

Flow Over Cylinder

Gönül Öykü Güngör
Department of Mechanical Engineering
Bilkent University
Ankara, Turkey

Abstract— CFD analysis of flow over a cylinder for different Reynolds Numbers is explained in this paper by using different turbulence models.

Keywords—Flow over cylinder; turbulence models; CFD; Reynolds vs Drag coefficient

I. INTRODUCTION

Computational fluid dynamics (CFD) is a tool to solve fluid flow problems; it has become an alternative to experiments in a lot of cases. In many ways CFD tools can be superior to experimentation. Although it is used mainly it is open to debate at what extend the data obtained can match experimental data. Flow over cylinder experiments has been conducted before; drag coefficients at different Reynolds numbers are tabulated. This papers aim is to model a 2D not wall bounded problem, with different turbulence models and see if the data matches the experimental values.

A. Hardware used for simulations

- Asus N56-VZ Intel Core i7-3630QM,2.40 GHz, 12 GB RAM, Win 8 x64
- HP-Z600 Intel Xenon CPU X5660, 2.67/2.66 GHz, 12GB RAM, Win 7 x64

B. Software used

- FlowVision Version 3.09.03

II. PROCESS&CFD MODEL

Steps consist of creating a model, verification of this model and executing detailed CFD's and calculations. When creating the model the documentation of Bilkent Universities wind tunnel was used, some dimensionally unspecified models were scaled accordingly.

The main idea of this paper was to observe effects different turbulence models, drag coefficients at different Reynolds numbers. To observe these at first the exact geometry of the wind tunnel at Bilkent University was first taken, so that the results from the analysis and experimental results could have been compared. Although as proceeded further with the analysis, it was observed that similar results are not reachable with that specific wind tunnel geometry. What was observed was that because of the tightness of the test section the flow characteristics were not able to fully develop.

System is assumed to be stable which means air flow is steady, uniform and unidirectional. When there is high

velocity flows over small geometries, the best is to start the analysis with time dependent solver, after the solutions reach a stable point, it is logical to switch to time independent, steady-state solver. The reason for this is; the initial inputs in steady-state solver are crucial, these guesses may not be correct even if it correct the analysis can give physically impossible solutions. With time independency computation increases but outcomes become more reliable.

A. Solid Geometries

The test section is drawn with AutoDesk Inventor, cylinder is imported from the FlowVision's library as a cone but then transformed to a cylinder, *.stl format. To find the flow characteristics 2D cylinder is used. Dimensions of each are as follows;

- Test section: 0.9(height) x 0.9(width)x 3(length) [m]
- Cylinder: 1(length) x 0.06(diameter)[m]

Dimensions of the cylinder were given according to the magnitude of the test section. If the geometries are too big when compare with the test section, accuracy of these results decrease.

The test section which is the flow volume is defined as the region; cylinder is defined as moving bodies. If moving body is not chosen, the geometry is sensed as a hole in the region which is not possible to move around. Moving body is able to translate, rotate, can have inertia, these can be set to zero still defining geometry as moving body gives flexibility.

B. Material Characteristics

Air is the fluid in this model and both modelled as incompressible and incompressible flow depending on the velocities.

- Incompressible region: Temperature is taken to be constant 300 Kelvin flow is assumed to be incompressible because it is smaller than 0.3 Mach, energy equations were not solved for this reason because there are no significant heat sources or similar physical phenomena. As a result computation time is decreased. Consequently air properties are only dependent on air pressure by this density is

1.177 kg/m³, dynamic viscosity is 1.846 x 10⁻⁵ m²/s, kinematic 1.568 x 10⁻⁵ m²/s.[2]

- Compressible region: After a critical Reynolds number Mach number becomes greater than 0.3, because of this fact flow becomes compressible and density is no longer stable. Changes in density create a disturbance in the thermodynamic equilibrium which leads to temperature variations. These temperature changes are no longer negligible, if these are not taken into account the accuracy of the solution can be affected drastically [1]. In order to obtain better solutions the heat transfer equation must be turned on. Ideally in the incompressible region, temperature is always 300K, in reality there can be slight temperature variations, using energy equation in this domain does not create a huge difference but increases the computational cost.

