Case Study

Delta Wing at Low Subsonic Speed

Prof. Dr. Philipp Epple, Coburg University of Applied Sciences

Edited by the Ansys Academic Development Team

education@ansys.com

Summary

Delta wings are designed for supersonic speeds. However, aircrafts with delta wings must take off and land at subsonic speeds. At subsonic speeds a pair of vortices form above the delta wing generating the so-called vortex lift, which adds to the potential flow lift. This case study demonstrates how the aerodynamic characteristics of a delta wing at subsonic speeds can be investigated using Ansys Discovery, including parametric geometry generation, meshing, simulation for the relevant range of angles of attack, and post-processing. The delta wing vortices are visualized and compared with experimental images [1] and the aerodynamic characteristics are compared with data of the literature [2], displaying a good qualitative and quantitative agreement, respectively.

Table of Contents

1. Introduction

The aim of this study is to investigate the aerodynamic characteristics of a delta wing at subsonic velocities, *i.e.* at Mach numbers smaller than one (in this case M=0.082<1), which is basically the condition at takeoff and at landing. At these velocities a high angle of attack is needed to generate strong vortices above the delta wing and in this way enough lift at low subsonic velocities to accomplish takeoff and landing. At takeoff and landing the angle of attack of the supersonic airliner Aérospatiale/BAC Concorde was about 18°, see Figure 1, as compared to about 10° to 12° of subsonic passenger aircraft. The Concorde could maintain supersonic cruise velocities of two times the speed of sound, *i.e.* Mach 2.0. At supersonic speeds, besides pressure and friction drag, also wave drag occurs. To reduce the wave drag as much as possible the wings at supersonic speeds have to be very thin and the leading edge swept back, as can be seen in the pictures of the Concorde in Figure 1.

Figure 1: Concorde at the Technik Museum Sinsheim in Germany (Pictures: © Philipp Epple), shown in the (a) side and (b) front view.

2. Method

In this study the lift and drag characteristics of a delta wing over the range of angles of attack from 0° to 40° will be investigated. To be able to compare the results with the literature, the results will first be compared qualitatively with experimental data (for more details see [1]). Furthermore, the delta wing planar form that has been investigated in [2] will also be explored using Ansys Discovery. In this way a quantitative comparison with the delta wing simulation results given in [2] will be performed. However, some small modifications, as compared to the planar wing form in [2], have been implemented. The first modification is the thickness of the delta wing, which in [2] is 6,4 mm and in the present study is 10 mm. Furthermore, instead of having sharp leading edges, as in [2], in this study round leading and trailing edges were chosen. The reason for these two modifications or adjustments is that the 'Live-Physics' GPU-solver in the Explore mode of Ansys Discovery is used. As this employs a fixed global grid size (in contrast to the Refine mode of Discovery), a large number of elements would have been required to accurately capture such a geometry which has a large aspect ratio. Likewise, the coarser grid in the Explore mode will not match a sharp edge very well.
Delta wings are so thin since they are designed for supersonic flows. In supersonic flows over wings, besides the pressure and the friction drag, a third kind of drag, the wave drag occurs and is substantial. Thin wings have way less wave drag then thick wings. Therefore, supersonic wings are thin. Since in this study the delta wing is investigated at subsonic speeds, there is no wave drag and one can thicken the wing a little bit to make sure the grid has at least two cells across the wing thickness.

This can be verified with the Size Preview command in Ansys Discovery(see Figure 2). After analyzing the flow in the Explore mode, it is always possible to run the Refine mode to resolve also thinner details of the geometry. In the Explore mode, however, it is already possible to capture quite well the main characteristics of the delta wing.

