Indian Journal of Engineering, Vol. 13, No. 34, October-December, 2016 ANALYSIS

ISSN 2319–7757
EISSN 2319–7765

Indian Journal of Engineering

Cooling Time Estimation on Rotating high temperature Cylinder using CFD technique and verification through Analytical method

Prabhat Kumar Hensh[1☼], Sachin Kumar Shrivastava[2], Narayan Teja A[3], Dakshina Murty M[4]

1. Deputy Manager, Computational Fluid Dynamics Laboratory, Corporate Research and Development Division, Bharat Heavy Electricals Limited, Vikasnagar, Hyderabad 500093, India
2. Manager, Computational Fluid Dynamics Laboratory, Corporate Research and Development Division, Bharat Heavy Electricals Limited, Vikasnagar, Hyderabad 500093, India
3. Sr. Engineer, Heat Transfer and Fluid Flow Laboratory, Corporate Research and Development Division, Bharat Heavy Electricals Limited, Vikasnagar, Hyderabad 500093, India
4. Additional General Manager, Heat Transfer and Fluid Flow Laboratory, Corporate Research and Development Division, Bharat Heavy Electricals Limited, Vikasnagar, Hyderabad 500093, India

☼Corresponding author: Computational Fluid Dynamics Laboratory, Corporate Research and Development Division, Bharat Heavy Electricals Limited, Vikasnagar, Hyderabad 500093, India, e-mail: pkhensh@bhelrnd.co.in

Publication History
Received: 12 June 2016
Accepted: 14 July 2016
Published: October-December 2016

Citation
Prabhat Kumar Hensh, Sachin Kumar Shrivastava, Narayan Teja A, Dakshina Murty M. Cooling Time Estimation on Rotating high temperature Cylinder using CFD technique and verification through Analytical method. *Indian Journal of Engineering*, 2016, 13(34), 617-624

ABSTRACT

Conjugate Heat Transfer study is carried out on a large rotating high temperature cylinder to estimate cooling time using Computational Fluid Dynamics (CFD) technique. Objective of the study is to estimate lowest time required to cool the cylinder from high temperature (750°C) to desired surface temperature (100°C) maintaining specified temperature differential between mean to surface on the rotating cylinder. To achieve the objective, it is proposed that cooling nitrogen gas flows axially through annulus surrounding on the rotating cylinder to cool down to desired temperature. The conjugate heat transfer study is proposed with five different combination of rotational speed, cooling gas pressure and axial velocity of fluid to determine the interdependence between the heat transfer mechanism and the structure of the secondary flows. Particular interest is the accurate prediction of the heat transfer from the cylinder to the flow field using CFD technique. An empirical formula is proposed on the rotating system to estimate cooling time maintaining specified mean to surface cylinder temperature. Surface and mean temperatures estimated by CFD simulation study are compared with analytical method.

Keywords: Heat Transfer, CFD, Cooling Time.

Nomenclatures:

ρ	= Fluid density
$\delta\rho/\delta t$	= the rate of increase of density in the control volume
$\nabla.(\rho V)$	= the rate of mass flux passing out of control volume
πij	= Stress Tensor
h	= Heat Transfer Co-efficient (HTC)
A	=Cross Sectional Area of rotor
Tb	= gas bulk temperature
T	= Rotor surface temperature
t	= time

1. INTRODUCTION

Convective heat transfer from a spinning cylinder is important in many practical applications, the most common being the cooling of conventional rotating machinery, such as electrical motors and steam turbines. The complexity of the structure of the spinning fluid motion around the rotating cylinder has drawn the attention of several researchers.

In spite of its technological importance, little work has been directed towards the study of heat transfer within rotating large cylinder. Convective heat transfer phenomenon is investigated experimentally from a cylinder for axial turbulent flows by variation of Reynolds number (8.9x104 and 6.17x105) at different turbulence intensity (<0.1% to 6.7%) by Roland Wiberg, et al. [1]. Effect of Reynolds number and Prandtl number on heat transfer around a circular cylinder is studied by Lattice Boltzmann Model and experimentally verified by Qing Chen [2]. The experimental study on rotating cylinder is also carried out by Baris Ozerden [3], S. Seghir-Ouali[4], K.S. Ball [5], An Vasil'Ev [6], Adinarayana [7] where maximum Reynolds number is 105.