C. Other Settings

- Gauge pressure is taken as 101.000 Pa which is a decision made for the sake of numerical efficiency
- Fluid is newtonian
- Heat transfer is turned off for incompressible, on for compressible
- Different turbulence models are checked. Turbulent intensity(pulsations) and scale are determined from Reynolds Numbers
- For external flow around an obstacle to be turbulent; Reynolds Number(Re) should be greater than 20.000[2]

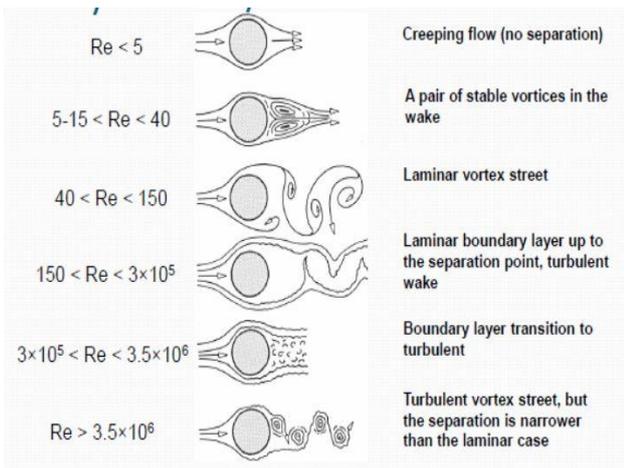


Figure : Relation between Reynolds number and flow characteristics

D. Turbulence Models&Important Parameters

There are six different turbulence models in FlowVision, different models can be applicable to a certain task the best is to try the best one or optimal combinations to obtain accurate and realistic results.

Another important thing to say is that laminar flows can also be modelled with certain turbulence models but using a turbulence model for a turbulent flow is a must.

Difference between these models is the number of equations; different kinds are “Eddy viscosity Models”, “Reynolds Stress Model” and “Kolmogorov Scale”[3]. From left to right the accuracy of the models increases along with the computational time.

Models in FlowVision fall in the “Eddy Viscosity Models” which have one or two equations [3]. Models used are as follows

- Model KES(standart k-ε)
- Model KEQ (nonlinear quadratic k-ε turbulence model)
- Model KEFV (k-ε turbulence model FlowVision)
- Model SA (Spalart-Allmarasa)
- Model SST (Shear Stress Transport)

There are two parameters that are crucial about using turbulence models, turbulence intensity and turbulent scale. If one is able to make good assumptions about these no matter what the model is the results will not be too unusual.

Turbulence flow properties such as velocity, pressure, temperature can be decomposed into mean and fluctuating parts, where \bar{u} indicates mean, $\hat{u}(t)$ indicates fluctuating part [4]

$$u(t) = \bar{u} + \hat{u}(t)$$

In practice measured velocity points represent a series of discrete points although, theoretically velocity data is continuous and the mean part can be found by integration.

$$\text{Mean velocity: } \bar{u} = \int_t^{t+T} u(t)dt = \frac{1}{N} \sum_1^N u_i$$

$$\text{Turbulent Fluctuation: } \hat{u}(t) = u(t) - \bar{u} \text{ OR } \bar{u}_i = u_i - \bar{u}$$

$$\text{Turbulence Strength: } u_{rms} = \sqrt{u'(t)^2} = \sqrt{\frac{1}{N} \sum_1^N \hat{u}_i^2}$$

$$\text{Turbulence Intensity: } u_{rms} / \bar{u}$$

By looking at the equations it can be said that turbulence intensity is the fluctuating part over the mean part of velocity, showing the level of fluctuations. As

Reynolds number increases the fluctuation part gets higher.

Obtaining this data is highly experimental but depending on the research done previously for low *Re* the intensity is between 0% - 3%, for moderate 3% - 5%, and for high 5% - 10% for external flows. Flows with intensities greater than 10% is not very common, for instance a percentage around 20% indicates mach number greater than 1[5]. Initial guesses can be seen in Table 1.

It may not be possible to give exactly the correct intensity for each Reynolds, depending on intensity choice values may not match the experimental data. To make sure user can start with an initial guess and check the “Turbulent Kinetic Energy” of the system. By using the following equation values given at the inlet can be checked and iteration can be made to get a correct value [6]

$$I = \frac{\sqrt{\frac{2k}{3}}}{V}$$

k = Turbulent Kinetic Energy

V = Inlet Velocity

I = Turbulence Intensity

The second important parameter is “Turbulence Length Scale” which defines the size of the energy containing eddies in the flow, eddies start as big and decay by the time they hit the obstacle. There are several ways to define this value; for internal flows the hydraulic diameter, for wind tunnel problems honeycomb diameter, for wall bounded flows a relation including boundary layer thickness can be used. For external flows there is no exact empirical correlation and this makes it hard to define a value, the best way is to conduct experiments and evaluate the long continuous data samples [7].

