Computational Aerodynamics Questions & Answers

Question by Student 201327133

Professor, i have a question about ximax. You showed us that ximax being similar as zero when state is steady. I run the code about Question #3, and i've got the result about time,iter,etc... but there is no more less than 1.0E+3 at ximax. but i can't believe that it could be zero... is it correct result? or are there more exact solution?

As I explained today, ximax probably won't go down to zero for your problems if you use the TVD Van Leer limiter or WENO. The residual will thus "hang". So, you can't rely on ximax to determine convergence. You need to proceed differently as explained in class.

Question by Student 201227111

Professor, I don't understand what the value from -opm command mean exactly. At first, I thought it was the average of values at x=x_station and y=(node1 to node 60). So, I extract data at i=2,10,20,30,40,50,60 and j=1 to 60 using "-pr" command. And averaging each value, And compare to value obtained from "-opm command" with varying x_station. (compare density, Temperature, velocity) When i is small, "-opm" value is similar to my average value. But the bigger the i, the bigger the difference. When i=60, "-opm command" value is similar to average value of i=60, j=1 to 20(only 1/3 nodes). So, I'm stuck here, I don't what to do after. Sorry about my poor explanation.

I need more information. Please show a plot of the pressure contours. Also, show your code in the Post() module. Further, how are you invoking warp with -opm? I hope you read in your data file... Otherwise, this will simply post-process the initial conditions..

Question by Student 201227111

Professor! I bring more explain about previous question. I wanted to know the meaning of Post() modules command to compute mass flux average stagnation pressure.
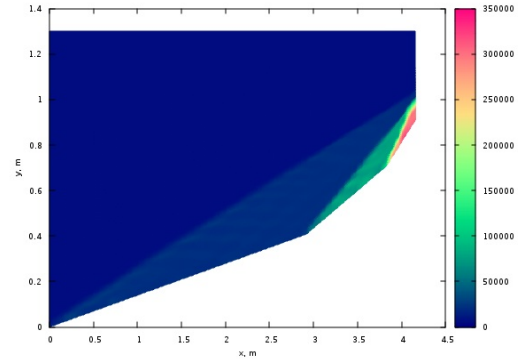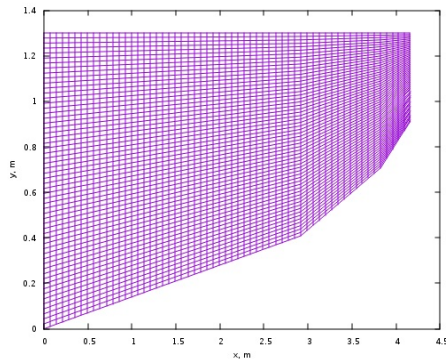
I simply gridding like this.(ie=60, je=60, dx is constant) Result is like this.

And My post code is same as original, only changing x_station[1] set to L-1e-5 (L is length of inlet)) and for loop range set 1 to 1.

```
#  X, Y, V[0], V[1], M[0], M[1], rho, P, T, a,
4.1585781115E+00  9.1104816400E-01  0.0000000000E+00  0.0000000000E+00  0.00
8.1710213132E+02
4.1585781115E+00  9.1764056800E-01  1.4327603644E+03  8.7202069975E+02  1.75
8.1710213132E+02
4.1585781115E+00  9.2423297200E-01  1.8688674662E+03  1.1369023634E+03  2.72
6.8495141257E+02
4.1585781115E+00  9.3082537600E-01  1.9144484891E+03  1.1672263446E+03  3.07
6.2309754554E+02
4.1585781115E+00  9.3741778000E-01  1.8569868290E+03  1.1360154632E+03  2.91
6.3760338939E+02
4.1585781115E+00  9.4401018400E-01  1.8829187596E+03  1.1600086610E+03  3.00
6.2621804714E+02
4.1585781115E+00  9.5060258800E-01  1.8885808498E+03  1.1567115341E+03  3.01
6.2728054526E+02
4.1585781115E+00  9.5719499200E-01  1.8866793073E+03  1.1426402424E+03  2.93
6.4183198872E+02
4.1585781115E+00  9.6378739600E-01  1.9168570097E+03  1.1355028504E+03  3.07
6.2288826298E+02
4.1585781115E+00  9.7037980000E-01  1.9283249661E+03  1.1261782347E+03  3.14
6.1285693704E+02
4.1585781115E+00  9.7697220400E-01  1.9475931664E+03  1.0887475754E+03  3.17
6.1255193448E+02
4.1585781115E+00  9.8356460800E-01  2.0128079347E+03  1.0455361008E+03  3.37
5.0670012146E.02
```

