VOL. 11, NO. 18, SEPTEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

NUMERICAL SIMULATION ON RECTANGULAR CONVERGENT AND DIVERGENT RIBBED CHANNELS

K. Sivakumar[1], E. Natarajan[2] and N. Kulasekharan[3]
[1]Valliammai Engineering College, Chennai, India
[2]Institute of Energy Studies, Anna University, Chennai, India
[3]Chrysler India Automotive Pvt. Ltd., Chennai, India
E-Mail: tjksiva2k@yahoo.co.in

ABSTRACT
Numerical simulation on the rib turbulated flow inside a convergent and divergent rectangular channel with square ribs of different rib heights and at different Reynolds numbers (Re = 20,000, 40,000 and 60,000). The ribs were arranged in a staggered fashion between the upper and lower surfaces of the channel walls. Computational investigations are carried out using computational fluid dynamic software ANSYS Fluent 14.0. Suitable solver settings like turbulence models were identified from the literature and the boundary conditions for the simulations on a solution of independent grid. Computations were carried out for both convergent and divergent channels with 0 (smooth duct), 1.5, 3, 6, 9 and 12 mm rib heights, to identify the ribbed channel with optimal performance, assessed using a thermo hydraulic performance (THP) parameter. The convergent and divergent rectangular channels show higher Nu values than the standard correlation values.

Keywords: CFD, rib turbulator, friction factor, thermo hydraulic performance parameter, correlation coefficient.

INTRODUCTION
Gas turbines play a vital role in today's industrialized world, and as the demands for power increase, the thermal efficiency and output generating power of gas turbine must also increase. The basic method of increasing both the power output and thermal efficiency of the engine is to increase the temperature of the rotator inlet gas temperature of the turbine. Numerical solutions of the problem have been obtained by a finite-difference method under the assumptions that the flow is steady and laminar with constant properties was done in the three fields such as flow, temperature and computational field performed by Webb and Ramadhyani (1), in this research computational field SIMPLER algorithm was used. Becker & Becker (2) investigate a detailed two dimensional numerical study of the fluid flow and heat transfer near a single turbulator within a short straight section of a turbine blade internal cooling passage is described in terms of axial distributions of local skin friction coefficient, Stanton number and Nusselt number, as well as profiles of velocity and temperature and contours of stream function were generated. Luy *et al* (3) presented a numerical study performed on the laminar forced convection in parallel plate channels with in-line and staggered arrangement of transverse fins.

Naimi & Gessner (4) to studied predictive capabilities of three transport-type turbulence models were analyzed by comparing predictions with experimental data obtained for fully developed turbulent flow in a square duct and in a 2: 1 aspect ratio rectangular duct each with two opposite rib-roughened. Heat transfer in a rib-roughened duct was numerically simulated by using the second-order finite difference method in coordinates fitted to transverse or angled ribs. For further information refer Kim & Kim (5) and Promovonge *et al* (6) have done a numerical investigation of laminar periodic flow and heat transfer in a three-dimensional isothermal-wall square channel fitted with 45° inclined baffles on one channel wall and 30° angled baffles on two opposite channel walls for two different works carried out. For both the works the same finite volume method is introduced and the SIMPLE algorithm has been implemented for all computations Wang *et al* (7) work presented the configuration optimization of regularly spaced short-length twisted tape in a circular tube for turbulent heat transfer in air using computational fluid dynamics modeling. Tang and Zhu (8) studied turbulent fluid flow and heat transfer through seven two pass channels with and without guide ribs or vanes which have been investigated numerically using computational fluid dynamics technique. From the literature survey reported above, it was found that the research work reported on the ribbed channels with varying cross section is limited. Hence a research work is envisaged to conduct computational investigations on the rib turbulated flow inside converging and diverging rectangular channels with different rib heights (e = 0 (smooth channel), 1.5, 3, 6, 9 and 12 mm) and at three different flow Reynolds numbers (Re = 20,000, 40,000 and 60,000). The Reynolds number based on the mean hydraulic diameter (Dm) of the channel was kept in a range of 20,000 to 60,000. Based on the literature survey, as explained in the physical domain considered for the present computational study was the convergent / divergent rectangular channel test section with or without ribs. In order to compare the present computational results, the test section geometry section was conceived taking hint and the Reynolds numbers were representative of any typical gas turbine cooling channel, which was also used by Wang *et al*. (9). Atmospheric air was passed through the rib roughened channel at chosen Reynolds number with the top and bottom test section at constant heat flux 3,800 W/m2 K.

