Some useful list of commands in XFOIL

1) **Loading an Airfoil**

- To load NACA airfoil

  Eg: `XFOIL c> NACA 2412`

- Some airfoil other than NACA
  Eg: `XFOIL c> load MH60.dat`, where MH60 is the filename consisting of airfoil co-ordinates (Trailing edge to Leading edge format)

Notice that Xfoil will return some of the specifications for the airfoil, including the location and magnitude of the maximum thickness, maximum camber, and other parameters.

2) **Setting the number of panels**

- This step is to ensure the airfoil has a decent number of panels to work on and does not have a very coarse distribution of panels.
- `XFOIL c> ppar`
- `XFOIL c> n`
- `XFOIL c> 240`

When you want to come out of the current level that you are in, press `Enter`.

3) **The OPER Sub-Level**

```
------------------
XFOIL c> OPER
```

This will produce the prompt

```
.OPERi c>
```

Type a "?" to see a list of available commands and a brief description of their use. This works on any level of XFoil.

In the OPER level this is what you will see after typing "?"

```
<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
```

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

4) Viscous Mode

Always start the viscous analysis at a good angle of attack. By good I mean, a small positive angle of attack, which does not show any convergence issues. It is always good practice to start at 0 or 2 degrees. I know you might have to analyse even for negative angle of attack but you will see how it is done.

However, before you even run the session create a polar file to save your data.

Type

```
.OPERi c> visc
```

This command will turn on the viscous mode. XFoil then prompts the user to input a Reynolds number. Notice that a “v” will now appear next to “OPER” in the prompt to indicate viscous flow.

```
.OPERv c> pacc
```

```
 Polar  1 newly created for accumulation
 Airfoil archived with polar: NACA 2412

Enter  polar save filename  OR  <return> for no file   s>
```

Enter filename, for example: n2412.pol.

Remember to save it with the extension ‘.pol’

Just press enter for the dump filename – you do not need to save that.

Change the iteration limit to 70

```
.OPERva c> iter
```

```
.OPERva   c>  iter
 Current iteration limit:          10

Enter new iteration limit   i>
```

Run it for an angle of attack first

```
.OPERva c> alfa 2
```

Now do it for a sequence of angle of attack

```
.OPERva c> aseq
```

Enter your first alpha, last alpha and incremental alpha (of around 0.2). See if it is converged at the last angle of attack you specified.

```
  8   rms: 0.3353E-04   max: -.3120E-03   C at   89  2
     a = 24.000      CL =  1.4241
    Cm = -0.0677     CD =  0.17595   =>   CDf =  0.00215    CDp =  0.17380
plradd ia,ip,cpol          111           0  5.62221510E-04

Point added to stored polar  1
Point written to save file  n2412.pol
Dump file unspecified or not available

OPERva   c>
```

This is how it would look when it is converged at all angles of attack that you specified and all this would be written to your polar file. However, as I said, we need to do for negative angles of attack as

well. Therefore, we have to initialize our BL parameters such that XFOIL can run for a lower angle of attack immediately after running for a high angle of attack.

To initialize, do the following

```
.OPERva c> vpar
```

```
..VPAR c> init
```

Here it will say BLs will be initialized to next point.

