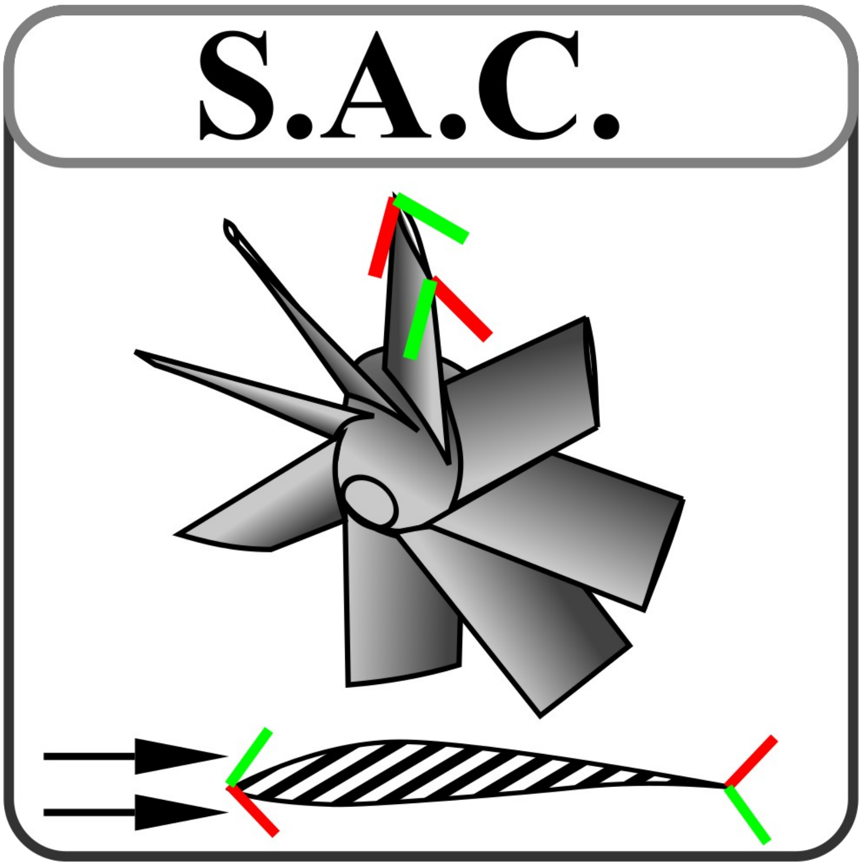
SUPERSONIC AEROFOIL CALCULATOR – S.A.C.

INSTRUCTION MANUAL



Created on 10/08/2019

Table of Contents

About S.A.C.

Supersonic Aerofoil Calculator (abbreviated as S.A.C.) is a cross-platform and open source tool for approximating common aerodynamic parameters around aerofoil cross-sections subjected to supersonic flow conditions. It allows rapid prototyping iterative cycles to find optimal aerofoil geometries for desired flow output conditions. Aerofoil cross-sections can be sketched in 2D space, or loaded from the built-in presets library or save file. The angle of attack (AoA), flow inlet speed and inlet gas gamma $\left(\frac{c_p}{c_v}\right)$ for the resulting geometry can be adjusted to simulate a range of aerodynamic scenarios (twisted turbine blades, supersonic aircraft wings, etc.). Power related coefficients, such as the drag coefficient (C_D) and lift coefficient (C_L), are computed for each simulated case, and oblique shocks and expansion fans are rendered using colour-coded primitive shapes. Continue...

Supersonic aerofoils are present in a range of engineering applications. A few examples include turbines in aero engines and wings of supersonic aircraft.Continue...

This manual aims to cover (almost) every aspect of the S.A.C. program. It will be assumed you have no knowledge of how S.A.C. works or how to use it. However, knowledge of supersonic flow may be useful (although it is not required to use the program). Continue...

This program is designed to operate in the supersonic region, and does not work when a bow shock is formed or in subsonic regions.

NOTE: This manual is a work in progress. There will be some differences between images you see in this manual and the current version of the program.

Essentials run-through

Introduction

This chapter will guide you through a quick but detailed run through of how to use the fundamental aspects of the program, from start to finish. From sketching/loading, to calculating and post-processing. This will be broken down into two major stages: **(1) Aerofoil preparation**, and **(2) Calculations**

(1) Aerofoil preparation

When the program is first launched, you are confronted with a menu similar to this:

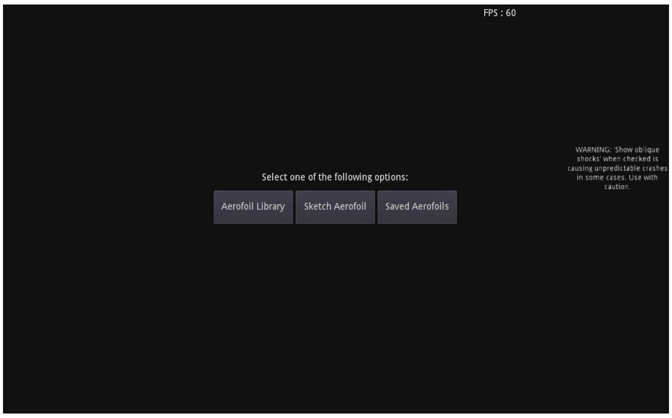


Figure 1: Start menu

There are three options to choose from:

- **Aerofoil Library** – choose from the built-in presets
- **Sketch Aerofoil** – sketch your own aerofoil
- **Saved Aerofoils –** load a saved aerofoil

Each option will be discussed in detail in the following subsections below.

(1.1) Aerofoil Library

As mentioned earlier, there are a number (two as of this writing) of pre-defined aerofoils which come out-of-the-box with S.A.C.. When the **Aerofoil Library** button is pressed, a popup list containing a list of presets will appear in the top left corner of the screen.

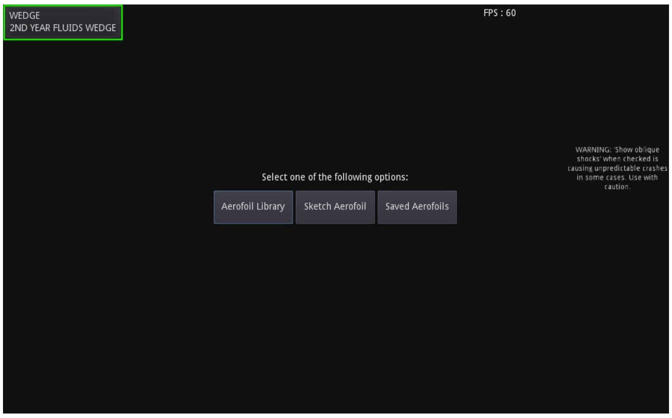


Figure 2: Aerofoil library popup

Select a preset from the menu (LMB click), and the aerofoil will be loaded in. The **2ND YEAR FLUIDS WEDGE** preset will be used throughout this manual.

