XFLR5 Tutorial

Contents

1 Installing XFLR5

Go to the XFLR5 website, click ’Downloads’, and open the folder with the highest version number. Download, extract, and run the executable corresponding to your operating system.

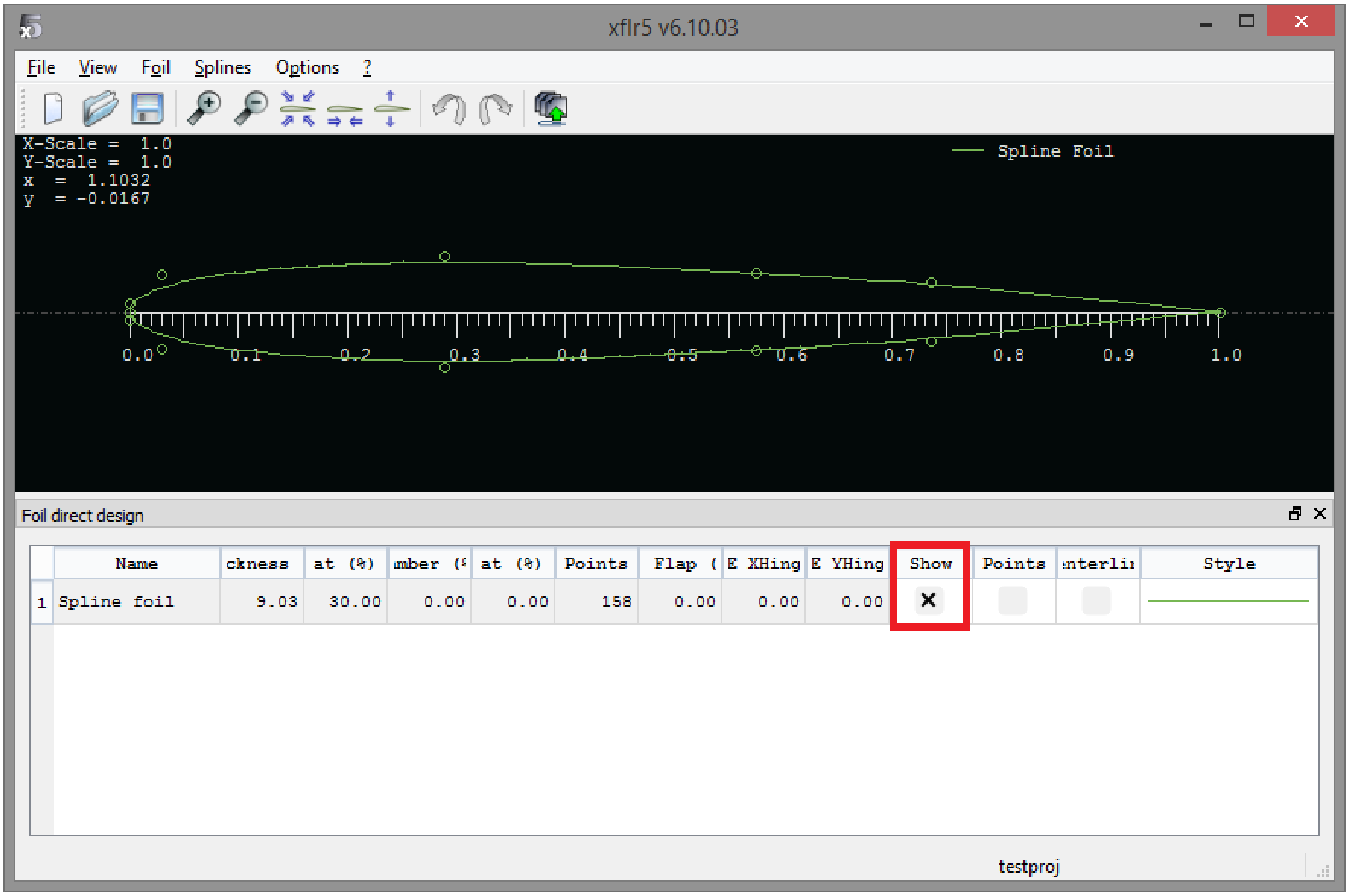
2 Creating a project

Project files in XFLR5 hold the airfoil profiles to be tested, as well as 2D and 3D test results. To create a project, go to File → New Project, then select File → Save Project As. Save the project in an appropriate folder, using a name like “<airfoil_name>_2D_tests”. XFLR5 does not auto-save, so you must **save your work on a regular basis**.

It is recommended to use separate project files for 2D and 3D testing, since 3D testing requires that data be collected at a range of Reynolds numbers, which clutters up the 2D result polars.