To make a guess about length scale, it was assumed as if there are honeycombs at the inlet. Honeycomb diameter was taken as the Bilkent Universities wind tunnel’s honeycombs, which were 8 mm’s, so the initial size of the eddies are 8mm. What is expected is a decrease

in size of the eddies by the time they come across the cylinder, the average of the initial size and size of the cylinder diameter was taken. Dissipation rate was significantly observed and it converged in each solution, so the guess made was suitable for this type of problem.

E. Initial Values

- Velocity= Differs according to different Reynolds Numbers, refer to Table 1
- Pressure = 0 (open to atmosphere, 1 atm)
- Pulsations(Intensity)= Differs according to different Reynolds Numbers, refer to Table 1
- Turbulence scale= 0.03

F. Boundary Conditions

Four boundary conditions are defined. When assigning these conditions first the type of the geometry should be chosen then other needed specifications should be assigned depending on the type.

Walls of the test section were first chosen as “Wall” then replaced with “Symmetry”. The reason is that the outer region is big enough, so it does not play any role in the characteristic of the flow. “Symmetry” is also defined as zero gradient.

Inlet is chosen as “Inlet/Outlet”. Variables are defined as;

- Velocity = Normal velocity with pressure
- Turbulence Energy= Pulsations (intensity). Check table for inlet values
- Turbulence Dissipation= Turbulent scale(length) is determined to be 0.008, as stated previously this is chosen from the 8 mm honeycombs

Outlet of the test section is chosen as “Free outlet”. This option applies a zero gradient to all variables along the boundary, excluding pressure. Now variables are;

- Velocity = Pressure. Region has atmospheric pressure, because the object is in the middle and small, assigning Pressure to Velocity gives numerically correct solutions.
- Turbulence Energy= Zero gradient
- Turbulence Dissipation= Zero gradient

Object is “Wall” and wall function was enabled according to the conditions.

<i>Re</i>	10 ⁻¹	1	10	10 ²	10 ³	10 ⁴	10 ⁵	10 ⁶
Velocity [m/s]	2,6139E-05	2,6139E-04	2,6139E-03	2,6139E-02	2,6139E-01	2,6139	26,139	261,39
Pulsation Initial	0.002	0.004	0.006	0.007	0.009	0.02	0.05	0.08
Pulsation Inlet	0.001	0.003	0.005	0.006	0.008	0.01	0.04	0.07

Table 1: Reynolds number and turbulence intensities at and at boundary conditions (initial) and at the inlet

G. Grid

Grid in FlowVision is constructed as rectangular parallelepiped, faces each cell are parallel to the axes of the global coordinate system.

“Law of the wall” is the main consideration when meshing the system. This law is used to describe turbulent velocity profiles universally. In order to get accurate solutions grid elements near wall are critical which depends solely on the dimensional distance y^+ . Prandtl’s analogy shows that the velocity profiles in the neighbor cells near the wall should follow[7];

$$u^+ = u^+(y^+)$$

This relation has been also confirmed numerically by many such as;

Author	Range of validity	Formulas
Prandtl	$0 \leq y^+ < 11.5$	$u^+ = y^+$
Taylor	$11.5 \leq y^+$	$u^+ = 2.5 \ln y^+ + 5.5$
von Karman	$0 \leq y^+ < 5$ $5 \leq y^+ < 30$	$u^+ = y^+$ $u^+ = 5 \ln y^+ - 3.05$
Reichardt	$30 \leq y^+$ $0 \leq y^+$	$u^+ = 2.5 \ln y^+ + 5.5$ $u^+ = 2.5 \ln(1 + 0.4y^+) + 7.8\{1 - e^{-y^+/11} - (y^+/11)e^{-0.33y^+}\}$
Deissler	$0 \leq y^+ < 26$ $26 \leq y^+$	$u^+ = \int_0^{y^+} \frac{dy^+}{1 + n^2 u^+ y^+ (1 - e^{-n^2 u^+ y^+})}$ $n = 0.124$ $u^+ = 2.78 \ln y^+ + 3.8$
van Driest	$0 \leq y^+$	$u^+ = \int_0^{y^+} \frac{dy^+}{1 + \{1 + 0.64y^{+2}[1 - \exp(-y^+/26)]^2\}^{1/4}}$
Rannie	$0 \leq y^+ < 27.5$ $27.5 \leq y^+$	$u^+ = 14.54 \tan h(0.0688y^+)$ $u^+ = 2.5 \ln y^+ + 5.5$

Figure: Different formulas for “law of the wall”[7]

y^+ of von Karman is commonly used, the first correlation defines viscous sublayer, second buffer layer/blending region, third fully turbulent/log-law region. What is suggested for this model is; if y^+ is chosen in the first region, it indicates enhanced wall treatment therefore wall function is not needed.