● ● ●

●
●
●

And I use "warp -r design3.wrp -i data.1500 -op post60.1 -pt gnuplot -pr 60 1 60 60" this command, I get properties that x is fixed in one line and y is vary.(like above picture) And, use matlab code, I compute average of each properties(ρ, T, q). Then, I compare this average value to value that obtained from "warp -r design3.wrp -i data.1500 -opm". (Because properties vary along y axis but using "warp -r design3.wrp -i data.1500 -opm" is give only one value)

```
INTROCFD.201227111:~> warp -r design3.wrp -i data.1500 -opm
Reading CFDWARP control file design3.wrp..
Grid..done;
Bdry..NavierStokesPerfect..done;
Reading data file data.1500 in CFDWARP binary format 010..done;
Model..NavierStokesPerfect..done;
Disc..NavierStokesPerfect..MUSCL..TVDRK..done;
Cycle..NavierStokesPerfect..PredictorCorrector..done;
Post..
Creating x-station..4.158568E+00..done.
x       = 4.158568E+00 m
Pback   = 8.965027E+03 Pa
Fpot    = 2.524588E+03 Ns/kg
mdot    = 2.160542E+02 kg/ms
htstar  = 3.443481E+06 J/kg
Tstag   = 3.440041E+03 K
Pstag   = 2.823567E+07 Pa
Pstar   = 1.853170E+05 Pa
T       = 7.891027E+02 K
q       = 2.297627E+03 m/s
rho     = 6.729664E-01 kg/m3
```

then my computed average value is $\rho = 0.2774$, $T = 467.5837$, $q = 2403.5$, but value obtained by "-opm command" is $\rho = 0.6730$, $T = 789.1027$, $q = 2297.65$.

I think value obtained by Post() module is average of value on fixed x line, but it isn't. It only work when i is very small(only 2 or 3..). So I want to understand about meaning of value that obtained by Post() module. or I think something wrong?

Everything seems fine except your matlab values. Are you doing a simple averaging using matlab? This will of course be wrong. You need to do a mass flux averaging (do $\frac{1}{m} \int P d\dot{m}$ in matlab not $\frac{1}{H} \int P dy$).

Question by Student 201327103

professor this is my counter plot for cylinder.
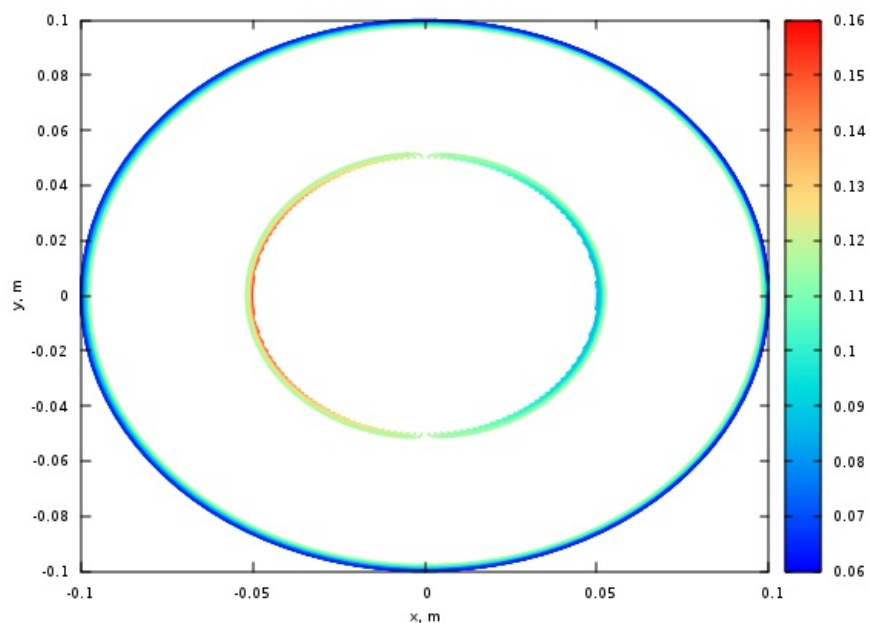
I can't get shock wave. I Think my boundary condition is wrong, but i don't know where is wrong. here is my code

The problem may be that you are imposing free stream boundary conditions on all boundary nodes..? You should check your boundary conditions are well imposed using the -on flag.

Question by Student 201327103

Professor, i check the boundary node and it looks like good. but the result still has same problem

```
FLUID NODE TYPES:

***************************
*99999999999999999999999999*<j=25
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*LL++++++++++++++++++++++LL*
*5555555555555555555555555*
***************************
```

This looks fine: your boundary conditions are well imposed. The contours you showed previously seem to have not been iterated at all. After the iteration process, are you reading in the data file (obtained after 1000 iterations or so) before outputting the post file?

Question by Student 201327132

Dear professor, I have a question about order of accuracy P. Assuming P>1, We obtained GCI_f. If P<1, Should we change to $\left|\frac{\triangle x_c}{\triangle x_f} - 1\right|$? I obtained P that is smaller than 1 and minus value. Did I do it wrong way? Thank you.

Very good question. We made this assumption only when deriving the GCI. So,

within the $\mathrm{GCI_f}$ equation, you should change the term to $\left|\left(\frac{\triangle x_c}{\triangle x_f}\right)^p - 1\right|$. That is, the GCI should always be positive. However, when determining order of accuracy p, we did not use the GCI and we did not assume that p should be greater than 1. So you should not change any of the equations used to determine p.