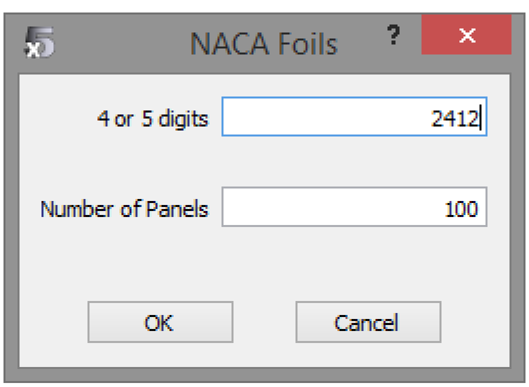
3 Creating and importing airfoils

To create an airfoil, first select File → Direct Foil Design. A split window will appear, with an airfoil outline in the top box and a table with one entry, ’Spline foil’, in the bottom box. Uncheck ’Show’ to hide the spline foil.



3.1 4 and 5-digit NACA airfoils

XFLR5 can automatically generate airfoil profiles for NACA 4 and 5-series airfoils (*e.g.* the NACA 2412). To do this, select Foil → Naca Foils, then enter the foil number and the number of panels desired. Click 'OK', then click 'OK' again in the 'Rename' dialog box that appears. The default 100-panel profile is appropriate, but additional accuracy can easily be gained by setting the number of panels to 200 or 300.



3.2 Other airfoils

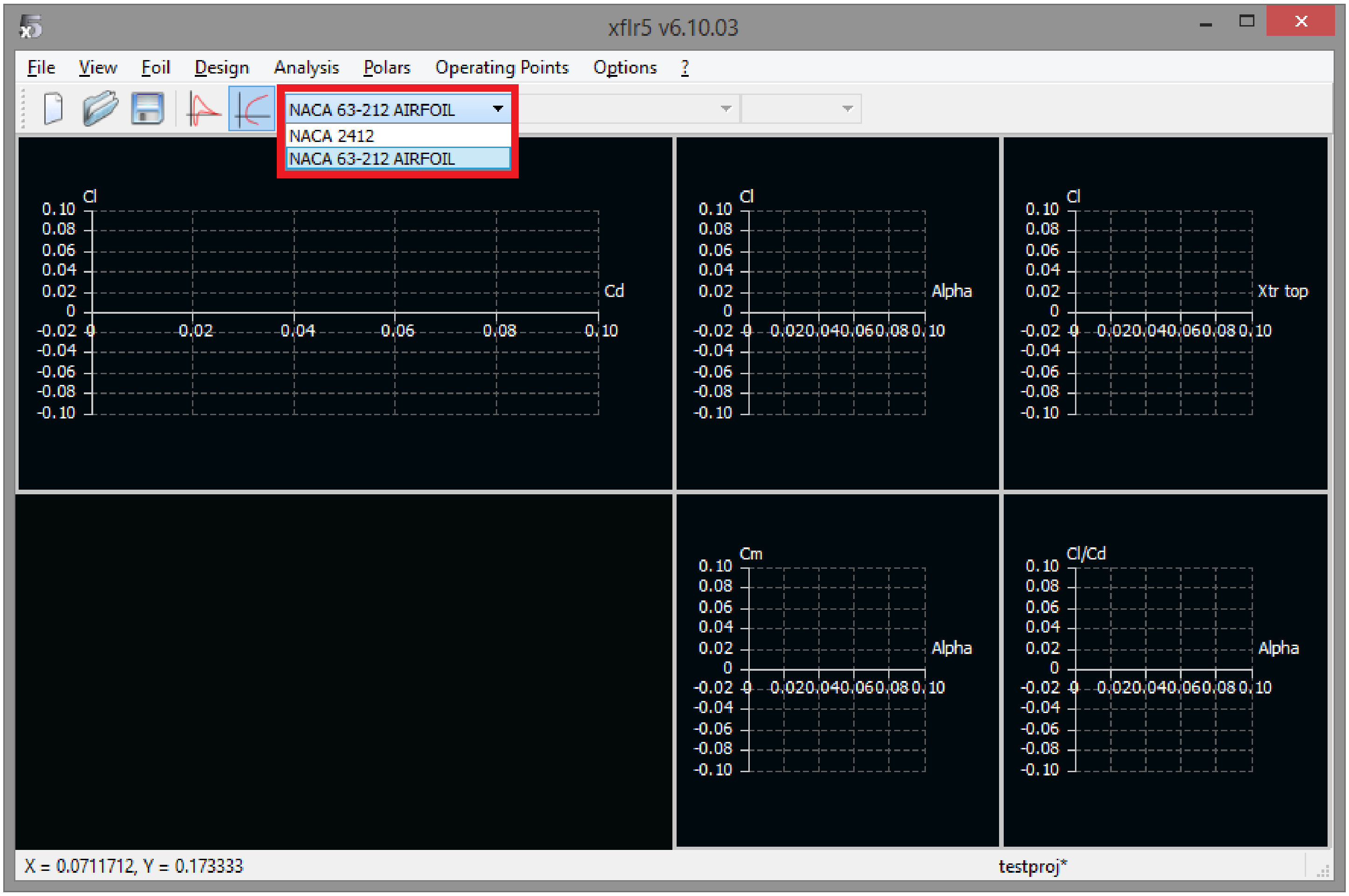
If an airfoil cannot be created automatically with XFLR5, it must be imported from a coordinate file. These files may be provided to you, but if they are not, they can be downloaded from many websites, including AirfoilTools. Coordinate files should be downloaded in the Selig format—that is, the first line should be the airfoil's name, and each following line should be in the format `<x coord> <y coord>`, as such:

```
NACA 63-212 AIRFOIL
  1.000000  0.000000
  0.950120  0.005660
  0.900250  0.012240
  0.850350  0.019470
  0.800420  0.026980
  0.750450  0.034510
  0.700430  0.041820
etc...
```

