International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

CFD Modeling for Pipes under Area Change and Application to Clogged Oil Coolers.

Amr Abbas

Uned University Spain

DOI: https://doi.org/10.55248/gengpi.4.1023.102652

ABSTRACT

The Research represent a CFD model for Flow across pipes after área change the study emphasis on the characteristics of the meshing affect the results and To observe the effect of the viscosity on sudden changes of geometry. To observe how the velocity profile varies along the duct. To observe how the temperature profile varies downstream of the heaters. To determine the variation of the pressure along the boundaries of the bars. The Study was applied on a clogged Fin Fan cooler and Good CFD results was concluded .

Introduction :

- The target of the study is to Analyze the temperature distribution at the outlet of a heater, which is represented as a two-dimensional duct for simplification, with a protrusion to access the area where the heat is supplied to the fluid. The problem can be considered as the result of applying a determined thermal energy to a fluid current, which in our case is done through some solid circles that emit a determined heat per unit volume. These circles can be considered as being the sections of steel bars, which are kept hot through a procedure outside of our problem. The fluid flows from the narrowest part to the widest part, establishing some determined velocity and temperature conditions at the narrow entrance of the duct. At the outlet, downstream in the wide part, the pressure has the same value as at the inlet. The heat transmission from the bars or circles to the fluid takes place by means of a convection phenomenon, although the influence of radiation on this transmission could also be tested.
- The convective heat transfer is initially tested and accordingly the Radiation will be introduced to the Model .
- The Model will be utilized to Analyze a forced draft cooler then studying clogged forced draft cooler with a status usually faced in industry causing imbalance in flow in the heater the clogging is represented by different areas in both left & right side of heater

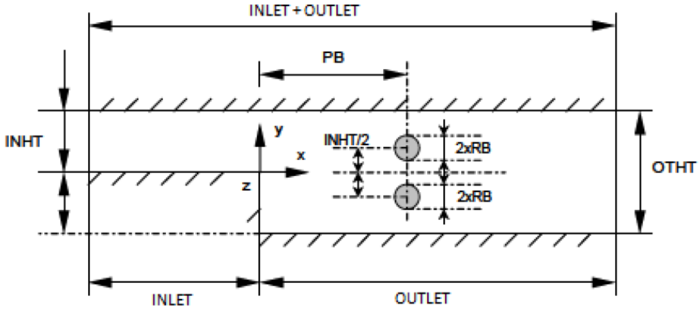
Problem Description

The trailing Drawing represents the problem Geometry and dimensions to study the effect of Area change and flow past cylinders with conjugate heat transfer .

The Geometry will be generated by Parabolized scheme in MSC Nastarn and will be exported to Mime for Meshing and accordingly to CFD++ for Analysis

Starting data

The geometric dimensions of the domain are shown in the figure below.



INLET = 0.250m

OUTLET = 1.000m

INHT = 0.050m

OTHT = 0.100m

PB = 0.250m

RB = 0.010m

The bars are considered to be steel; therefore the thermal characteristics of this material.

To facilitate the study, the following values can be considered:

• Conductivity: 67.4 W/m - °K

• Density: 7882.0 Kgr / m^3

• Specific heat: 511.0 J/Kgr - °K

The fluid can be considered to be water, air and air whose density is only function of the temperature.

In the case of water, the values of the properties to be considered are:

• Density: 961.30 Kgr / m^3

• Viscosity: 8.64 E – 4 Kgr / m – S

• Conductivity: 0.628 W / m - °K

• Specific heat: 4178.0 J / Kgr - °K

The Mathematical Model

2d segregated, k-ε solver is used to solve the governing differential equations for the conservation of the mass, momentum and energy equations. For incompressible (for water case) flow, the Navier-Stokes equations, the continuity, momentum and energy equations can be written as

(1)

$$\frac{\partial V_i}{\partial X_i} = 0.0$$

$$\frac{\partial V}{\partial t} + V_j \frac{\partial V_i}{\partial X_j} = -\frac{1}{\rho}\frac{\partial P}{\partial X_i} + \nu \nabla^2 V_i$$

$$\frac{\partial T}{\partial t} + V_j \frac{\partial T}{\partial X_j} = K \nabla^2 T$$