If y^+ is in third region user has to solve using “Wall Function”. If this function did not exist user would have to

resolve grids of the turbulent boundary layer near the solid object. Existence of this decreases great amount of computational effort. Choosing dimensionless distance in the buffer region presents challenge [9] [10].

The cell count increases drastically as y^+ values approach to zero, values change depending on the Reynolds number and length(L) of the solid object. Relation below is used to approximate the size of the mesh in external flows[4]

$$\frac{\Delta y}{y^+} = 8.6 L Re^{-13/14}$$

Although it is suggested that y^+ should be in strictly in certain regions some CFD tools such as FlowVision blend the previously stated “law of the wall” formulas to eliminate the uncertainty in the buffer region. It is stated that in the new versions of the CFD tool, y^+ greater than five with wall functions are still applicable [6].

Whether this works or not for this particular problem will be stated under the Results&Discussion part.

Initial grids according at different Reynolds numbers can be seen at the Table 2 below, the main was to keep the y^+ values between 0-5 ranges as possible to get the best results

H. Solver Settings

Mathematical calculations are done based on core fluid mechanics equations discretization and numerical solutions. The steps followed are important because the outcome of these calculations manipulate the algebraic and functional residuals of variables, such as Pressure, Velocity, and Turbulent Energy. If .If they are not small enough more iterations are needed.

Another variable that has to be determined is the time step. CFL (Courant-Friedrichs-Lewy) is chosen for this. CFL number represents the ratio of the time step of the time for which the perturbation of the flow stream is transferred within the cell [12]. User can change CFL but ideal value is one for this dimensionless number and, as this number increases, time to solve the problem decreases. In this model CFL number is taken as one.

Re	10^{-1}	1	10	10^2	10^3	10^4	10^5	10^6
$y^+ \pm$	0.002± 0.001	0.008± 0.005	0.029± 0.022	0.21± 0.15	1.24± 0.76	1.46± 1.02	5.07± 3.69	7.00± 4.99
disper.								
Cell count	9412	10116	10116	11460	11536	50188	99408	387204
Wall Function	Off	Off	Off	Off	Off	Off	On	On

Table 2: Grid characteristics according to different Reynolds numbers

III. RESULTS

Drag coefficients at different Reynolds values that can be seen at Table 1 & 2 was calculated by using different turbulence models. SA, SST, KES, KEQ, KEFV was used all for the range $0,1-10^5$. To see the decrease at point where Mach becomes greater than 0.3, critical Reynolds was solved for once with KEQ, which gave accurate results, Table 3 below shows the actual drag coefficient values, data are extracted from extraction software.

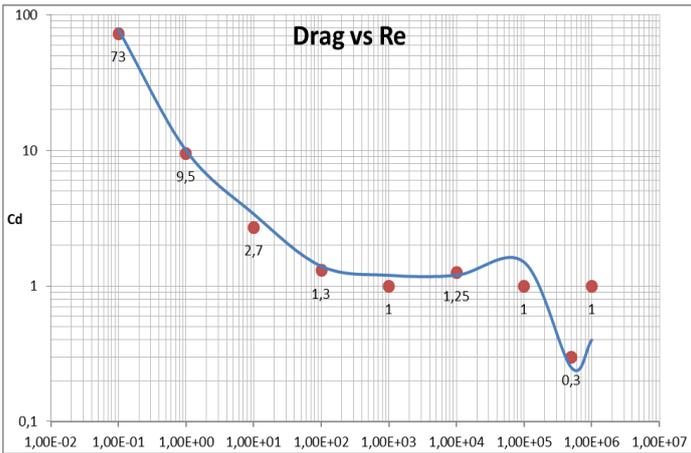
10^6 was only solved only using KES and KEQ this region falls into Mach number~1 so the computational effort it takes to solve is almost half a month. For the models KEFV, SST and SA; KEQ value was used in the plots.