VOL. 11, NO. 18, SEPTEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

NUMERICAL INVESTIGATION

In the present article a computational investigation is conducted to analyze the three-dimensional incompressible Navier-Stroke flows through the rectangular convergent and divergent square ribbed channel. The commercial finite-volume based CFD code ANSYS Fluent 14 has been used to simulate fluid dynamics and heat transfer. Computational domain and grid generation, governing equations, Turbulence Model selection, boundary conditions and solution method convergence criteria is presented in detail in the following sub-sections.

COMPUTATIONAL DOMAIN AND GRID GENERATION

In the computational, three-dimensional turbulent flow through inside the rectangular convergent channel having different square sectioned transverse rib roughness on the top and bottom surface of the channel is simulated in staggered arrangement. Hence, in the present analysis, 3-dimensional computational domain of convergent and divergent rectangular channel with different ribs has been adopted which is similar to the computational domain of Yadav & Bhagoria (10). The computational domain is a simple convergent rectangular channel of length 500 mm, 100 mm width and height at the convergent entrance section 80 mm and exit section 74 mm. In a divergent section same cross sectional parameters are used and the flow of air is passed in reverse direction. The total length of the test section is divided in to three regions, 1. Entrance section (L1=100 mm), test section (L2 = 300 mm) and exit section (L3 = 100 mm). The thickness and material of the total test section has 5 mm and copper material of thermal conductivity 386W/m2K. The test section with inlet and outlet straight portions will form the boundaries for the computational analysis. The wall thickness for the test section and the solid volume of the ribs are considered for the present computational analysis, which was carried out as a conjugate heat transfer analysis, where the conduction within the solid domain and the convection of heat from the heated plate to the air flowing inside will be carried out. Figure-1 shows the view of 9 mm rib height channel with transparent wall surfaces. In the present computational, 3-dimensional square sectioned transverse ribs have been considered as roughness element.

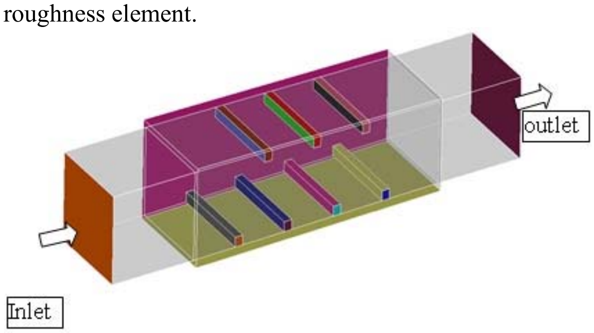


Figure-1. Computational domain with extended inlet and outlet channel.

The geometry parameters of the test section are mentioned as shown in Table-1.

Table-1. Dimension for test section geometry (convergent and divergent channel).

Section	W	e	H	W/H	e/D_m	P/e	Re.
Inlet	100	0	80	1.25	----	----	20000 40000 60000
	100	1.5	80	1.25	0.0185	40	
	100	3	80	1.25	0.0345	20	
	100	6	80	1.25	0.0694	10	
	100	9	80	1.25	0.1040	6.6	
	100	12	80	1.25	0.1387	5	
Outlet	100	For all ribs	74	1.35	Same as above	Same as above	Same as above

The square sectioned transverse ribs are considered on the underside of the top inner surface and topside of the bottom inner surface. The square sectioned ribs are varied from 1.5, 3, 6, 9 and 12 mm and also smooth surface (without ribs) have to use. The mesh for the computational domain is made with tetrahedral cells, as the geometry is complex.