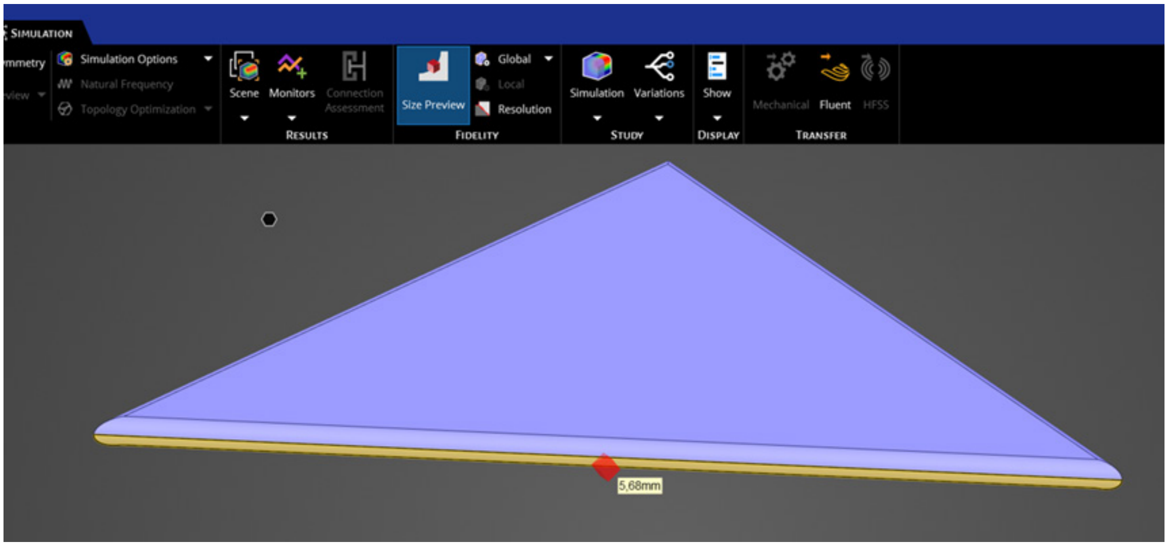


Figure 2: Size preview in the simulation ribbon, providing insights into the size of the voxel mesh elements in the Explore mode.

2.1 Geometry Preparation

The delta wing investigated has a span of b=306 mm and a height of also h=306 mm, *i.e.* h/b=1, as in [2]. These proportions are quite close to the ones of the Concorde wing, which has a span of b=25.60 m and a height of h=27.66 m, *i.e.* h/b= 1.08. However, the Concorde has an ogival wing planform. The wing was generated using the sketching functionalities in the Design ribbon. All the steps were recorded so that the geometry generation can also be repeated by the reader (see geometry file attached). The reader can also reproduce the geometry according to the drawing in Figure 2 or use the ready geometry file as shown in Figure 2.

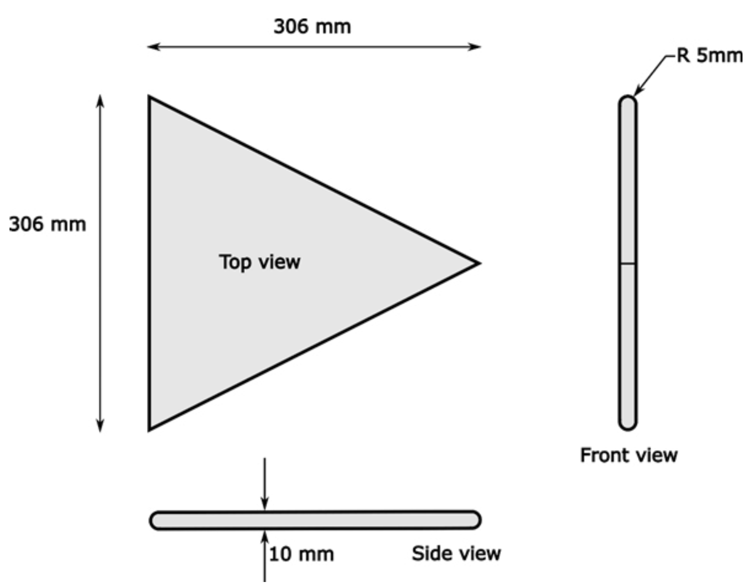


Figure 3: Dimensions of the delta wing (drawing not in scale)

2.2 Setting the Angle of Attack as a Parameter

In this study the angle of attack will be varied between 0° and 40°, which exceeds a little bit the usual angle of attack range of a delta wing. This can be done by parameterizing the angle of attack and running the simulation for a prescribed series of angles of attack specified as a parameter. To set the angle of attack as a parameter, switch on the history tracking prior to creating the wing. Select the move tool and rotate the delta wing 20° (or some other value, just as a placeholder) about the y-axis, see Figure 4(a). Subsequently, click on the double arrow in the history tracking window and choose the angles as a parameter, see Figure 4(b). In the History Tracking window click at 'rotate about y handle' and set the angle of attack to 20° as a parameter (by right clicking as shown in Fig. 4(c)).