Re	10^{-1}	1	10	10^2	10^3	10^4	5×10^5	10^5	10^6
Actual C_d	78	10	3.4	1.4	1.25	1.2	0.25	0.25	0.4

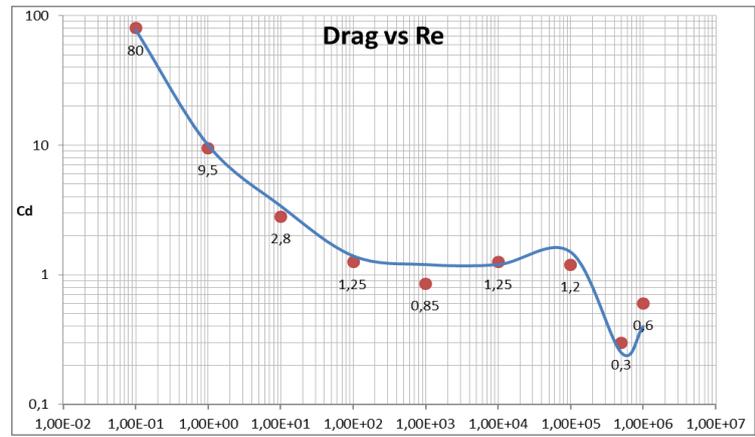
Table 3: Experimental Drag Coefficients vs Reynolds Number

A. Plots Comparing Different Models and Experimental Values

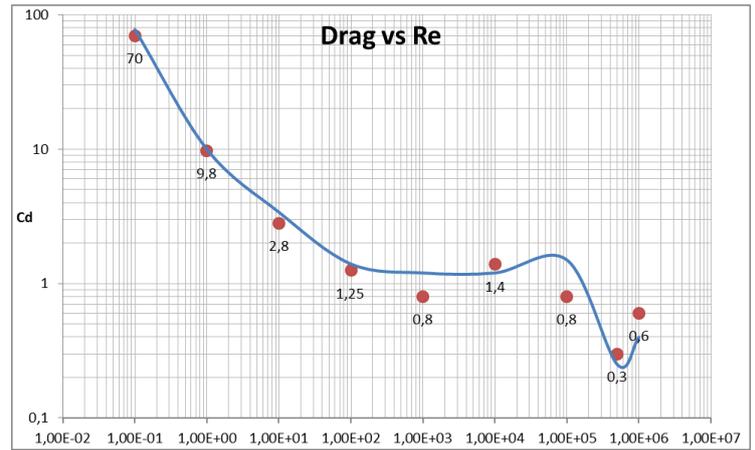
The line indicates the original plot with experimental values while the dots indicate CFD solutions.



