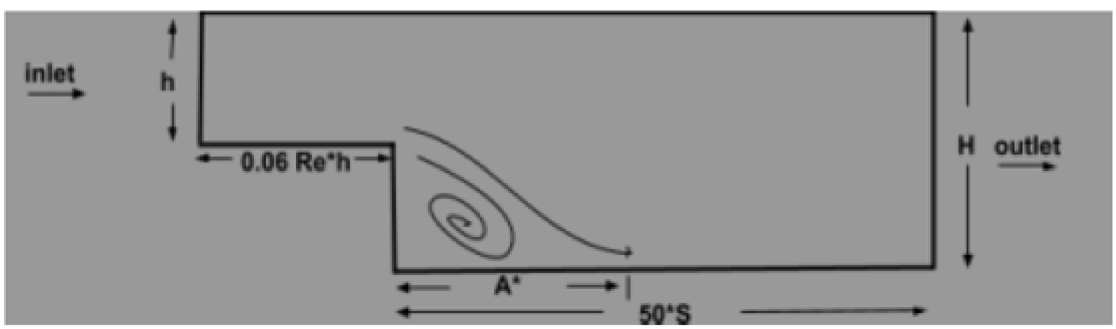
Reattachment Length and Velocity Profiles in a Back-Facing-Step using OpenFoam

Anish Oturkar

April 30, 2024

Problem Statement

Given a BFS (Back-Facing-Step) of dimensions and having a flow of air of Reynolds Number $Re = 200$, kinematic viscosity $\nu = 1.76 \times 10^-5$, density $\rho = 1.247$ kg/m^3, $S = 15$ mm, $H = 30$ mm and $H/h = 2$. We need to find the Reattachment length (A^*), Velocity profile on, before and after reattachment point. Also how will A* change on turbulent flow? Let's answer!!



1 Calculations and Models

1.1 So Geometry calculations,

$H/h = 2$
$H($ Outlet diameter $) = 30$ mm $= 0.03$ m
$h($ Inlet diameter $) = 0.015$ mm
$S = 15$ mm $= 0.015$ m
$50 \times S = 0.75$ m
$0.06 \times Re \times h = 0.18$ m

1.2 boundary condition Calculations,

$\mu = \nu \times \rho$, $\mu(nut\ value) = 2.245 \times 10^{-}5$
Free stream velocity $V = \frac{Re \times \rho}{h} = 0.234$ m/s, $\mu($ nut value $) = 2.245 \times 10^{-}5$

I Used **Wolf Dynamics Turbulence Calculator**, to calculate the k, omega, nut, and Epsilon values. two major considerations have been taken while doing the simulations :

Turbulence Intensity T_u **= 5**% and **Eddy Viscosity Ratio** $\frac{\mu_t}{\mu}$ **= 10**%

$$\mu_t = 10 \times \mu \ , \ \mu_t = 2.245 \times 10^{-}4$$

These values are taken as mentioned in the **Altair inlet turbulence parameter** website listed in references section.More the Turbulence intensity more the turbulence. Therefore here is the pic of the following Values on Wolfdynamics and The vertices of the BFS to visualize it better!

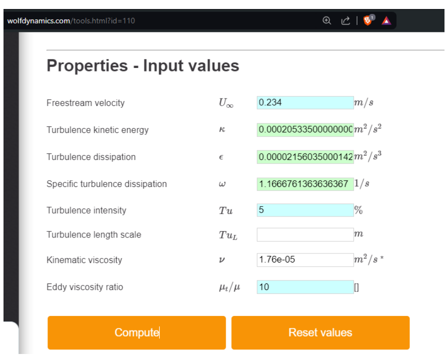


Figure 1: WolfDynamics Calculations

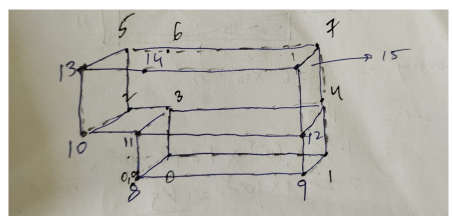


Figure 2: vertices of a BFS

The casestudy files with the **allrun command file** has been provided. There are two Models that I have run the Simulations on. One being **K-epsilon model** and the other being **K-omegaSST model**. According to some reports, both model work fine with BFS problem. K- omega overestimates A* and K-epsilon underestimates it.Both have their pros and cons. But I see k-omega SST is always preferred for turbulent and boundary wall conditions.
I have **not used** any external meshing or cad software to achieve results. I copied the **pitzDaily** folder from the **pimpleFoam directory** and made the respective changes in the files:

1. changed blockMeshDict: changes were done to the vertices of the geometry (removed the lowerwall and converted upperwall to walls) and improved suitable grading of the three blocks. The grading or Mesh size is concluded on the basis of how accurate i want the results and how much run time i am suitable with. The vertices were put after the geometric calculations were done.