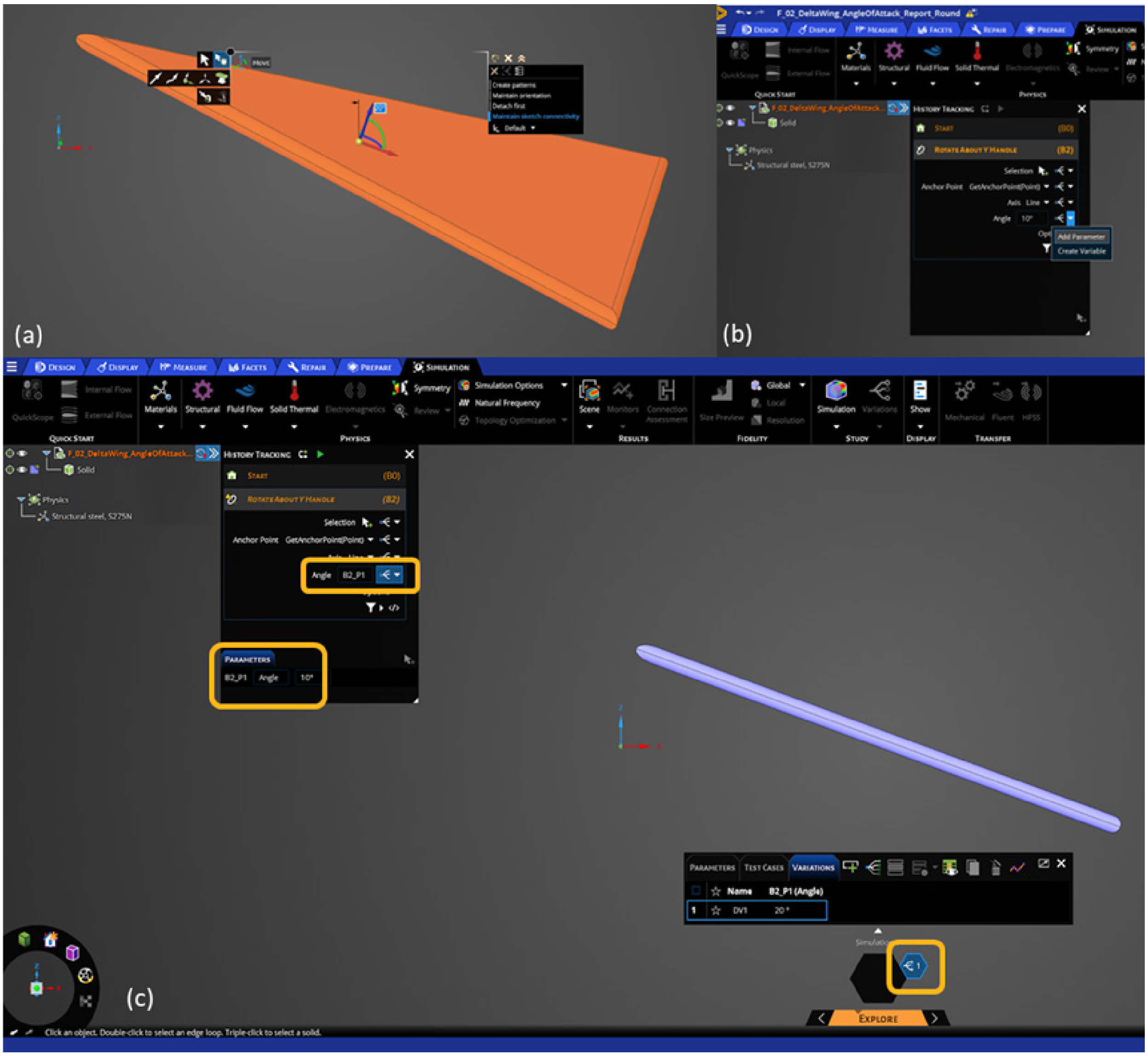


Figure 4: (a) Setting the angle of attack as a rotation about the y-axis and subsequently (b/c) set a rotation angle as parameters in the history tracking window.

Click at the parameter symbol at the bottom of the main window to open the list or parameters (see Fig. 4(c)). To check if the angle of attack parameter is properly implemented, change it in the parameters window to some other value and press the green play button right from the History Tracking and check if the angle of attack is changing properly, the wing must rotate to the new angle of attack.

2.3 Prepare the Flow Domain

The flow domain is created through an enclosure around the delta wing, which can be accessed via the Prepare ribbon. Set the default cushion to 100% and the dimension in front (-x direction) to 400 mm and the dimension behind (+x direction) to 800 mm, as shown in Figure 5. These are approximate but well working dimensions for this geometry. However, the reader is encouraged to experiment with different dimensions of the flow domain and observe the corresponding variation of the lift and drag forces. The flow domain is large enough when these quantities don't change anymore. Additional information on how to prepare the fluid domain for simulation can be found here: Geometry Preparation for Fluids Simulation | Ansys Courses.

Figure 5: (a) Creating the enclosure via the Prepare ribbon and (b) setting the dimensions of the fluid domain enclosure.