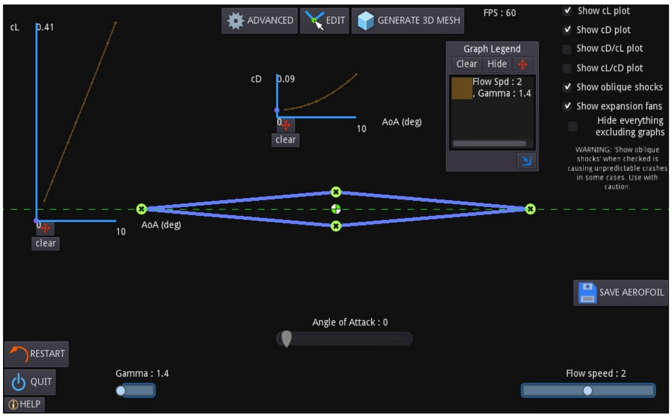


Figure 3: Post-selection stage. (Note: aerofoil will look different to this image based on which preset you select)

Once the aerofoil has been selected and loaded in, you are ready to move onto the next stage (put section number here)

(1.2) Sketch Aerofoil

Aerofoil's can be manually constructed in 2D space using the sketch function. This works by adding points (green circles with black cross) in screen-space pixel coordinates, called *nodes*, which connect to form line segments (light blue lines), called *plates*. These plates form the surface of the aerofoil.

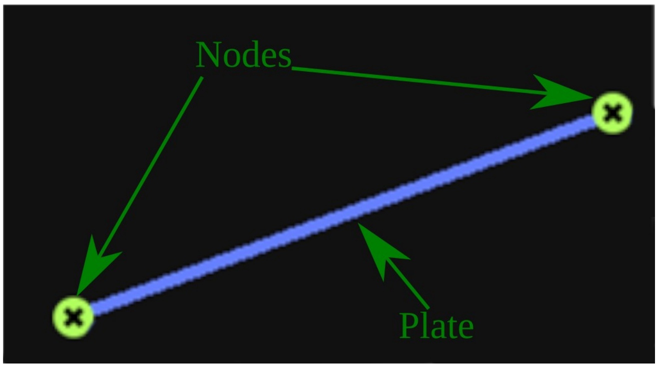


Figure 4: Nodes and plate

To create a sketch, select the **Sketch Aerofoil** button from the start menu (see Figure 1). This will display a series of new GUI elements and text boxes, since sketch mode is entered.

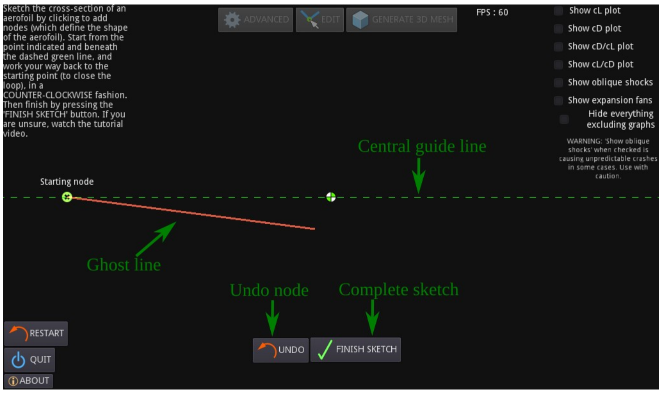


Figure 5: Sketch mode

There a few things of importance to notice:

- Blinking orange ‘ghost’ line - acts as a visual representation of how the line segment (or *plate*) will look like once the LMB is released
- **UNDO** button – goes one step back if you make a mistake when sketching.
- **FINISH SKETCH** button – completes the sketch by closing the aerofoil, and exits sketch mode
- Central dashed guide line – acts as a visual guide which helps during sketching and visualising AoA

Guidelines for good sketches

In order for the simulation to run, an aerofoil must have an ‘acceptable’ sketch. This depends on multiple factors, and the most important being the relative deflection angles between plates. These relative deflection angles should be kept relatively small. Currently, there is no easy way of accurately defining good or bad deflection angles, since parameters such as flow speed and gamma also have an impact.

(1.3) Saved Aerofoils

A saved aerofoil (see section….) can be loaded in using the file manager. To access and load a saved file, select the **Saved Aerofoils** button from the start menu (see Figure 1). This will open the saved files dialogue box, and a list of saved aerofoil files will be displayed.

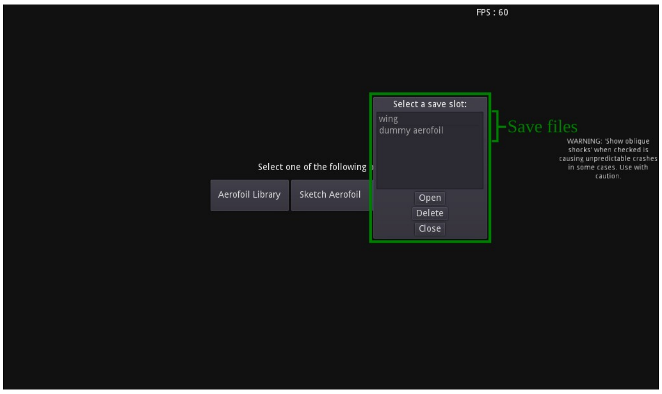


Figure 6: Loading a saved file

To select a save file, simply LMB click on the name of the file. There are three options to choose from:

- **Open** – Loads the aerofoil file
- **Delete** – Deletes the aerofoil file from memory
- **Close** – Closes the dialogue box

Once you have loaded an aerofoil (Select file > **Open**), you are ready to move onto the next stage (put section number here)

(2) Calculations

After you have sketched or loaded your aerofoil, it is time to move onto the calculations phase. This is where plots are generated and aerodynamic parameters are computed. This section is broken down into two categories: **(2.1) Running simulation**, and **(2.2) Displaying results**.

