AERO97051 Applied Computational Aerodynamics

Computer Laboratory Guides

Jan N. Rose, Jan R. Eichstädt and Joaquim Peiró

2019-2020

Chapter 1

XFOIL

1.1   Introduction to XFOIL

XFOIL is an interactive program for the design and analysis of subsonic isolated airfoils[1] written by Mark Drela (Aeronautics and Astronautics department at MIT) and Harold Youngren (Aerocraft, Inc.). It consists of a collection of menu-driven routines which perform various useful functions such as viscous (or inviscid) analysis of an existing airfoil, airfoil design and redesign by interactive modification of surface speed distributions or geometric parameters, and blending of airfoils. It also provides facilities for writing and reading of airfoil coordinates and polar save files and for plotting of geometry, pressure distributions, and multiple polars.

XFOIL is an open-source code and is available to download from `http://web.mit.edu/drela/Public/web/xfoil/` should you want to use it in your own computer. Documentation is available in that page and also in the Blackboard Learn page of the course.

This tutorial will focus mainly on the description of some of the commands available in XFOIL for the analysis of airfoils. Commands for the design of airfoils are also available but will not be covered in this short guide. However, once you master the following sections, it should be relatively straightforward to acquaint yourself with these additional options.

1.2   Opening XFOIL

This section is specific to the MS Windows installation of XFOIL in the PC computer clusters of Imperial College London. To start the process, go to the software hub (`https://softwarehub.imperial.ac.uk/`) and download/install "XFOIL".

You can now launch XFOIL from the launcher window (it should also be in the start menu).

1.3   Running XFOIL

In launching XFOIL you will be presented with some text on the screen that, in Version 6.97, looks like this:

```
===================================================
  XFOIL Version 6.97
  Copyright (C) 2000   Mark Drela, Harold Youngren

  This software comes with ABSOLUTELY NO WARRANTY,
    subject to the GNU General Public License.
```

---

[1]Note that in this document we follow the American spelling of wing sections, i.e. airfoil, and not aerofoil as used in the Lecture Notes, to be consistent with the original documentation and online help for the XFOIL code.

```
  Caveat computor
 ===================================================

 File  xfoil.def  not found

   QUIT      Exit program

  .OPER     Direct operating point(s)
  .MDES     Complex mapping design routine
  .QDES     Surface speed design routine
  .GDES     Geometry design routine

   SAVE f  Write airfoil to labeled coordinate file
   PSAV f  Write airfoil to plain coordinate file
   ISAV f  Write airfoil to ISES coordinate file
   MSAV f  Write airfoil to MSES coordinate file
   REVE    Reverse written-airfoil node ordering
   DELI i  Change written-airfoil file delimiters

   LOAD f  Read buffer airfoil from coordinate file
   NACA i  Set NACA 4,5-digit airfoil and buffer airfoil
   INTE    Set buffer airfoil by interpolating two airfoils
   NORM    Buffer airfoil normalization toggle
   HALF    Halve the number of points in buffer airfoil
   XYCM rr Change CM reference location, currently  0.25000 0.00000

   BEND    Display structural properties of current airfoil

   PCOP    Set current-airfoil panel nodes directly from buffer airfoil points
   PANE    Set current-airfoil panel nodes ( 160 ) based on curvature
  .PPAR     Show/change paneling

  .PLOP     Plotting options

   WDEF f  Write  current-settings file
   RDEF f  Reread current-settings file
   NAME s  Specify new airfoil name
   NINC    Increment name version number

   Z       Zoom    | (available in all menus)
   U       Unzoom  |

 XFOIL   c>
```

This text tells you the version of XFOIL you are using and list the commands that are available *at this level*. The commands prefixed by a dot "." will take you to another level where a different set of commands will be accessible.

Notice that there are three columns, the first is the command, the second one gives an indication of other inputs the command needs. These arguments mean the following: "`r`" means that the command expects a real number; "`i`" means that the command expects an integer; "`f`" means that the command expects a filename; and "`s`" that the command expects a string. If the input is not typed after the command XFOIL will prompt the user.

The message "`File xfoil.def not found`" indicates that a file of start-up defaults is not present.[2]

A word of warning: XFOIL is not foolproof. Read carefully the section on "caveats" at the end of the MIT manual to

---

[2]XFOIL has hardwired parameters controlling the paneling, plotting, and viscous execution. Most of these can be changed at runtime in the various menus. A default file `xfoil.def` can be saved to avoid the need to change the parameters everytime XFOIL is executed. This file is read at any time with the `RDEF` command.

avoid potential pitfalls.

1.3.1   Defining the airfoil geometry

The geometry of the airfoil can be defined using the "`LOAD`" or "`NACA`" commands. Initially we will consider a NACA 2412 airfoil. To define this airfoil type

```
XFOIL c> NACA 2412
```

XFOIL then returns some of the specifications for the airfoil, including the location and magnitude of the maximum thickness, maximum camber, and other parameters relating to the paneling used, namely

```
Max thickness =     0.120034  at x =   0.301
Max camber     =    0.020000  at x =   0.398

Buffer airfoil set using 239 points

Blunt trailing edge.  Gap =  0.00252

Paneling parameters used...
  Number of panel nodes        160
  Panel bunching parameter    1.000
  TE/LE panel density ratio    0.150
  Refined-area/LE panel density ratio    0.200
  Top    side refined area x/c limits  1.000 1.000
  Bottom side refined area x/c limits  1.000 1.000
```

This is consistent with the NACA notation where the first "`2`" indicates that the maximum camber is 2%, the "`4`" says it is located at 4% of the chord and the final "`12`" tells us its thickness-to-chord ratio is 12%.

Note: The syntax of the commands is not case-sensitive, i.e. "`naca 2412`" will work too, but the names of the files that XFOIL reads are.

1.3.2   Inviscid flow past the NACA 2412 airfoil

To obtain flow variables and aerodynamic coefficients for the flow past this airfoil, type

```
XFOIL c> OPER
```

This will produce the prompt

```
.OPERi c>
```

Notice the letter "`i`" next to the symbol ".`OPER`" in the prompt. This indicates that XFOIL is in inviscid mode. The flow is also incompressible ($Ma = 0$) by default. Typing a question mark "?" at the prompt will display a list of available commands and a brief description of their use. This works on any level of XFOIL. For instance, at this level XFOIL shows:

```
  <cr>      Return to Top Level
  !         Redo last ALFA,CLI,CL,ASEQ,CSEQ,VELS

  Visc r   Toggle Inviscid/Viscous mode
 .VPAR     Change BL parameter(s)
  Re   r   Change Reynolds number
  Mach r   Change Mach number
```

```
  Type i    Change type of Mach,Re variation with CL
  ITER      Change viscous-solution iteration limit
  INIT      Toggle BL initialization flag

  Alfa r    Prescribe alpha
  CLI  r    Prescribe inviscid CL
  Cl   r    Prescribe CL
  ASeq rrr  Prescribe a sequence of alphas
  CSeq rrr  Prescribe a sequence of CLs

  SEQP      Toggle polar/Cp(x) sequence plot display
  CINC      Toggle  minimum Cp  inclusion in polar
  HINC      Toggle hinge moment inclusion in polar
  Pacc i    Toggle auto point accumulation to active polar
  PGET f    Read new polar from save file
  PWRT i    Write polar to save file
  PSUM      Show summary of stored polars
  PLIS i    List stored polar(s)
  PDEL i    Delete stored polar
  PSOR i    Sort stored polar
  PPlo ii.  Plot stored polar(s)
  APlo ii.  Plot stored airfoil(s) for each polar
  ASET i    Copy stored airfoil into current airfoil
  PREM ir.  Remove point(s) from stored polar
  PPAX      Change polar plot axis limits

  RGET f    Read new reference polar from file
  RDEL i    Delete stored reference polar

  GRID      Toggle Cp vs x grid overlay
  CREF      Toggle reference Cp data overlay
  FREF      Toggle reference CL,CD.. data display

  CPx       Plot Cp vs x
  CPV       Plot airfoil with pressure vectors (gee wiz)
 .VPlo      BL variable plots
 .ANNO      Annotate current plot
  HARD      Hardcopy current plot
  SIZE r    Change plot-object size
  CPMI r    Change minimum Cp axis annotation

  BL   i    Plot boundary layer velocity profiles
  BLC       Plot boundary layer velocity profiles at cursor
  BLWT r    Change velocity profile scale weight

  FMOM      Calculate flap hinge moment and forces
  FNEW rr   Set new flap hinge point
  VELS rr   Calculate velocity components at a point
  DUMP f    Output Ue,Dstar,Theta,Cf vs s,x,y to file
  CPWR f    Output x vs Cp to file
  CPMN      Report minimum surface Cp
  NAME s    Specify new airfoil name
  NINC      Increment name version number
```

These commands allow you carry out flow simulations about this airfoil. Here "`<cr>`" stands for the return key (or carriage return as used in the old-fashioned typewriters) which is most computers looks like this: ⏎ . For instance, to obtain the flow around the airfoil for a given angle of attack, say $\alpha = 3$ degrees, type

```
.OPERi c> alfa 3
```

A window pops up showing the pressure distribution, the section lift coefficient, the section moment coefficient, the angle of attack and the airfoil name. This is shown in Figure 1.1.
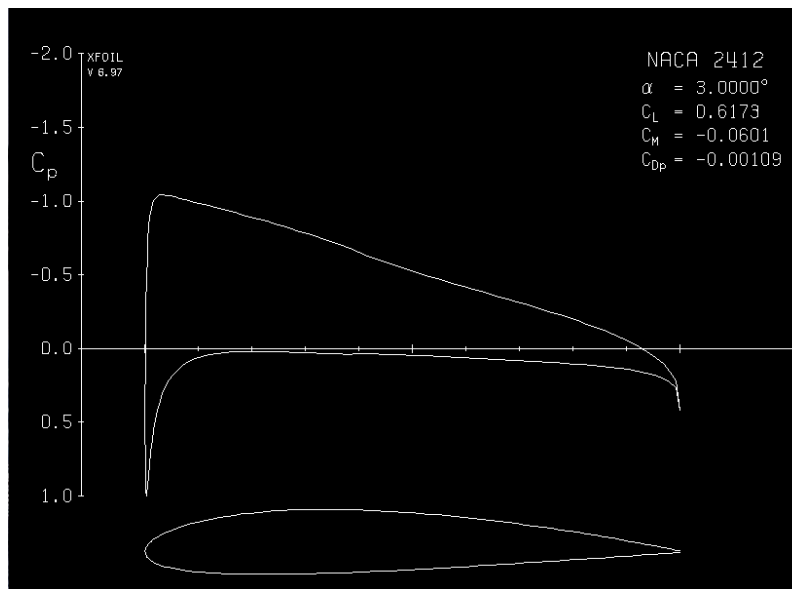

Figure 1.1: Screenshot of the *inviscid incompressible* C_p distribution about a NACA 2412 airfoil at an angle of incidence $\alpha = 3$ degrees.