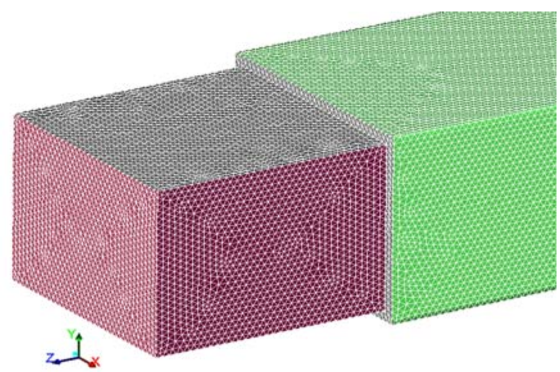
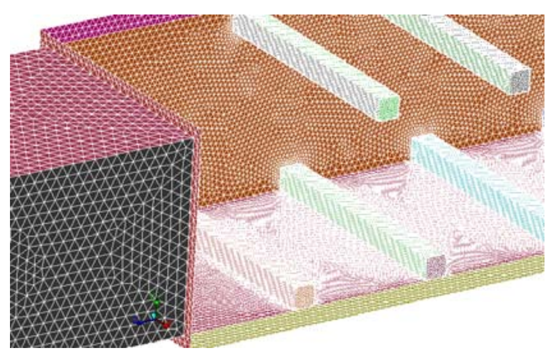
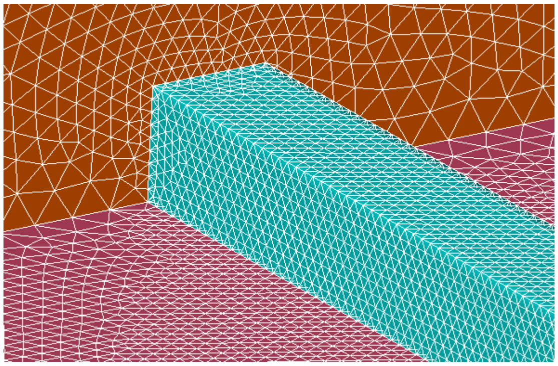


Figure-2. Meshed computational domain for the smooth channel.



(a) domain inner with ribs



(b) Mesh near the ribs with finer cell size

Figure-3. Meshed computational domains with 9 mm ribs.

The numerical grid contains a finer near wall and rib surfaces, with the first node placed at 0.1 mm and with a cell size growth factor of 1.1, normal to the wall. After a careful grid independency study, the optimum grid chosen for the present computational fluid dynamics study for the smooth channel is shown in Figure-2 which has an overall mesh size of 6,91,303 cells (0.69 million cells). The ribbed channel with the 9 mm rib size shown below has a mesh size of 11,03,754 cells (1.1 million cells).

Figure-3(a) domain inner with ribs and (b) Mesh near the ribs with finer cell size. A coarse mesh was chosen for the K-ε turbulence model variants to satisfy the y+ requirements as indicated in the solver manual Ansys Fluent 14.0. The pressure based solver, chosen by with implicit and absolute velocity formulation. SIMPLE algorithm is used for coupling the pressure and velocity terms. The second order upwind differencing scheme is used for convective terms and for the turbulence quantities in order to enhance the numerical accuracy of the computational results.

GOVERNING EQUATIONS

The governing equations used in the present computations are the continuity, momentum, energy and turbulence equations. These equations were solved with the help of the computational fluid dynamics software Ansys Fluent 14.0, which is based on the finite volume approach. The computations are treated as steady, incompressible, three dimensional with a single phase approach for the air flow inside the channels. The governing continuity, momentum and energy equations adopted for the present computational fluid dynamics equations as follows:

$$\frac{\partial}{\partial x_i}\left(\rho u_i\right) = 0 \tag{1}$$

Momentum:

$$\frac{\partial}{\partial xi}\left(\rho uiuj\right) = -\frac{\partial p}{\partial xi} + \frac{\partial}{\partial xj}\left[\mu\left(\frac{\partial ui}{\partial xj} + \frac{\partial uj}{\partial xi}\right)\right] - \frac{2}{3}\mu\frac{\partial uk}{\partial xk}\delta ij \tag{2}$$

Energy:

$$\frac{\partial}{\partial xi}\left(\rho ujCpT - \mathrm{k}\frac{\partial T}{\partial xj}\right) = \mathrm{uj}\frac{\partial p}{\partial xj} + \left[\mu\left(\frac{\partial ui}{\partial xj} + \frac{\partial uj}{\partial xi}\right) - \frac{2}{3}\mu\frac{\partial uk}{\partial xk}\delta ij\right] \tag{3}$$

TURBULENCE MODEL SELECTION

The Reynolds averaging of the Navier – Stokes equations (momentum equations) is done by modelling the applicable Reynolds stresses, using an appropriate turbulence model. Present computations are carried out using Realizable *k* and turbulent model available within the solver. The governing equations for the turbulent kinetic energy (*k*) and turbulent dissipation rate (ϵ) as given below:

Turbulent kinetic energy (k):

$$\frac{\partial}{\partial xi}\left(\rho k ui\right) = \frac{\partial}{\partial xj}\left[\left(\mu + \frac{ut}{\sigma k}\right)\frac{\partial k}{\partial xj}\right] + \mathrm{Gk} - \rho\varepsilon \tag{4}$$

Turbulent dissipation rate (ϵ):

$$\frac{\partial}{\partial x_i}\left(\rho\varepsilon u_i\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{ut}{\sigma k}\right)\frac{\partial\varepsilon}{\partial x_j}\right] + \mathrm{C}_{1\varepsilon}\frac{\varepsilon}{k}\mathrm{G}_k - \mathrm{C}_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{5}$$

Where G_k is the rate of generation of the turbulent kinetic energy, while ρ is the destruction rate and C_1 , C_2 is expressed by the following equation:

$$G_k = \left(-\rho\overline{u_i'u_j'}\right)\frac{\partial uj}{\partial xi} \tag{6}$$

The Boussinesq hypothesis is normally used in the eddy viscosity models in flows with turbulence, to relate the Reynolds stresses $-\rho\overline{u_i'u_j'}$ to the mean velocity gradients by the equation as given below:

$$-\overline{\rho u_i'u_j'} = \mu\left(\frac{\partial ui}{\partial xj} + \frac{\partial uj}{\partial xi}\right) \tag{7}$$

The turbulent viscosity term μ_t is to be computed from the turbulence model selected using the following equation:

$$\mu_t = \rho C\mu\frac{k^2}{\varepsilon} \tag{8}$$

The other relevant non-dimensional parameters of interest in the present CFD investigation are the Reynolds number, Nusselt number, Friction factor and thermo-hydraulic performance parameter.

Reynolds number is defined as

$$Re_m = \frac{U_m D_m}{\nu} \tag{9}$$

The local and channel average Nusselt numbers were defined as

$$Nu_z = \frac{h_z D_m}{k} \tag{10}$$

$$Nu = \frac{(Q - Q_{loss})\, D_m}{Ak(T_w - T_m)} \tag{11}$$

The friction factor is obtained for a convergent channel, as shown in Figure-4**,** as detailed below. Let u, v, w be the velocity components in the x, y, z directions, U_{in}, U_{out} the average axial velocity at inlet and outlet cross sections, and A_{in}, A_{out}, the cross section along the flow direction is not significant, it is assumed that the velocities 'u' and 'v' are very small compared with the mean velocity U_m and the static pressure is uniform at each cross-section.

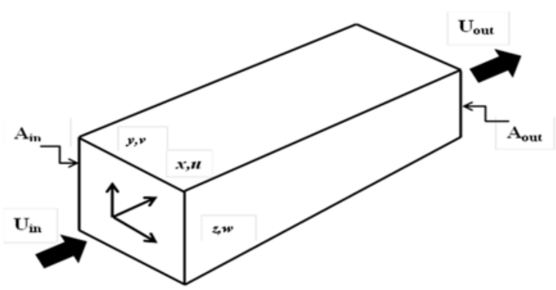


Figure-4. Diagram for derivation of effective pressure drop.

The effective pressure drop is expressed as follows:

$$\Delta p_e = p_{in} - p_{out} + \frac{1}{2}\rho U_{in}^2 - \frac{1}{2}\rho U_{out}^2 \tag{12}$$

The friction factor of the convergent / divergent channel is calculated by effective pressure drop, across the length of test section and can be obtained by

$$f = \frac{U_{in}^2}{U_m^2}\frac{D_m}{L}\lambda\left[1 - \left[\frac{A_{in}}{A_{out}}\right]^2\right] \tag{13}$$