Save the downloaded file as "<airfoil_name>.dat", preferably in the same directory as your project file. In XFLR, select File → Open, then select your downloaded coordinate file. It should appear in the Direct Foil Design screen. If the downloaded coordinate file gave only a small number of points (fewer than 90), additional points can be added[1]. Select Foil → Refine Globally, then enter the desired number of panels (between 100-300). Select 'OK', then select 'Overwrite' in the Rename window that appears.

[1]The law of diminishing returns applies here—going overboard on the number of panels will only serve to make data collection slower

4 2D airfoil testing

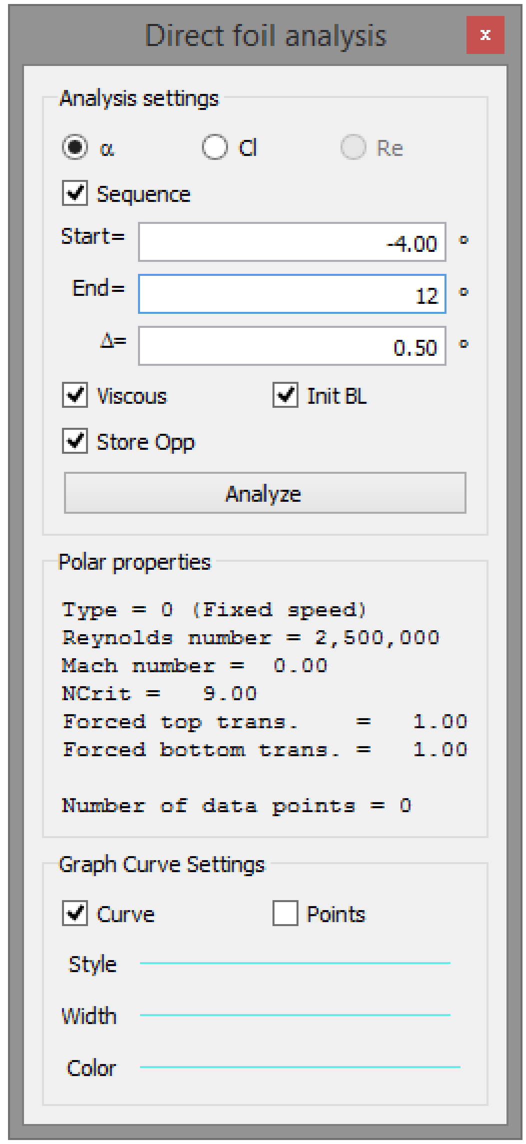


To perform 2D airfoil testing, select File → XFoil Direct Analysis[2]. If the project file includes multiple airfoils, select the airfoil you want to test from the drop-down list at the top of the screen.

[2]XFLR5 uses XFOIL for 2D airfoil analysis. XFOIL was developed by Mark Drela at MIT beginning in the 1980s and was last updated in 2013. A large part of XFLR5's value is as a modern interface to XFOIL.

4.1 Testing a single airfoil at a single Reynolds number

Select Analysis → Define an Analysis. Enter the Reynolds number, then select 'OK'. Note that it is possible to enter large Reynolds numbers using scientific notation, *e.g.* "2e6". It isn't necessary to enter the Mach number unless you know the airfoil will be operating above Mach 0.3. If you know the airflow will be tripped to turbulence at a certain fraction x/c of the chord, you can enter that fraction (as a decimal) in the TripLocation boxes at the bottom of the Analysis window. 'NCrit' is a parameter associated with surface roughness—qualitatively, lower NCrit (~ 7) corresponds to higher surface roughness, and higher NCrit (~ 12) corresponds to a fairly smooth surface. Keep the default setting of NCrit $= 9$ if you don't know the desired surface roughness.