1.3.3   Viscous flow past the NACA 2412 airfoil

To switch on to the viscous mode, type

```
.OPERi c> visc
```

XFOIL then asks the user to input a Reynolds number:

```
Enter Reynolds number   r>
```

Let us use a numerical value for the Reynolds number of "1e05" or "100000" both of which are acceptable. After entering this value, the prompt is now ".OPERv c>" where the letter "v" indicates viscous flow.

Let us repeat the calculation of the flow at an angle of incidence of three degrees. Type

```
.OPERv c> alfa 3
```

Upon pressing return, XFOIL displays some values at the steps of the iterative procedure to match the viscous and inviscid solutions. The last step in the iteration provides information on the residuals and the final aerodynamic coefficients, namely

```
 Side 1  free  transition at x/c =  0.7138   68
 Side 2 forced transition at x/c =  1.0000   76

   6   rms: 0.5320E-06   max: -.5429E-05   C at   76  2
       a =  3.000      CL =  0.6091
      Cm = -0.0624      CD =  0.01583   =>   CDf =  0.00760     CDp =  0.00823
```

This gives the location of transition to turbulent flow in the upper and lower surfaces, the values of the lift (`CL`), moment (`Cm`) and drag (`CD`) coefficients. The last one is split into friction (`CDf`) and pressure (`CDp`) drag.

The graph on the screen changes to that depicted in Figure 1.2 which now shows two pressure distributions. The dashed lines represent the inviscid flow distribution and the solid lines the viscous one. This permits the comparison of the viscous and inviscid flow solutions and identify those regions where viscous effects are important, for instance the region $0.5 \leq x/c \leq 0.8$ in Figure 1.2. The figure also outlines the boundary layer around the airfoil and gives the lift-to-drag ratio, the drag coefficient, and the value of N_{crit} used in the e^N model of transition.
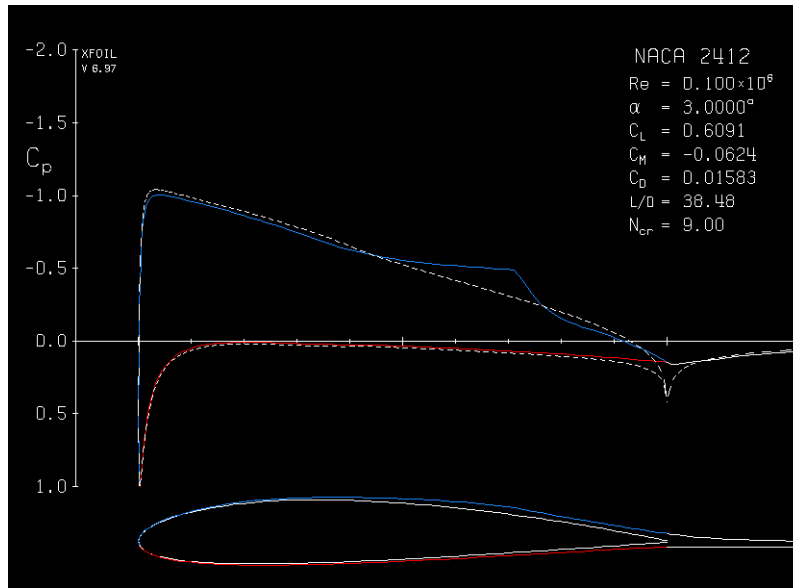

Figure 1.2: Incompressible viscous C_p distribution about a NACA 2412 airfoil at an angle of incidence $\alpha = 3$ degrees and a chord Reynolds number $\mathrm{Re} = 10^5$.

1.3.4 Boundary-layer variables

To investigate the viscous effects in the boundary layer of the aerofoil, we use the sub-command ".`VPlo`" of the command "`OPER`. Type

```
.OPERv   c>  vplo
```

The prompt now shows

```
..VPLO   c>
```

The two dots "`..`" indicate that we are two levels down from the main command menu. Invoking the help as it is customary by typing "?" shows us a new menu:

```
    <cr>    Return to OPER menu

    H       Plot kinematic shape parameter
    DT      Plot top    side Dstar and Theta
    DB      Plot bottom side Dstar and Theta
    UE      Plot edge velocity
    CF      Plot skin friction coefficient
    CD      Plot dissipation coefficient
    N       Plot amplification ratio
    CT      Plot max shear coefficient
```

```
RT      Plot Re_theta
RTL     Plot log(Re_theta)

DUMP f Write current plot variable to file
OVER f Overlay current plot variable from file

X  rrr Change x-axis limits
Y  rrr Change y-axis limits on current plot

BLOW    Cursor blowup of current plot
RESE    Reset to default x,y-axis limits
SIZE r Change absolute plot-object size
.ANNO    Annotate plot
HARD    Hardcopy current plot

GRID    Toggle grid plotting
SYMB    Toggle node-symbol plotting
LABE    Toggle label plotting
CLIP    Toggle line-plot clipping
FRPL    Toggle TS frequency plotting
```

The discrepancies in the values of the pressure distribution for the inviscid and viscous solutions in the region $0.5 \leq x/c \leq 0.8$ shown in Figure 1.2 could be due to flow separation, transition to turbulence, or both. To elucidate if separation is present we look at the skin friction coefficient by typing

```
..VPLO   c>  cf
```

XFOIL displays the results shown in Figure 1.3. Negative values of C_f are evidence of separation, therefore we can observe that the flow in the pressure side of the airfoil is attached, and that the suction side shows a separation bubble in the region $0.4 \leq x/c \leq 0.75$, approximately. We could also verify if transition to turbulence flow has
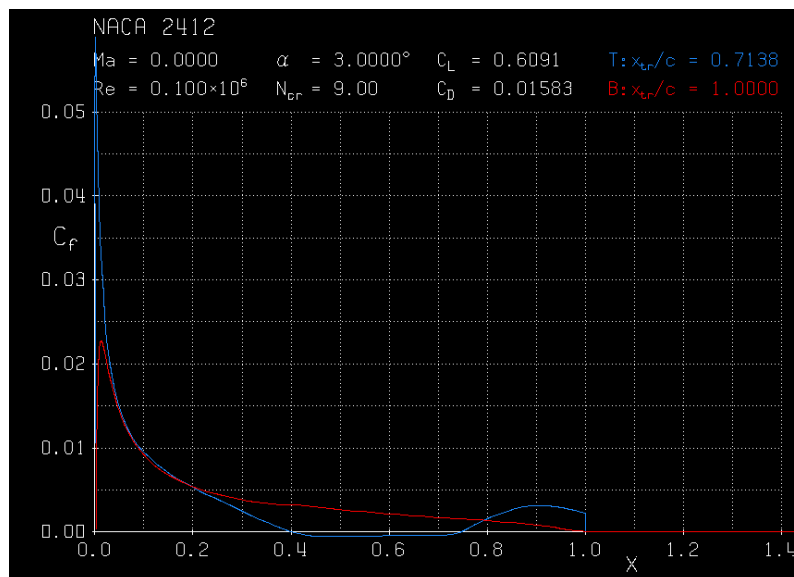

Figure 1.3: Skin friction coefficient distribution for the incompressible flow about a NACA 2412 airfoil ($\alpha = 3°$ and $\mathrm{Re} = 10^5$).

occurred. The legend in Figure 1.3 shows a value of $x_{\mathrm{tr}}/c = 0.7138$ in blue (or top surface) and $x_{\mathrm{tr}}/c = 1.0000$ in red (or bottom surface) which indicates the flow is laminar throughout the bottom surface and laminar on the top surface between the leading edge ($x/c \sim 0$) and the transition point $x_{\mathrm{tr}}/c = 0.7138$. Another way of analysing this behaviour in the boundary layer is to plot the displacement and momentum thicknesses in the top suface by typing

```
..VPLO   c>  dt
```

The results are shown in Figure 1.4 where we can observe an increase in δ^* due to separation followed by a reasonably steep decrease near the transition point. The sudden change of slope of θ at $x/c \sim 0.7$ provides further evidence of transition to turbulent flow there. Note that the graphs for δ^* and θ are not especifically identified. To help you do this, they are the solid lines in the plot and furthermore you know that $H > 1$.
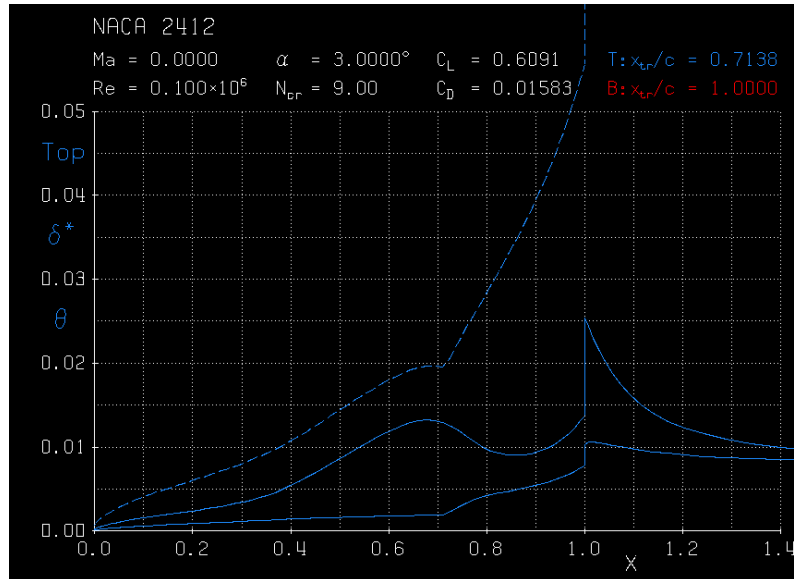

Figure 1.4: Top-surface boundary-layer displacement and momentum thickness distributions for the incompressible flow about a NACA 2412 airfoil ($\alpha = 3°$ and $\mathrm{Re} = 10^5$).