However, the work of the paper is focused to estimate lowest time required to cool the cylinder from high temperature to desired temperature maintaining specified temperature difference between mean to surface on the rotating cylinder where Reynolds number is more than 106. The CHT study is also indicated the requirement of flow parameters to meet the temperature different to cool the hot cylinder. Finally, the results are compared with analytical method.

2. WORK DESCRIPTION

A large cylinder of aspect ratio 1:1 is considered for the study. Nitrogen, an inert gas, is considered suitable for cooling of the cylinder as it has favorable thermodynamic properties over the operating range and also due to its availability and economics. Five cases, with combination of different cylinder rotation, flow velocity and fluid medium pressure, are carried out to establish cooling methodology for the large cylinder using CFD technique. The cooling behavior of the cylinder is evaluated through CFD study and verified through analytical approach. Importance of the study is that it is required to maintain differential of mean temperature and surface temperature of the solid cylinder throughout the cooling process as per ASME guideline [8].

3. CONJUGATE HEAT TRANSFER (CHT) STUDY

Transient Conjugate Heat Transfer (CHT) study is carried out using CFD techniques for estimation of cooling time for the high temperature rotating cylinder. Solid cylinder and fluid flow path is modeled and meshed for the CHT study. Continuity, Navier-Stokes and energy equation are solved in CFD code on the fluid flow path and only energy equation is solved on the solid geometry for CHT study.

Continuity Equation:
The continuity equation in partial differential equation form is given by,

$$\delta\rho/\delta t + \nabla.(\rho V)=0 \quad (1)$$

Momentum Equation:
Newton's Second Law applied to a fluid passing through an infinitesimal, fixed control volume yields the following momentum equation:

$$\delta/\delta t(\rho v)+\nabla.\rho V\, V=\rho f + \nabla.\pi_{ij} \quad (2)$$

The first term {δ/δt(ρv)} represents rate of increase of momentum per unit volume. The second term (∇.ρV V) represents the rate of momentum lost by convection through the control surface. Right hand side equation shows the first term (ρf,) as the body force per unit volume, while the second term (∇.πIj) represents the surface force per unit volume.

Energy Equation:
The first law of thermodynamics applied to a fluid passing through an infinitesimal, fixed control volume yields the following energy equation in terms of enthalpy.

$$\rho(De/Dt) = Dp/Dt + \delta Q/\delta t - \nabla.q + \phi \quad (3)$$

The term {ρ(De/Dt)} in the left hand side is the rate of change of energy inside the fluid element. Right hand side, first two terms (Dp/Dt +δQ/δt) are the rate of work done on element due to body and surface forces; third term (∇.q) is net flux of heat in to element. The last term (ϕ) on the right hand side is known as dissipation function and represents the rate at which mechanical energy is expended in the process of deformation of the fluid due to viscosity.

CFD model for conjugate heat transfer study is shown in figure.1. A segment of the rotor and the flow passage is considered for CHT study. Rotational speed is applied on the cylinder and the nitrogen is passed through the fluid flow passage at different operating conditions as mentioned in table 1.

In rotating systems, the existence of hydrodynamic instabilities may lead to a variety of secondary flows as the parameters describing the system are varied. Along with each transition in a flow, the transport mechanisms are altered, and usually result in markedly changed rates of heat and momentum transport. The objective of five different combination of rotational speed, cooling gas pressure and axial velocity of fluid is to determine the interdependence between the heat transfer mechanism and the structure of the secondary flows. Particular interest is the accurate prediction of the heat transfer from the cylinder to the flow field using CFD technique for estimation of cylinder cooling time.

Numerical simulation is carried out with initially the cylinder at its uniform hottest temperature. The transient numerical simulation is continued for that time duration up to when the core temperature of the cylinder is reached to the desired temperature limit. CHT study for all the cases is carried out using CFD techniques and the cylinder temperature variation with time as shown in Figure.2 to Figure. 6. Temperature and cooling time are shown in 0 to 1 scale as normalized value. The highest mean and surface temperature difference is observed for case 1, which is more than the desired value. Due to higher speed of the rotor, high turbulence effect is generated on fluid near the rotor surface and the surface is cooling relatively faster compared to core. For case 5, turbulence effect on the fluid due to rotation is less due to lower speed of the rotor. So, surface temperature is not falling drastically compared to core temperature of the cylinder. The temperature difference is very low compared to desire value as estimated through CFD study for case 5. The desired value of the temperature difference is achieved for Case 2 and Case 4. The temperature difference for case 3 is lower than desired value but higher than case 5.