It has been found that the artificial roughness in the form of square sectioned rib on the convergent test section results in substantial enhancement of heat transfer also increase in friction factor. Therefore to choose the roughness geometry such that the heat transfers is maximized while keeping the pumping losses at the least possible value. To analyze overall performance of a convergent rectangular ribbed channel, thermo-hydraulic performance should be evaluated by simultaneously consideration of thermal as well as hydraulic performance. Webb and Eckert (12) suggested a thermo-hydraulic performance parameter. A thermo hydraulic performance parameter is introduced to study the overall benefit of a particular channel by estimating a ratio of the incremental benefit of enhanced heat transfer to the corresponding increase in pressure drop with the ribbed channels, when compared to the smooth channels. The thermo-hydraulic performance parameter is used to estimate how effectively a rib roughened surface enhances the heat transfer under constant pumping power constraints. A value of thermo-hydraulic performance parameter greater than one ensures the effectiveness of using an enhancement device and can be used to compare the performance of number of sized ribs and arrangement of the ribs to decide the best among these. The value of $\overline{Nu}$ and f from the smooth channel at Re = 20,000 is considered as the baseline reference values. Hence the THP (η) parameter showed a value of unity for this case.

Thermo-hydraulic performance parameter:

$$\frac{(\overline{Nu} / \overline{Nu_o})}{(f / f_o)^{1/3}} \tag{14}$$

BOUNDARY CONDITION

The boundary conditions for the computational analysis are velocity inlet for the channel extended inlet whose value is deduced from the corresponding Reynolds number value and the fluid properties of air. The air inlet temperature for the particular channel configuration is taken from the ambient conditions. The channel outer wall is specified with a constant heat flux value is equal to 3800 W/m^2. This heat flux value is calculated by geometry of the test section as per Wang *et al* (9). The outlet of the flow domain is considered as exiting to the atmosphere and hence provided with a constant static pressure value of '0'.

SOLUTION METHOD

Three dimensional model of the flow domain used for numerical analysis is built and grid generated using ANSA 13, then exported to ANSYS TRID 14 for generating volume meshed model. Finally volume mesh is exported to ANSYS FLUENTv.14 for analysis. The continuity equation, energy equation and the Navier-Stroke equations in their steady, incompressible form, along with the associated boundary conditions are solved using the multipurpose finite volume based CFD software package, ANSYS FLUENT v14. In the present numerical study RNG K-ε turbulence model with 'enhanced wall treatment' is used. In the discretization of governing equations, SIMPLE (semi-implicit method for pressure linked equations) algorithm is used in pressure-velocity coupling and second order upwind discretization scheme is for all the transport equations as suggested by Kumar and Saini (11).

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

CONVERGENCE CRITERIA

The convergence criteria for the computational solution are determined based on scaled residuals for the equations of continuity, momentum, energy and turbulence quantities specific to the respective models. A convergence criteria value of 1 x 10^{-10} was set for all the flow properties in the solution such as continuity, x, y, z velocities. For the turbulent properties turbulent kinetic energy and dissipation rate, the convergence criteria were set as 1 x 10^{-4}. For energy equation, the convergence criterion was set as 1 x 10^{-10}. All the cases were made to run till the convergence criterion were met. For cases where the convergence oscillates and cases where convergence was not achieved, the iterations are continued for 1000 more iterations and stopped to achieve iterative convergence.

RESULTS AND DISCUSSIONS

The results extracted from the computational convergent and divergent channels with different rib heights and at different Reynolds numbers are discussed within these two categories. Flow and heat transfer properties extracted from the computations. A thermo hydraulic performance parameter is introduced to study the overall benefit of a particular channel by estimating a ratio of the incremental benefit of enhanced heat transfer to the corresponding increase in pressure drop with the ribbed channels, when compared to the smooth channels.

VARIATION OF THERMO-HYDRAULIC PERFORMANCE PARAMETER (η) FOR CONVERGENT CHANNEL – CFD

Table-2, shows the value of thermo-hydraulic performance parameter (η) with *Re* for computational fluid dynamics analysis in a convergent rectangular channel with different rib heights in the non-dimensional number and Figure-5, shows the corresponding tabulated values graphically. The values of *Nu* and *f* from the smooth channel at *Re* = 20,000 is considered as the baseline reference values. Hence the THP (η) parameter showed a value of unity for this case. THP values for the smooth converged channel increases with increase in *Re* and this indicates that this channel performs better at higher *Re.* All ribbed channels also showed an increased values of THP with *Re.* However, compared to the smooth channel, the channel with 3 mm rib height (*e*) showed THP values greater than the other ribbed channels.

Table-2. Thermo hydraulic performance parameter values for the convergent channel – CFD analysis.