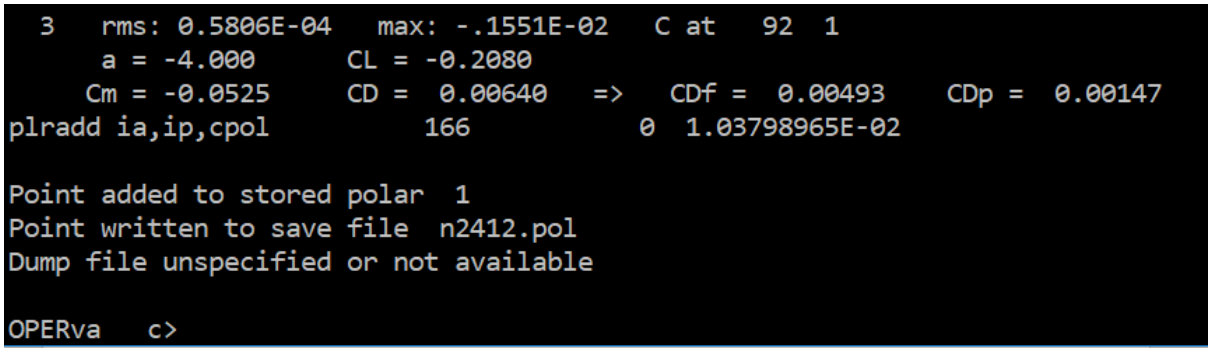
```
..VPAR c>
```

Press Enter to come out of VPAR and run it for a low alpha (say 2 degrees) and then run it for sequence of negative angles of attack. Remember to give increments as (-0.1 or -0.15) because you want for lower range of alphas.

```
.OPERva c> alfa 2
```

```
.OPERva c> aseq
```

Once that is done, it would show something like this

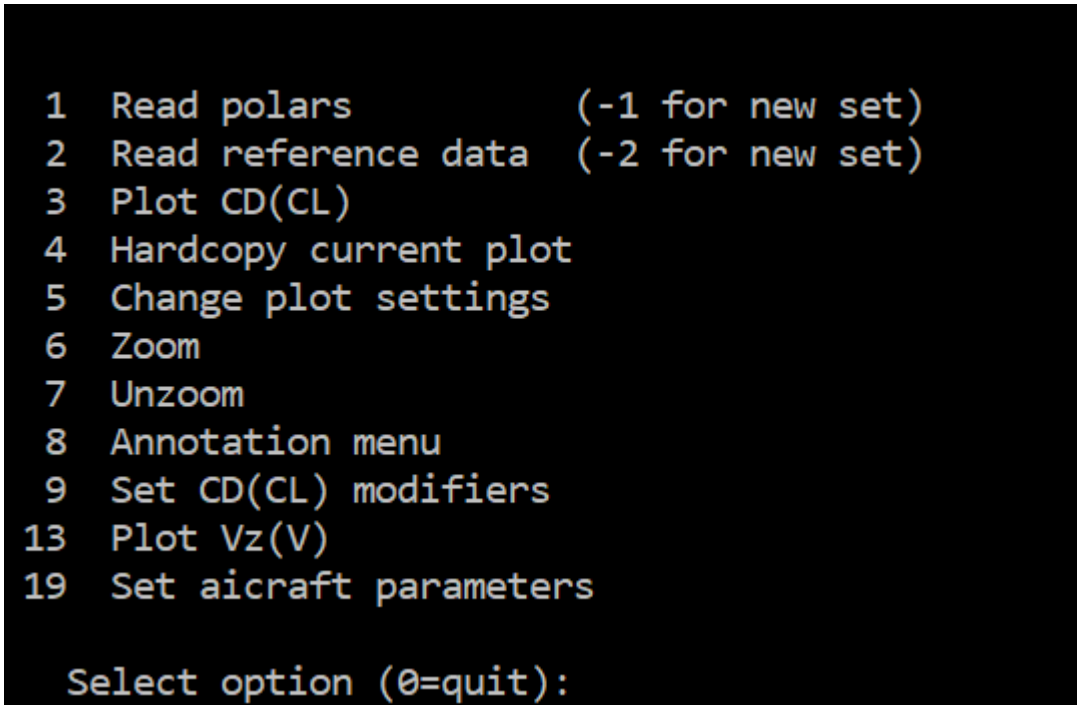


You basically have all the data that you need to analyse for your lab reports. Take your .pol file, read it in MATLAB, and compare it with experimental data.

If something in the above is not clear, ask me. You need to have an understanding of this to analyse Lab 3 results.

BELOW IS JUST FOR YOUR INFORMATION IF YOU WANT TO USE XFOIL IN FUTURE

Once you have generated your .pol file, exit xfoil and type pplot.



It shows all these options. To activate an option, type the number next to it. So if you want to read polars type 1. Enter your polar data filename (if you have more than 1, just enter those names as it keeps prompting)

When you are finished press Enter and a new plot window should pop up which shows the drag polar, Cl-alpha curve and transition curves. To get a copy of this plot, choose Option 4, which saves it as a postscript file (.ps). To change scaling of your axes, use Option 5.

This part is for your understanding as to how XFOIL data can be obtained using pplot. This is not required for your MAE 253 Lab reports.

If there is anything else you want to know, we can always talk.