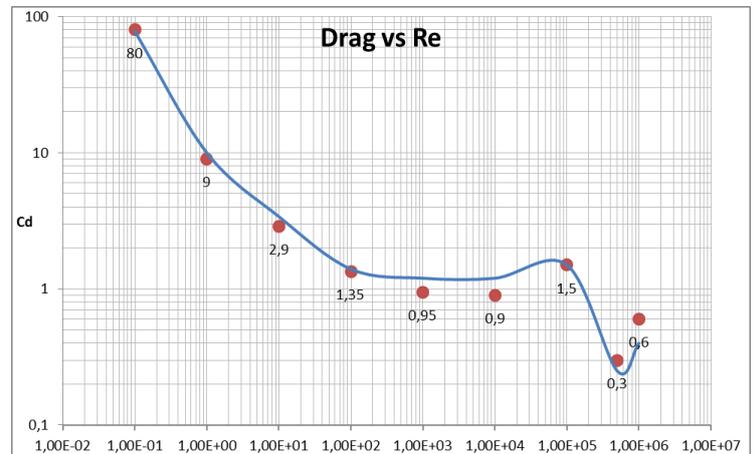
Graph 1: Comparison with KES



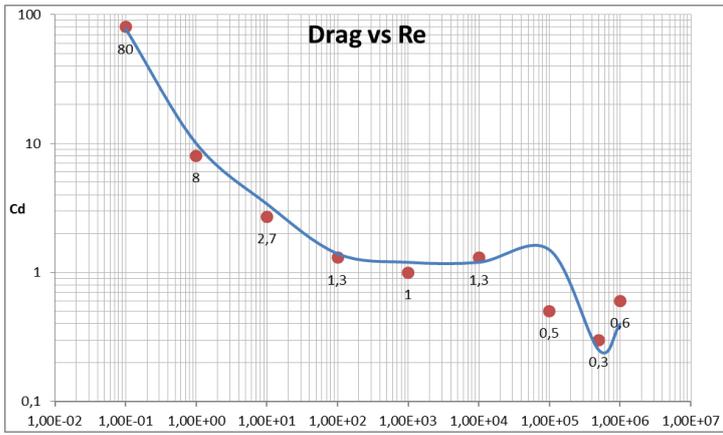
Graph 2: Comparison with KEQ



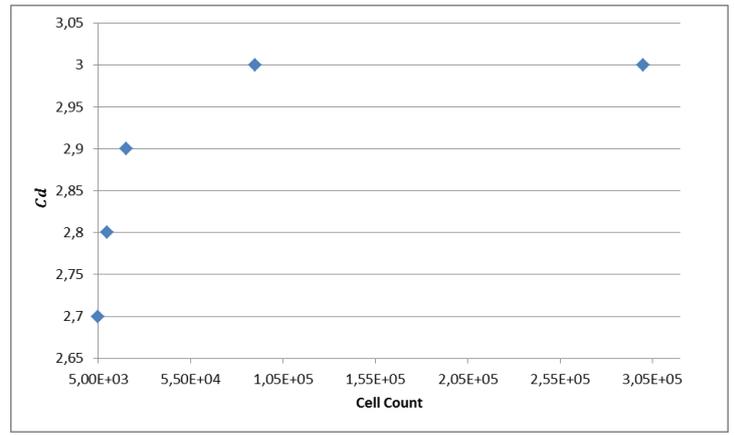
Graph 3: Comparison with KEFV



Graph 4: Comparison with SST



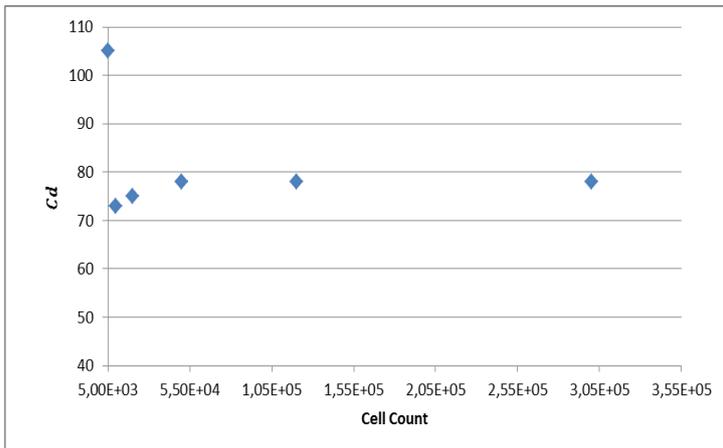
Graph 5: Comparison with SA



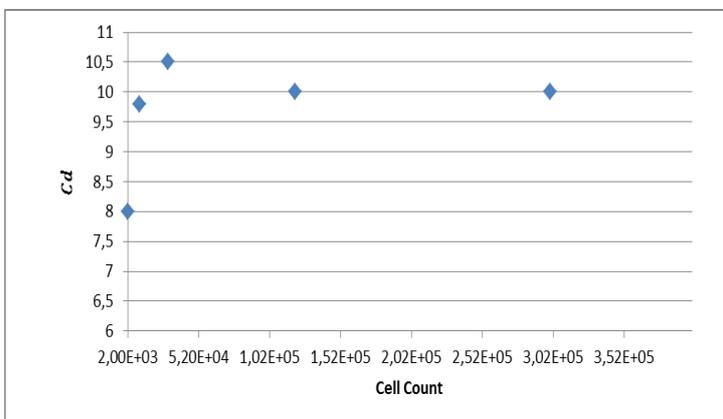
Graph 8: Reynolds 10 with SST . Converges to actual value after ~90000 cell count

B. Mesh Independence

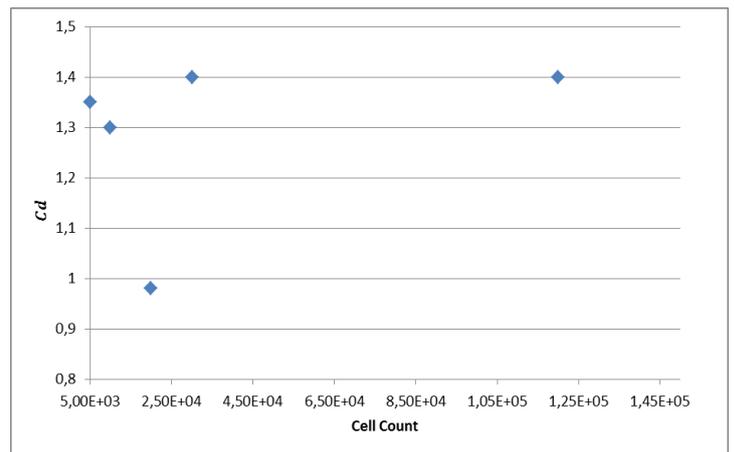
For all Reynolds numbers the turbulence models that gave the closest results were chosen for mesh independence study.