1.4 Getting XFOIL output to your report

Figures in the previous sections have used the screen output of XFOIL, that uses a black background, for the purpose of illustrating what the screen output of XFOIL looks like. Such choice is suitable for screen displays, but you are strongly advised not to use screenshots in your report. As you might have noticed here, such background makes it very difficult to read text and interpret graphs in the relatively small format of the figures used in this report. Here there are two alternative options. Choose the one that suits you best.

1.4.1 Postscript output

To get a copy in postscript format of the displayed plot, i.e. the last issued command, type

```
..VPLO   c> hard
```

A postscript file named `plot.ps` will be produced[3]. You will not be able to open this file until you exit XFOIL. However, any other files that you hardcopy *will be appended* to the file `plot.ps`. You will not be able to fully access all the plots until you quit the XFOIL program. As an example, the same graph of Figure 1.4 is reproduced in this format in Figure 1.5. Here a white background with black lines makes it easy to read in this report format.

1.4.2 Saving to a file

The same information can be output to a file in plain text format, say "`DT.dat`" by typing

---

[3]This file goes to the directory you are running XFOIL from. You will need programs such as "GSview/Ghostscript" to read it and display it, or "Acrobat distiller" to translate it onto pdf format.
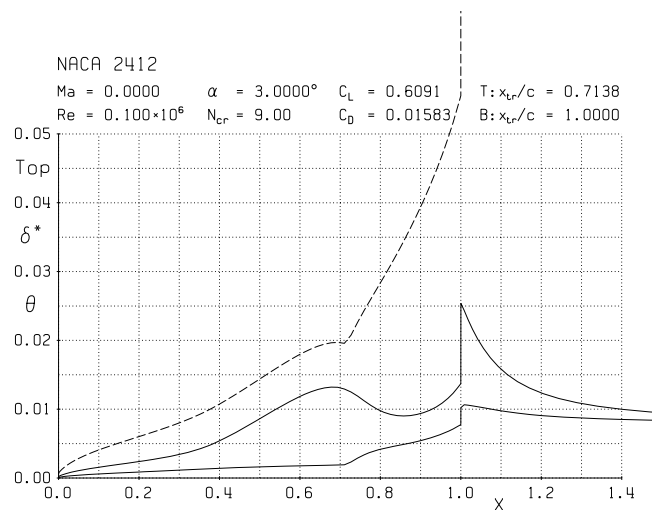
Figure 1.5: Postcript output of the variables in Figure 1.4 using now the `HARD` command from XFOIL.

```
..VPLO  c>  dump DT.dat
```

This file looks like this:

```
#   NACA 2412
#   alpha =    3.0000
#   Mach  =    0.0000
#   Reyn  =    100000
#   Ncrit =    9.0000
#
#    x           dT
   0.122133E-02  0.235530E-03
   0.563520E-03  0.223739E-03
      .
      .
      .
    1.86201       0.865715E-02
    1.99994       0.850685E-02


   0.122133E-02  0.105642E-03
   0.563520E-03  0.100875E-03
      .
      .
      .
    1.86201       0.815508E-02
    1.99994       0.810165E-02
```

There are two sets of x–y value separated by a blank line. The first group corresponds to the displacement thickness, i.e. δ^* vs x/c, and the second to the momentum thickness, i.e. θ vs x/c. The intermediate points, denoted by the vertical dots ":" have been omitted for the sake of brevity. Use your favourite x–y plotting program to display the data.

In the OPER menu of commands there are similar option to write output to file. One could use the command "`DUMP f`" to output the values of velocity U_e, displacement thickness δ^*, displacement thickness θ, and skin friction coefficient C_f against the arc length s and the (x, y) coordinates to the file "`f`". The command "`CPWR f`" will write the coordinate x and pressure coefficient C_p to the file "`f`".

1.5   Failure to converge

Let us try to stretch XFOIL further by looking at a design point far from the previously specified angle of incidence ($\alpha = 3°$). Type

```
.OPERv c> alfa 12
```

You will notice that XFOIL does not converge, see Figure 1.6(a). This is because it reached the maximum number of iterations. There are two different things that can be done. You can either further iterate from the last computed solution by repeatedly typing "`!`" until XFOIL converges, or by increasing the default maximum number of iterations, which is 20. For the second option, type

```
.OPERv c> iter
```

A prompt will ask you to enter the number of iterations, type "`100`" and run at the same angle of incidence again, i.e. type

```
.OPERv c> alfa 12
```

Now XFOIL has converged after a number of additional iterations (~ 11) to get the graph in Figure 1.6(b).
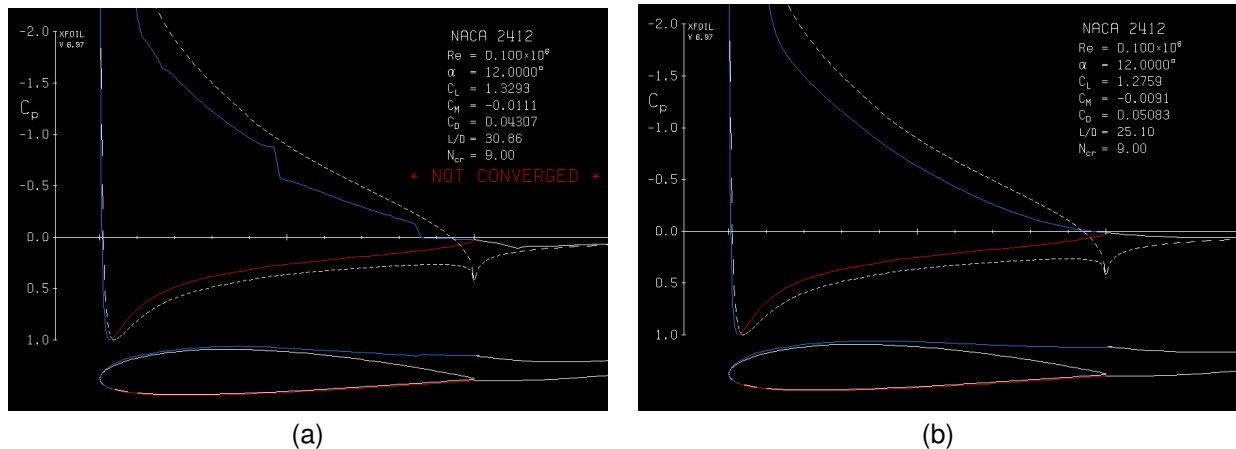

Figure 1.6: Flow at high incidence ($\alpha = 18$ degrees) requiring additional iterations: (a) default; (b) `ITER 100`.

1.5.1   Graph out of the screen?

The C_p graph in Figure 1.6 is out-of-bounds. It is desirable to change the length of the y–axis before including it in the report.

This is done by issuing the command "`CPMIN -6`" followed by "`CPX`" that leads to the new graph shown in Figure 1.7 where the y–axis now represents $-6 \leq C_p \leq 1$.

You should consult the online help in the various XFOIL command menus to check what is required to adjust other plots.

1.6   Running XFOIL over a sequence of angles of incidence

To analyse the performance of the airfoil we would like to obtain its polar curve. For this, we need to sample a range of incidences. As a starting point, type
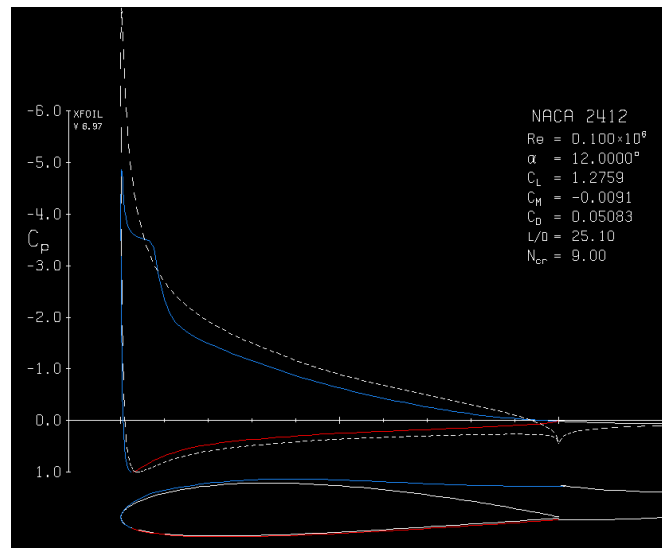
```
.OPERv   c>  pacc
```

Figure 1.7: The same variables shown in Figure 1.6(b), but with an appropriate length of its axes this time.

This tells XFOIL to accumulate values to be calculated in the next stages. It will create a file to which the section lift coefficient, the section drag coefficient, the upper and lower transition points and other data will be saved. You will be prompted to enter filenames for a polar save file and for a polar dump file. Let us use `naca2412.pol` and `naca2412.dmp` here. XFOIL displays the following:

```
Polar  1 newly created for accumulation
Airfoil archived with polar: NACA 2412
Enter  polar save filename  OR  <return> for no file   s>  naca2412.pol
New polar save file available

Enter  polar dump filename  OR  <return> for no file   s>  naca2412.dmp
New polar dump file available
Polar accumulation enabled
```

Now type

```
.OPERva c> aseq -2.0 10.0 1.0
```

This command will run XFOIL for a series of angles of attack, from -2 to 10 degrees at increments of 1 degree. Display messages providing information on the progress of the iteration will appear on the command window looking (roughly) like this:

```
Calculating wake trajectory ...
Calculating source influence matrix ...
Solving BL system ...

Point added to stored polar  1
Point written to save file  naca2412.pol
Point written to dump file naca2412.dmp
```

Note that some of the messages have not been included. Do not ignore these as they might tell you there might be some issues with the iteration. During the sweep over the angles of incidence, the screen will display pressure distributions for the sample angles of incidence. To stop XFOIL accumulating more points, type again

```
.OPERva c> pacc
```

The file `naca2412.pol` is now available, its contents are in text format and can be read and displayed using your x–y plotting program of choice. Alternatively, you could use XFOIL to display this information by typing

```
.OPERva   c>  pplo
```

which will display the graphs in Figure 1.8 representing the polar and lift and moment coefficients curves and also the position of the transition point.
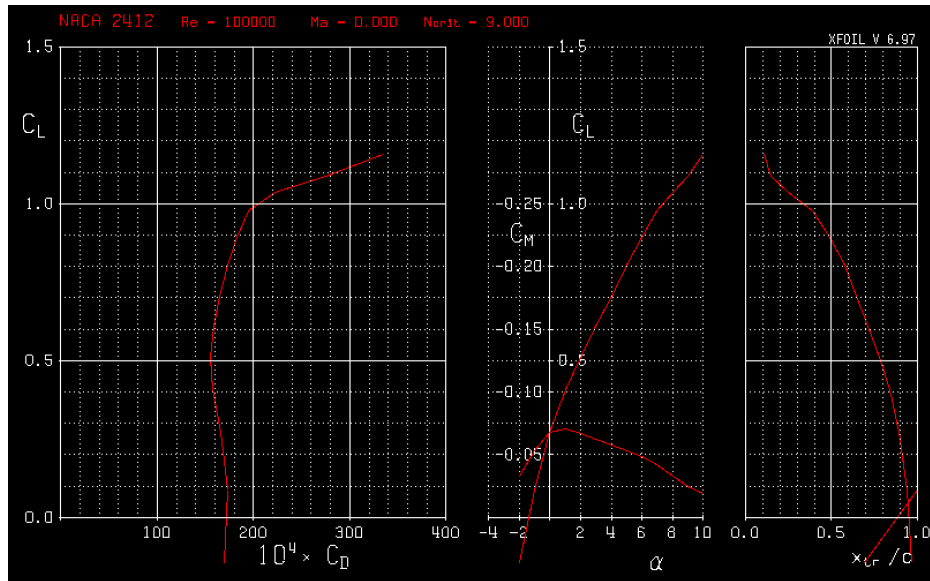

Figure 1.8: XFOIL graphs for C_l vs C_d, C_l and C_m vs α, and C_l vs x_{tr} (the location of the transition points).