2.4 Boundary conditions

In this simulation the inlet velocity is set to 28.16 m/s (alternatively it can be defined as mass flow rate, pressure or swirling flow) and the outlet is set as a pressure outlet with 0 Pa, as shown in Fig. 6(a). All other boundaries are left as wall (default boundary condition, *i.e.* walls with no slip condition). For more information of setting up a fluid flow analysis in Discovery, please refer to Fluid Flow Simulation in Ansys Discovery | Ansys Courses.

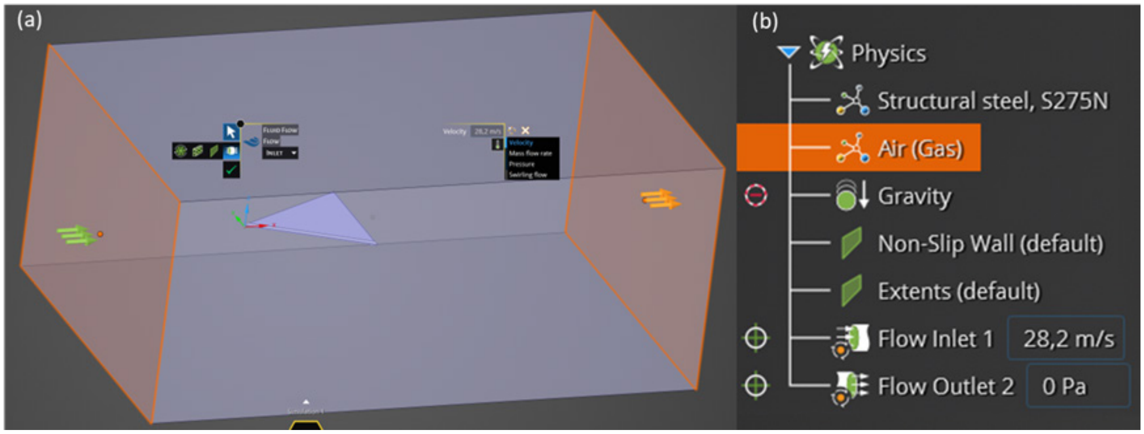


Figure 6: (a) Fluid Flow heads-up display for setting the boundary conditions for the CFD analysis. (b) Populated physics tree including the adapted fluid medium (default is water).

2.5 Physics – Flow Condition

By default, water is assigned the fluid media, which can be changed in the Physics Tree to air (see Figure 6(b)). Note that neither at the inlet nor at the outlet a temperature was specified and hence an incompressible simulation was set up. This is justified since at a velocity of 28.16 m/s compressibility effects are negligible. In case the reader wants also to investigate higher velocities, *i.e.* Mach numbers higher than 0.3, it is necessary to solve for compressible flow. In this study, however, the Mach number is below 0,1 and hence compressibility effects can be neglected. This can be readily seen assuming a temperature of T=20°C= 293.15 K, an isentropic coefficient of the air of k=1.4 and an ideal gas constant for the air of R=287 J/kg K. Hence, the speed of sound is $\mathbf{a=\sqrt{kRT}}$= 343 m/s and the Mach number **M=U/a**=0.082. The Reynolds number $\mathbf{Re=(\rho U c_{root})/\mu=5.9\cdot10^5}$, where $\mathbf{\rho=1.16\ kg/m^3}$ is the density, **U=28.16 m/s** the velocity, $\mathbf{c_{root}=306\ mm}$ is the root chord and $\mathbf{\mu=1.83\cdot10^{-5}\ Pa\cdot s}$ the viscosity. These results are summarized in Table 1.

Table 1: Reference quantities and flow conditions with values identical to [2].

Quantity	Symbol	Value	Unit
Root Chord	c_{root}	0.306	*m*
Freestream Velocity	U	28.16	*m/s*
Density	ρ	1.16	kg/m^3
Viscosity	μ	$1.83\text{x}10^{-5}$	$Pa\cdot s$
Reynolds number	Re	$5.9\text{x}10^5$	-
Mach number	Ma	0.082	-

2.6 Grid Size Check and Refinement

In the Explore mode click on the simulation tab and select Size Preview and hover the mouse over the flow domain and check the grid size, see Figure 7. In the left picture the mesh size is 8.73 mm. One has to keep in mind that the delta wing thickness is 10 mm. The grid size must be smaller than the wing thickness, better two times smaller or less. Therefore, on the right side the grid was refined pulling the slider at the bottom of the simulation information display (SID) to the right (see Fig. 7). The refined grid size is 5.52 mm, which is about half of the thickness of the wing. It is important to note that the voxelized mesh in the Explore mode is geared toward providing almost instantaneous results for quick and easy design exploration and the default fidelity will be dependent on the individual computer performance.