A separate toolbar window labeled 'Direct foil analysis' should be visible. If it is not, select Options → Restore Toolbars. In the toolbar, if testing is only needed at a single angle of attack, enter that angle of attack. If you're testing over a range of α, check the 'Sequence' box, enter the upper and lower bounds for α, and set $\Delta\alpha$ to a reasonable value (the default of $0.50°$ is fine). Note that the flow solver used to determine aero results assumes the flow is not separated, so the solver will fail to converge at α somewhat greater than $\alpha_{C_\ell,\max}$ as flow becomes separated.

Save your project file in case a crash occurs, then click 'Analyze'. A window labeled 'XFoil Analysis' should appear and show the solver's progress at each angle of attack. Once all angles of attack are solved to convergence or the iteration limit (100 by default), a set of solution plots will appear. You can click and drag the plots to change the view area, or use the scroll wheel to zoom.

4.2 Batch analysis

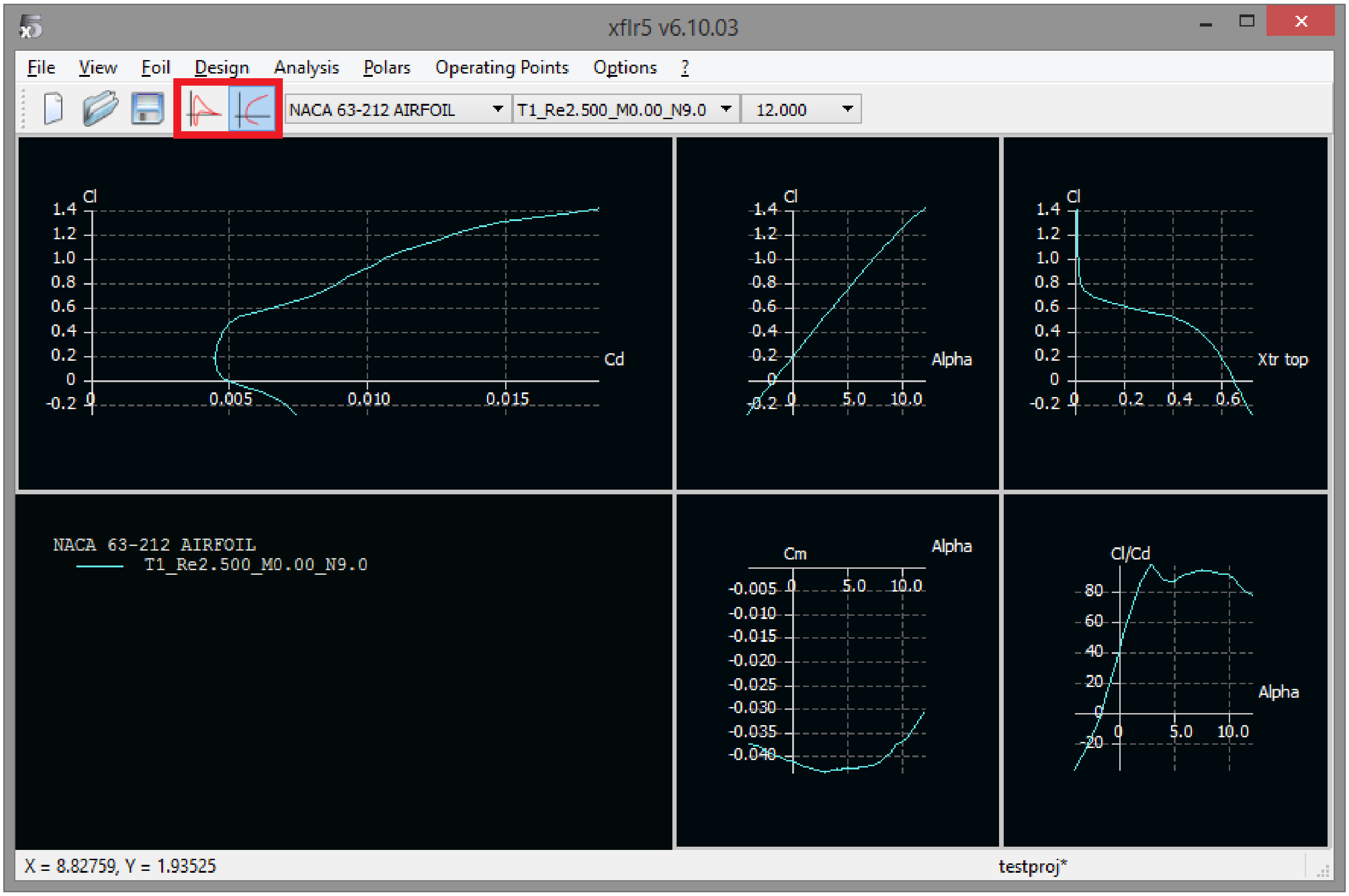
It is possible to compare the performance of several different airfoils, or test one airfoil at several Reynolds numbers, or compare several airfoils at several Reynolds numbers. Note that testing a large number of cases will lead to cluttered result plots, making interpretation of results difficult.

