ANALYSIS OF CSTR TECHNIUQUE FOR STUDY OF NATURAL CONVECTION OVER HORIZONTAL SMOOTH AND GROOVED SURFACES

Ritika Tripathi, Dr. Satyendra Singh, Shailesh Ranjan Kumar

Abstract This study investigates natural convection with smooth and grooved surface plates in various combinations. A total of five cells were used in the simulation. Both the global Nu and the Nu for each plate (or the associated boundary layer) are measured. The results reveal that the smooth plates are insensitive to the surface (grooved or smooth) and boundary conditions (i.e. nominally constant temperature or constant flux) of the other plate of the same cell. The heat transport properties of the rough plates, on the other hand, depend not only on the nature of the plate at the opposite side of the cell, but also on the boundary condition of that plate. It thus appears that, at the present level of MATLAB simulation resolution, the smooth plate can influence the rough plate, but cannot be influenced by either the rough or the smooth plates experimental and CSTR is an easily constructed, versatile and cheap reactor, which allows simple catalyst charging and replacement. Its well - mixed nature permits straightforward control over the temperature.

1 Introduction

Convection heat transfer occurs between a surface and a moving medium when they are in different temperature. Two mechanism involves in heat convection, diffusion in which energy transfer from hot surface to a adjacent fluid by random molecular motion and transport of energy by bulk movement of the fluid from higher temperature region to lower temperature region. Convection heat transfer classified into two categories:

Natural convection or free convection is an important branch of fluid mechanics and heat transfer, depending on how the fluid motion is initiated. Natural convection is induced by a density difference or density gradient in the flow field. Another condition that is necessary for natural convection to occur is body force that is proportional to density. Natural convection is a process, in which the fluid motion results from heat transfer. When a fluid is heated or cooled, its density changes and the buoyancy effects produce a natural circulation in the affected region, which causes itself the rise of warmer fluid and the fall of colder fluid: Therefore, energy transfer from hotter region to colder region and such process is repeated as long as the temperature difference in the fluid exists.