2. k,epsilon,omega,nut, p, U : Changes were made to these files by putting the respective values from wolf dynamics calculator. This set up the boundary conditions complete and the model was ready to simulate. In k-omegaSST case I had to make omega file which I copied from **simpleFoam motorbike case** and subsequent changes were made to **fvSolution** and **fvSchemes** files.Seperate **run time changes through controlDict file** were made for it.

3. Changed controlDict : changes were done to the controlDict file in the cases of both models due to **timestep** and **Courant number** considerations. The Courant number(co) is kept **below 1 by using maxCo function**. **maxTime** is decided on convergence and thus numerical stability is ensured.

2 Results and Conclusions

I used **pimpleFoam** which is a **transient solver for turbulent flows in openFoam** and is based on Reynolds average stress modelling **RAS**. As I mentioned earlier I solved this problem in two different models and here are the results.Both models were run on **parallel computing using decomposePar**.

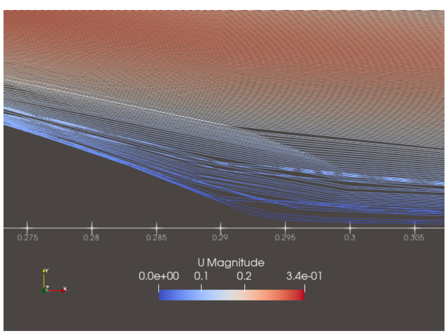


Figure 3: A* from k-omegaSST model

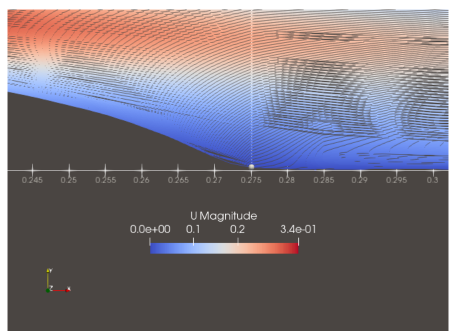


Figure 4: A* from k-epsilon model

Question1 : Thus from the above Images Reattachment length (A*) is achieved as **0.295m** for **K**-omegaSST model and **0.275m** for **K**-epsilon model.

Question2 : According to what I have read so far...for higher Re number like $Re > 4000$ (turbulent flow) some resources tell the that the A* shrinks on the basis that turbulence is created which creates better mixing of fluids leading to shorter A*. Also shrinkage of the recirculation zone is told to be done due to inertial forces (probably it doesn't consider $Re > 4000$). Some research papers claim to have lengthening and shrinkage of A^* over a range of Re. Some claim it to Length. I stay put on it will lengthen in this case. Reason being more turbulent flow, more it will travel distance from the step due to higher velocities and the reattachment process is disturbed. I did some analysis:

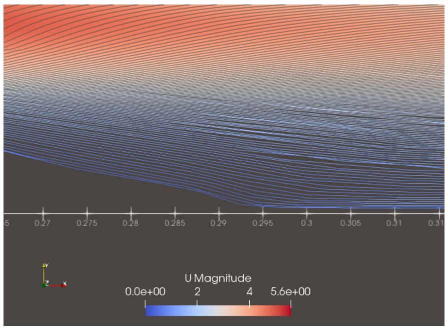


Figure 5: A* from kepsilon model (at turbulent intensity = 4)

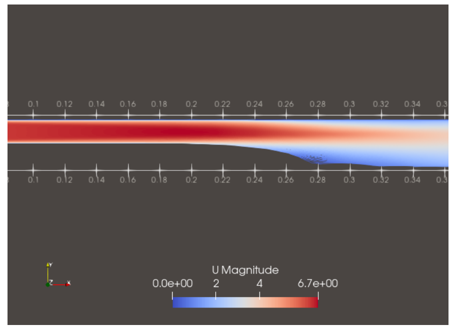


Figure 6: A* from k-epsilon model at Turbulent intensity =5% and Re = 5000

Both simulations that i run conclude that A* increases(even though i decreased turbulent intensity less in one from 5% to 4%).Probably this is due to the solver doesn't read turbulent intensity effect well. I have serious doubts on how much these two simulations have converged and the numerical stability of both.
Question3. Ooh now comes the exciting part.Velocity profiles obtained while running K-epsilon model (fig 7,8,9) and K-omegaSST model (fig 10) are displayed below:

- **In Fig 7** which is before A^*(0.25 m) we get quite a rise and dip in velocity initially due to the recirculation region. The velocity is higher in the middle of the vertex, this is signified by the small bump in velocity initially. due to **no slip boundary condition** velocity at the boundary is 0 m/s and at approx 0.019 m it is highest as the flow passes at high velocity through there. as shown in the figures 11 and 12

- **In Fig 8** which is @ A*(0.275 m) due to no slip boundary condition velocity at the boundary is 0 m/s and at approx 0.019 m it is highest as the flow passes at high velocity through there.