1.7   Another useful commands

The following are just a few commands that you might find useful for both your coursework and, hopefully, in any future use of XFOIL.

1.7.1   Reading airfoil geometry from a file

XFOIL reads a text file describing the geometry of the aerofoil which contains a first line with a title, which will be used as the airfoil name in subsequent screen displays, followed by a set of x–y coordinates that start at the trailing edge of the airfoil, move towards the leading edge over the top surface, and return to the trailing edge via the bottom surface. An example of this is the file `airfoil.dat` available in the course Blackboard Learn page,[4] which looks like this:

```
AIRFOIL EXAMPLE
     1.000000        0.0000000E+00
    0.9962500        6.2000001E-04
    0.9855600        2.7500000E-03
    0.9689800        6.3399998E-03
    0.9469800        1.0760000E-02
    0.9194700        1.5750000E-02
    0.8866400        2.1550000E-02
    0.8491000        2.8290000E-02
```

---
[4]Under the title "Input data file of aerofoil coordinates for XFOIL"

```
0.8074700       3.5950001E-02
0.7624600       4.4429999E-02
0.7148000       5.3569999E-02
0.6652000       6.3120000E-02
0.6143700       7.2820000E-02
0.5630000       8.2330003E-02
0.5117200       9.1329999E-02
0.4610900       9.9469997E-02
0.4116400       0.1064300
0.3638300       0.1119300
0.3180400       0.1157000
0.2746300       0.1175600
0.2338900       0.1173500
0.1960900       0.1149500
0.1613700       0.1103100
0.1298900       0.1035100
0.1017500       9.4549999E-02
7.6820001E-02   8.3600000E-02
5.5059999E-02   7.1160004E-02
3.6680002E-02   5.7730000E-02
2.1849999E-02   4.3710001E-02
1.0720000E-02   2.9490000E-02
3.4200000E-03   1.5520000E-02
9.9999997E-05   2.3900000E-03
0.0000000E+00   0.0000000E+00
0.0000000E+00   0.0000000E+00
1.5200000E-03  -8.9199999E-03
8.5800001E-03  -1.9110000E-02
2.0810001E-02  -2.9270001E-02
3.7379999E-02  -3.8840000E-02
5.7999998E-02  -4.7330000E-02
8.2500003E-02  -5.4290000E-02
0.1108300      -5.9330001E-02
0.1429600      -6.2150002E-02
0.1791000      -6.2470000E-02
0.2196300      -6.0550001E-02
0.2644300      -5.7119999E-02
0.3128500      -5.2730002E-02
0.3642200      -4.7699999E-02
0.4178500      -4.2289998E-02
0.4729900      -3.6720000E-02
0.5288900      -3.1199999E-02
0.5848000      -2.5869999E-02
0.6399400      -2.0889999E-02
0.6935500      -1.6349999E-02
0.7448900      -1.2340000E-02
0.7932400      -8.9199999E-03
0.8378900      -6.1100000E-03
0.8782000      -3.9200000E-03
0.9135500      -2.3099999E-03
0.9434000      -1.1800000E-03
0.9673500      -3.8000001E-04
0.9851200       5.9999998E-05
0.9962100       7.9999998E-05
 1.000000       0.0000000E+00
```

The command to read the file in XFOIL is

```
XFOIL   c>  load ./airfoil.dat
```

Note that this is assuming that the file exists in the directory you are running XFOIL from, otherwise you might have to write the name including the whole directory path.[5]

If successful, XFOIL will show some of the airfoil geometrical parameters in the following message:

```
Labeled airfoil file.  Name:  AIRFOIL EXAMPLE


Number of input coordinate points:  63
Counterclockwise ordering
Max thickness =     0.177278  at x =    0.220
Max camber    =     0.032096  at x =    0.364


 LE  x,y  =   0.00000   0.00000  |   Chord =   1.00000
 TE  x,y  =   1.00000   0.00000  |

Current airfoil nodes set from buffer airfoil nodes (  62 )
```

1.7.2 Inspecting the airfoil and its paneling

You might want to see what the airfoil looks like using the command to show/change paneling by typing

```
XFOIL   c>  ppar
```

which displays the following information on the parameters describing the discretization of the loaded airfoil into panels:

```
   Present paneling parameters...
 N  i    Number of panel nodes        160
 P  r    Panel bunching parameter     1.000
 T  r    TE/LE panel density ratio    0.150
 R  r    Refined area/LE  panel density ratio   0.200
 XT rr  Top    side refined area x/c limits    1.000 1.000
 XB rr  Bottom side refined area x/c limits    1.000 1.000
 Z oom
 U nzoom

Change what ? (<cr> if nothing else)    c>
```

The screen also graphically displays this information as seen in Figure 1.9(a). To change the number of panels from the existing 160 to, say, 200 we enter

```
 Change what ? (<cr> if nothing else)    c> n 200
```

and press return again to finish the changes at this level of the menu. XFOIL then displays the new paneling which is shown in Figure 1.9(b).

1.7.3 Changing the point of transition to turbulent Flow

Following with the loaded airfoil, let us calculate the pressure distribution for $\alpha = 3°$ and $Re = 100000$ with the new number of panels N=200. The sequence to type is

---

[5]In Windows and MacOS, this could be achieved by dragging the file icon onto the command window.

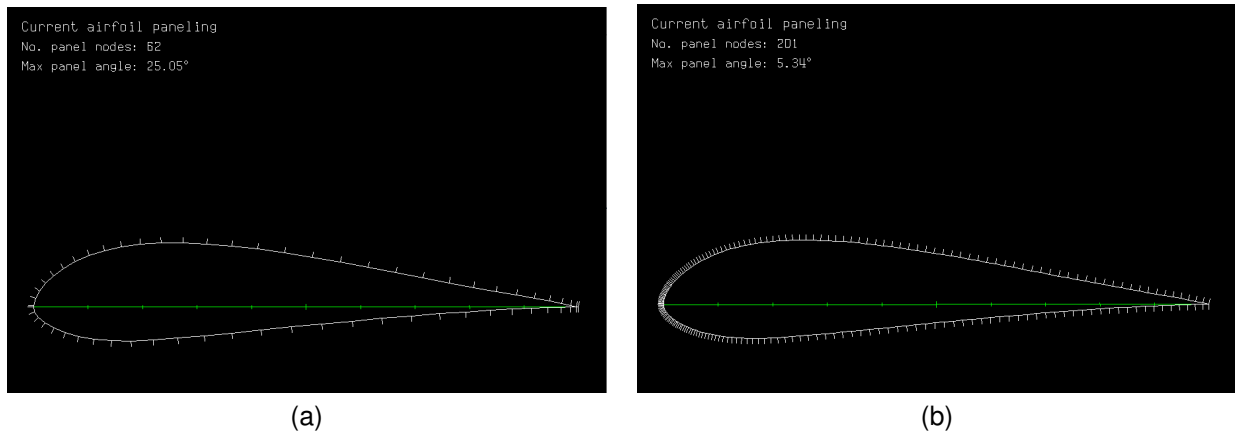(a)   (b)

Figure 1.9: Paneling for the airfoil data contained in the file `airfoil.dat`: (a) Default number of panels (`N=160`); (b) `N=200`.

```
XFOIL    c>   oper
.OPERi   c>   visc
Enter Reynolds number    r>   100000
.OPERv c> alfa 3
```

The last iteration will look like this:

```
Side 1  free  transition at x/c =   0.4436    74
Side 2 forced transition at x/c =   1.0000    94

   7   rms: 0.1914E-04   max: -.7995E-04    n at   80  2
        a =   3.000        CL =   0.6692
       Cm = -0.0570        CD =   0.02275    =>   CDf =   0.00811     CDp =   0.01464
```

XFOIL has calculated transition at $x/c = 0.4436$ and the pressure distribution shown in Figure 1.10(a). The distribution shows all the hallmarks of a separation bubble: an approximately constant C_p in the region $0.25 \leq x/c \leq 0.35$ followed by a sharp rise of its value after $x/c \sim 0.35$ where it reverts back to values of the inviscid C_p.[6] The location

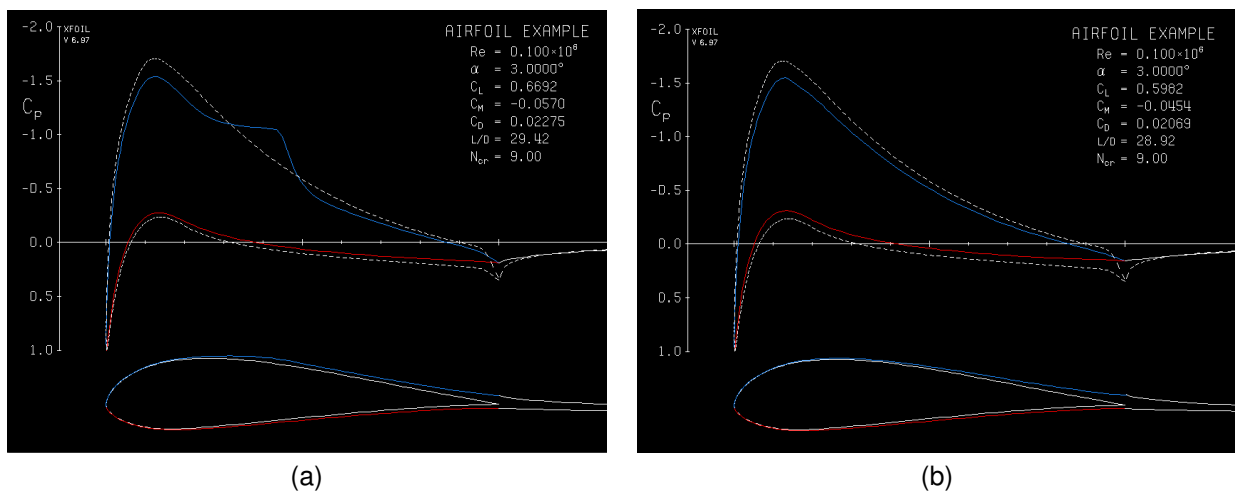(a)   (b)

Figure 1.10: Calculation of transition: (a) Default; (b) Transition point fixed at x/c.

of transition on the upper surface (Side 1) is at $x/c = 0.4436$.