There are large number of studies for natural convection in closed cavities; known as Rayleigh-Benard convection. [1] Lyi and hasan (2014) Numerical study of airflow and heat transfer in an enclosure containing staggered arrangement of blockages. Temperature difference between two vertical walls is 42.2^{0}C, characteristic Rayleigh number of 1.45×109. Two types of blockage are considered for analysis, and these consist of In-line and staggered arrangement object. At the mid height and mid width steady flow and wall heat transfer data of the cavity are presented. They concluded from the numerical results, high temperature at the top and low temperature at the bottom region of enclosure. Due to pattern of arrangement 16% average heat transfer reduce in the enclosure.[2] Kumar and Subudhi (2015) This study investigates turbulent free convection over horizontal smooth or grooved surface in an open cavity top of the water surface exposed to ambient. Experiments for smooth surface was done values of AR,3,2,1 and 0.5 and for grooved surface values of AR are 3,2 and modified Rayleigh number in the range of $2.5\times10^8 \leq Ra \leq 3\times10^{12}$. They concluded from the results structure of plumes are more inclined with increasing heat fluxes and in case of grooved surface plumes are in disorder manner as compared to smooth surface. [3]Alzwayi and Paul (2014) Numerical simulations to study the transition of free convection flow between two isothermal vertical plates. They investigate channel width effect on the transition of the flow under various plate temperatures to obtain the numerical simulation of flow and thermal fields in the channel they use K-ε turbulence model with an enhanced wall function. Results indicated that if channel width increase the location of the velocity peaks moved very close to the heated plate. At the transition stage single boundary layer developed along the heated plate and flow in the adiabatic case takes early compared to isothermal case.[4] Pallares and Grau (2011) Numerical simulations of particle dispersion in the turbulent natural convection flow between two vertical walls. It is assumed that the particles do not affect the flow and stokes numbers in the range of $0.843 \leq St \leq 17.45$. They concluded that if stokes number is increased for this natural convection flow is comparable to the measured in an isothermal turbulent pipe or channel flow and near the wall the velocity fluctuations away from the wall are more probable than the velocity fluctuations towards the wall.[5] Abdellahoum et al. (2015) Numerical analysis of turbulent force convection of nano-fluids confined in a shallow cavity heated from all sides with a uniform temperature. Simulations are performed for a Reynolds number ranging from 4×10^4 -10^5 . They concluded from the results evolution of nusselt number of the flow system depending on the Reynolds number for different models of the conductivity.[6]Carvalho and Lemos (2014) Study on turbulent free convection in a porous square cavity using two temperature model with a Newtonian fluid. They employed two energy models which is one and two temperature models. According to the double-decomposition concept governing equations were time and volume average. Discretization of governing equations was obtained with the control-volume approach and system of algebraic equation was eased via the SIMPLE method. They concluded if the ratio of thermal conductivities k_s/k_f equals unity both result give similar results. [7] Ganesan et al. (2015) Compares five type of computational fluid dynamics (CFD) models, in which two single-phase models and three two-phase models, to investigate turbulent force convection of Cu-water nanofluid in a tube with a constant heat flux on the tube wall. The Reynolds number is taken between 10,000 and 25,000 and volume fraction of cu particles varied between the range 0% to 2%. In this CFD results compare with experimental results concluded mixture model gives a maximum error of 15% and Non-Newtonian single phase model in general, does not show good agreement with Xuan and Li's correlation.[8] Zhang and Hao (2015) Experimentally studied the turbulent thermal free convection over a horizontal heated plate in an open top cavity. Air used as a working fluid at atmospheric pressure and temperature conditions corresponds to Rayleigh number of order 10^8. Power spectra density results at the same locations indicate the presence of characteristic angular frequency, whereas secondary peaks observed at higher frequencies provide further support to the concept of an intermittent process.[9] Subudhi and Arakeri (2012) Discuss the flow visualization in turbulent free convection over horizontal smooth and grooved surfaces and the top of water surface exposed to ambient. The experiment is carried out with Rayleigh number in the range , 1.3×10^7- 4×10^7 . Analyze flow structure with the help of plumes dynamics and temperature variations with time. They concluded from the experiment for smooth surface, no separation of plumes and for grooved surface separation of plumes is observed. [10] Benim et al. (2011) Computational analysis of turbulent forced convection in a channel with a triangular prism investigated by a cascade of modelling strategies. Reynolds number in the range of 2500,5000 and 25,000. In all computations Prandtl number is equal to 0.7. For Re=2500 three dimensional URANS (3D URANS) and Re=5000 large eddy simulations (LES) are also performed. Results indicate for triangular prism much strong heat transfer and peak time averaged Nusselt number in downstream region of the prism by LES much higher than the URANS.[11]Cheng and Liu (2014) Numerical simulation investigate effect of cavity inclination on mixed convection heat transfer in lid-driven cavity flow is induced by a shear force in two-dimensional (2-D) cavity. Numerical results indicate in three aspect ratio investigate flow structures does not affect by the increase of inclination angle. For the case of A=0.2 and Ri=0.01 generate a primary clockwise recirculation vortex, if A=1 and Ri=0.01 flow field similar to A=0.2 and Ri=0.01 and entire cavity with two small vortices. For A=5 and Ri=0.01 in this unit of cavity height cavity inclination has no effect on the flow patterns, isotherms and local heat transfer. Nusselt number indicates that the increase of cavity inclination has no effect on the overall heat transfer process.

From the literature survey it is revealed that, there are large number of studies for natural convection in closed cavities; known as Rayleigh-Benard convection. Very few had investigated convection in open cavity, cavity with one side open, there is no clear consequence of CSTR functions for study of natural convection.