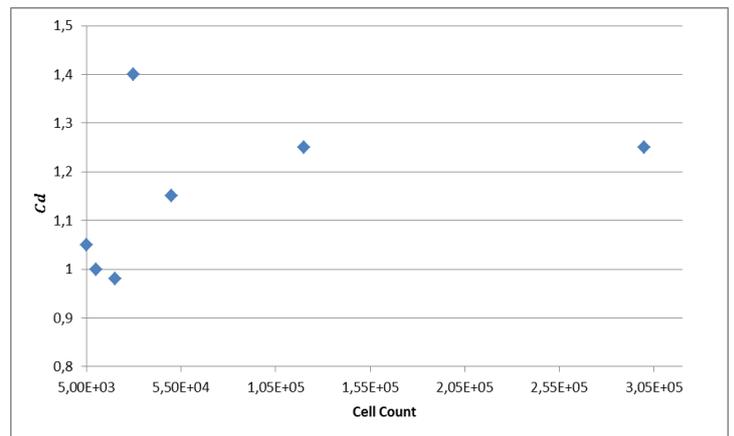
Graph 6: Reynolds 0,1 with KES. Converges to actual value after ~50000 cell count



Graph 7: Reynolds 1 with KEFV. Converges to actual value after ~120000 cell count



Graph 9: Reynolds 10² with SST . Converges to actual value after ~30000 cell count



Graph 10: Reynolds 10³ with SST . Converges to actual value after ~120000 cell count

IV. DISCUSSION & CONCLUSION

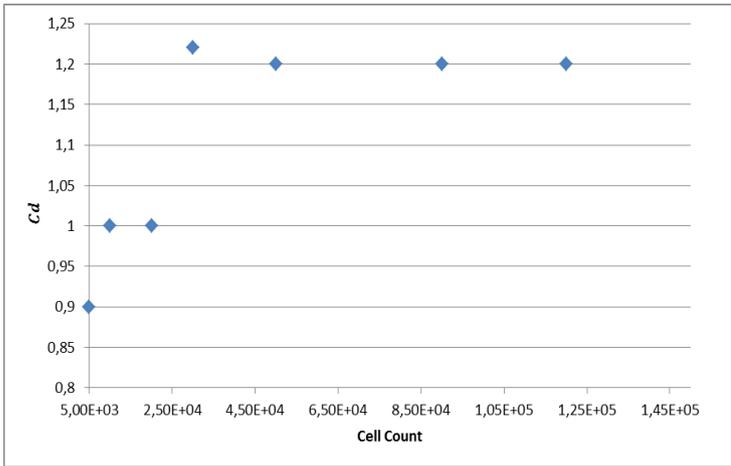
According to graphs the most successful models for each Reynolds numbers are;KES-KEFV-SST-SST-KES-KES-KEQ. Depending on the results for low Reynolds Numbers; smaller than 10, best is to keep y^+ under one. For moderate Reynolds y^+ can be chosen at any interval, of course to obtain closer results is should be taken in viscous sublayer; however for some cases; too small y^+ can cause negative effects. For flows greater than 0.3 Mach y^+ must be in viscous sublayer region, for high speed flows keeping y^+ between 0-5 increases the cell count drastically along with computational time. For instance solving 10^6 with ~1000000 cell count has taken approximately three weeks with the specified hardware, so a computer with superior specifications is a must.

As stated in this paper before FlowVision had a claim that in the new versions of the software, it would be able to solve problems which fall inside the buffer layer. When doing mesh independence study it was seen that y^+ values between 5-30 also give fine values, accuracy increases as y^+ taken closer to 5. Unfortunately this claim is not applicable for compressible flows. What can be said is that; for some problems at some velocity or Reynolds range taking y^+ inside buffer layer and using wall function gives acceptable results.