---

[6]Obviously, the skin friction should confirm this, but we leave it as an exercise for you.

Assume that we would like to trip the boundary layer at $x/c = 0.1$ to force transition and affect the boundary-layer behaviour. What would you expect to happen to the separation bubble? Let us find out. Type

```
.OPERv c> vpar
..VPAR c> xtr 0.1 1
```

The last command will force transition at $x/c = 0.1$ for the top surface, and specifying $x/c = 1$ at the bottom surface is the same as free transition, i.e. XFOIL will calculate it as usual.

Now let us evaluate the pressure coefficient with the trip at $x/c = 0.1$. Press "<cr>" to move up to the "OPER" menu and proceed as before by typing

```
.OPERv c> alfa 3
```

Now the last iteration will now look like this:

```
 Side 1 forced transition at x/c =  0.1000   45
 Side 2 forced transition at x/c =  1.0000   94

   8   rms: 0.1428E-05   max: 0.1372E-04   C at   94  2
       a =   3.000       CL =  0.5982
     Cm = -0.0454       CD =  0.02069   =>   CDf =  0.01110    CDp =  0.00959
```

Transition now occurs at $x/c = 0.1$ as prescribed, and the pressure distribution is shown in Figure 1.10(b) that shows no sign of the separation bubble. Further the total drag coefficient, C_d, has been reduced from 0.02275 to 0.02069. This has been achieved because the decrease in pressure drag by avoiding flow separation is larger that the friction drag increase due to having a larger region of turbulent flow.

1.8 End of your XFOIL session

Press "<cr>" until you return to the top level and there type

```
XFOIL c> quit
```

and that is it. Go ahead and later explore these and other capabilities of XFOIL.

Chapter 2

AVL

2.1 Introduction to "Athena Vortex Lattice"

AVL is a vortex-lattice method implemented by Mark Drela and Harold Youngren at MIT. Mark Drela is a very well-respected aerodynamicist, who also wrote the 2D aerofoil tool XFOIL, presented in the previous chapter. The AVL source code, precompiled executives, example input files, and a documentation can be found at `http://web.mit.edu/drela/Public/web/avl/`.

The tool can compute the aerodynamics of lifting surfaces using the vortex-lattice method. Fuselages and nacelles are modelled using a slender-body model. These two geometric types can be arbitrarily combined to represent any aircraft configuration. The tool can further be used to calculate the flight dynamics of any such configuration, by specifying control surfaces and mass properties. These flight dynamics analyses are outside the scope of this course, though.

You will notice that the operating commands and the general feel of the tool is similar to that of XFOIL.

2.2 Opening AVL

Go to the software hub (`https://softwarehub.imperial.ac.uk/`) and launch "AVL".

2.3 Familiarising with the AVL workflow

2.3.1 Open a new session

After opening AVL, you find yourself in the start menu, which should look like the following:

```
 This software comes with ABSOLUTELY NO WARRANTY,
   subject to the GNU General Public License.

 Caveat computor
 ==================================================

 =========================================================
   Quit    Exit program

  .OPER    Compute operating-point run cases
  .MODE    Eigenvalue analysis of run cases
  .TIME    Time-domain calculations
```

```
   LOAD f  Read configuration input file
   MASS f  Read mass distribution file
   CASE f  Read run case file

   CINI    Clear and initialize run cases
   MSET i  Apply mass file data to stored run case(s)

  .PLOP    Plotting options
   NAME s  Specify new configuration name

 AVL   c>
```

2.3.2   Load a configuration

As a first step you need to load a configuration file, containing your wing or full aircraft configuration. These need to have the file extension `.avl`. For the beginning we provide you with a simple configuration file of a rectangular wing: `wing_ini.avl`. Create a new folder in a suitable location in your personal folder tree and download the file from Blackboard Learn. You can also copy the content of the file given hereafter into a new text file.

Lines beginning with a pound symbol (#) are comments. While not required, you are encouraged to use them to enhance readability of the parameters.

```
Initial Wing
#Mach
 0.0
#IYsym    IZsym    Zsym
 0        0        0.0
#Sref     Cref     Bref
90.0      3.0      30.0
#Xref     Yref     Zref
0.75      0.0      0.0
#CDp
0.0
#
#=================================================================
SURFACE
Wing
#Nchordwise  Cspace    Nspanwise    Sspace
2            1.0       8            1.0
#
YDUPLICATE
0.0
#
ANGLE
0.0
#-------------------------------------------------------------
SECTION
#Xle    Yle    Zle     Chord   Ainc  Nspanwise  Sspace
0.      0.     0.      3.0     0.0   0          0
#-------------------------------------------------------------
SECTION
#Xle    Yle    Zle     Chord   Ainc  Nspanwise  Sspace
0.0     15.0   0.0     3.0     0.0   0          0
```

Type into the command line:

```
AVL   c> LOAD C:\[path To File]\wing_ini.avl
```

or just type `LOAD` and drag the file onto the program window. You will see the command line output:

```
 Reading file: wing_ini.avl  ...

 Configuration: Simple Wing

   Building surface: Wing

   Building duplicate image-surface: Wing (YDUP)

 Mach =     0.0000  (default)
 Nbody =   0      Nsurf =   2      Nstrp =  16      Nvor  =  16

 Initializing run cases...

 AVL   c>
```

The output gives you the name of the configuration file and summarises the surfaces and bodies created—in this case one lifting surface and its image duplicated along the y-axis. The Mach number has been set to $\mathrm{Ma} = 0.0$. If a Mach-number greater than zero is specified, a Prandtl–Glauert correction is applied. The last line summarises the number of bodies, surfaces, vortex-strips (or vortex lines), and the total number of vortices.

Type `?`, if you would like to see the options of the current menu again. Similarly to the XFOIL convention, commands are usually given by a four letter string. If the command leads to another menu, it is proceeded by a dot. To get back to the parent-level menu, just press the return-key. If the command is followed by `f`, a file name needs to be given in addition to the operating command.

2.3.3 The operation menu

For our exercise we just need the operations menu. To get there, just type into the command line:

```
AVL   c> OPER
```

The menu shows a selection of commands and should look like:

```
Operation of run case 1/1:   -unnamed-
=======================================================

  variable          constraint
 ------------      ------------------------
 A lpha        ->  alpha       =   0.000
 B eta         ->  beta        =   0.000
 R oll  rate   ->  pb/2V       =   0.000
 P itch rate   ->  qc/2V       =   0.000
 Y aw   rate   ->  rb/2V       =   0.000
 ------------      ------------------------


  C1  set level or banked  horizontal flight constraints
  C2  set steady pitch rate (looping) flight constraints
  M odify parameters

"#" select  run case          L ist defined run cases
 +  add new run case          S ave run cases to file
 -  delete  run case          F etch run cases from file
 N ame current run case       W rite forces to file
```
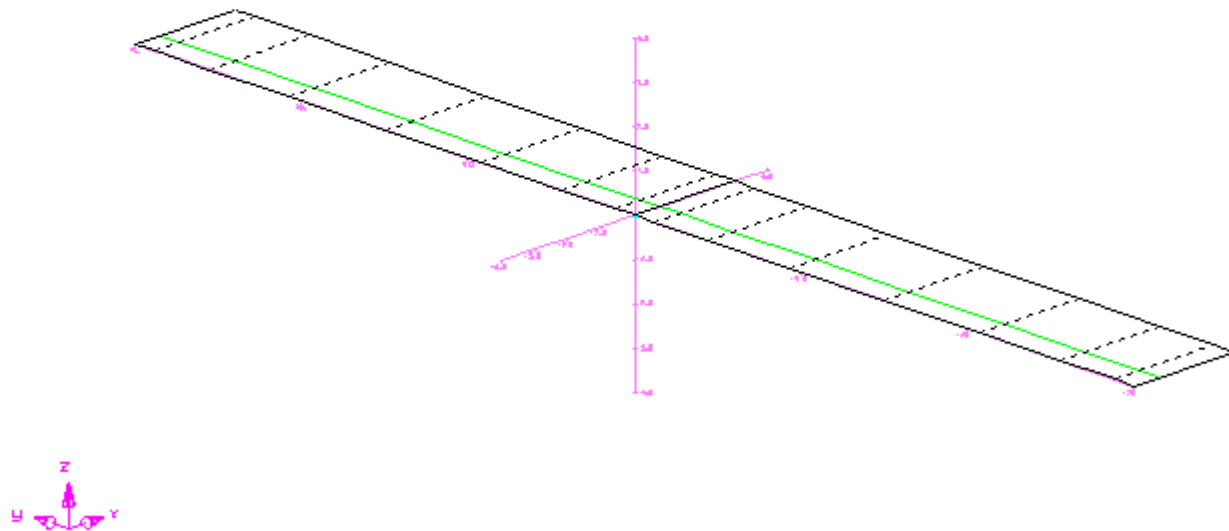
Figure 2.1: AVL Geometry plot of initial wing configuration (colour inverted)

```
eX ecute run case          I nitialize variables

 G eometry plot            T refftz Plane plot

 ST  stability derivatives   FT  total   forces
 SB  body-axis derivatives   FN  surface forces
 RE  reference quantities    FS  strip   forces
                             FE  element forces
 DE  design changes          VM  strip shear,moment
 O ptions                    HM  hinge moments

.OPER (case 1/1)   c>
```

We will discuss the important commands in the following sections.