Table 1 Parameters for the five case studies

Case	Rotor Speed	Fluid velocity	Operating Pressure
1	N	V	P
2	0.67N	0.1V	P
3	0.4N	0.52V	P
4	0.33N	0.052V	2P
5	0.167N	0.52V	P

Table 2 Comparison of CFD study results for temperature difference with desired value and cooling time with analytical method

Case	Temperature difference compared to desired value	Time required for cooling compared to analytical method
1	Higher	10% Low
2	Achieved	20% Low
3	Low	Matched
4	Achieved	20% Low
5	Lower	50% More

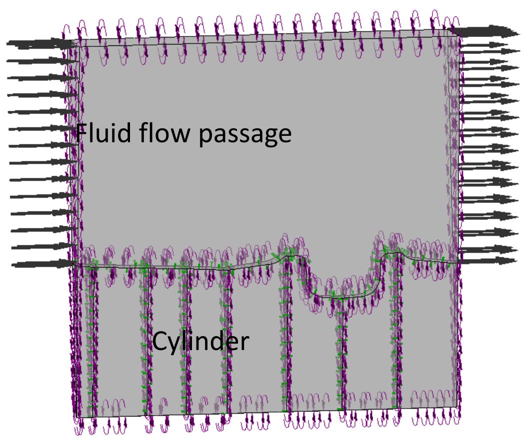

Figure 1 CFD Model for CHT Study

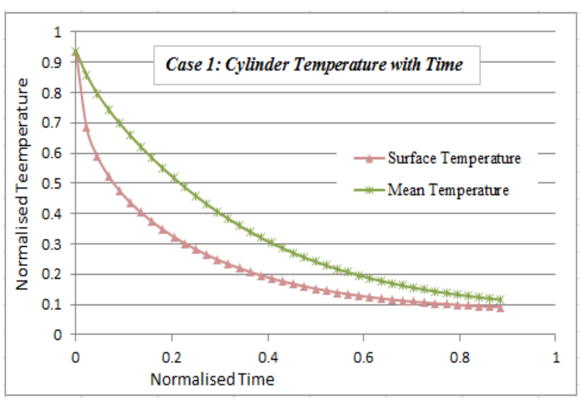

Figure 2 Cylinder temperature variations with time through CFD study for Case 1

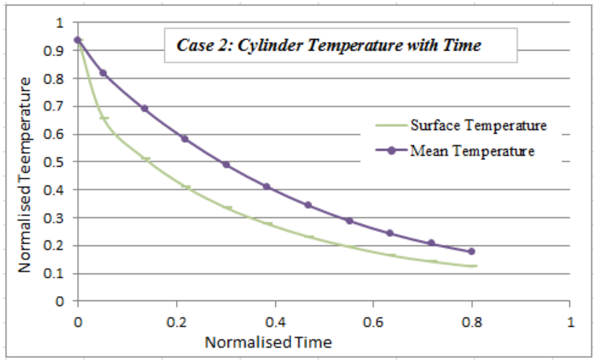

Figure 3 Cylinder temperature variations with time through CFD study for Case 2

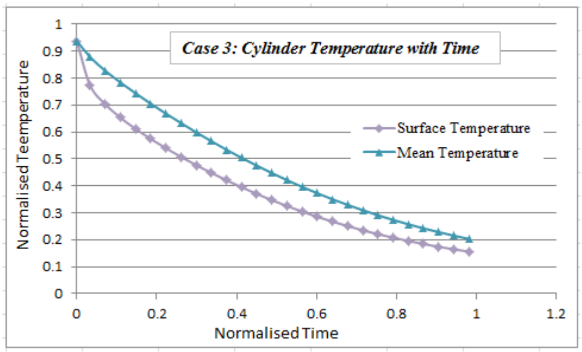

Figure 4 Cylinder temperature variations with time through CFD study for Case 3

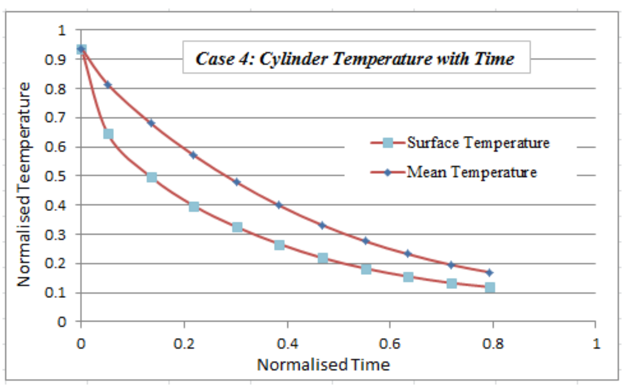

Figure 5 Cylinder temperature variations with time through CFD study for Case 4

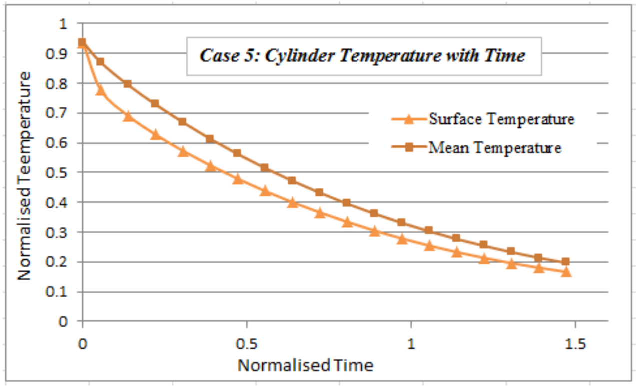

Figure 6 Cylinder temperature variations with time through CFD study for Case 5

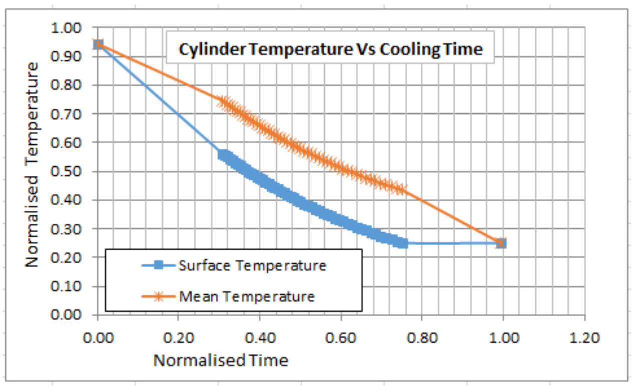