First make sure that all airfoils to be tested have been imported into XFLR5 (see §3). Then, go to Analysis → Batch Analysis, and a window labeled “Batch foil analysis” should appear. If multiple foils are to be tested, select the “Foil list” button, then click the “Foil list” box and select the desired airfoils. If you’re only testing a single airfoil, use “Current foil only”. The current foil can be set in the drop-down box in the menu bar. Choose the default Type 1 analysis.

The Reynolds number can either be tested at evenly-spaced points spanning a specified range, or at values in a list. Choose whichever is appropriate. Choosing “Re List” also allows you to determine the aerodynamic effects of Mach number or NCrit by setting the Reynolds numbers in the list to some reasonable constant ($e.g.\ 10^6$), then varying Ma or N_{crit}. Remember to save the project before running analysis.

If no results appear when you click ‘Analyze’, you may not have selected any airfoils. Make sure the desired airfoils are highlighted in the foil list.

4.3 2D lift curve and polar visualization

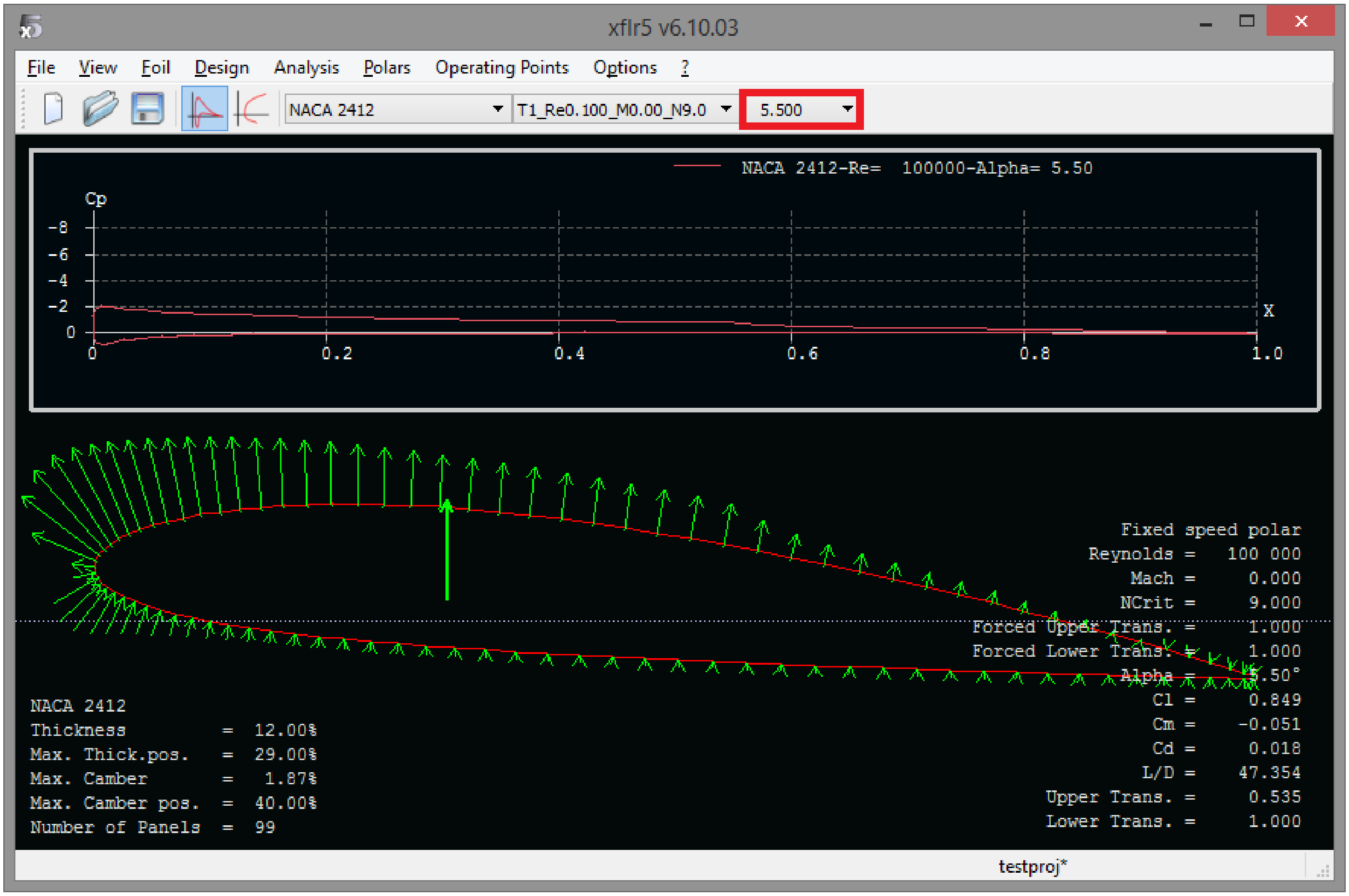


Clockwise from top left:

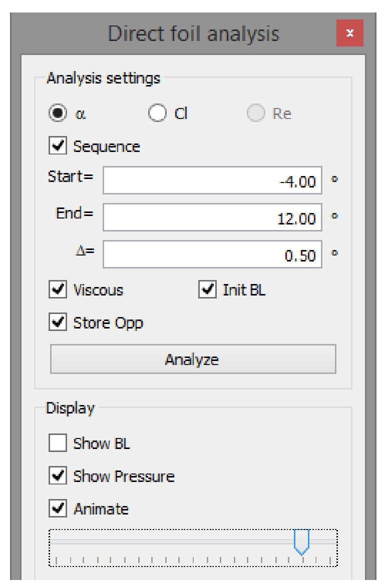
1. C_l *vs.* C_d: lift versus drag polar. It's important for C_d to remain acceptably small across the expected operating range of C_l.
2. C_l *vs.* α: lift versus angle of attack. Critical points include the angle of attack where the plot crosses the x-axis (called $\alpha_{L=0}$) and the maximum coefficient of lift point ($\alpha_{C_{L,\max}}$), beyond which XFLR5 results will become unreliable.
3. C_l *vs.* $X_{\text{tr,top}}$: lift versus turbulence transition point. $X_{\text{tr,top}}$ is the fraction of the chord at which the flow over the top of the airfoil transitions to turbulence.
4. C_l/C_d *vs.* α: Lift to drag ratio (*i.e.* glide slope) versus angle of attack. Ideally, the glide slope should be maximized near the operating angle of attack.
5. C_m *vs.* α: Moment coefficient versus angle of attack. A negative slope means good pitch stability.
6. Plot key. The key is grouped by airfoil, and each line represents a set of operating conditions. The naming format is <Analysis type>_Re<Reynolds number/10^6>_<Mach number>_<NCrit>

These results can be exported for analysis in Excel or MATLAB. To export the data for only one airfoil and set of operating conditions, select the desired airfoil and operating conditions from the drop-down boxes in the menu bar, then select Polars → Current Polar → Export and save as csv.

4.4 2D pressure coefficient visualization



To view plots of pressure coefficient *vs.* span, click the corresponding icon (which looks like a c_p plot) in the menu bar. The icon's location is marked with a red box in the screenshot above. A pressure coefficient plot is available at each tested operating point. Note that each operating point is defined by both the operating conditions (selectable with the second drop-down box) and angle of attack (selectable with the third drop-down box, outlined in the screenshot on page 8). To get a sense for how pressure coefficient varies with angle of attack, check the 'Show Pressure' and 'Animate' boxes in the 'Direct foil analysis' window.



To export pressure coefficient results, go to Operating Points → Current OpPoint → Export. The exported file will have the operating point parameters at the top, then five columns of data. The first column is the x-location of the data point, then the pressure coefficient (calculated for both inviscid and viscous flow) and the dynamic pressure Q, also calculated for inviscid and viscous flow.