2.3.4 Show configuration geometry

From within the operation menu you can type G to show a geometry plot of your configuration.

A new window should open, which displays the geometry of the loaded configurationas shown in Figure 2.1.

You then have multiple options to modify your plot. You can type K to go into keystroke mode and then rotate the

view. You can type `A` to annotate the plots with arrows and strings. You further have 8 different options to enable or disable plot elements like chord line or camber line, normal vectors, or the aerodynamic loading. Your choice is indicated by `T` for true or `F` for false.

Take some time to explore the different options.

When you are happy with your plot you can type `H` to generate a postscript plot called `plot.ps`, that will be saved in the directory in which the AVL binary is located.

2.3.5   Set run parameters and constraints

The two variables that we want to set are the angle of incidence α and the yaw angle β. The other variables are only meaningful for the flight dynamic cases.

There are two options for setting each variable; you can either specify a certain angle of attack α, or specify a constraining lift coefficient C_L. AVL will then iterate over the angle α to establish which value will satisfy the constraint given by the lift coefficient.

For this exercise set the angle of incidence to 5 degrees by first indicating that you want to adjust the variable `ALPHA`. Type into the command line: `A`

Secondly, you need to state the new constraint for `ALPHA`, so type in the command line: `A 5.0`

You can choose to name this run-case by typing: `N [NameOfCase]`

2.3.6   Execute calculation

The calculation is executed by typing: `X`

2.3.7   Output visualisation: Trefftz Plane

To visualise the results of the vortex-lattice calculation you can employ the Trefftz Plane feature. The Trefftz plane is perpendicular to the inflow, so in a Y-Z plane, but infinitesimally-far behind the lifting surfaces. The results of the vortex-lattice method are most accurate at this location.

To show the Trefftz plane type in the command line: `T`

A second window is opened that should look like Figure 2.2.

In the upper part you see the overall resulting values of your run case. In the first column you find the angle of incidence α, the yaw angle β, and the Mach number. Remember, a non-zero value will have turned on the Prantl-Glauert correction.

The second columns provides some flight dynamics results, which we will ignore.

In the third column, C_L will give the overall lift coefficient, C_Y will give the overall side-force coefficient, and C_D the overall drag coefficient. These are defined as:

$$C_L = \frac{F_x}{QS_{ref}}$$
$$C_Y = \frac{F_y}{QS_{ref}}$$
$$C_D = \frac{F_z}{QS_{ref}}$$

with the dynamic pressure $Q = 0.5\rho V^2$ and the reference surface area S_{ref}.

The drag coefficient C_D is the sum of an induced drag coefficient C_{Di}, as calculated by summing the forces over the lifting surfaces, and the profile drag coefficient C_{Dp}. The profile drag coefficient cannot be calculated with the vortex-lattice method and can only be specified as an input parameter in the configuration file.
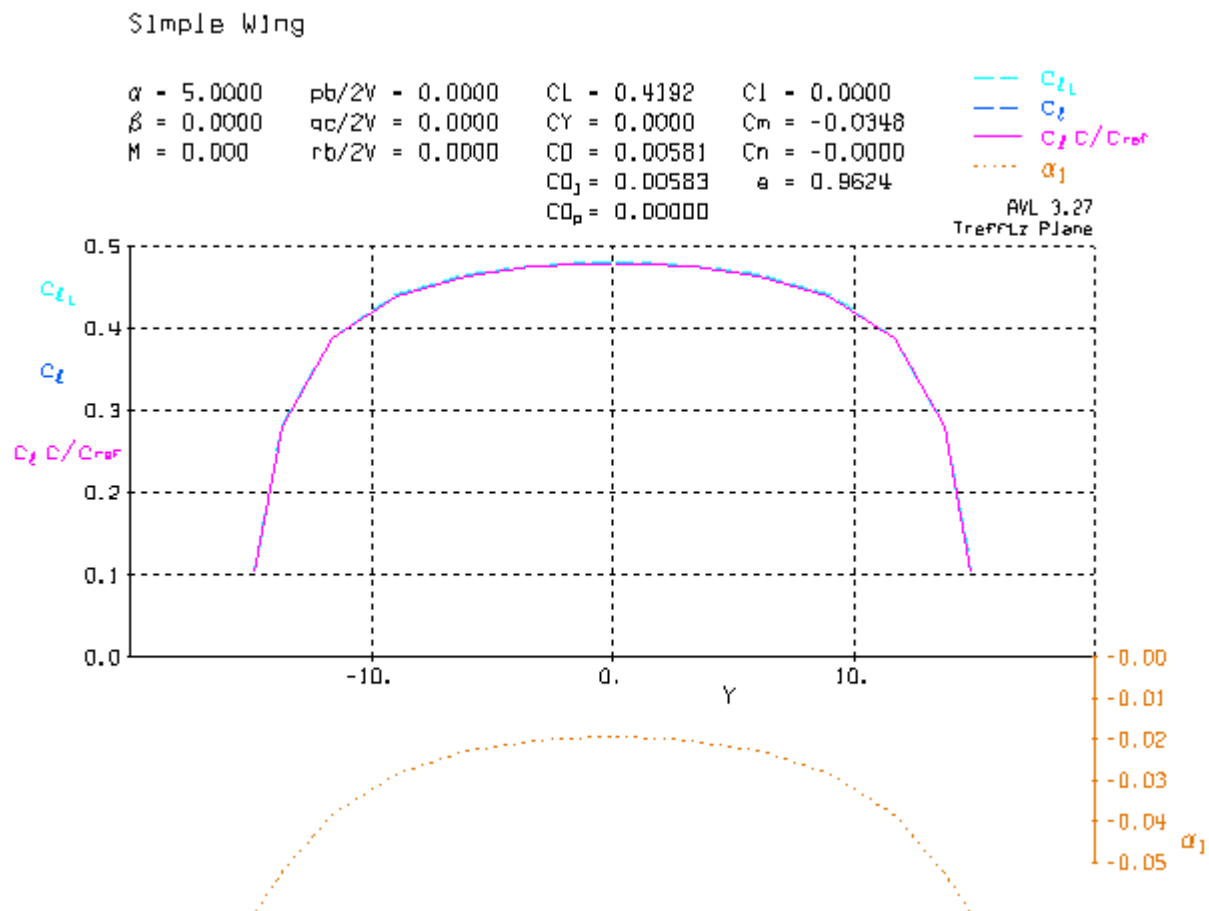
Simple Wing

α = 5.0000   pb/2V = 0.0000   CL = 0.4192   Cl = 0.0000
β = 0.0000   qc/2V = 0.0000   CY = 0.0000   Cm = -0.0348
M = 0.000   rb/2V = 0.0000   CD = 0.00581   Cn = -0.0000
                                   CDi = 0.00583   e = 0.9624
                                   CDp = 0.00000

Figure 2.2: AVL Trefftz plane of initial wing configuration (colour inverted)

The induced drag coefficient C_{Di}, however, is calculated based on the wake trace in the Trefftz plane, which should be more accurate than the integration over the lifting surfaces.

The fourth column specifies the three moment coefficients, with the roll moment c_n, the pitch moment c_m, and the yaw moment c_l. These are defined as:

$$c_n = \frac{M_x}{QS_{ref}B_{ref}}$$

$$c_m = \frac{M_y}{QS_{ref}C_{ref}}$$

$$c_l = \frac{M_z}{QS_{ref}B_{ref}}$$

with the reference span B_{ref} and chord C_{ref}, respectively.

Lastly, the Oswald efficiency is given, which is calculated as:

$$e = \frac{C_L^2 + C_Y^2}{\pi \Lambda C_{Di}}$$

with the aspect ratio $\Lambda = B_{ref}^2 / S_{ref}$.

The lower part of the plot shows the Trefftz plane with up to four data series.

The perpendicular sectional lift coefficient $c_{l\perp}$ is calculated using the local chord length c and the flow speed perpendicular to the local sweep at the quarter cord line. The sectional lift coefficient c_l is calculated based on the local chord length c, that is parallel to the x-axis. The lift per span loading L' or $c_l c/c_{ref}$ is the relevant result for indicating the actual lift distribution. Lastly, we find the downwash angle α_i over the span.

Back in the main window you can set plot options, like turning on or off some of the mentioned curves, do annotations, or create a hardcopy in postscript format.

2.3.8 Text Output

You can output the summarising results to a textfile by typing: `W [NameOfFile.txt]` The output options can be changed by typing: `O`

If you need to plot the lift distribution over the span, similar to the output in Trefftz plane, use the command `FS` to extract the strip forces either to the screen or to a textfile.

2.4 Adapting the geometry

Let us now haves a more detailed look into how to set-up a more complex wing configuration using the AVL configuration file.

2.4.1 Overall variables

Each configuration file has to start with 10 parameters, to be given in the following format:

```
SIMPLE WING
#Mach
 0.0
#IYsym   IZsym   Zsym
 0       0       0.0
#Sref    Cref    Bref
 90.0    3.0     30.0
#Xref    Yref    Zref
```

```
 0.75     0.0      0.0
#CDp
 0.0
```

The Mach number is utilised within AVL to apply a Prandtl-Glauert correction to account for the compressibility of the air. Be careful, however, in which range of Mach numbers this will provide a reasonable result!

`IYsym` specifies whether the calculation should be done using symmetry in the Y-axis. Set it to `1` to turn it on, or to `0` to turn it off. It should not be confused with setting up the geometry in a symmetric manner! The only justification to use it is to save computing resources, which shouldn't really be necessary these days.

`IZsym` set to `1` would create a symmetry plane normal to the Z axis, which would allow to simulate ground effect. The variable `Zsym` would then specify the z-coordinate of the ground.

`Sref`, `Cref`, and `Bref` provide the values for the reference surface, chord, and span for the wing configuration. They are used by AVL to calculate the aerodynamic coefficients. This means you need to calculate them yourself, according to the wing that you will specify further below in the configuration file.

`Xref`, `Yref`, and `Zref` specify the coordinates around which aerodynamic moments and rotation rates are calculated.

Using `CDp` you can specify an additional profile drag coefficient for the overall geometry.

2.4.2   Lifting surfaces

Lifting surfaces are defined using two or more sections. The space between any two sections will be linearly interpolated by AVL.

In the initial configuration file you can see the simplest case, already being set-up.

```
SURFACE
Wing
#Nchordwise  Cspace    Nspanwise    Sspace
1            1.0        8            1.0
```

The definition of a new lifting surface is started by the keyword `SURFACE` and the name of the surface in the succeeding line. In the third valid line (i.e. not being a comment line starting with `!` or `#`) the number of chordwise and spanwise vortices and their respective spacing parameter have to be specified. We will discuss the spacing parameters in a later section.