Turbulence Model

Two-Equation Realizable k-epsilon Model

In CFD++'s realizable k - ϵ model, the Boussinesq relation is again used to obtain Reynolds-stresses (algebraically) from the modeled eddy viscosity (μ_t) and the available mean-strain tensor:

$$\rho\overline{u_i u_j} \;=\; \frac{2}{3}\delta_{ij}\rho k - \mu_t S_{ij},$$

where

$$S_{ij} = \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3}\frac{\partial U_k}{\partial x_k}\delta_{ij}\right).$$

The model consists of the following transport equations for k and ϵ :

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}(U_j \rho k) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\epsilon,$$

$$\frac{\partial(\rho\epsilon)}{\partial t} + \frac{\partial}{\partial x_j}(U_j \rho\epsilon) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + (C_{\epsilon 1}P_k - C_{\epsilon 2}\rho\epsilon + E)T_t^{-1},$$

in which the rate of production of turbulence energy, $P_k = -\rho\overline{u_i u_j}\partial U_i/\partial x_j$. T_t is a realizable estimate of the turbulence timescale:

$$T_t = \frac{k}{\epsilon}\max\{1, \zeta^{-1}\}, \quad \zeta = \sqrt{R_t/2},$$

and the turbulence Reynolds number, $R_t = \rho k^2/(\mu\epsilon)$. The additional term, E , in the dissipation-rate equation is designed to improve the model response to adverse pressure-gradient flows. This term has the form:

$$E = A_E \rho\sqrt{\epsilon T_t}\Psi\max\{k^{\frac{1}{2}}, (\nu\epsilon)^{\frac{1}{4}}\}, \quad \Psi = \max\{\frac{\partial k}{\partial x_j}\frac{\partial \tau}{\partial x_j}, 0\}, \quad \tau = k/\epsilon.$$

The model constants are given by:

$$C_\mu = 0.09, \quad C_{\epsilon 1} = 1.44, \quad C_{\epsilon 2} = 1.92, \quad \sigma_k = 1.0, \quad \sigma_\epsilon = 1.3, \quad A_E = 0.3.$$

The eddy viscosity, μ_t , is obtained from:

$$\mu_t = \min\{C_\mu f_\mu \rho k^2/\epsilon, 2\rho k/3S\}$$

where $S = \sqrt{S_{kl}S_{kl}/2}$ is the dimensional strain magnitude and f_μ is a low-Reynolds number function, designed to account for viscous and inviscid damping of turbulent fluctuations in the proximity of solid surfaces:

$$f_\mu = \frac{1 - \exp^{-0.01R_t}}{1 - \exp^{-\sqrt{Rt}}}\max\{1, \left(\frac{2}{R_t}\right)^{\frac{1}{2}}\}$$