(2.1) Running simulation

There are three sliders which can be individually adjusted:

- **Angle of Attack –** rotates the aerofoil about the pivot point. In degrees
- **Flow speed** – changes the ratio of inlet flow speed (in m/s) to speed of sound in air at inlet (in m/s). A dimensionless quantity
- **Gamma** – changes the ratio of C_p (pressure coefficient) to C_v (volume coefficient) at inlet. A dimensionless quantity

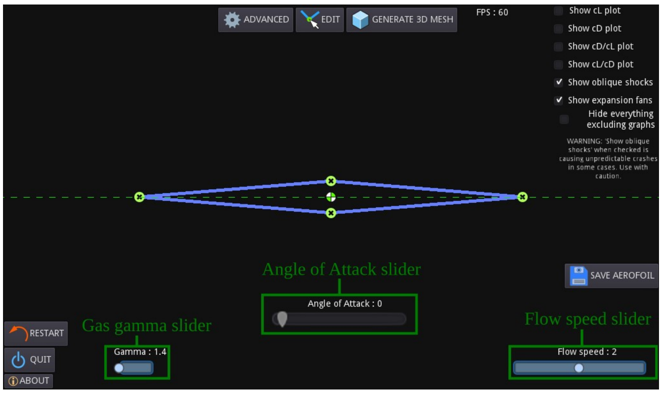


Figure 7: Sliders

Changing any of these sliders, by clicking and dragging the handles, causes the simulation to update. However, different sliders cause different updates. Adjusting the **Angle of Attack** slider causes flow speeds, pressure ratios, coefficients, etc. to be computed at the new angle, at fixed values of gas gamma and inlet flow speed. But adjusting either **Gamma** or **Flow speed** sliders causes computation of coefficients at 0.1 degrees increments up to bow shock angle. In the case of **Gamma**, the coefficients are computed at 0.1 degrees increments (increasing AoA up to bow shock) at a fixed inlet flow speed. And in the case of **Flow speed**, the coefficients are computed at 0.1 degrees increments (increasing AoA up to bow shock) at a fixed gas gamma. Any updates to the simulation can be visualised through the oblique shock lines and expansion fan triangles, provided the checkboxes are ticked (which they are by default). A detailed explanation of reviewing simulation update parameters will be discussed next (section 2.2).

(2.2) Displaying results

There are a number of ways to visualise various output parameters calculated by S.A.C.. These include colour-coded primitive shapes, numerical data tables, and coefficient plots. A detailed explanation for each type will be discussed below.

(2.2.1) Visualising oblique shocks and expansion fans

The magnitude and angle for oblique shocks and expansion fans for each plate can be visualised by enabling **Show oblique shocks** and **Show expansion fans** checkboxes (they should be enabled already by default).

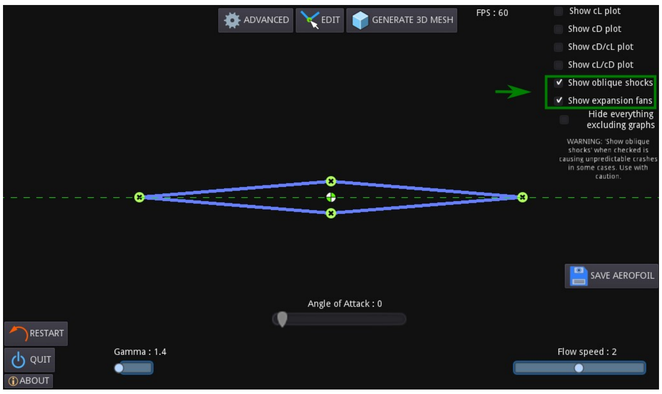


Figure 8: Oblique shock and expansion fan checkboxes

To visualise oblique shocks and expansion fans, simply change any of the three sliders, i.e. **Angle of Attack**, **Gamma** or **Flow speed.** Adjusting the **Angle of Attack** slider calculates the relevant parameters (i.e., C_L, C_D, gamma, pressure ratios, etc.) for fixed values of **Gamma** and **Flow speed**.

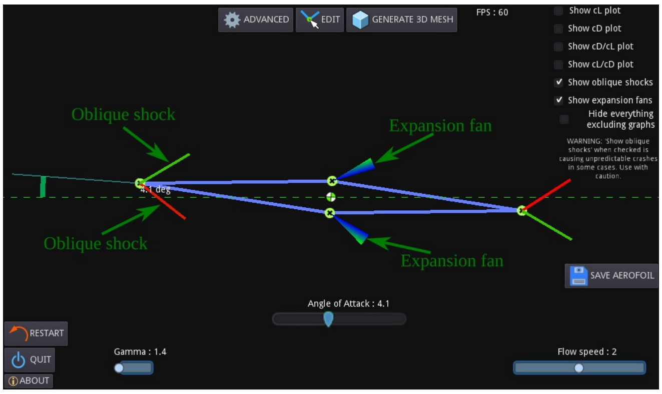


Figure 9: Oblique shocks and expansion fans

Oblique shocks are represented as solid coloured lines, with their colour representing the magnitude of the pressure ratio on that plate (pressure on plate divided by inlet pressure) relative to the largest pressure ratio acting on the aerofoil. A solid red colour indicates that the pressure ratio on the respective plate is the largest acting on the aerofoil. Whereas a solid blue colour indicates that the pressure ratio on the respective plate is the smallest in magnitude acting on the aerofoil. Any colour in between red and blue (i.e. orange, green, cyan, etc.) indicates the magnitude lies in between maximum and minimum pressure ratios (e.g. green represents a pressure ratio that lies exactly in between maximum and minimum).

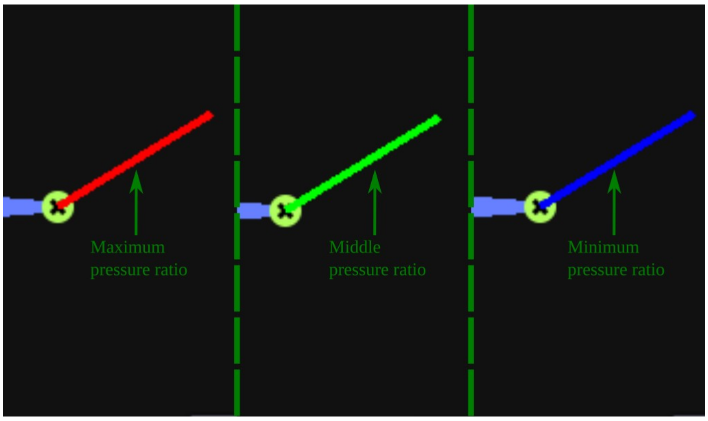


Figure 10: Pressure ratio colour guide

Expansion fans are rendered as triangles having a green-to-blue colour gradient. This gradient does not represent any magnitude or parameter; it's merely chosen to clearly display the expansion triangle (useful when triangles begin to overlap on aerofoils with a dense node concentration). The width of the triangle represents the angular range in which expansion fans exists. As the **Angle of Attack**, **Gamma** or **Flow speed sliders** are adjusted, the width of the triangles may (or the expansion range), in some cases, increase.