Figure 7 Cylinder temperature variations with time

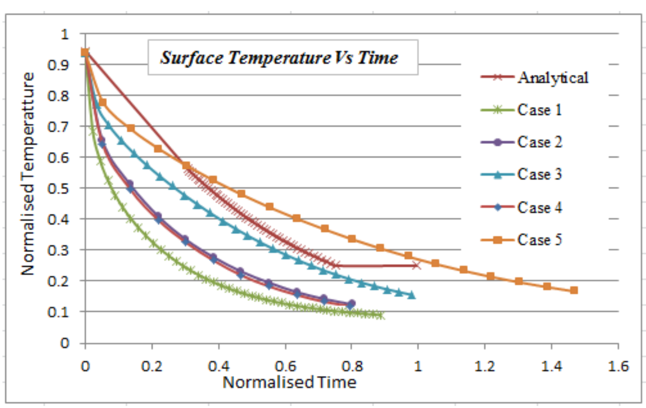

Figure 8 Comparison of Cylinder surface temperature variation with time

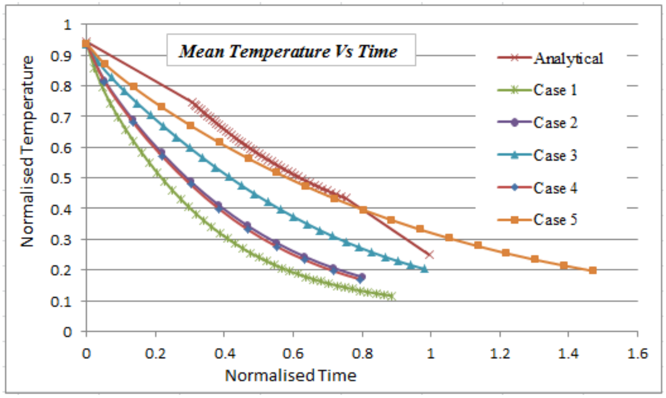

Figure 9 Comparison of Cylinder mean temperature variation with time

4. ANALYTICAL METHOD

Analytical formulation is adopted to study the cooling behavior on the large cylinder. The following equation is considered for computation of cooling time analytically.

$$hA(T - T_b) = \rho C_p(dT/dt) \qquad (4)$$

Fixed bulk gas temperature of nitrogen has been considered for the analytical study. Further HTC is considered in analytical study while maintaining required temperature difference between mean and surface temperatures of the cylinder. The HTC value is obtained through FE simulation for cooling. Using the above mentioned parameters in equation (4), cooling time of the cylinder is calculated to cool the cylinder to desired temperature. Figure 7 shows the cylinder temperature variation with respect to time.
Theoretical study over a rotating cylinder was carried out by K. Abdella [9], A.A. Kandoush [10]. All the research activity was limited to estimate heat transfer for low Reynolds number.

