

## NUMERICAL SIMULATION OF FORCED-CONVECTION HEAT TRANSFER IN A RIBBED CHANNEL WITH HIGH BLOCKAGE

L.Marocco, P.Gramazio, D.Fustinoni, A.Niro

Dipartimento di Energia, Politecnico di Milano  
Campus Bovisa, via Lambruschini, 4 – 20156 Milano, Italy

alfonso.niro@polimi.it

### ABSTRACT

Turbulence promoters in form of ribs are commonly used in various equipment such as turbine cooling channels, heat exchangers, nuclear reactors and solar air heaters. This artificial roughness surfaces modify the fluid dynamics by various mechanisms, such as periodic interruption of the boundary layer growth or periodic streamline deflection and in addition they promote turbulence development as their characteristic size is close to the turbulent microscales (the lower the Reynolds number, the larger the size of dissipative structures).

The present study is focused on the turbulent forced convection in a channel with continuous transverse rib on both walls. The rib to channel height ratio here investigated result in a high blockage ratio. A constant temperature is applied on the lower and upper wall whereas the ribs are adiabatic. The calculations are carried out using a RANS approach. The ribs have a square cross section and are transverse to the main direction of the flow. Different pitch-to-height ratios ( $p/e=10, 20$  or  $40$ ) and rib dimensions ( $e=2$  mm or  $4$  mm) are considered. Three Reynolds numbers, based on bulk velocity and duct height, has been chosen (1771, 2942 and 4119) in order to have a turbulent flow regime typically encountered in the internal cooling channels of the turbine blades.

### INTRODUCTION

Ribs of various cross section are commonly used to increase the heat transfer rate in ducts, e.g., compact heat exchangers and ducts used in the internal cooling of gas turbine blades. The ribs disturb the boundary layers, thereby promoting turbulence and enhancing heat transfer. Their presence leads to a complex velocity field with regions of flow separation upstream and downstream of the ribs, and it contributes to the level of mixing of the cooler part of the air stream with the warmer air close to the walls.

The authors are currently involved in an investigation on fluid dynamics and heat transfer characteristics of forced convection inside periodic ribbed channels of various configurations [1], [2]. In previous numerical works [1], [3], it was found that the  $k-\epsilon$  model gave better predictions than  $k-\omega$  or RSM of forced convective turbulent flow in a two-dimensional ribbed channel.

Scope of the present work is the simulation of forced convection inside a ribbed channel with high blockage ribs installed on both walls for three different pitch-to-height ratios (10, 20 and 40), two ribs dimensions (2 and 4 mm) and three Reynolds numbers (1771, 2942 and 4119).

### PROBLEM CONFIGURATION

The investigated geometry is a slender (Depth/Height = 10) rectangular section duct, partially blocked by successive square ribs. A previous study showed that a periodic condition in the streamwise direction is achieved [1]. This allows confining the numerical simulation to a module consisting of one single rib on both walls and periodic boundary conditions in the streamwise direction:

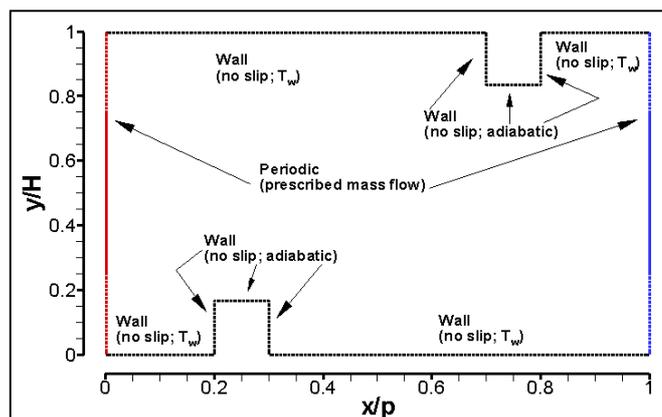
$$\begin{aligned} u(x) &= u(x + p) \\ \theta(x) &= \theta(x + p) \end{aligned} \quad (1)$$

where

$$\theta(x) = \frac{T(x, y) - T_w}{T_{b,in} - T_w} \quad (2)$$

The computational domain together with the boundary conditions is illustrated in Figure 1 and is summarized in Table 1 together with the thermophysical properties of the flowing air.

**Table 1:** Channel and ribs configuration according to Figure 1; Thermophysical properties of fluid and flow conditions.



**Figure 1:** Computational domain of the ribbed channel

<b>Geometry</b>	
Channel height, H	12 mm
Square Rib size, e	2 mm = 0.167H 4 mm = 0.333 H
Pitch-to-height ratio, p/e	
	10, 20, 40
<b>Thermophysical properties</b>	
Fluid	Air
Ambient temperature, $T_0$	22.73°C
Ambient pressure, $P_0$	101325 Pa
Density @ $T_0$ , $\rho$	1.193 kg/m <sup>3</sup>
Dynamic viscosity, $\mu$	1.85 10 <sup>-3</sup> Pa s
Specific heat @ $T_0$ , $c_p$	1005 J/kgK
Thermal conductivity, $k_f$	0.0262 W/mK
<b>Flow conditions</b>	
Reynolds number, $\rho u_b H / \mu$	1771, 2942, 4119

The numerical simulations have been performed according to the following assumptions:

- two-dimensional fluid flow and heat transfer
- incompressible Newtonian fluid
- constant thermophysical properties
- negligible body forces and viscous dissipation
- negligible radiation heat transfer

The last assumption is justified by the small bulk temperature increase across the module.