To refine the grid furthermore one can proceed to the Refine mode (single chevron to the right in the SID), which is utilizing the CPU solver of Ansys Fluent or use the in-build exporters to run the simulation in Ansys Fluent. For further information on best practices on mesh generation in CFD analysis, the reader is referred to: Best Practice Guidelines for CFD Simulations | Ansys Courses. In this report, the aim is to focus on the rapid design capabilities in the Explore mode of Ansys Discovery. Therefore, in this study, the delta wing was built a little bit thicker than in reference [2] and the leading edges were rounded instead to have a thin chamfer. The thin leading edge would not be captured by the grid, and it also can be neglected to perform preliminary design studies.

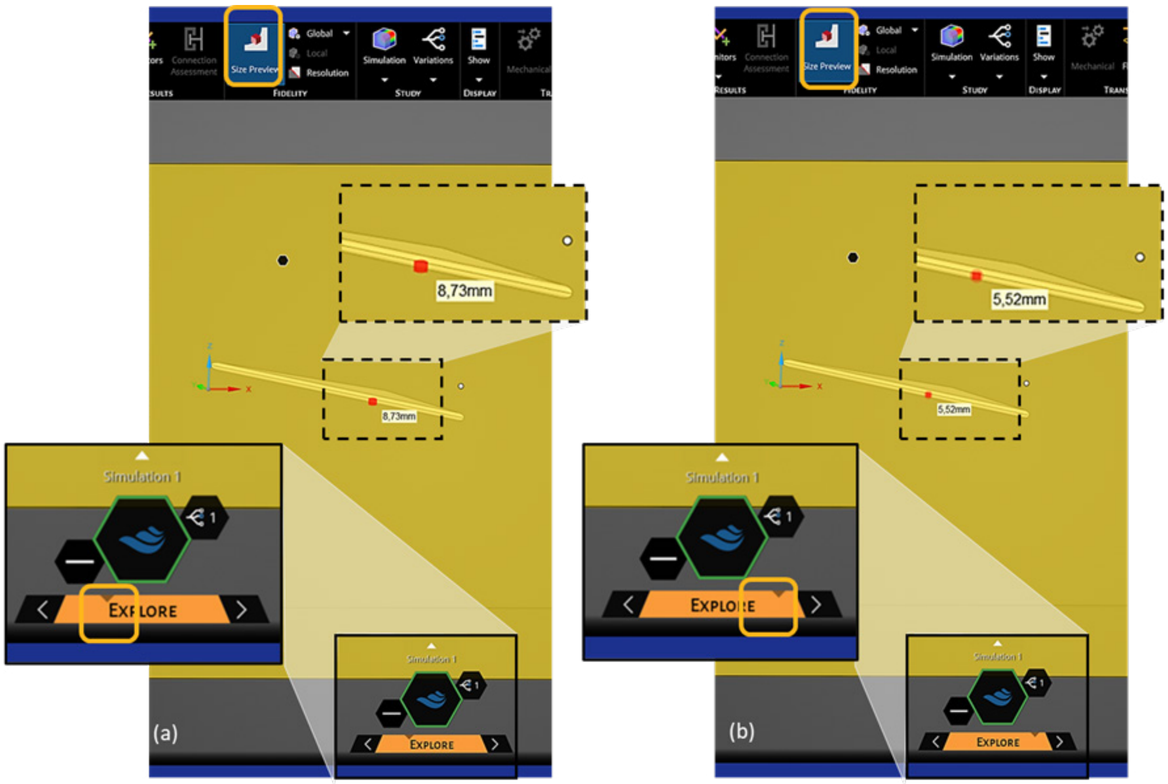


Figure 7: Grid refinement and size preview in the Explore mode shown for (a) a low fidelity mesh and (b) a high-fidelity mesh.

2.7 Lift and Drag Monitors

After box-selecting the internal delta wing geometry within the fluid domain (note, the actual delta wing geometry must be hidden and excluded from the simulation first), a custom monitor (Figure 8) can be created from the simulation tab to record the forces on the wing in x- and z-direction, *i.e.* the drag and lift forces.

Figure 8: Drag and lift monitor applied via box-select to the internal surfaces of the fluid domain which represent the delta wing geometry.