2 Results and Discussion

Continuous stirred tank reactors (CSTR) are common used because of their technological arrangement. Reaction inside flows continuously and we can control this reaction. Simulation usually consists of steady-state analysis which observe behavior of the system in steady-state and dynamic analysis which shows dynamic behavior after the step change of the input quantity.

Fig 1 shows concentration A of inlet feed stream define at initial point increase after that it get constant and define some decrement in concentration level after that it will be constant and at end point concentration level similar to initial point. For inlet feed stream temperature variation with respect to time shows increment and decrement in some points and at end point temperature of inlet feed stream temperature similar to initial point. For jacket coolant temperature variation with respect to time shows high variation in temperature.

Fig 2 shows time response comparison for reactor temperature and for a concentration level. In this smooth and grooved plate is used for same cell which is defining maximum value of stirred tank reactor (STR) -67.42. Time response comparison in A concentration at initial point concentration high but after that it get constant. Reactor temperature at initial point high as compare to end point.

Fig 3 shows concentration level increases with same time with minimum -102.7 stirred tank reactor (STR). Temperature level gets constant above of the theoretical line.

Fig 4 shows time minimization curve for smooth plate and grooved plate. Same temperature used in both case it shows that time response in case of smooth and grooved plate less as compare to grooved plate and water.

Fig 5 shows heat transport natural convection Rayleigh number with respect to cost at 0.5-1.5 get minimize, when 1.5 the Rayleigh number remain constant.

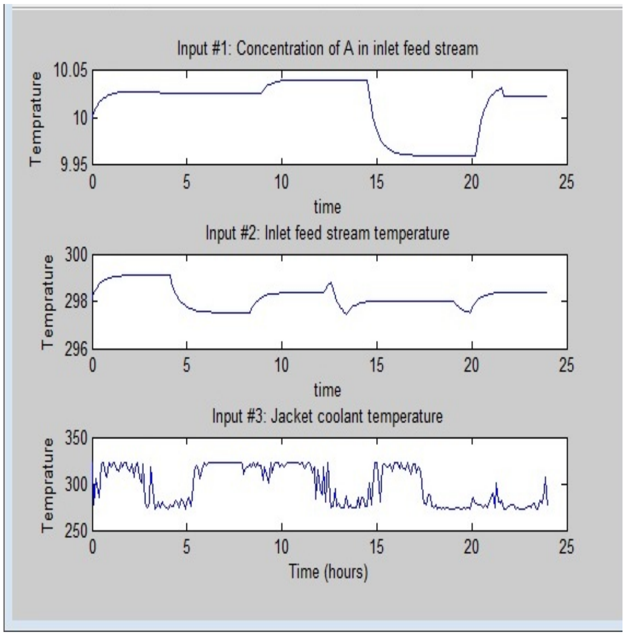

Fig.1 Temperature Variation with Time for inlet feed stream and jacket coolant temperature

