Waudkaston Graphs Arfol Tool Graph Options Window Halp
Nie Opin Site	First Config Config <thconfig< th=""> Config <thconfig< th=""> <thconfig< th=""> <thconfig< th=""></thconfig<></thconfig<></thconfig<></thconfig<>
ShaljSurface 1: (0.0000.0.0000.0.0000	OK Delete Cancel

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.



Sette (CD Solver Generate Grid Solver Solve Flow Solve Flow So	ie Edit	View	Design	Flow Field	CFD Solver	Aerodynamic Data	Visualization	Graphs	Airfol	Tool	Graph Opti	ons Windo	w Help
Nor Union Save The Generate Gold (Solve) Generate Gold (Solve Perv Solve Perv (From Intel Conditions) Perume Sakkan Update Sakkan Save CPD Results Open CPD Results Open CPD Results Update	0			1 =	Setup CFD	Solver	1 1	c.	9	1	X=-19.95 Y=5.9	0	
Sole Flow (From Intel Candoma) Resure Solton Patter Resure Solton Patter Patter Patter Patter P	New	Open	Save	Flow	Generate (Grid (Only) Grid /Solve Flow	• 2	ulio"	Hep				
B-D Surface Viewe Resume Solution Update Solution STOP Solution STOP Solution Store Problem Some GPD Results Open GPD Results		6			Solve Flow	(From Initial Conditions)	- 11					
STOP Solution Save CPD Results Open CFD Results T T T T Result Result Update			-D Surfa	ce Viewe	Resume Si Update So	olution Iution						Bo	tate
Save CFD Results Open CFD Results					STOP Solu	tion							
7/ 7/ 7/ 7/ 7/ 7/ 7/ 7/ 7/ 7/					Save CFD	Results						+	×.
Update						10.3003						4	4
			4									14 14 14 14 18	
												1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	- - - - ssot
												Here Upp	-

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.

open save Pow	Desilo 30	Zoone A440			_
1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1	 Setup Flow Fleid Solv 	/er			
3-D Surface Viewer	Numerics Model Dimensio	ns Boundary Restart			- 🗆 🗙
	Model Size :	Small: < 240K Cells	•		Rotate
	Initial Grid Resolution:	2	•] ⇔×
	Near-Body Cells to Split	3	-		•••
	Select Solver.	2nd Order FVS	-		4
	Number of Resistons:	2500	_		1
		and the second			• -
	Use extra layer of cells	on boundary.			Reset
			· · · · · · · · · · · · · · · ·		Update
		ОК	Cancel		
				_	

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).



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.





Setting the Flow Field

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



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



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



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.



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

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).





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



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

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.



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



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
CI	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
CI	0.608	0.616
Cd	0.0456	0.0444
L/D	13.33	13.87
Cm	-0.286	-0.290

Angle of A	ttack = 10	Degrees
------------	------------	---------

<u> </u>		
	Stallion 3D	MSA
CI	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.