3. Results and Discussion

3.1 Flow Visualization

After having generated the grid and previewed the grid size, the simulation can be run (green central icon on the result arc at the bottom right) and the flow visualized for further inspection. In Figure 9(a) the streamlines around the delta wing are shown and as a comparison in Figure 9(b) the experimental visualization of the flow around a delta wing is shown. In the experimental visualization, smoke is released in stream tubes over the central plane of the wing and with two line-lasers. Consequently, the flow is visualized in the longitudinal and transversal directions showing how the delta wing vortex forms over the leading edge. The transversal laser after the delta wing shows the core of the delta wing vortex (compare Figure 9(b). For more details on the flow visualization around a delta wing see [1].

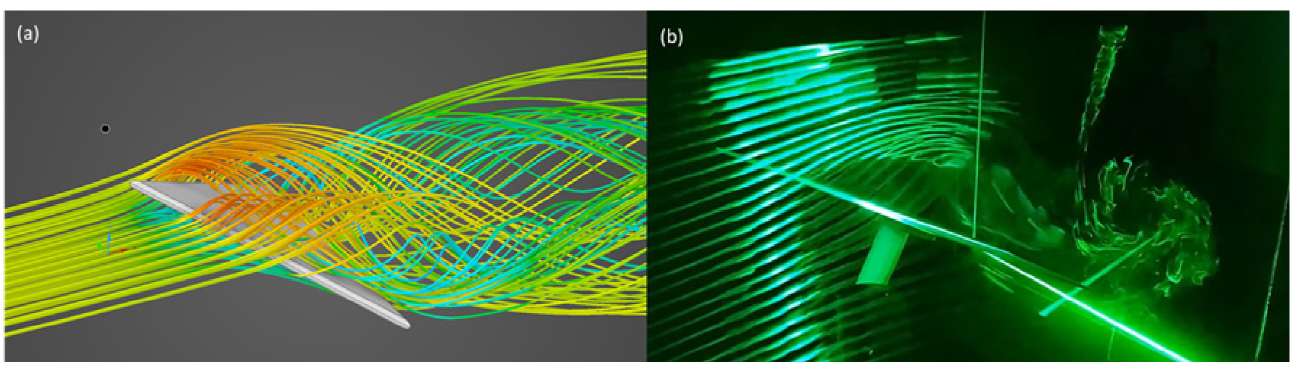


Figure 9: Flow visualization around the delta wing. (a) Streamlines, as obtained by Discovery, showing the delta wing vortex and (b) the corresponding experimental flow visualization in the wind tunnel (Source: by the author Epple).

The results of the Ansys Discovery simulation are in very good qualitative agreement with the experimental observations. Figure 10(a)/(b), a velocity isosurface and a vertical direction field were added in the numerical model, respectively. These highlight the two delta wing vortex cores forming. Figure 10(c) represents the experimental counterpart to the simulation, visualizing the two delta wing vortices by using two transversal laser sheets. This shows that the simulation results and the experimental flow visualization are in very good qualitative agreement.