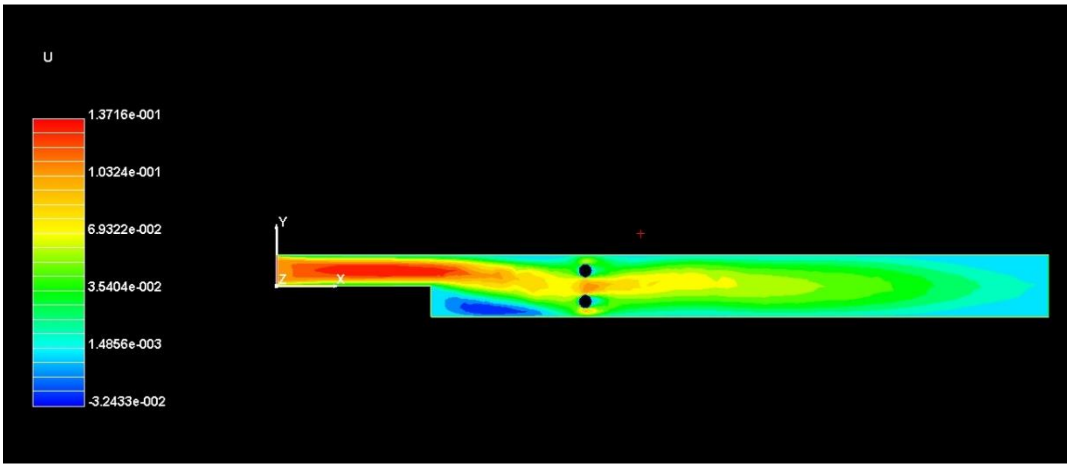
The CFD Model building Steps

- **The Geometry was Created in Patran**
- **The Geometry Is exported as parabolized**
- **The Geometry is Mished by Mime based on the selected global Mesh Size**

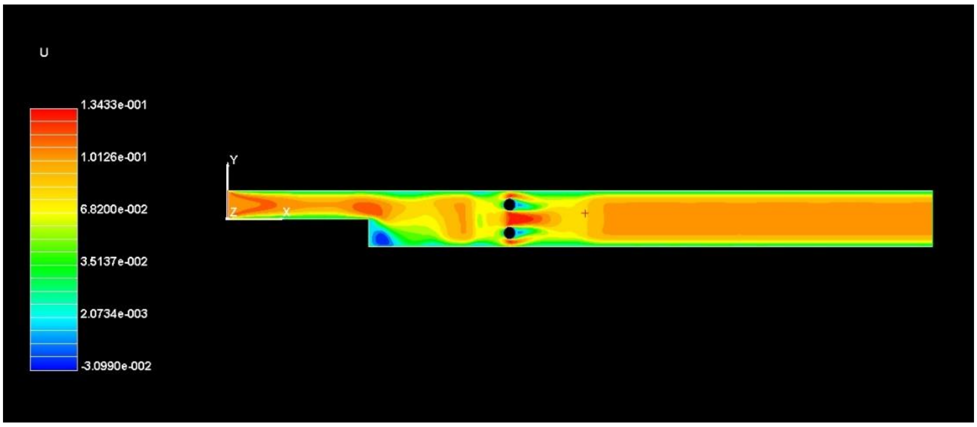
- **The Mished geometry is submitted to CFD++ Analysis Platform**
- **The Boundary Conditions are**
- **Inlet with variable velocity based on re Number and Temperature of 300 K**
- **Outlet back flow pressure of value zero**
- **Adiabatic Wall**
- **Heat Source with heat equal to 1.06 E6 X 2 X 0,01 X0,01 X 3.14 =665Watt**
- **The Equation used are Navier stokes PG for Compressible & incompressible**
- **The pressure, temperature and velocity were studied across the duct at different velocities**
- **The Turbulence model Kept constant as mentioned**
- **Dt or time steps are kept to 250 steps this had been tested and proved to be ok in correspondence to computation time**
- **The Physical properties feed as per given in the project**