Rib height ➡ / Re ⬇	0 mm (Smooth)	1.5 mm	3 mm	6 mm	9 mm	12 mm
20000	1.00	1.82	1.91	1.81	1.57	1.42
40000	1.86	2.90	3.10	2.76	2.41	2.19
60000	2.39	3.24	3.58	3.34	2.94	2.65

This indicates that the converged channels with rib heights larger than 3 mm do not show an overall good performance when compared to the smooth channel. Even though there may be a heat transfer enhancement with these channels (e > 3mm), this positive is nullified by the increased pressure drop across those channels. It must be noted that the converged channel with rib height of 3 mm show THP parameter values very close to that of the smooth channel, particularly at higher Re. The highest value of THP parameter for the ribbed channel with e = 3 mm is 3.58 which could be interpreted that this ribbed channel at Re = 60,000 is 358% better than the smooth channel at Re = 20,000.

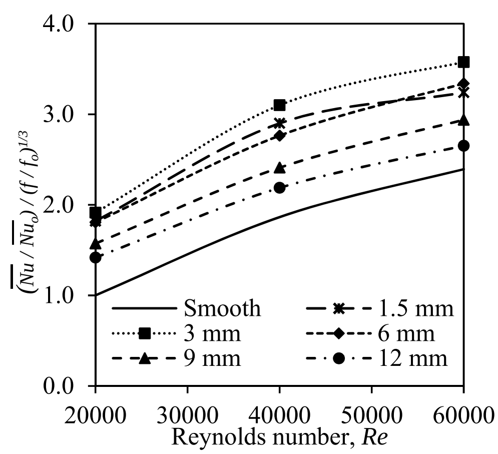


Figure-5. Variation of thermo-hydraulic performance parameter with Re for convergent channel.

VARIATION OF THERMO - HYDRAULIC PERFORMANCE PARAMETER (H) FOR DIVERGENT CHANNEL – CFD ANALYSIS

Table-3, shows the value of thermo-hydraulic performance parameter (η) with Re for computational fluid dynamics analysis in a divergent rectangular channel with different rib heights in the non-dimensional number and Figure 6 shows the corresponding tabulated values graphically. The THP parameter showed a value of unity for the smooth channel at Re = 20,000, whose values are considered as the baseline reference. THP values for the smooth converged channel increases with increase in Re

VOL. 11, NO. 18, SEPTEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

and this indicates that this channel performs better at higher Re.

Table-3. Thermo- hydraulic performance parameter values for the divergent channel.

Rib height ➡ Re ⬇	0 mm (smooth)	1.5 mm	3 mm	6 mm	9 mm	12 mm
20000	1.00	1.60	1.85	1.79	1.60	1.43
40000	2.05	2.55	2.85	2.70	2.43	2.30
60000	2.60	3.33	3.60	3.40	3.00	2.80

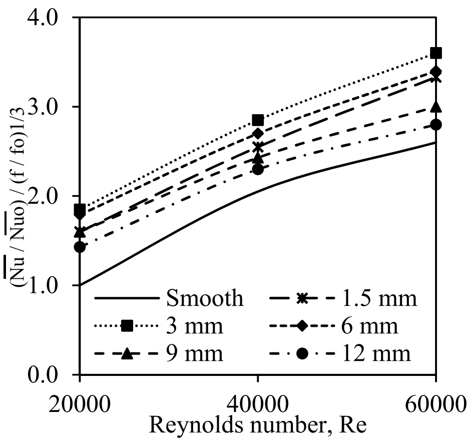


Figure-6. Variation of thermo-hydraulic performance parameter with Re for divergent channel.

Computational results show that the 3 mm rib is outperforming all the other ribbed channels with highest values of THP. The highest value of THP parameter for the ribbed channel with e = 3 mm is 3.60 which could be interpreted that this ribbed channel at Re = 60,000 is 360 % better than the smooth channel at Re = 20,000.