In the next lines more parameters for the whole surface can be given.

`YDUPLICATE` followed by a parameter giving the Y position of the X-Z mirror plane will geometrically mirror the whole aerodynamic surface. The flow over the mirrored surface can still by unsymmetrical though!

```
YDUPLICATE       | (keyword)
0.0              | Ydupl
```

The `ANGLE` keyword can change the angle of incidence for the entire surface. It is an offset added to the angle of incidence specified for the individual sections of that surface.

```
ANGLE      | (keyword)
2.0        | dAinc
```

There are further `SCALE` and `TRANSLATE` keywords to scale or translate the entire surface in a more concise way than changing the parameters for each defining section.

There are three further keywords: `NOWAKE`, `NOALBE`, and `NOLOAD`. Respectively, these specify that the relevant surface is not shedding a wake, is not affected by the freestream direction changes, or does not add to the overall forces and moments calculation. For our simple wing configuration these three keywords will not be required.

2.4.3   Airfoil sections

Then the individual section of the surface are specified one after the other, each starting with the `SECTION` keyword.

```
SECTION
#Xle    Yle    Zle     Chord   Ainc  Nspanwise  Sspace
0.      0.     0.      3.0     0.0   0          0
```

The next line gives the leading edge coordinate in x, y, and z direction, the chord length—so that the trailing edge coordinate is at (`Xle+Chord,Yle,Zle`)—, and the section's angle of incidence. The two last keywords are optional and only considered if they have not already been set for the overall surface.

N.B. `Ainc` will not rotate the geometry, but change the tangency boundary condition on the surface. Within linear theory, these are equivalent.

As mentioned, at least two sections for the root and the tip need to be specified. The local angle of incidence and chord length are linearly interpolated between two sections.

The aerofoil camberline is a flat plate by default. However, different camberlines can be defined using the 4-digit NACA airfoils with the following notation:

```
NACA       X1  X2          | (keyword)    [ optional x/c range ]
4300                       | section NACA camberline
```

It is also possible to define a custom camberline using an input file containing the coordinates. Such a file can be created by XFOIL.

```
AFILE      X1  X2          | (keyword)    [ optional x/c range ]
filename                   | filename string
```

The x/c range is optional and could be applied when the camberline does not cover the entire chord, such as in cases where a control surface is to be defined.

To obtain more accurate results, two more parameters can be set. Using the keyword `CDCL` a profile drag profile can be specified:

```
CDCL                        | (keyword)
CL1 CD1  CL2 CD2  CL3 CD3   | CD(CL) function parameters
```

The six parameters specify three pairs of lift and drag coefficients based on the aerofoil polar. The pair `CL2 CD2` should specify the point of minimum drag, the pair `CL1 CD1` specifies a point on the negative stall region, and the pair `CL3 CD3` defines a point on the positive stall region. AVL uses two parabolic functions between `CL1` and `CL2` and between `CL2` and `CL3` to interpolate extrapolate the data. The polars are linearly interpolated between aerofoil section, so make sure to include these parameters for none or all section of one surface. Please also consult the documentation for more details.

Using the keyword `CLAF` the lift-curve slope of the aerofoil can be altered, using the following notation:

```
CLAF        | (keyword)
CLaf        | dCL/da scaling factor
```

This allows to better incorporate the effect of thick aerofoils. AVL recommends to set the coefficient to

$$Claf = 1 + 0.77t/c$$

with t/c being the thickness to chord ratio of the aerofoil. By default this factor is set to 1.0, so the usual lift curve slope of 2π is obtained.

2.4.4   Slender bodies

Slender bodies like fuselages and nacelles can be modelled using the `BODY` keyword. This again is outside the scope of the exercises of this course.
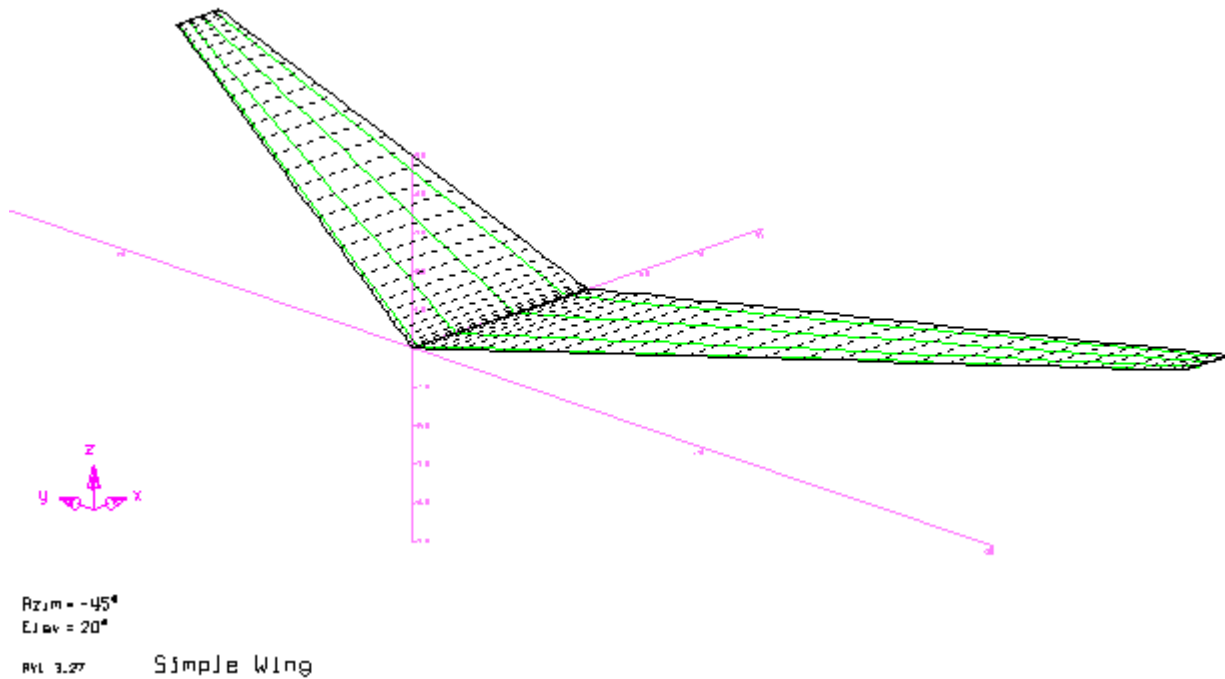
Figure 2.3: AVL Geometry of A320-like wing (colour inverted)

2.5 Creating a swept, tapered and twisted wing with NACA aerofoil sections

After having been exposed to the configuration file keywords and settings, it is now time to create a more complex wing configuration. It is advised to start with the simple initial wing configuration file. Make a copy of it and rename it and subsequently modify its content.

Try to reproduce the wing of an A320-like aircraft. We will assume a span of 35.0m, a root chord of 6.0m, a taper ratio of 0.25, a sweep at the quarter chord line of 25deg and a dihedral angle of 5deg. We will assume straight leading and trailing edges over each half of the wing.

Make the calculations to adapt the reference wing area, reference wing span, reference chord, and section definitions accordingly.

Save the new values in the new configuration file and have a look at the new wing configuration using the Geometry plot.

You can also introduce section camberlines by specifying a NACA 4-series aerofoils.

The resulting wing will look like Figure 2.3.

2.6 Convergence study

To improve the numerical accuracy of your AVL results whilst not utilising more computational resources than necessary, you should conduct a convergence study.

It is advisable to scale both spanwise and streamwise numbers of vortices with the same ratio. Surfaces near the leading and the trailing edge of a lifting surface need a finer spacing of vortices. The same applies for stronger discontinuities in spanwise direction, be that the wing root of a highly swept wing, or the kink between the main wing and its winglet.

To adapt the spacing or distribution of a fixed number of vortices on a surface, you can adapt the parameter value

of the keywords already introduced with the SURFACE definitions, `Sspace` and `Cspace`. The cosine function is generally a good starting point. For a straight wing without a discontinuity along the root, and defined by a root and a tip section, a sine distribution will be more applicable, so that the overall wing again has a cosine vortex distribution. It is possible to specify intermediate values also, a value of `0.5` will result in a distribution interpolated between equal and cosine spacings.

```
parameter                        spacing
---------                        -------

   3.0       equal       |   |   |   |   |   |   |   |   |

   2.0       sine        ||  |   |   |     |     |      |      |

   1.0       cosine      ||  |     |         |       |    ||

   0.0       equal       |   |   |   |   |   |   |   |   |

  -1.0       cosine      ||  |     |         |       |    ||

  -2.0       -sine       |       |      |     |     |    |  ||

  -3.0       equal       |   |   |   |   |   |   |   |   |
```

2.7   Extra: Adding a winglet

You can try adding winglets to the tip of the existing wing. You can either add another section to the wing and place the leading edge coordinate at a location that will give a large dihedral angle. Alternatively, if you want to introduce a winglet over a fraction of the tip chord only create a new surface using two more sections. You then also need to add the `COMPONENT` keyword to both wing and winglet surface to tell `AVL` to join them. It will be beneficial to also use the `TRANSLATE` keyword to specify the inboard winglet coordinate and subsequently all winglet sections relative to each other.

Vary the geometry and especially the incidence angle to yield an improved Oswald efficiency.

Chapter 3

STAR-CCM+

This chapter aims to clarify nomenclature and serve as a quick start guide for STAR-CCM+. It contains brief explanations of the various operations that can and should be performed to create and process a CFD simulation within STAR-CCM+. While the focus is on this particular software, the general concepts apply to other CFD packages as well.

3.1 Opening a new session

Go to the software hub (https://softwarehub.imperial.ac.uk/) and download/install "STAR-CCM+ 13.04.011". You can now launch STAR-CCM+ from the launcher window; it should now also be available in the start menu.

After the initial loading screen, you will be presented with an empty window like in fig. 3.1. Create a new simulation by clicking on the corresponding icon in the toolbar or select `File` ⟩ `New...` in the menu bar.

In the window, select the "Power-On-Demand" license type. Depending on available hardware, you can change the process options from "Serial" to "Parallel on Local Host" and specify the number of compute cores you want STAR-CCM+ to use.