Results

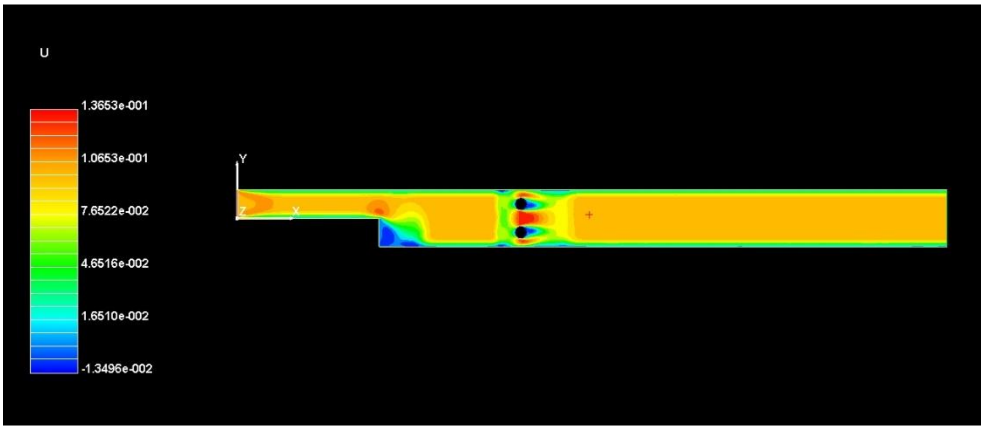
Mesh Analysis



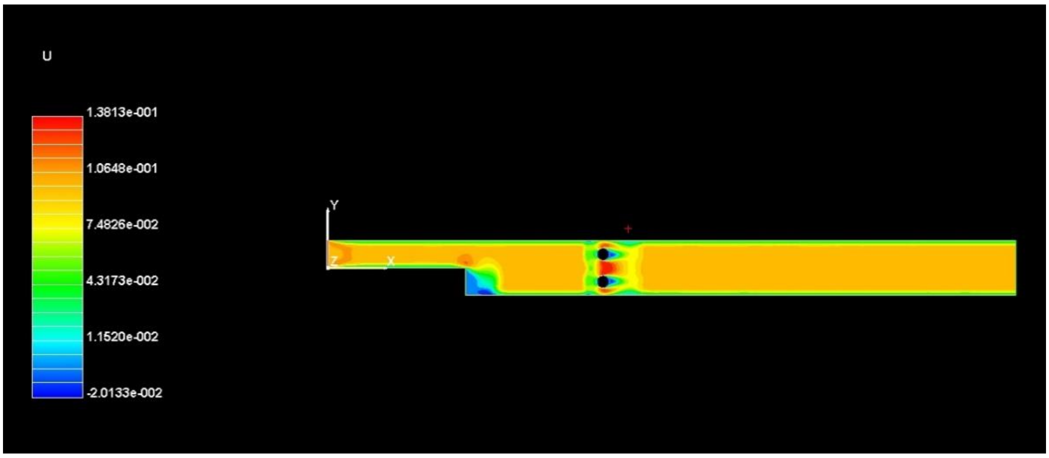
Global Mesh size = 5e-4 @ velocity = 0.1 m/sec water



Global Mesh size = 5e-5 @ velocity = 0.1 m/sec water



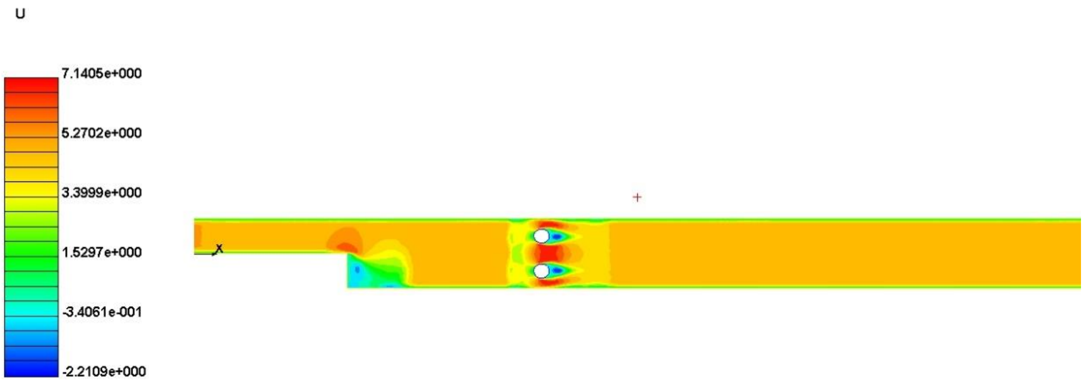
Global Mesh size = 3e-5 @ velocity = 0.1 m/sec water