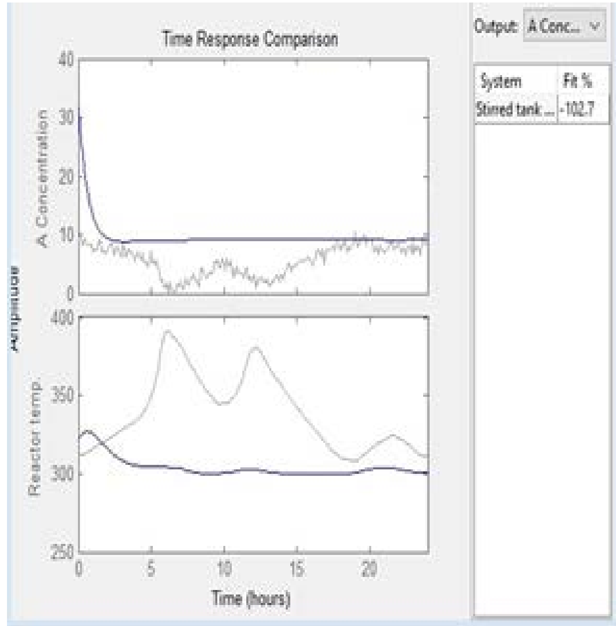

Fig.3 Time Response Comparison with STR - 102.7

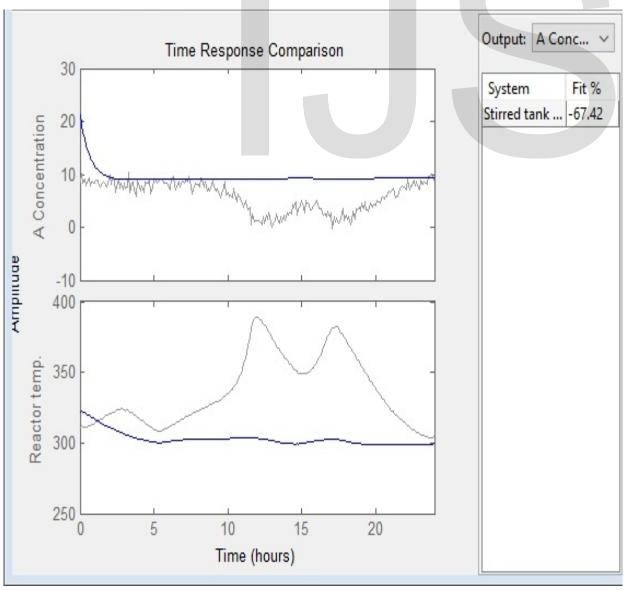

Fig.2 Time Response Comparison with STR - 67.42

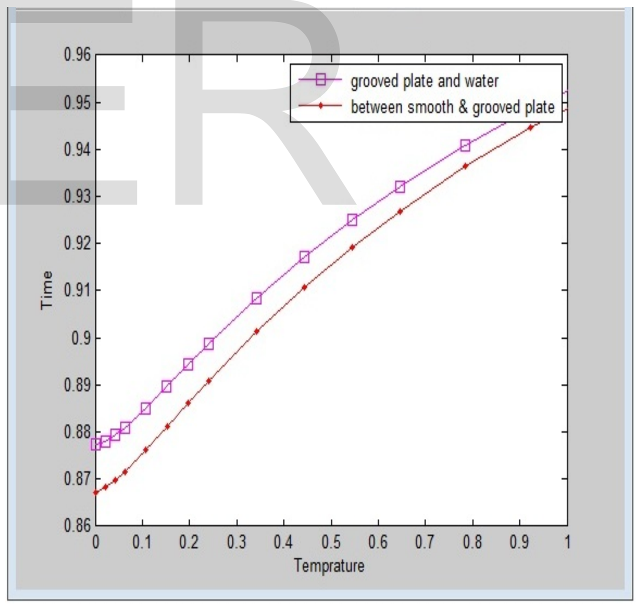

Fig.4 comparison of time with temperature for smooth grooved plate and grooved plate and water