4.5 Saving and printing plots

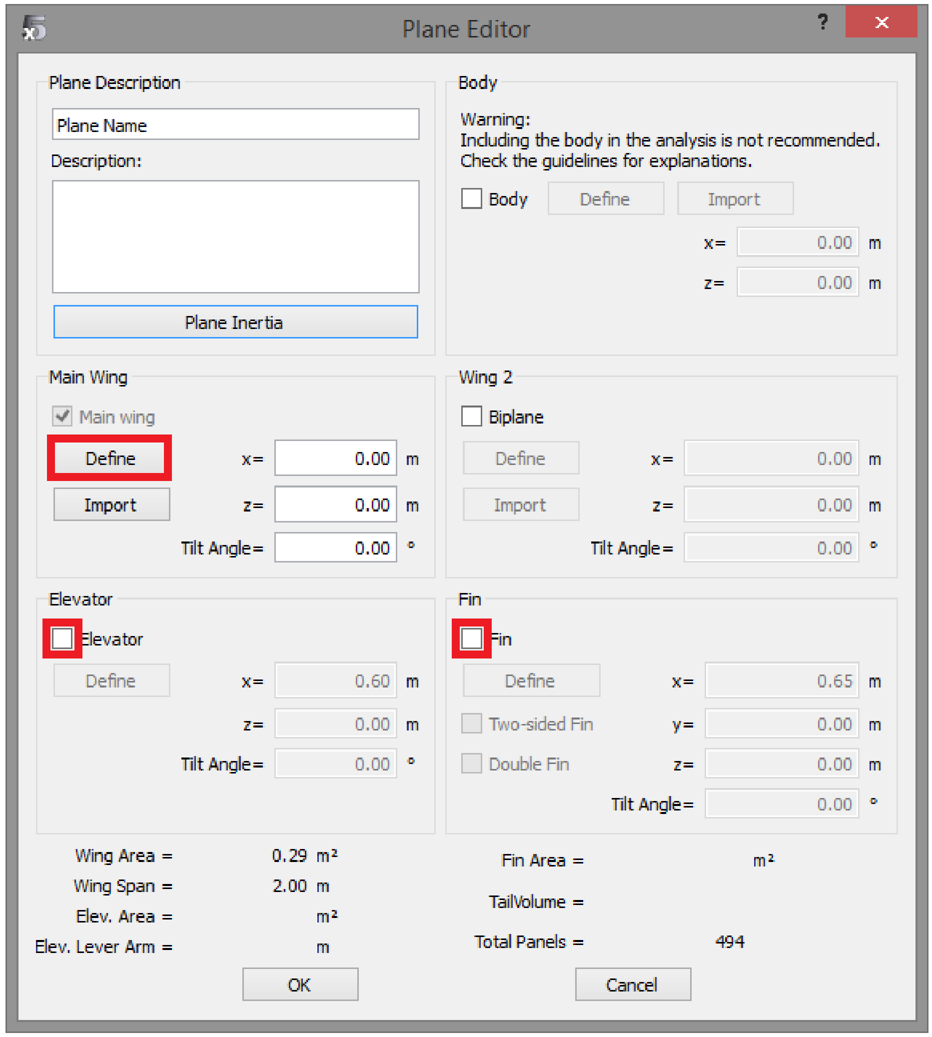
The polars and pressure coefficient plots can be saved as images using View → Save View to Image File[3]. If you plan to print the plots directly, you may want to invert the background and text colors to reduce ink/toner consumption and improve readability. To do so, go to Options → General Display Settings, and click All Graph Settings. In the Fonts and Background tab, change the title and label color to white, and the graph background to black.

When formally publishing XFLR5–produced data, it's better to export the results as discussed in §4.3, then create the desired plots using 3rd–party plotting software (*e.g.* MATLAB) capable of producing high-quality, scalable plots.

[3]The png format offers a good tradeoff between image file size and quality

5 3D wings

XFLR5 uses an AVL[4] clone for 3D flow analysis. AVL models 3D flow as a system of horseshoe vortices, and is well-validated for most typical subsonic flow conditions. Explanations of the code's assumptions and limitations are available in the XFLR5 user guide[5]–this discussion will focus on practical usage.



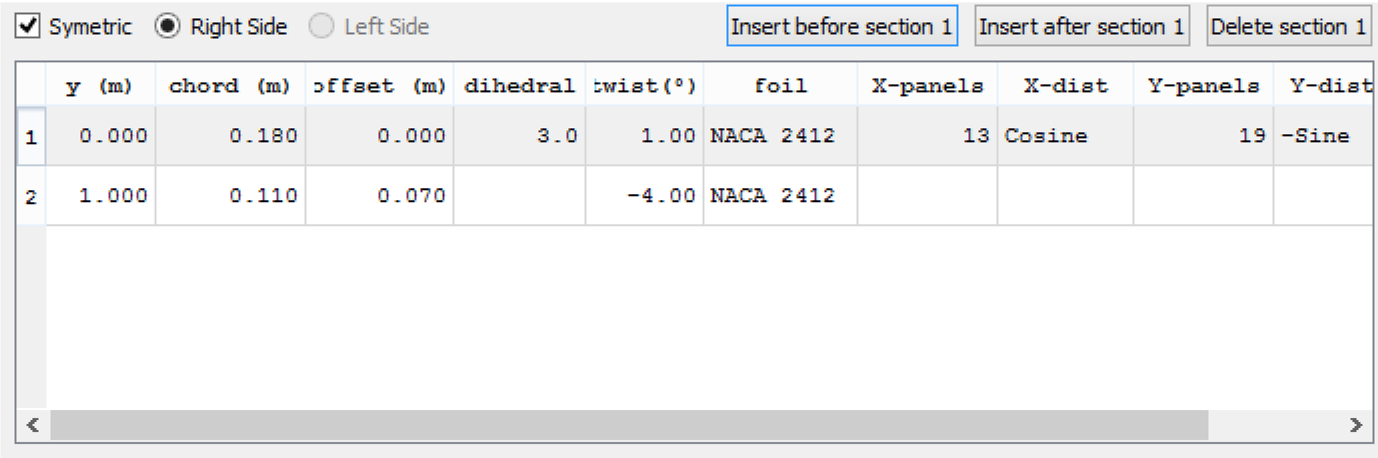
5.1 Wing aerodynamics

The 3D wing analysis mode is accessed via File → Wing and Plane Design. To define a wing, first make sure that the airfoil(s) that define the wing have been loaded into the project per §3. Then, go to Plane → Define a New Plane, and the Plane Editor window will appear.