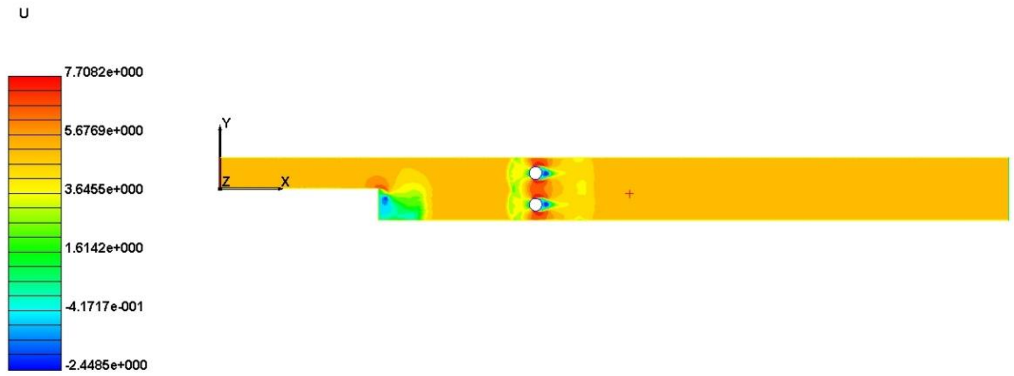
Global Mesh size = 2e-5 @ velocity = 0.1 m/sec water

<u>Based on the above we will fix our study Global Mesh size to 3e-5 , in order to save the computation time as stability in solution had been noticed between 2e-5 & 3e-5</u>

<u>Effect of density and viscosity on the area Change " Velocity parameter" with application on Air & water</u>



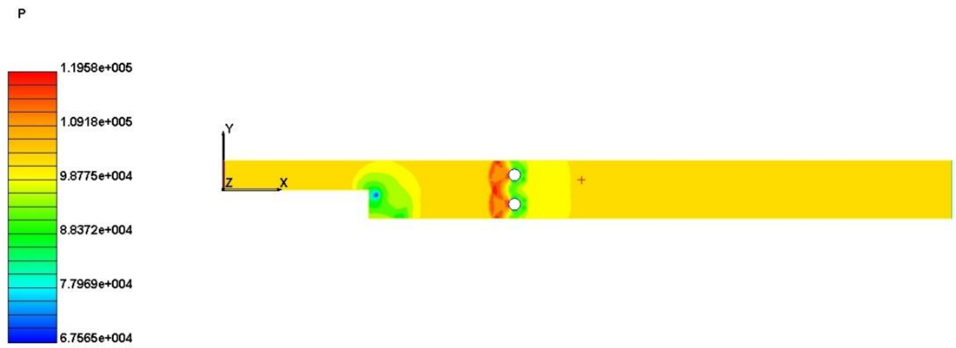
Air @ velocity =5m/sec at entrance



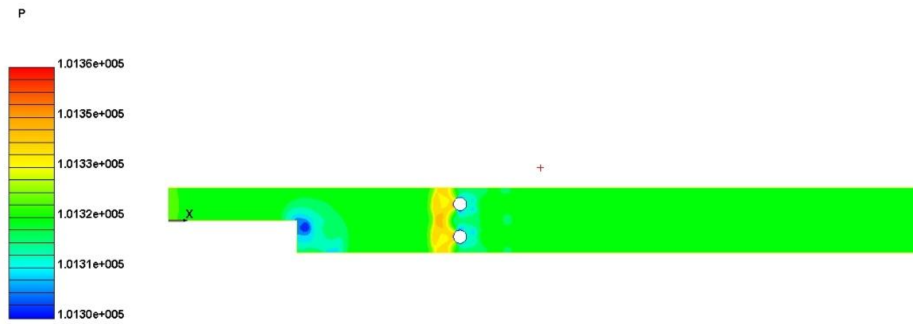
Water @ velocity =5m/sec at entrance

It is noticed clearly the drop in velocity due to the area increase and noticed that Air drops more and requires more distance to come back to normal velocity , the wake and velocity drop in back of cylinders is also clear .