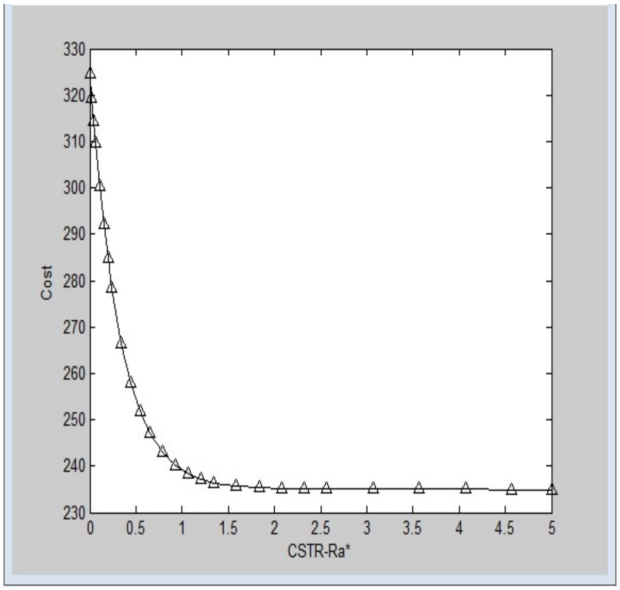

Fig .5 cost reduction with modified rayleigh number

3 Conclusion

System identification simulation design for many nonlinear systems is typically much more involved than for linear systems. This is also true for the CSTR, where on one hand it is desired that the controllable input 1, 2, 3, and 0.5 aspect ratio is such that it excites the system sufficiently.

The cavity of smooth or grooved surface heat flux varies from 150-1250 W/m^2 is observed by using Continuous stirred tank reactor with low variation of aspect ratio of longer horizontal shape due to the higher water layer height for the case of lower heat flux.

Higher water layer height for the case of lower heat flux for tank reactor process for smooth surface thermal boundary thickness is reduce with the heat flux for all aspect ratio, by using our proposed technique we minimizing overall aspect ratio at 1250 W/m^2 as compared to base method. And modified Rayleigh number slightly improvement on heat transport and its structure.

In the case of grooved surface experiments, this thickness is increased first, and then reduced for the case of aspect ratio 3, 2, 1. The velocity boundary layer near the rough surface is shifted upward by a distance d, because the horizontal shear flow is blocked partially by the rough elements on the surface, resulting in a stagnant layer of fluid deep inside the grooves.

Define inlet feed stream temperature and jacket coolant temperature with respect to time and time response comparison for concentration level and for reactor temperature and time response minimization for smooth and grooved plate and cost reduction for modified Rayleigh number.

References

1.Lyi, D., Hasan, D., (2014),” Natural convection flow and heat transfer in an enclosure containing staggered arrangement of blockages” ICTE 176-183

2.Kumar, G.K., Kumar, S.R., Subudhi, S., (2015)” Experimental study of the turbulent free convection over horizontal surface in an open cavity” Heat Mass Transfer 015-1559-6

3.Alzwayi, A.S., Paul, M.C., (2014)” Study transition of free convection flow between two isothermal vertical plates” International Journal of Heat and Mass transfer 307-316

4.Pallares, J., Grau, F.X., (2011)” Particle dispersion in the turbulent natural convection flow in two vertical walls” Journal of Aerosol Science 45-56

5. Abdellahoum, C., Mataoui, A., Oztop, H.F., (2015)” Effects of different models of thermal conductivity on turbulent nanofluid flow” Journal of Molecular Liquids 915-921

6. Carvalho, P.H.S., Lemos, M.J.S., (2014)” Turbulent free convection in a porous cavity using two-temperature model and high Reynolds closure” International Journal of Heat and Mass transfer 105-115

7.Ganesan, P., Sivasanskaran, S., Behroyan, I., (2015)” Turbulent force convection of cu-water nanofluid CFD model comparison” International Communications in Heat and Mass transfer 163-172

8. Zhang, X., Hao, P., He, F., Yu, J., (2015)” Statistical analysis of turbulent free convection over a small heat source in a large open cavity” Applied Thermal Engineering 446-455

ISSN 2229-5518

9. Subudhi, S. and Arakeri, J.H., (2012)" Flow visualization in turbulent free convection over horizontal smooth and grooved surfaces" International Communications in Heat and Mass

10. Benim, A.C., Nahavandi, A., Chattopadhyay, H., (2011)" Computational analysis of turbulent force convection in a channel with a triangular prism" International Journal of thermal sciences 1973-1983

[11] Cheng, T.S., Liu, W.H., (2014)," Effect of cavity inclination on mixed convection heat transfers in lid-driven cavity flows" Computer and Fluids 108-122