Surface and mean temperature on the cylinder for CHT study results for all the five cases is compared with analytical results. The comparison of the surface temperature and mean temperature between CFD study and analytical method is shown in Figure. 8 and Figure. 9 respectively.

Table 2 shows the temperature difference estimated through CFD study compared to desired value of the same and comparison of the cooling time between CFD study and analytical method.

5. CONCLUSION

Conjugate heat transfer study using CFD technique for five cases show the time required to cool the cylinder at desired temperature. Analytical formulation is also adopted for estimation of desired temperature difference between mean and surface temperature of the large cylinder and time required for cooling. Temperature difference and the cooling time estimated through transient conjugate heat transfer study using CFD techniques is verified with analytical solution.

Temperature difference estimated through CFD study is higher compared to desired value for case 1 due to higher rotational speed and lower for case 5 due to lower rotational speed. Maintained the desired temperature difference is essential for structural requirement. The conditions used for case 1 and case 5 are not suitable for high temperature cylinder cooling requirement.

Cooling time estimated through CFD study is in well agreement for case 3 with the analytical method, but the temperature difference for the case is lower than the desired which is more essential.

Desired temperature difference estimated for case 2 and case 4 using CFD techniques shows a good agreement with analytical study.

Time estimated for cooling using CFD technique is 20% lower compared with analytical study for case 2 and case 4.

Parameters selected for case 2 and case 4 are more suitable for cooling time estimation. The estimated cooling time achieved through the study will be further verified through experimental study.

ACKNOWLEDGEMENT

The authors wish to acknowledge the considerable contributions of the employees from Experimental Mechanics Lab and Design Analysis Group on the work making this paper possible.

REFERENCES

1. Roland Wiberg, Noam Lior 2005, Heat transfer from a cylinder in axial turbulent flows; International Journal of Heat and Mass Transfer 48 1505 – 1517.
2. Qing Chen, Xiaobing Zhang and Junfeng Zhang, Effects of Reynolds and Prandtl numbers on Heat transfer around a circular cylinder by simplified Thermal Lattice Boltzmann model; Commun. Comput. Phys., xx (201x), pp.1-23.
3. BarisOzerdem 2000, Measurement of convective heat transfer coefficient for a horizontal cylinder rotating in quiescent air, Int. Comm. Heat Mass Transfer, Vol 27, No. 3 pp 389 – 395.
4. S. Seghir-Ouali, D. Saury, S. Harmand, O. Philipart, D. Laloy 2006, Convective heat transfer inside a rotating cylinder with an axial air flow, Int. J. of Thermal Sciences 45 1166 – 1178.
5. K.S. Ball, B. Farouk, V.C. Dixit 1989, An experimental study of heat transfer in a vertical annulus with a rotating inner cylinder; Int. J. Heat Mass Transfer, Vol. 32, No. 8, pp. 1517 – 1527.
6. AN Vasil'Ev, VV Goludev 1981, Heat transfer from a rotating cylinder under forced convection, Journal of Engineering Physics and Heat transfer.
7. Adinarayana, V.M.K. Sastri 1996, Estimation of Convective HTC in Industrial steam turbines; Journal of Pressure Vessel Technology Vol. 118/247.
8. ASME, Section VIII, Division. II
9. K. Abdella, P.S. Nalitolela 2008, Approximate analytic solutions for forced convection heat transfer from a shear flow past a rotating cylinder; Applied Mathematical Sciences, Vol. 2, no. 11, 497 -527.
10. A. A. Kendoush 1996, An approximate solution of the convective heat transfer from an isothermal rotating cylinder, Int. J. Heat and Fluid Flow 17: 439 – 441.
11. Suhas V. Patankar 1798, Numerical Heat Transfer and Fluid Flow, Taylor & Francis Publication