Effect of density and viscosity on the area Change " Pressure parameter"

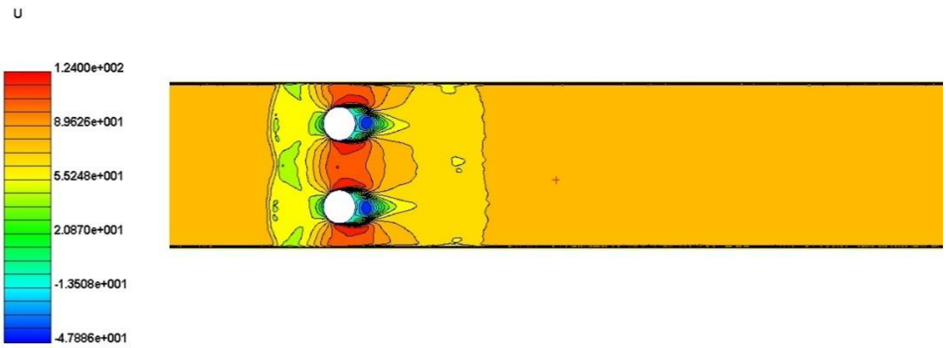


Water @ velocity =5m/sec at entrance



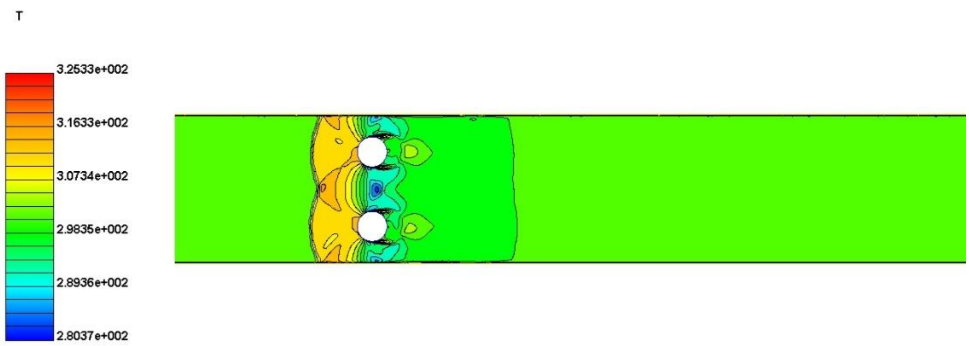
Air @ velocity =5m/sec at entrance

The effect of area change apparent also the stagnation in front of cylinders with increase in pressure is clear