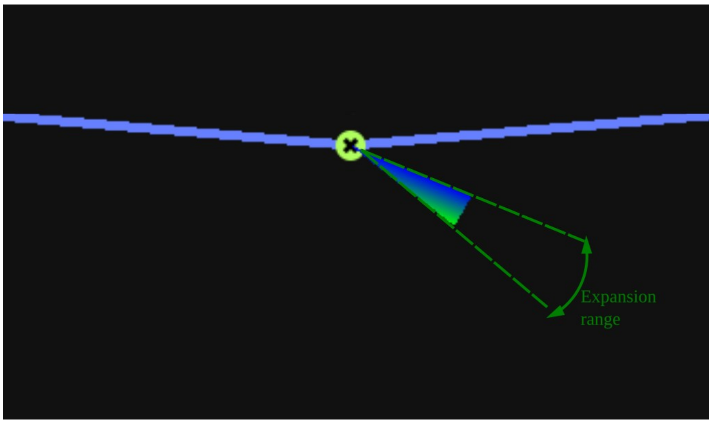


Figure 11: Expansion range

(2.2.2) Coefficient plots

(2.2.3) Numerical data tables

Numerical tables display computed parameters in numerically in decimal form for each plate. Different values can be shown, including but not limited to, pressure ratio, flow speed, C_L, C_D, etc.. To display a table for a specific parameter, click on the **ADVANCED** button in the top bar. This will open a popup list with options to choose from, highlighted in the dashed orange box.

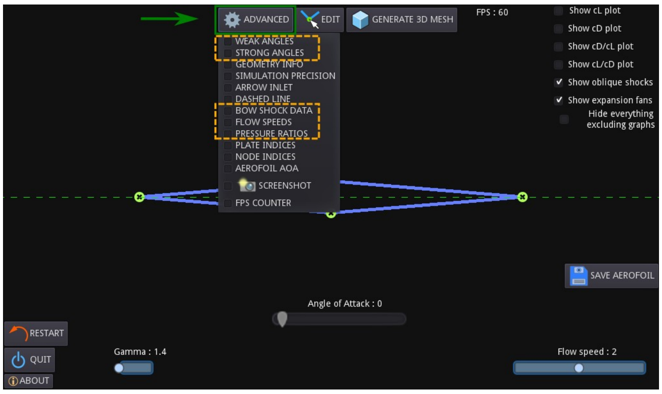


Figure 12: Numerical table popup list

As of the current S.A.C. version (v0.2), there are a total of five data tables which can be displayed:

- **WEAK ANGLES** – shows values for weak shock angles (in radians) for oblique shocks. A value of 0 means no oblique shocks for that plate.
- **STRONG ANGLES** – shows values for strong shock angles (in radians) for oblique shocks. A value of 0 means no oblique shocks for that plate. **Note:** this function is not fully complete, since it currently only calculates strong shocks for all regions excluding the trailing edge. Also note that enabling this option will decrease performance (by different magnitudes based on the hardware used), since extra calculations have to be carried out to compute the strong angles, and these calculations are computationally expensive.
- **BOW SHOCK DATA** – relays data about when a bow shock has just formed. As mentioned earlier, this program is not designed to operate in bow shock territory, so this is the

maximum limit that can be calculated for all aerodynamic parameters. There are four parameters:

- **AoA** – is the angle of attack (in degrees) when a bow shock is just formed
- **cL** – is C_L when a bow shock is just formed
- **cD** – is C_D when a bow shock is just formed
- **cD/cL** – is the ratio $\left(\frac{C_D}{C_L}\right)$ when a bow shock is just formed

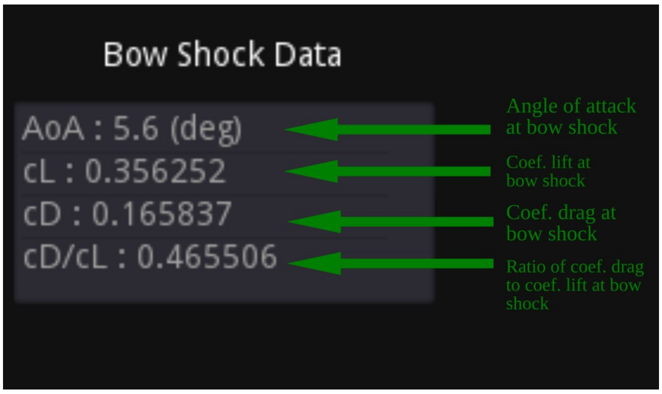


Figure 13: Bow shock data table

- **FLOW SPEEDS –** displays the flow speeds (relative to speed of sound in air) for each plate. A dimensionless parameter.
- **PRESSURE RATIOS** – displays the pressure ratios (pressure on plate divided by inlet pressure $\left(\frac{P_{plate}}{P_{inlet}}\right)$) for each plate. A dimensionless parameter.

When a parameter is selected from the popup list, its data table will show in the left-hand section of the display. To hide that data table, click on the **ADVANCED** button and select the name which corresponds to the table from the popup list.

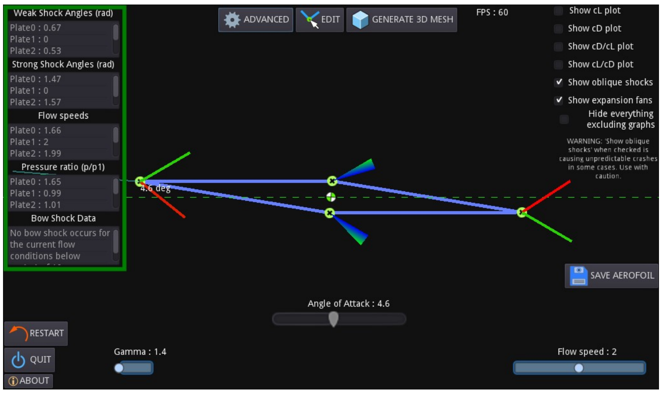


Figure 14: Data tables

Edit Aerofoil Tools

Once an aerofoil is sketched or loaded in, you may wish to change the geometry of the aerofoil in certain ways. S.A.C. provides a built-in geometry editor which provides basic editing functionality to adjust an aerofoil to conform to the users' specification. To access the edit tools, click on the **EDIT** button in the top bar. This will open a popup list with a number of options to choose from:

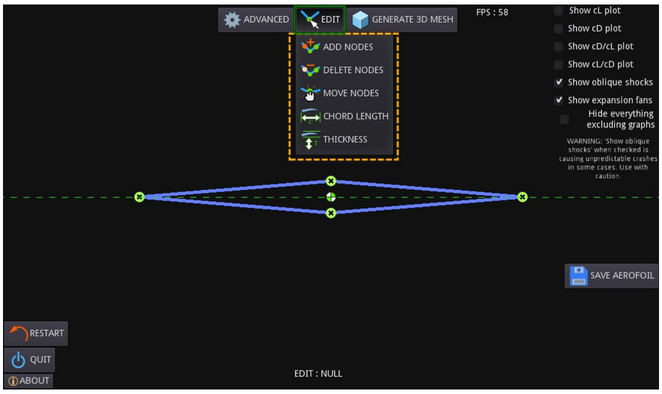


Figure 15: Edit Tools menu

- **ADD NODES** – allows the manual placement of extra nodes along plates
- **DELETE NODES** – manually delete selected nodes (excluding the starting node and provided the total node count exceeds 3)
- **MOVE NODES** – manually adjust the position of nodes
- **CHORD LENGTH** – adjust the global length of the chord (acts on all nodes)
- **THICKNESS** – adjust the global thickness of the aerofoil (acts on all nodes)

Use of each mode will be discussed in detail below.

ADD NODES

Once the **ADD NODES** option is selected from the popup list, the aerofoil will start to blink light green, indicating the regions which nodes can be added along.

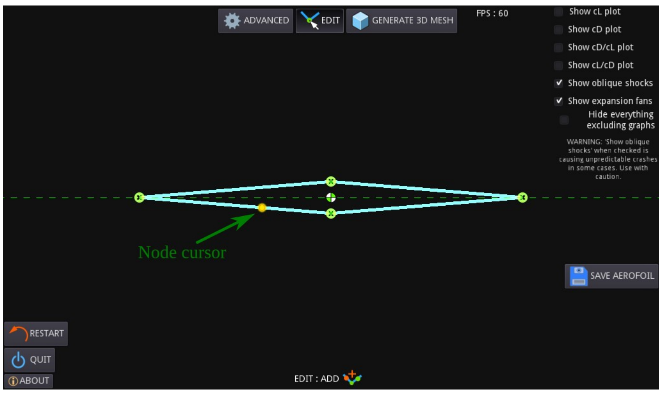


Figure 16: ADD NODE mode

Hovering over a plate of the aerofoil reveals an orange ‘Node cursor’. This acts as a visual representation of where the node can be placed along the aerofoil. LMB click to add a node at that location (provided the mouse cursor is on a plate).

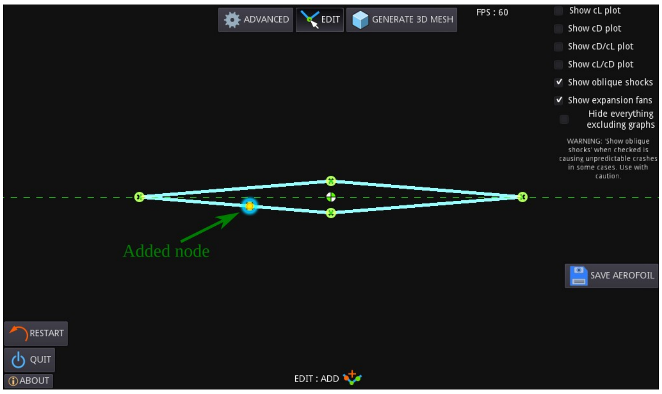


Figure 17: Added node on plate

Once you are complete, press the **EDIT** button to exit.

DELETE NODES

Once the **DELETE NODES** option is selected from the popup list, a node can be deleted by hovering over it and LMB clicking. Hovering over a node changes its appearance from green to blue.

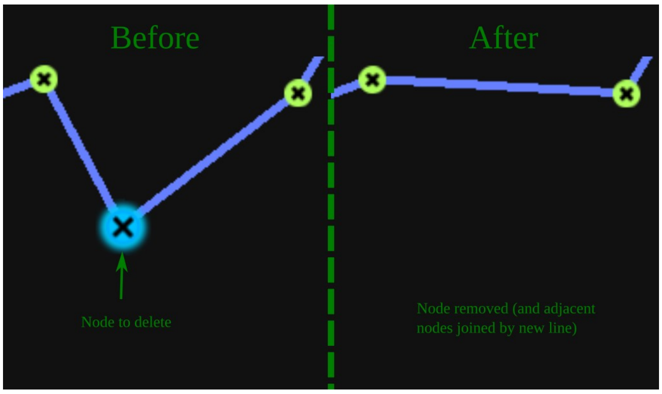


Figure 18: Figure 18: Delete node illustration

Once you are complete, press the **EDIT** button to exit.

MOVE NODES

Once the **MOVE NODES** option is selected from the popup list, a node can be moved by LMB clicking it and dragging. **Note:** it is easy to cause steep plate gradients when adjusting nodes. This will give an error message, since bow shock(s) are usually formed as a result (try and keep plate gradients closer to the horizontal than vertical).

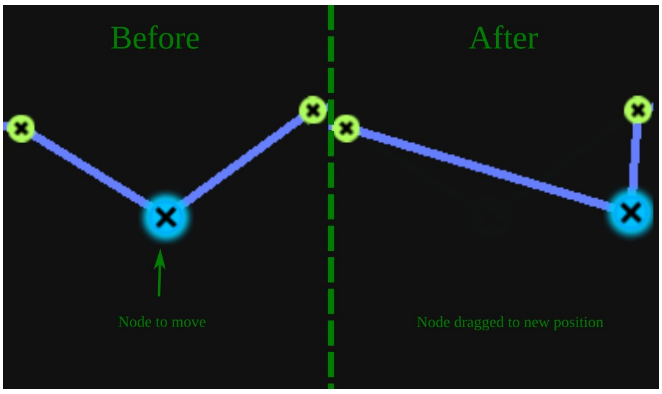


Figure 19: Figure 19: Move node illustration

Once you are complete, press the **EDIT** button to exit.

CHORD LENGTH

Once the **CHORD LENGTH** option is selected from the popup list, a new **Chord Slider** shows up next to the **SAVE AEROFOIL** button.