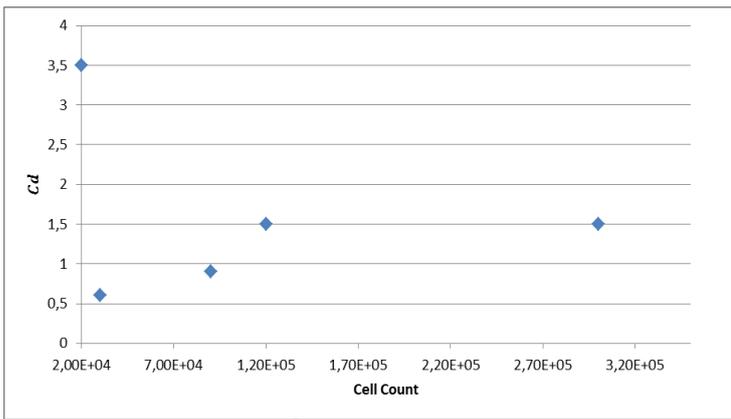
From the graphs; there is no ordered pattern which makes it possible to say use this particular model for this particular Reynolds number. There are many suggestions about the application areas of these turbulence models, checking those and applying them directly may not be a good decision. Before solving best is to read a lot and try to find similar papers, of course this never means what is written on the paper can be directly applicable to your problem. Even if everything is the same a slight change can make a model a bad choice. Choosing y^+ appropriate as possible as the computational power allows, length scale and intensity are the most vital parameter when turbulence modelling. There is no such superior model for all flows.

REFERENCES

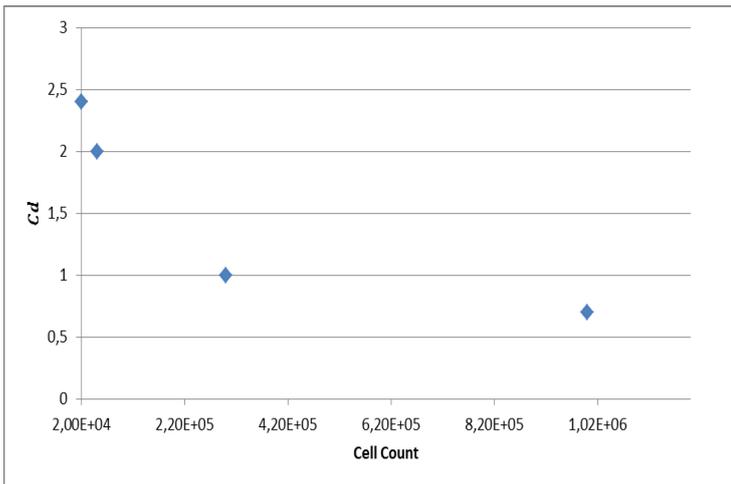
- [1] D.Wilcox,“Turbulence Modelling for CFD”, DCW Industries, 1993
- [2] H. Aksel, “Notes on Fluid Mechanics”, Vol 1-2
- [3] J. Sodja, “Turbulence Models in CFD”, Universt of Ljubljana, Faculty for mathematics and physics, March 2007
- [4] Chen, Yaw. “Fundamentals of Turbulence Modeling”, Taylor & Francis, 1998
- [5] Versteegi, Malalasekera. “An Introduction To Computational Fluid Dynamics the Finite Volume Method”, Longman Scientific&Technical, 1995
- [6] FlowVision Documentation, Capvidia
- [7] T.R Camp, H.W Shin, “Turbulence Intensity and Length Scale Measurements in Multistage Compressors”,
- [8] D.B Spalding, “A single Formula for the Law of the Wall”, p455-458, September 1961
- [9] G. Kalitzin, G. Medic, G. Iaccarino, P. Durbin, “Near-wall Behaviour of RANS Turbulence Models and Implications for Wall Functions”, Journal of Computational Physics 204(2005) 265-291
- [10] T. Nishino, G.T. Roberts, X. Zhang, “Unsteady RANS and Detached eddy-simulations of Flow Around a Circular Cylinder



Graph 11:Reynolds 10^4 with KES. Converges to actual value after ~50000 cell count



Graph 12:Reynolds 10^5 with SST. Converges to actual value after ~120000 cell count



Graph 13:Reynolds 10^6 with KEQ. No convergence to real value at ~1000000 cells

- in Ground Effect" *Journal of Fluids and Structures* 24 (2008) 18–33
- [11] H. Chanson, G. Carosi, "Turbulent Time and Length Scale Measurements in High-Velocity Open Channel Flows." *Experiments in Fluids* 42 (2008), pp. 385-401
- [12] H. Schlichting, K. Gersten, "Boundary Layer Theory", McGraw Hill, 2000