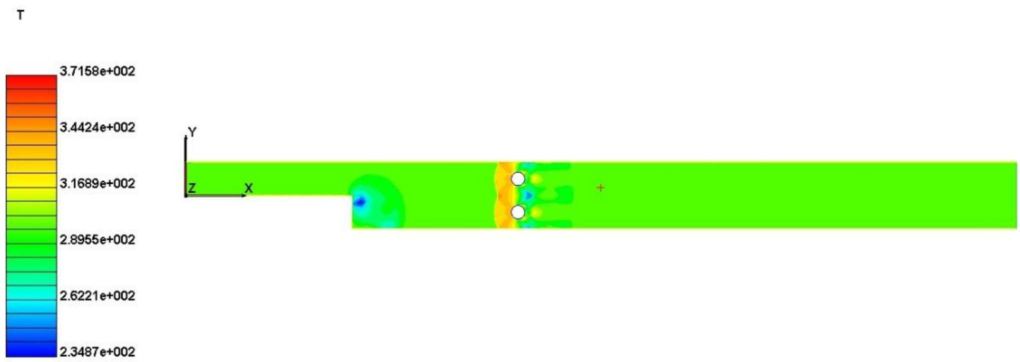


Velocity @ 80 m/sec with focus on circles and with contours

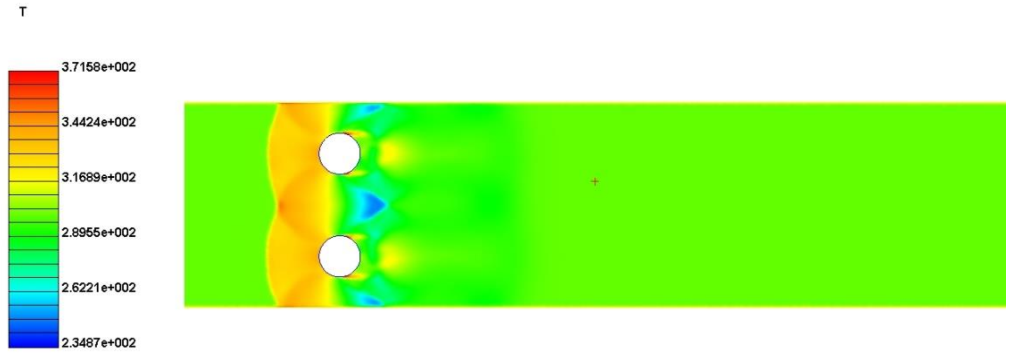
Temperature Air



Temp @ 150 m/sec with focus on cylinders with contours



Temp @ 250 m/sec



Temp @ 250 m/sec with focus on cylinders

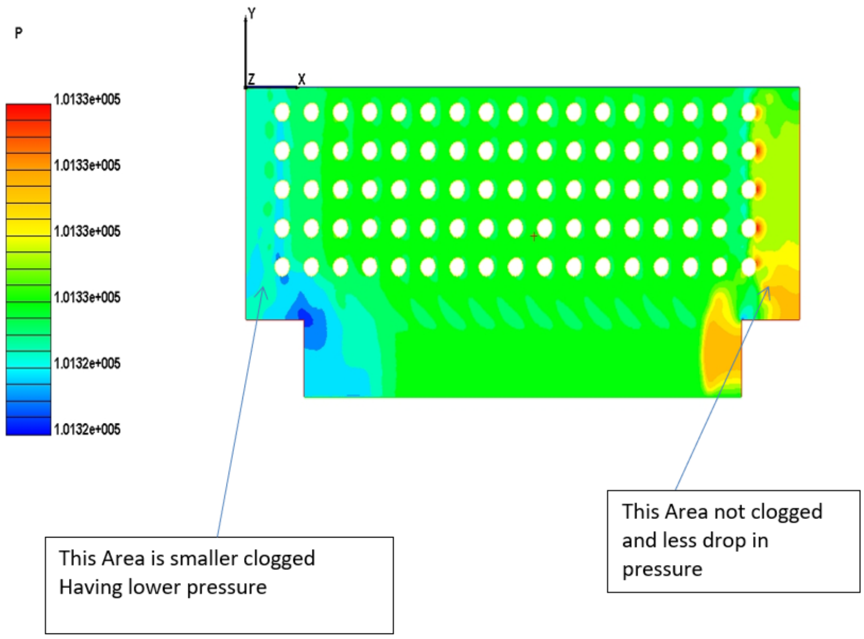
It is clear that the temperature increases in the stagnation period in front of cylinders , the fast flow is faster than the temperature propagation .

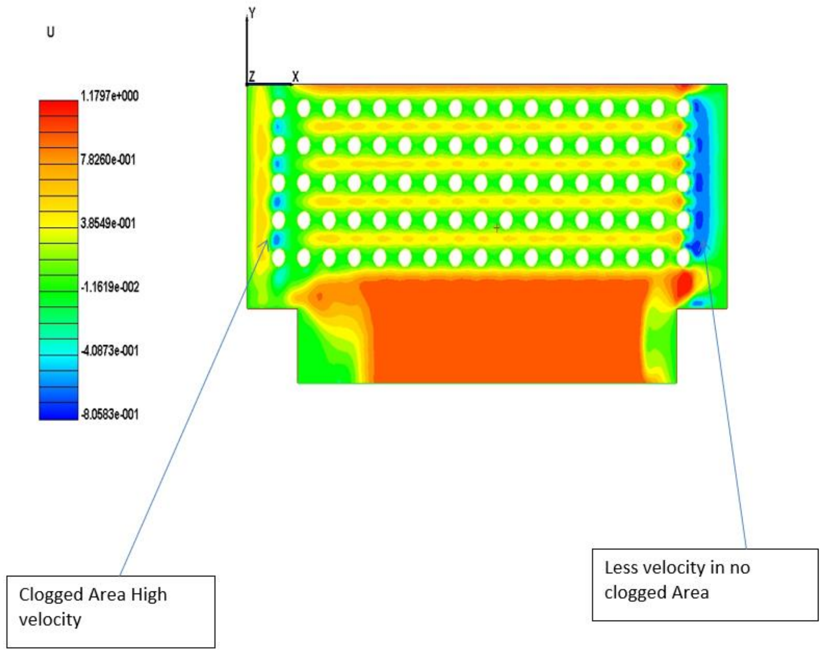
<u>Case study for Alpha laval cooler with blocked Area and effect on flow pattern</u>