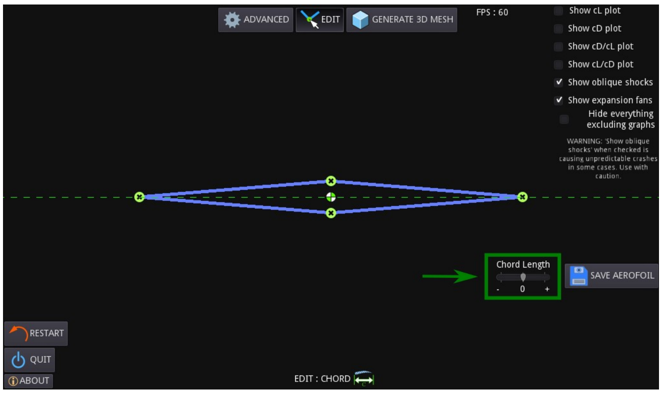


Figure 20: Chord Slider

Moving the slider knob towards the **+** increases the chord length. Whilst moving it towards the **-** decreases the chord length. A value of **0** means no changes made to the chord length.

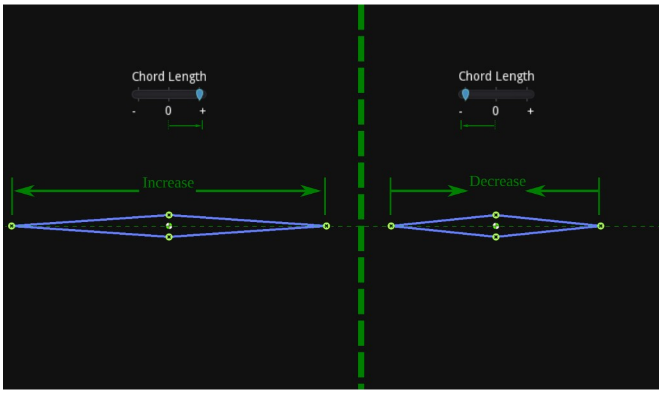


Figure 21: Adjust chord length

Once you are complete, press the **EDIT** button to exit.

THICKNESS

Once the **THICKNESS** option is selected from the popup list, a new **Thickness** slider shows up near the **SAVE AEROFOIL** button.

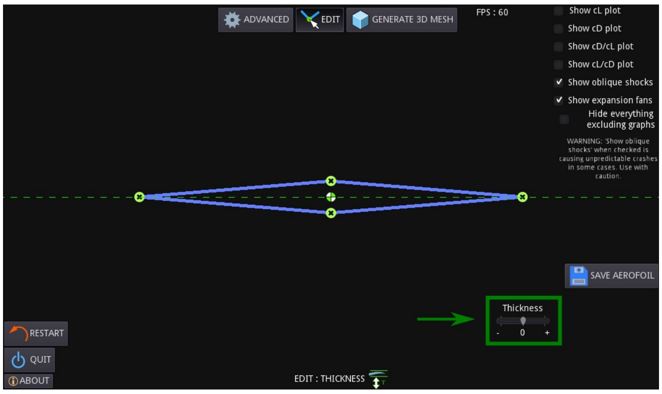


Figure 22: Thickness slider

Moving the slider knob towards the **+** increases the global thickness of the aerofoil. Whilst moving it towards the **-** decreases the global thickness. A value of **0** means no changes made to the chord length. **Note:** in the current version (v0.2), moving the slider towards **-** can result in the aerofoil being inverted for some geometries. This is an issue that is yet to be fixed, however use with caution.

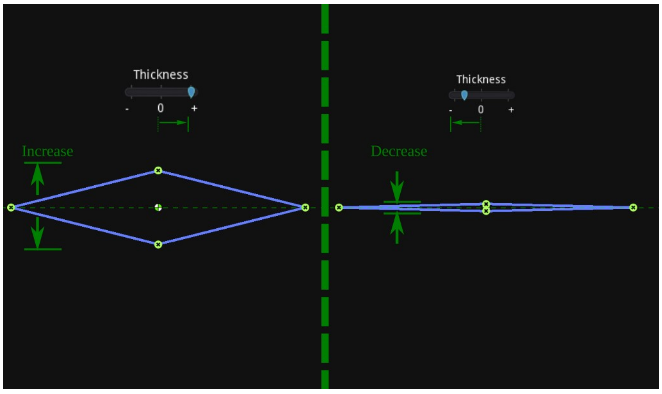


Figure 23: Adjust global thickness

Generating 3D Meshes

S.A.C. provides an integrated mesh viewer for generating and visualising aerofoils in 3D space. Any sketch or preset can be converted to an extruded 3D mesh, which can either be visualised as a rotating turbine, or a single blade. To generate a mesh, click the **GENERATE 3D MESH** button.

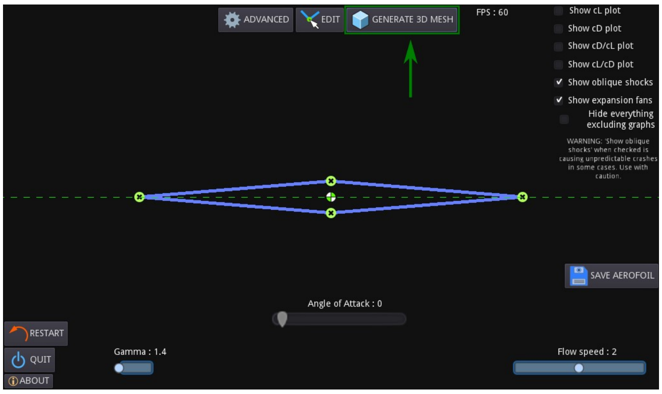


Figure 24: GENERATE 3D MESH button

This will take you to the **MESH VIEWER** scene, where you are greeted with a rotating turbine.

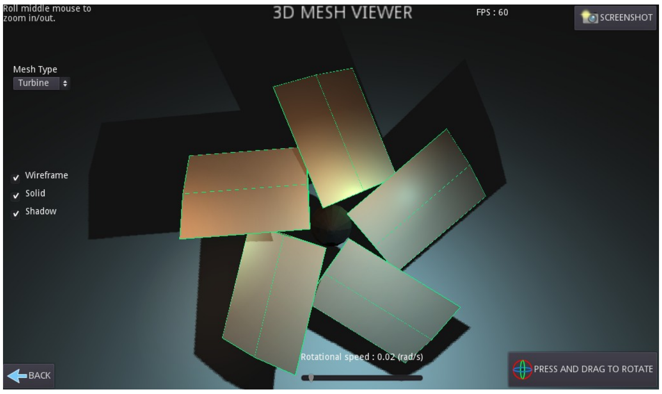


Figure 25: MESH VIEWER scene

To rotate around the mesh, LMB click and drag on the **PRESS AND DRAG TO ROTATE** button. This will gimbal rotate the camera around the mesh. To move closer or further away from the mesh, roll the middle mouse wheel up/down. To change the rotational speed of the turbine, simply adjust the **Rotational Speed** slider.