XFLR5 is capable of delivering approximate aerodynamic results (including flow around the fuselage and empennage) for conventionally-shaped aircraft, but this section will focus solely on wing aerodynamics. To disinclude other aero surfaces, uncheck the 'Elevator' and 'Fin' boxes, then click 'Define' for the main wing.

[4]Athena Vortex Lattice, developed by the creators of XFOIL (Mark Drela and Harold Youngren of MIT). Documentation can be found at http://web.mit.edu/drela/Public/web/avl/avl_doc.txt

[5]Located at sourceforge.net/projects/xflr5/files/ as 'Guidelines.pdf'



A window labeled 'Wing Edition' should appear. The table in the top of the window is used to define the wing. Non-symmetric wings can be created by unchecking the 'Symetric' [*sic*] box, then defining the left and right sides separately, but this is not recommended, since non-symmetric flows quadruple the time required by the 3D flow solver.

Each row in the table defines the wing at a station along the chord. At least two are needed to fully define a wing, and more stations can be added to describe more-complex wing geometry. The table entries follow (note that distances are given in metric units).

y Distance from the origin to the station along the y-axis. The furthest station should have **y** equal to half the wingspan.

chord Distance from the leading to the trailing edge of the airfoil at the station.

offset x–component (fore–aft) of the distance between the station's leading edge and the y–axis

dihedral Angle between the xy–plane (top plane) and the quarter-chord line extending tipwards from the station. A positive angle at the first station gives a v-shaped wing.

twist Angle between the station chord line and the x–axis. Positive twist appears clockwise from the tip.

foil The station airfoil. It is possible to use different airfoils along the span–say, a 2412 at the root and a 2308 at the tip. Even if you only have one airfoil loaded in the program, you must still click these cells and choose the foil from the drop-down box that appears.

panels Don't edit these. Counterintuitively, increasing the number of panels makes convergence more difficult and results less accurate.

Once the wing is defined to your satisfaction, hit 'Save and Close', then 'OK' in the Plane Editor window. The wing you defined should now be visible in the main program screen.

To begin analysis, go to Analysis → Define an Analysis. In the Analysis Definition window that appears, note that several tabs are present. The first tab, Polar Type, defines the analysis regime. Fixed Speed is the most useful for basic analysis—the freestream speed can be set to replicate wind tunnel conditions, or set to expected cruise speed for wing design analysis. The second tab, Analysis, controls the 3D flow solver method. Each method is best-suited for a different type of analysis.

LLT (lifting line theory) is most accurate for unswept, slender (high aspect ratio) wings. XFLR5's implementation of LLT is a viscous nonlinear solver that uses interpolation of 2D results to allow for accurate results at high angles of attack. Unlike the other solvers, it can't account for ground effect and has a difficult time dealing with unconventional wing geometry and large ($\gtrsim 10°$) dihedral.

VLM (vortex lattice method) models flow using a distribution of horseshoe vortices. Unlike LLT, it assumes linearity, which makes results at very high and low angles of attack unreliable. It should be used for wing analyses with low aspect ratio, large dihedral, and/or significant sweep.

3D Panels Can be used to solve flow around near–arbitrary (aerodynamic) geometry, including fuselage shapes. While LLT and VLM model the wing as a thin sheet placed at the camber line, the 3D panel method models the wing in its actual shape, with finite thickness. Solutions are more computationally–intensive, but finding a solution at a single operating point should still take less than one second on modern computers.

Once an appropriate solver is chosen, go to the 'Aero data' tab and enter air density and viscosity values for the expected flight conditions, then hit OK. You can then use the 'Plane analysis' window to run analysis over one or more angles of attack.

After analysis is initiated, one of two things might occur. The first is successful flow solution on the first try—this is indicated by the appearance of a listing of the values of several coefficients in the lower right corner of the main window.

The second case is the appearance of a status window labeled "3D Panel Analysis", which will probably say "Panel Analysis completed... Errors encountered" at the bottom. This error occurs because XFLR5 relies on interpolation of 2D results to produce 3D data, and the project file doesn't have data across all necessary operating points. To fix this, take note of the range of lift coefficients and Reynolds numbers at which data is not available. Return to the 2D analysis mode (File → Xfoil Direct Analysis) and run a batch analysis (per §4.2), setting the Reynolds number and C_l ranges to a bit larger than the range of missing data. Smaller range increments will produce more accurate 3D results, but will take longer to collect data. After 2D analysis is completed, go back to File → Wing and Plane Design, and re-run the analysis that failed previously. It should converge now, but if it doesn't, check the error output to see if you're still missing data at any points.

Pressure coefficient determination: If information on the spanwise distribution of pressure is desired, the '3D Panels' flow solver should be used. To view the C_p distribution, select the menu bar icon that resembles a C_p plot, or use the F9 key. Note that XFLR5 may crash if you attempt to view the C_p plot after running an 'LLT' analysis.