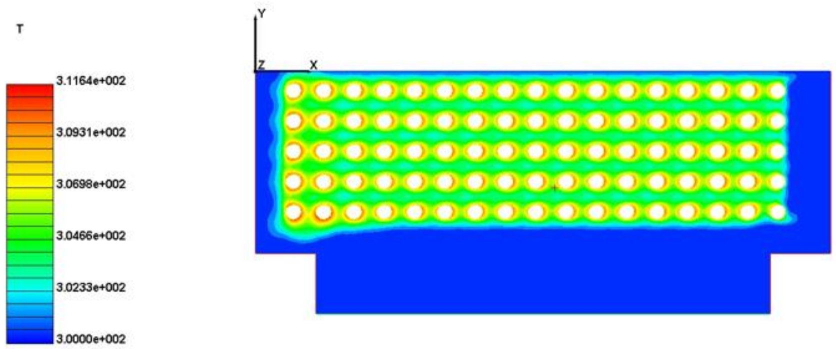
Data Velocity 1 m/sec

Q added 1.06e3 watt/sq m

Pressure







Temperature distribution across the cooler

Conclusion

A CFD Model is built to represent the effect of area change on pipes in stream line and it is included as above that the velocity and pressure had clear effects and the model had been extended to present a clogged Fin fan cooler and it showed excellent acceptable results.

References

1. Uriel C. Goldberg (2016): "A $\sqrt{k} - l$ Turbulence Model for Fluids Engineering Applications", Studies in Engineering and Technology Vol. 3, No. 1; August 2016 .
2. https://www.metacomptech.com/index.php/features/icfd CFD++
3. CFD++ Manual
4. Engineering Equation Solver
5. Uriel C. Goldberg, Paul Batten, Oshin Peroomian & Sukumar Chakravarthy (2015): "The R-γ transition prediction model", International Journal of Computational Fluid Dynamics .
6. Uriel C. Golberg, Paul Batten (2015): "A wall-distance-free version of the SST turbulence model", Engineering Applications of Computational Fluid Mechanics

7. P.Batten, U.Goldberg, E.Kang and S.Chakravarthy, "Smart Sub-Grid-Scale Models for LES and hybrid RANS/LES", AIAA-2011-3472, 2011.
8. U. Goldberg, S. Palaniswamy, P. Batten, V. Gupta, "Variable Turbulent Schmidt and Prandtl Number Modeling", Engineering Applications of Computational Fluid Mechanics, Vol. 4, No. 4, pp. 511-520, 2010.
9. P.Batten, U.Golberg, O.Peroomian and S.Chakravarthy, "Recommendations and best practice for the current state of the art in turbulence modelling", International Journal of Computational Fluid Dynamics, vol. 23, no. 4, pages 363-374, 2009.
10. U.Goldberg, O.Permoomian, P.Batten and S.Chakravarthy, "The k-epsilon-Rt Turbulence Closure", Engineering Applications of Computational Fluid Mechanics, vol. 3, no. 2, pages 175-183, 2009.
11. "An Introduction to Fluid Dynamics 20" by G. K. Batchelor
12. "Physical Fluid Dynamics 5" by D. J. Tritton
13. "Fundamentals of Aerodynamics 4" Book by John D. Anderson
14. "Modern Compressible Flow: With Historical Perspective 3" 2nd Edition Book by John D. Anderson
15. "Modern Compressible Flow: With Historical Perspective 1" by J. D. Anderson
16. "Elements of 3Gasdynamics 3" by H. A. Liepmann and A. Roshko. Liepmann and Roshko
17. "Boundary Layer Theory 6" by H. Schlichting
18. "Turbulent Flows 4" by Stephen B. Pope (got that as well, very good book!)
19. "Thermodynamics, 2Schaum's 2 Outline Series 2" by M. M. Abbott and H. A. van Ness