COMPUTATIONALRESULTS COMPARISON WITH – CORRELATION OBTAINED FROM LITERATURE

The values of Nusselt number (Nu) obtained from the computaional convergent and divergent channel were compared with the standard correlation. Dittus-Boelter correlation (Smooth duct and unifiorm cross section-Heating) for estimating the Nusselt number for the smooth ducts of uniform cross section from the inlet to the outlet is considered.

$$Nu_d = 0.023\ \mathrm{Re}^{0.8}\ \mathrm{Pr}^{0.4} \tag{15}$$

The surface averaged Nusselt numbers estimated from the top and bottom wall are compared with the correlation values for the corresponding Re. The comparison is plotted in Figure-7 which shows the standard correlation values form a straight line with a constant slope. The convergent and divergent rectangular channels shows higher average (Nu) values than the standard correlation values. This indicates that when there is an area change from higher to lower (convergent Channel), due to heating of air, air molecules were expaned but there is no space for expantion of air. In case of (Divergent Channel) area changes from lower to higher, for heated air, air molecules were expaned over the ribbs. The rib surfaces having more surface area and more amount of heat transfer can takes place. Among the presently tested channels, the divergent channels showed higher average Nu values at all Re than the corresponding convergent channels.

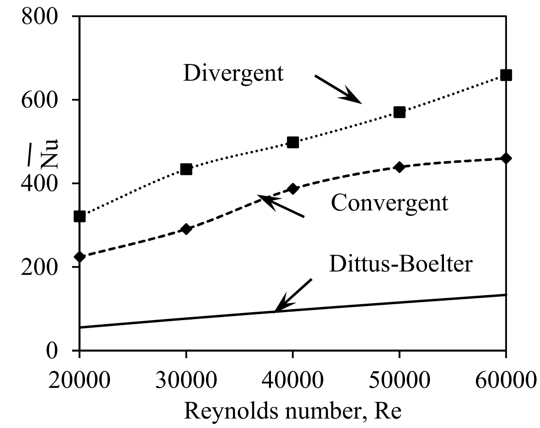


Figure-7. Comparison of average Nu of the smooth channels.

REGRESSION ANALYSIS OF THERMO-HYDRAULIC PERFORMANCE PARAMETER (η)

Correlation coefficients were estimated using regression analysis for the computational results obtained from our research work. Regression analysis generates an equation to describe variable in the statistical relationship between one or more predictors and the response variable and predict new observations. Regression generally uses the ordinary least squares method which derives the equation by minimising the sum of the squared residuals. Regression results indicate the direction, size and statistical significance of the relationship between a predictor and response. Sign of each coefficient indicates the direction of the relationship. Coefficients represent the mean change in the response for one unit of change in the predictor while holding other predictors in the model constant.

The correlations were developed for the thermo hydraulic performance parameter (□). The details of the regression analysis for each of the sets of results from the computations for the converging and diverging channels are discussed below. P-value for each coefficient tests the null hypothesis that the coefficient is equal to zero (no effect). Therefore, low P- values suggest the predictor is a meaningful addition to the model. The equation predicts new observations given specified predictor values.

Computational analysis - convergent channel

The correlation coefficients from the regression analysis for convergent channel with computational fluid dynamics analysis as in Table-4, indicates that both the predictors Reynolds number and rib height are significant because of their low p-values. The R- Square value for this correlation was found to be 68.41%.

Table-4. Correlation coefficients from the regression analysis- Convergent channel.

Predictor	*Coefficient*	*Standard Error*	*T - Stat*	*P-value*	*Lower 95%*	*Upper 95%*
Constant	0.9922	0.3009	3.2973	0.0049	0.3508	1.6335
Reynolds Number *(Re)*	$3.94 x 10^{-5}$	$6 x 10^{-6}$	5.6893	$4 x 10^{-5}$	$2 x 10^{-5}$	$5 x 10^{-5}$
Rib Height *(e)*	-0.008	0.0244	-0.338	0.7403	-0.06	0.0437

The correlation equation for the thermo-hydraulic performance parameter (η) in terms of the Reynolds number and Rib height of convergent channel with computational fluid dynamics equation as given below:

$$\mathbf{THP\ (\eta) = 0.99215 + 3.94 \times 10^{-5}\,(Re) - 0.00823 \times (e)} \quad (16)$$

Computational analysis - divergent channel

The correlation coefficients from the regression analysis for divergent channel with computational analysis as in Table-5, indicates that both the predictors Reynolds number and rib height are significant because of their low p-values. The R- Square value for this correlation was 81.39%. The correlation equation for the thermo-hydraulic performance parameter (η) in terms of the Reynolds number and Rib height of divergent channel obtained from CFD results equation is given below:

Table-5. Correlation coefficients from the regression analysis- divergent channel.