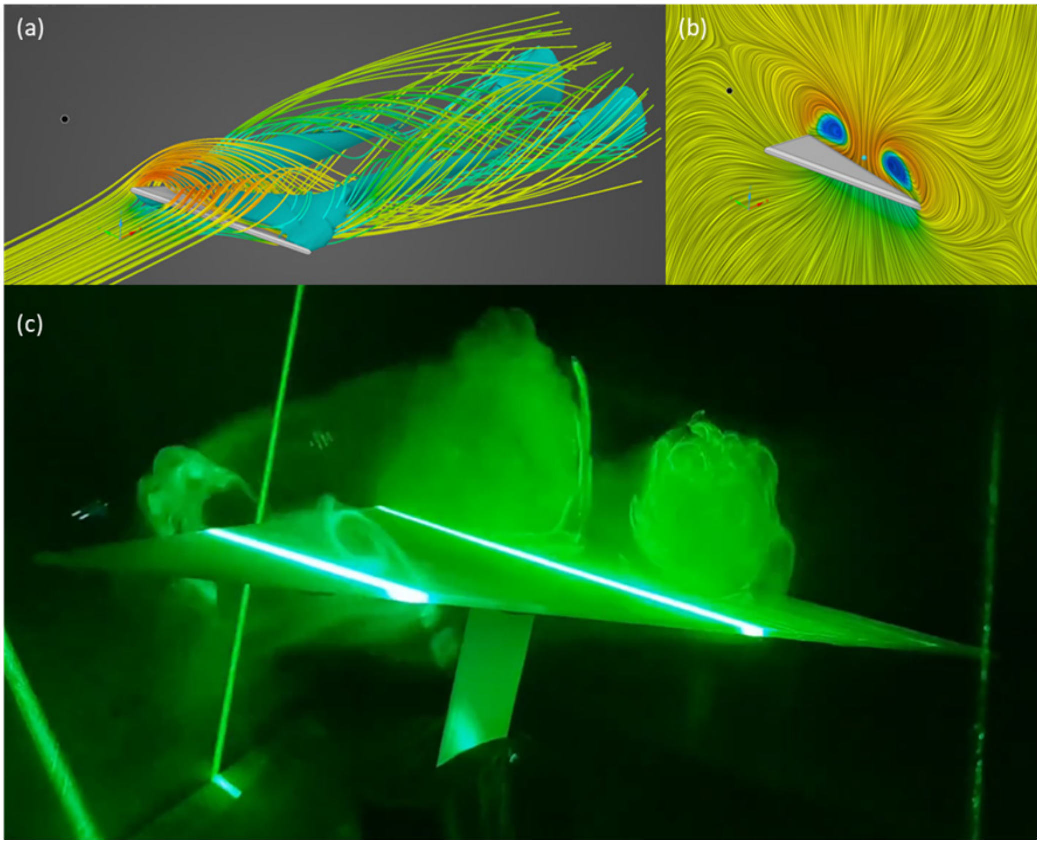
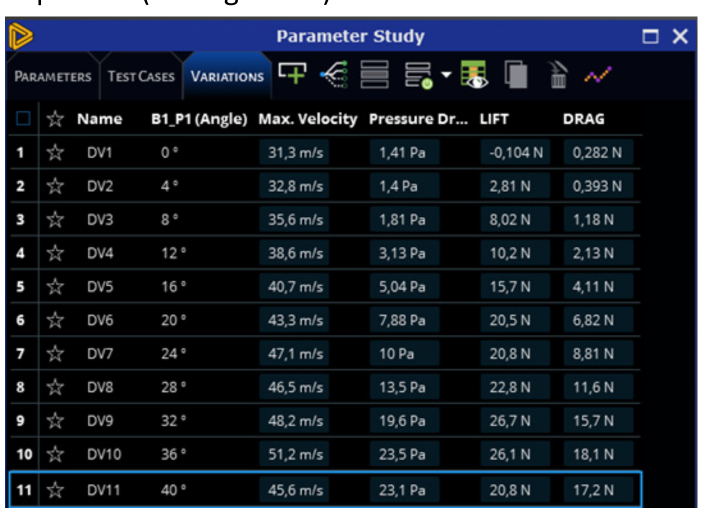


Figure 10: (a) Streamlines with velocity isosurface showing the delta wing vortex core and (b) vertical direction field showing the delta wing vortices over the wing, which are in good visual agreement with (c) the vortices obtained during the experimental analysis using two vertical laser sheets (Source: by the author Epple).

3.2 Running the simulation for a full characteristic (range of angles of attack)

After validating the simulation qualitatively for one angle of attack, the full range of angles of attack can be simulated easily through a parametric study in Discovery, in which the angles of attack, from 0° to 40° are analyzed in steps of 4° (see Figure 11).



	Name	B1_P1 (Angle)	Max. Velocity	Pressure Dr...	LIFT	DRAG
1	DV1	0 °	31,3 m/s	1,41 Pa	-0,104 N	0,282 N
2	DV2	4 °	32,8 m/s	1,4 Pa	2,81 N	0,393 N
3	DV3	8 °	35,6 m/s	1,81 Pa	8,02 N	1,18 N
4	DV4	12 °	38,6 m/s	3,13 Pa	10,2 N	2,13 N
5	DV5	16 °	40,7 m/s	5,04 Pa	15,7 N	4,11 N
6	DV6	20 °	43,3 m/s	7,88 Pa	20,5 N	6,82 N
7	DV7	24 °	47,1 m/s	10 Pa	20,8 N	8,81 N
8	DV8	28 °	46,5 m/s	13,5 Pa	22,8 N	11,6 N
9	DV9	32 °	48,2 m/s	19,6 Pa	26,7 N	15,7 N
10	DV10	36 °	51,2 m/s	23,5 Pa	26,1 N	18,1 N
11	DV11	40 °	45,6 m/s	23,1 Pa	20,8 N	17,2 N

Figure 11: Table of the parameter study showing the variation of the angle of attack (parameter B1_P1) and the resultant state variables.

Export these results in a report (by right-clicking on the monitor window) and copy the table to a spreadsheet, for example Microsoft Excel. To compute the lift and drag coefficients the following equations are applied:

$$C_L = \frac{L}{qA} = \frac{L}{\frac{1}{2}\rho U^2 A} \qquad C_D = \frac{D}{qA} = \frac{D}{\frac{1}{2}\rho U^2 A}$$

where **ρ=1.16 kg/m²** is the density of the air, **U=28.16 m/s** is the flow velocity, **A=(0.316²)/2=0.046818 m²** is the area of the delta wing, and **L** and **D** are the lift and drag forces, respectively. In such a way the results of Table 2 are obtained.