- **In Fig 9** which is after A^*(0.29 m) due to **no slip boundary condition** velocity at the boundary is 0 m/s and at approx 0.019 m it is highest as the flow passes at high velocity through there.you will observe a small variation initially
  this signifies the flow is reattached and the velocity is more at that height as compared to at A*.

- **In Fig 10** which is @ A* (0.295 m) as per K-omegaSST model due to no slip boundary condition velocity at the boundary is 0 m/s and at approx 0.019 m it is highest as the flow passes at high velocity through there.you will observe a small a bump at the right side this time. This is due to as K-omega is a good solver for turbulent flows it also simulated a recirculation zone on the wall above the flow (which you can observe in fig 11,12) this causes the bump like in fig 7 , same case..quite interesting right?

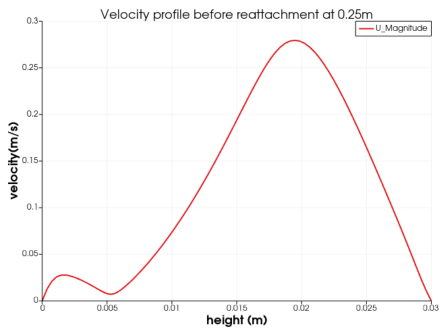


Figure 7: Velocity profile before A*

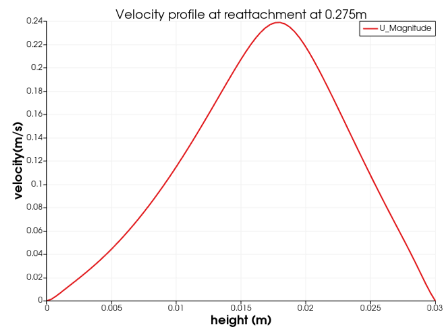


Figure 8: velocity @ A*

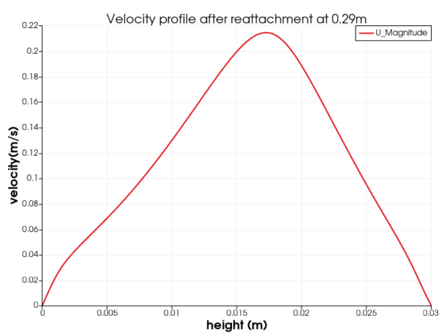


Figure 9: Velocity profile after A*

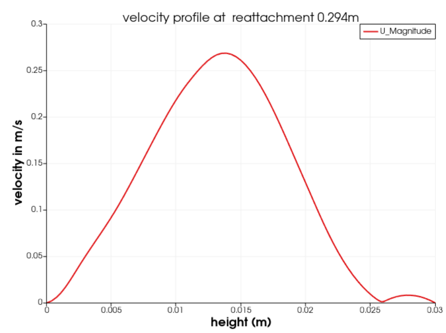


Figure 10: velocity @ A* for K-omegaSST

And finally for what you have been waiting for **BFS velocity flows!!**
As we can observe the k-omegaSST has better flow characteristics on boundary side than k-epsilon model.We can see how the flow later in the tube goes to a constant velocity. The max velocity is 0.34 m/s in both the simulations. We can also watch the inlet velocity to be 0.234 m/s by colour code.

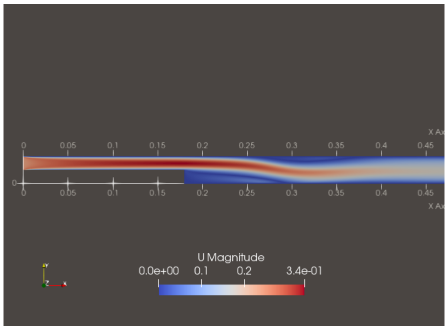


Figure 11: BFS for K-omegaSST model

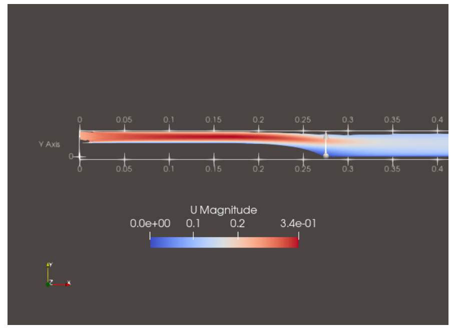


Figure 12: BFS for k-epsilon model

References

1.wolfDynamics turbulence properties calculator

2.Backward-Facing Step Flows for Various Expansion Ratios at Low and Moderate Reynolds Numbers by G. Biswas*, M. Breuer, F. Durst

3.Altair inlet turbulence parameters

4.Spoken Tutorial

5.OpenFoam backfacingstep simulation on pimpleFoam- Pawel Lojek

Closing Lines

I Thank FOSSEE for providing such an opportunity. This past few days I have played a lot with openFoam trying different parameters, models and etc. I hope to see you soon with new challenges and a lot playing!!