Predictor	*Coefficients*	*Standard Error*	*T-Stat*	*P-value*	*Lower 95%*	*Upper 95%*
constant	0.806	0.2324	3.469	$34 x 10^{-4}$	0.3108	1.3013
Reynolds No.*(Re)*	$3.94 x 10^{-5}$	$5 x 10^{-6}$	8.098	$7 x 10^{-7}$	$3 x 10^{-5}$	$5 x 10^{-5}$
Rib Height*(e)*	$-9.4 x 10^{-5}$	0.0188	$-5 x 10^{-3}$	0.996	-0.04	0.04

$$\mathbf{THP\ (\eta) = 0.86047 + 3.94 \times 10^{-5}\,(Re) - 9.4 \times 10^{-5}\,(e)} \quad (17)$$

CONCLUSIONS

Research work was carried out on rectangular converging and diverging channels roughened with rib turbulators. Computations were conducted for different ribbed channel configurations and the optimal rib was found the present computational methodology.

The convergent and divergent rectangular channels show higher Nu values than the standard correlation values. This indicates that when there is an area change in the channel, the heat transfer increases than the channel of uniform cross section. Among the presently tested channels, the convergent channels showed higher Nu values at all Re than the corresponding divergent channels.

From the results, however the THP parameter which includes the effect of friction factor indicates that the divergent channels are more efficient than the convergent channels in exchanging heat from the hot wall to the fluid, thus cooling the duct.

Correlations for the thermo hydraulic performance parameter (*Re)* are developed for the computational and experimental investigations as a function of (Re) and rib height (*e).*

The equivalent roughness ratio as defined from Moody's chart for my rib heights various from (0-Smooth, 1.5, 3, 6, 9, and 12) 0.02 to 0.05.

REFERENCES

[1] Webb, B.W., and Ramadhyani, S. 1985. Conjugate heat transfer in a channel with staggered ribs. International Journal Heat Mass Transfer. 28:1679-1687.

[2] Becker, B.R., and Becker, C.S. 1990. The flow field and heat transfer near a turbulator. International Communication Heat Mass Transfer. 17: 455-464.

[3] Luy, C.D., Chaeng, C.H., and Huang, W.H. 1991. Forced convection in parallel-plate channels with a series of fins mounted on the wall. Applied Energy. 39:127-144.

[4] Naimi, M., and Gessner, F.B. 1997. Calculation of fully developed turbulent flow in rectangular ducts with two opposite roughened walls. International Journal Heat and Fluid Flow. 18:471-481.

[5] Kim, K.Y., and Kim, S.S. 2002. Shape optimization of rib-roughened surface to enhance turbulent heat transfer. International Journal of Heat Mass Transfer. 45:2719-2727.

[6] Promvonge, P., Sripattanapipat, S., Tamna, S., Kwankaomeng, S., and Thianpong, C. 2010. Numerical investigation of laminar heat transfer in a square channel with 45° inclined baffles. International Communication in Heat and Mass Transfer. 37:170-177.

[7] Wang, Y., Hou, M., Deng, X., Li, L., Huang , C., Guang, H., Zhang, G., Chen, C., and Huang, W. 2011. Configuration optimization of regularly spaced short-length twisted tape in a circular tube to enhance turbulent heat transfer using CFD modelling. Applied Thermal Engineering. 31:1141-1149.

[8] Tang, X.Y., and Zhu D.S. 2013. Flow structure and heat transfer in a narrow rectangular channel with

ARPN Journal of Engineering and Applied Sciences

different discrete rib arrays. Chemical Engineering and Processing. 69:1-14.

[9] Wang, LB, Wang, QW, He, YL & Tao, WQ 2002, Experimental and numerical study of developing turbulent flow and heat transfer in convergent / divergent square ducts. Heat and Mass Transfer. 38:399-408.

[10] Yadav, A.S., and Bhagoria, J.L. 2014. A numerical investigation of square sectioned transverse rib roughened solar air heater. International Journal of Thermal Sciences. 70:111-131.

[11] Kumar, S., Saini,R.P. 2009. CFD based performance analysis of a solar air heater duct provided with artificial roughness. Renewable Energy. 34:1285–1291.

[12] Webb, R. L., Eckert, E. R. G. 1972. Application of rough surface to heat exchanger design. International Journal Heat Mass Transfer. 15(9):1647–1658.