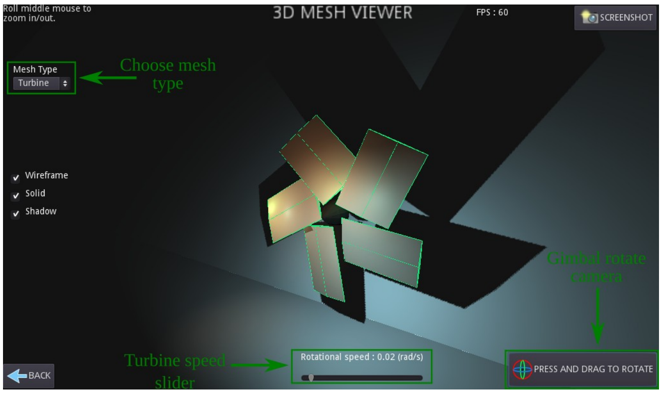


Figure 26: Interact with mesh GUI

To visualise a single blade, click the drop-down button under **Mesh Type**, and select **Single Blade**.

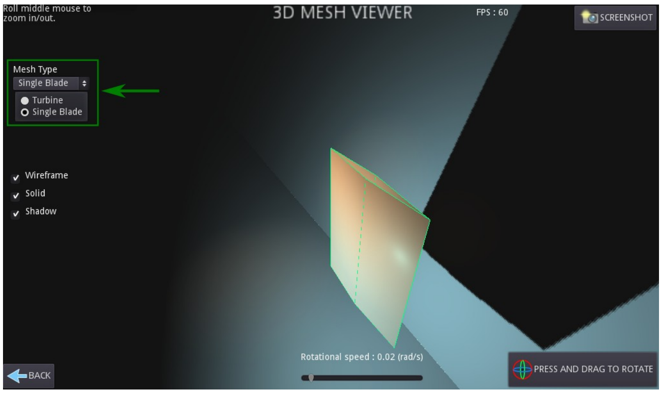


Figure 27: Mesh Type selector

Meshes can also be viewed using **Wireframe** or **Solid** geometry mode using the checkboxes near the left edge of the screen.

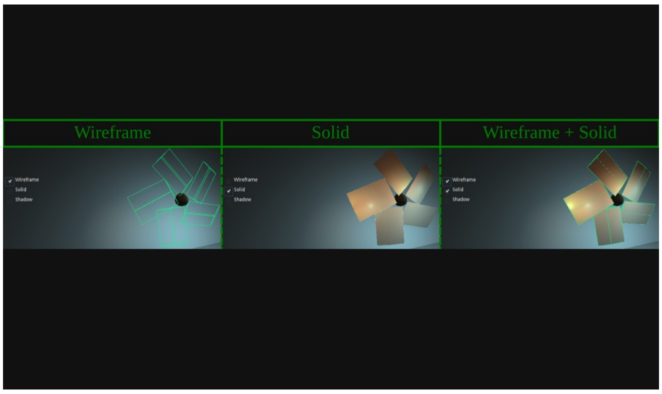


Figure 28: Wireframe and Solid modes

Saving an Aerofoil

Any aerofoil can be saved by pressing the **SAVE AEROFOIL** button. This will open a save dialogue where you can name your save file. Enter a name for the save file. Names cannot contain any of the following symbols **\ / < > * ? : |**, and they cannot start with a number. Press **Save** once finished.

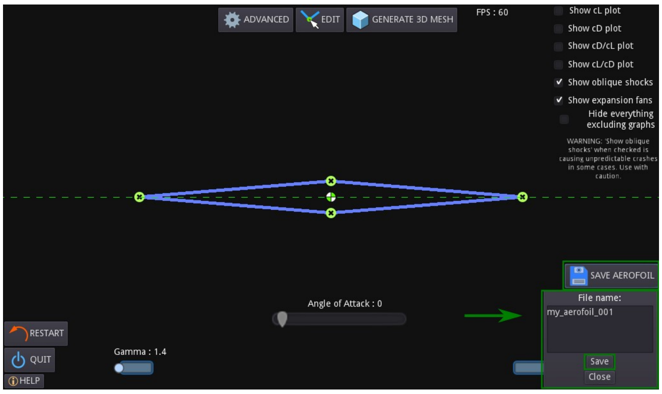


Figure 29: Saving an Aerofoil

To load the saved aerofoil, refer to section 1.3.

Extra tools

There are various optional functions which can be grouped together as being miscellaneous. These are currently all located via the **ADVANCED** button.

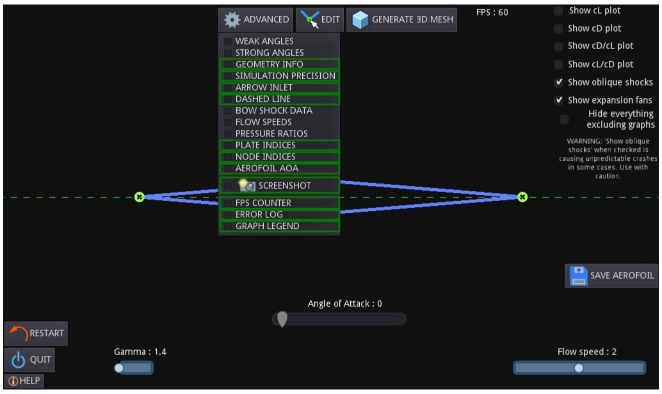


Figure 30: Extra options

- **GEOMETRY INFO** – Selecting this option displays an option button named **Aerofoil Info**. Clicking on the **Aerofoil Info** option button to reveals geometric data:
  - **T/C** – is the ratio of maximum thickness (**T**) to chord length (**C**)
  - **Thickness (px)** – maximum thickness in pixels
  - **Chord Length (px)** – chord length in pixels
  - **Total number of nodes** – cumulative total node count

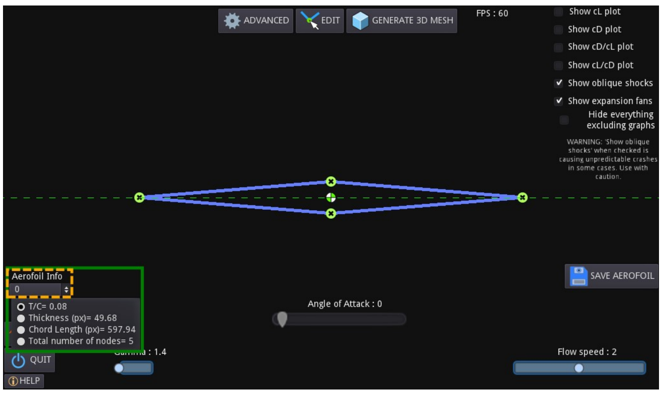


Figure 31: GEOMETRY INFO

- **ITERATION RESOLUTION –** Selecting this option displays a slider which allows the user to directly change the convergence criteria used by the numerical bisection method (used to solve oblique shocks). Minimum value is 0.01, maximum value is 0.0000001. **NOTE**: it is recommended to keep the default value. Decreasing the slider will reduce solution accuracy, and produce rough plots and jitter.