As soon as a new simulation is created it is recommended to save it and to keep saving regularly. Unlike for example ANSYS Fluent, STAR-CCM+ projects are contained fully[1] in .sim files. Nonetheless, as with all projects, it is wise to keep a clean folder structure. Alongside the .sim file will be other files like input geometries, results figures, and possibly scripts for automation.

3.2 Help

Here we describe how to get information on how to use STAR-CCM+. For conceptual CFD questions, https://www.cfd-online.com/ is a good place to visit.

3.2.1 Offline Documentation

STAR-CCM+ ships with extensive documentation that explains (almost) every button in the program. It does not however provide help on best practises. See section 3.2.2 for more.

Open the documentation in STAR-CCM+ by either pressing `F1` or clicking `Help` ⟩ `Help` in the menu bar.

To get help on specific items in the simulation, press the question mark (📖) in the property window.

---

[1]With the exception of a simulation history (.simh) file, which can be created to store time history in case of unsteady simulations
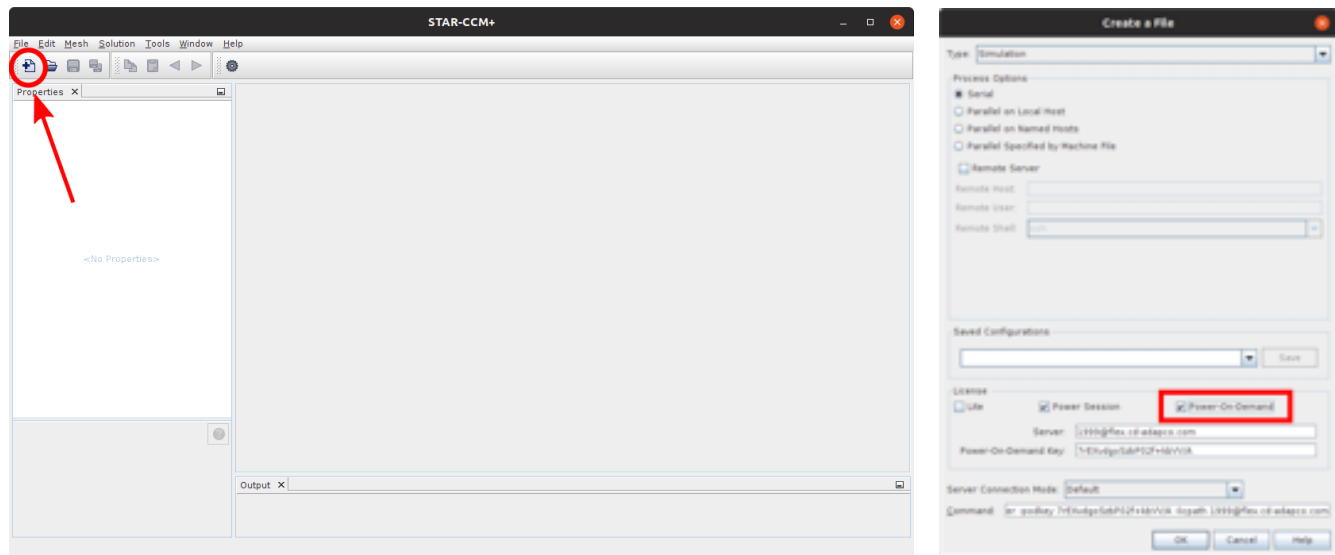
Figure 3.1: Launching a new instance of STAR-CCM+

Overview of the relevant sections of the documentation

Simcenter STAR-CCM+ 〉 User Interface	How to interact with the simulation
Simcenter STAR-CCM+ 〉 Pre-Processing	From geometry to mesh
Simcenter STAR-CCM+ 〉 Simulating Physics	Physics models and boundary conditions
Simcenter STAR-CCM+ 〉 Post-Processing	Extracting results from the simulation
Simcenter STAR-CCM+ 〉 Tutorials	Worked examples, supporting files can be found on Steve Portal

3.2.2 Online Support – The Steve Portal

The Steve Portal is the online companion for STAR-CCM+. Here you can find blog-like articles and videos, covering examples and specific how-to questions. A list of useful links is given here:

- My divergence proof simulation checklist and best practice
- How to use the 2D-meshing Operation in a pipeline
- NACA4 trade off study
- Subsonic Wing Simulation Tutorial and Best Practices
- Transonic/Supersonic Wing Simulation Tutorial and Best Practices

3.3 Tree

The simulation is broken down into a tree, whose major branches and their order correspond to common concepts in the CFD workflow.

The section Simcenter STAR-CCM+ 〉 User Interface 〉 What Are the Important Concepts? in the documentation offers an extensive explanation of these concepts and how they relate to the elements in the tree. Here we provide a short summary of the most relevant elements for this course:

Geometry Meshing operations on single parts or assemblies, either from CAD or tesselated objects.

- Define the relevant geometries. Define a new CAD geometry under `Geometry` ⟩ `3D-CAD-Models` using the built-in CAD engine. Create new Geometry Parts based on the CAD models or based on primitive geometries (block, cylinder), to appear under `Gemetry` ⟩ `Parts`.

- Manipulate the Surfaces or Curves of individual Parts, e.g. split them or give them descriptive names.

- Apply boolean operations under `Geometry` ⟩ `Operations` to create more complex geometries.

- Use the "Badge for 2D Meshing" operation in case you need to prepare Parts for 2D meshing. Use Automated Mesh or Automated Mesh (2D) to create a mesh based on a Part. Do not forget to assign that part to a Region first.

- Change the mesh type under `Geometry` ⟩ `Operations` ⟩ `Automated Mesh` ⟩ `Meshers`. Manipulate the global mesh parameters under `Geometry` ⟩ `Operations` ⟩ `Automated Mesh` ⟩ `Default Controls`. Manipulate the mesh based on specific surfaces or curves using `Geometry` ⟩ `Operations` ⟩ `Automated Mesh` ⟩ `Custom Controls`.

Continua Physics continuum models (and e.g. gas properties) and reference/initial conditions.

Regions The computational domain and its boundaries, including boundary conditions.

Derived Parts Subsets of the domain, e.g. cut planes or streamlines, to use elsewhere in the simulation.

Solvers Settings for the numerical solver, primarily the time step and CFL number.

Stopping Criteria Termination criteria for the simulations, e.g. maximum number of iterations.

Reports Integral quantities, such as force coefficients, volume integrals, surface averages, or custom expressions.

Plots XY plots, such as residuals or report monitors vs iteration/time, or C_P vs x-coordinate.

Scenes Geometry, Mesh, or Scalar Scenes to visualise results.

3.4   Pre- and Post-processing steps

3.4.1   CAD for 2D aerofoil

Obtain an aerofoil definition, e.g. from http://airfoiltools.com/airfoil/naca4digit. If necessary, convert the coordinates to a (headerless) two- or three-column CSV file.

Right-click on `Geometry` ⟩ `3D-CAD Models` and select `New`. This will change the tree to that of the 3D-CAD tool and show you the coordinate system in the scene "3D-CAD View 1". Return to the main simulation menu any time through the tab `Simulation`.

Right-click the top of the tree `3D-CAD Model 1` and select `Import` ⟩ `3D Curves` (or 2D Curves). Navigate to and `Open` the CSV file that you downloaded from Blackboard Learn. Select *polyline* (to avoid error-prone interpolation) and *close the curve* to make sure the aerofoil geometry is watertight.

Right-click on the created sketch and `Extrude` it. Double-check the defaults so that the direction is in z and the Extrusion Options are set to "One Way". Choose a distance of a few chord lengths to ease in visualising the part. Set the Body Interaction to `None`. Confirm with `OK`.

Inspect the geometry! Exit CAD by pressing `Close 3D-CAD`.

In the main tree, right-click on `Geometry` ⟩ `3D-CAD Models` ⟩ `3D-CAD Model 1` ⟩ `Bodies` ⟩ `Body 1` and select `New Geometry Part`. Leave all defaults and confirm. This imports the CAD part into `Geometry` ⟩ `Parts`. Rename[2] "Body 1" to "Aerofoil".

Create a new solid primitive part. Right-click on `Geometry` ⟩ `Parts` and select `New Shape Part` ⟩ `Cylinder`. Specify appropriate start and end coordinates (respective centres of the endcaps) and radius. To separate the domain boundary from the rest, right-click on `Cylinder` ⟩ `Surfaces` ⟩ `Cylinder Surface` and split it by part curves. Create a geometry scene if you have not done yet, and rename the appropriate surface to "farfield".

---

[2]right-click, or select and press `F2`

3.4.2 Visualising flow quantities in the domain

Create a new scene. In the tree, right-click on [Scenes] and select [New Scene ⟩ Scalar] to create "Scalar Scene 1" (or 2, 3, ...). To hide the default outline, double-click on [Scalar Scene 1 ⟩ Displayers ⟩ Outline 1].

Leave the scene open and create a new derived part on which to display your desired variable. In the tree, right-click on [Derived Parts] and select [New Part ⟩ Section ⟩ Plane...]. In the [Edit] menu, input the x, y, and z components of the plane origin and normal vector. Tailor this to your specific geometry. Under [Display], select [Existing Displayer ⟩ Scalar 1] to add this part to the displayer. Ensure the correct region has been added under [Input Parts]. Click [Create] and [Close]. This is not required for 2D simulations, where we can use the flow domain itself. Instead, select [Scalar 1 ⟩ Parts] and assign your region to the [Parts] property.

Assign the field function of the variable of interest to the displayer. Select [Displayers ⟩ Scalar 1 ⟩ Scalar Field] and assign a [Function] in the [Properties] menu.

3.4.3 Plotting the pressure coefficient over the aerofoil surface

Follow the same steps as outlined in section 3.4.2 to create a new plane section. You do not need to create a separate plane for this. Change the [Input Parts] to be *only* the wing/aerofoil surface; deselect the region if necessary. This is not required for 2D simulations, where we can use the region boundary of the aerofoil directly in subsequent steps.

Create a new plot. In the tree, right-click on [Plots] and select [New Plot ⟩ XY Plot] to create "XY Plot 1" (or 2, 3, ...). Select this plot in the tree and assign the above-mentioned aerofoil section to the [Parts] property.

Ensure your x direction is correct under [X Type ⟩ Vector Quantity]. Assign the "Pressure Coefficient" field function under [Y Types ⟩ Y Type 1 ⟩ Scalar Function]. If desired, select [Reverse] under [Axes ⟩ Left Axis]. Change the normalisation of C_P under [Tools ⟩ Field Functions ⟩ Pressure Coefficient] in the tree.