Table 2: Lift and drag coefficients of the delta wing.

Name	α [°]	L [N]	D [N]	C_L [-]	C_D [-]
DV1	0	-0.1	0.3	-0.005	0.013
DV2	4	2.8	0.4	0.130	0.018
DV3	8	8.0	1.2	0.372	0.055
DV4	12	10.2	2.1	0.474	0.099
DV5	16	15.7	4.1	0.729	0.191
DV6	20	20.5	6.8	0.952	0.317
DV7	24	20.8	8.8	0.966	0.409
DV8	28	22.8	11.6	1.059	0.539
DV9	32	26.7	15.7	1.240	0.729
DV10	36	26.1	18.1	1.212	0.841
DV11	40	20.8	17.2	0.966	0.799

These results were compared with the highly accurate simulation results of Seraj and Martins [2] and are shown in Figure 12. One can see that the Ansys Discovery results are in quite good quantitative agreement with these values of the literature. Since the grid in the Explore mode is rather coarse and the flow is detached one can see some small oscillations and overshooting at higher angles of attack, where the flow is detached. Even so, the order of magnitude is well reproduced. Hence, Ansys Discovery is very well suited for preliminary studies, delivering very good preliminary qualitative and quantitative results.

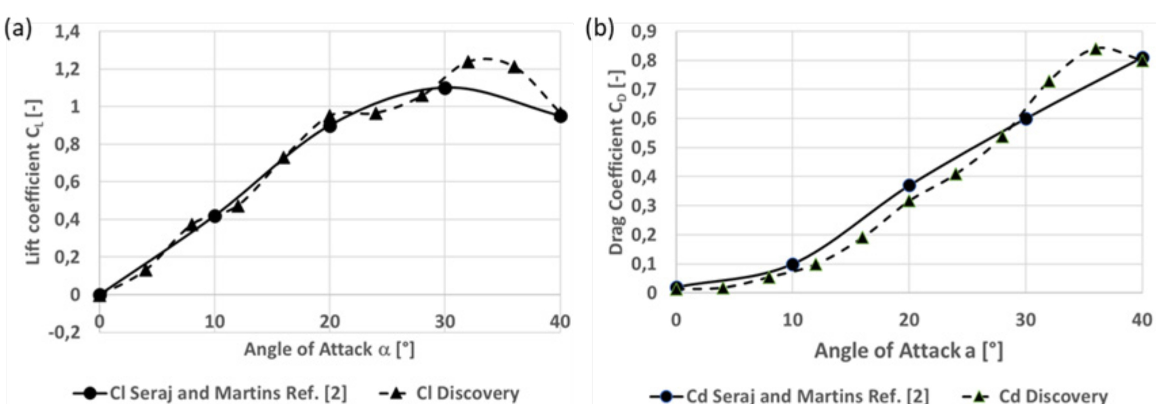


Figure 12: Comparison of Ansys Discovery results for (a) lift and (b) drag coefficient with literature values [2].

4. Conclusion

It was shown that Ansys Discovery enables the fast and easy parametric exploration of a delta wing at subsonic speeds under considerations of a few geometric adaptations (*e.g.* the thickness and the rounding-off of the leading edge) and thus helps in obtaining preliminary results for the lift and drag values for different angles of attack. This study has shown that the simulations agree qualitatively very well with experiments and quantitatively also well with more elaborate and precise CFD simulations, with some minor deviations. The order of magnitude of the Ansys Discovery results, however, is the same. Therefore, Ansys Discovery lends itself as an up-front simulation tool to study the main characteristics of a delta wing at low subsonic speeds. Likewise, it is conceivable to investigate similar delta wing parameters (*e.g.*, nose angle, leading edge, chord lengths, etc.).

5. References

[1] Epple, Ph., Kramer, D. and Steppert, M.: Flow Visualization of Wing Vortices of Delta Wings, J. Fluids Eng. June 2023, 145(6): 060904, https://doi.org/10.1115/1.4057070

[2] Seraj. S. and Martins, J.R.: Predicting the High Angle of Attack Characteristics of a Delta Wing at Low Speed, Conference AIAA Aviation 2021 Forum, https://doi.org/10.2514/6.2021-2612

Use and Reproduction
The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information
This case study is part of a set of teaching resources to help introduce students to structures, fluids, or heat transfer (physics areas supported by Ansys Discovery).

Ansys Education Resources
To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback
If you notice any errors in this resource or need to get in contact with the authors, please email us at education@ansys.com.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com