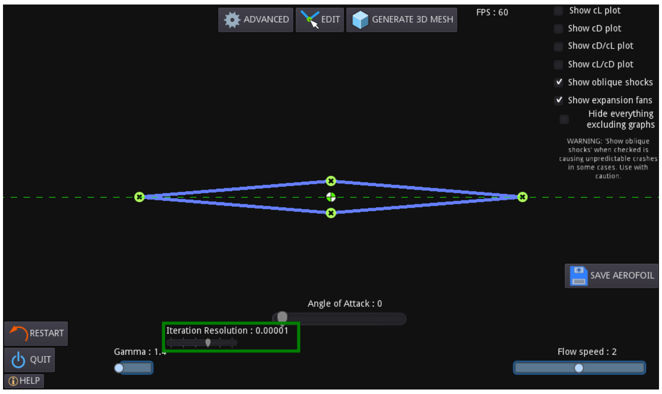


Figure 32: ITERATION RESOLUTION

- **DASHED LINE** – turns on/off the visibility for central green line
- **PLATE INDICES** – turns on/off index labels for each plate. Each plate has an index of the form ***xp*** , where ***x*** represents the number, and **p** means ‘plate’.

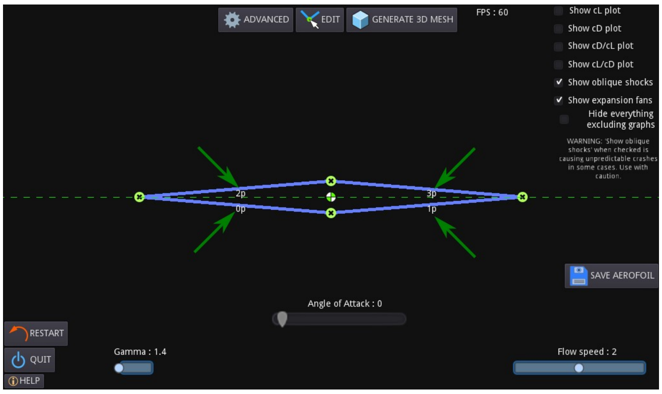


Figure 33: PLATE INDICES

- **NODE INDICES** – turns on/off index labels for each node. Each node has an index of the form ***x*** , where ***x*** represents the number, and **n** means ‘node’.

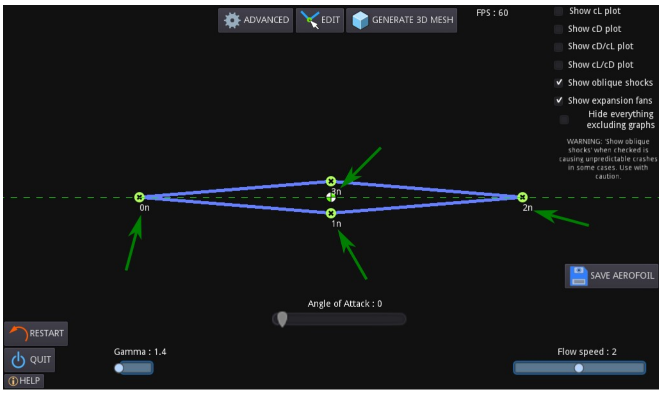


Figure 34: NODE INDICES

- **FPS COUNTER** – turns on/off the frames per second (FPS) label.
- **AEROFOIL AOA** - turns on/off the angle of attack visualiser

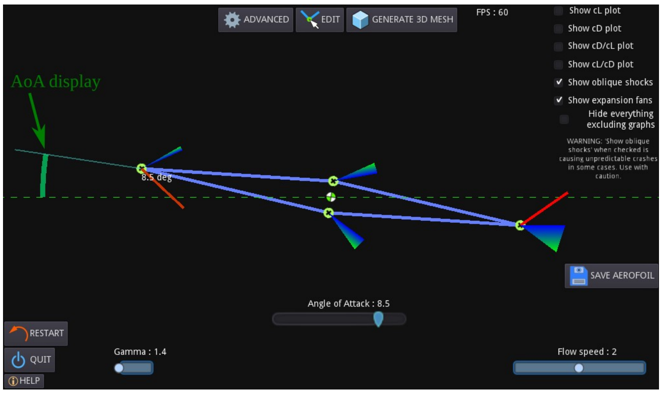


Figure 35: AEROFOIL AOA

- **SCREENSHOT** – allows snapshots to be saved as PNG files. Press to reveal the **SCREENSHOT** button near the right edge of the display. Click this button to reveal the save dialogue box. Choose a directory, then enter a name (for the image) and click OK.

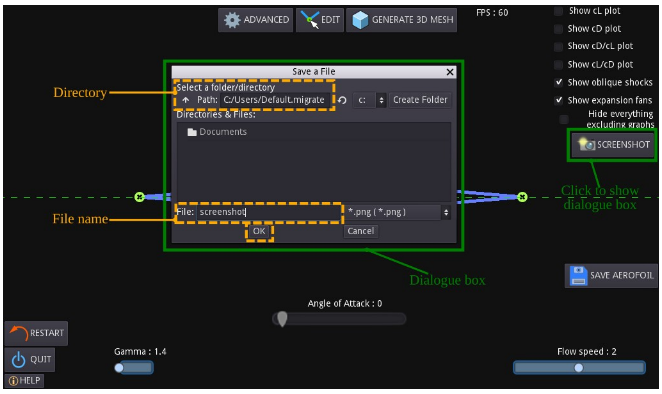


Figure 36: SCREENSHOT

Help Menu

The **HELP** menu contains various resources covering the basics of using S.A.C.. It can be accessed continue...