Note that the Reynolds number in [2], defined with the same bulk velocity but with the hydraulic diameter of the experimental channel, corresponds to  $D_h/h \cong 1.81$  times the Reynolds number of the present work.

## PROBLEM SETUP

### Boundary conditions

The upper and lower walls are maintained at a constant temperature  $T_w=40^\circ\text{C}$  while the wooden ribs are considered adiabatic. No-slip boundary conditions are enforced at all walls and rib sides. Streamwise periodic conditions for velocity and temperature are imposed.

### Turbulence models

The heat transfer enhancement and thermal performance in a channel with ribs in different geometries and arrangements has been already numerically studied, but different authors do not agree about the best turbulence model to be used. According to a previous study by these authors [1], the  $k-\varepsilon$  models provided better results with respect to the  $k-\omega$  models. Even though the  $k-\varepsilon$  Realizable and  $k-\varepsilon$  RNG models gave quite similar results, the  $k-\varepsilon$  model with the RNG formulation has been chosen for this study because of its known improved accuracy for strain flows and for low Reynolds numbers.

### Numerical simulation method

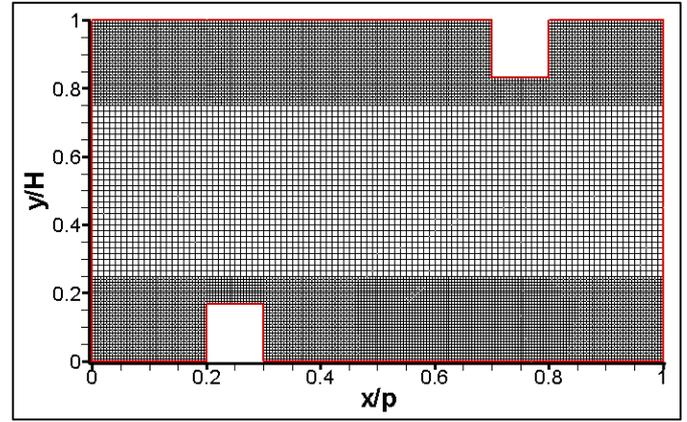
The steady state time-averaged continuity, momentum, energy and turbulence equations are solved numerically, sequentially (Pressure based solver), by a finite volume method using the commercial CFD code ANSYS FLUENT v.13. The convective terms are discretized with a second order upwind scheme. The SIMPLE algorithm handles the coupling between pressure and velocity. Underrelaxation is applied to all equations.

A convergent solution is considered to be reached when all the following conditions are satisfied:

- constant average drag coefficient on the channel walls
- constant average heat transfer coefficient
- unscaled residual values lower than  $10^{-7}$  for all variables

**Table 2:** Grip properties  $N_{xy}/1000$

s	Re = 1171			Re = 2942			Re = 4119		
	p/e 10	p/e 20	p/e 40	p/e 10	p/e 20	p/e 40	p/e 10	p/e 20	p/e 40
2	14.2	18.7	37.9	14.2	29.2	59.2	25.3	51.9	105.3
4	15.3	18.9	35.5	38.8	80.8	164.8	73.8	74.7	100.4



**Figure 2:** Rectangular mesh for  $p/e=10$ ,  $e=2$  and  $Re=4119$

As shown in Figure 2, a structured rectangular mesh has been used. The cell size is such to obtain a value of  $y^+ \approx 1$  in order to resolve the viscous sublayer. Table 2 summarizes the total number of cells ( $N_{xy}$ ) for the different configurations.

## FLOW STRUCTURE

Figure 3 shows the contour plots of the  $x$ -velocity component, together with the streamlines, for  $Re=4119$ ,  $e=2\text{mm}$  and pitch-to-height ratios of 10, 20 and 40. Upstream of the rib, a small clockwise rotating vortex is clearly visible. In the low pressure region behind the ribs, a large clockwise rotating recirculating vortex as well as a smaller counter-clockwise rotating vortex is correctly predicted. Anyway, the reattachment point varies for the different configurations and Reynolds numbers, as summarized in Table 3. The value of the  $x$ -velocity gradient with respect to the normal wall coordinate gives the position of the reattachment point of the flow. Indeed, at the separation point ( $\partial u/\partial y$ ) becomes zero and then decreases until it reaches a minimum. After this point it starts to increase, becoming again zero where the flow reattaches.

The recirculating region behind the rib decreases with increasing  $p/e$  and the flow profile tends to become symmetric in the inter-rib region.

**Table 3:** Reattachment point  $x^*/H$  of the flow after the rib.  $x^*$  is evaluated from the downstream edge of the rib

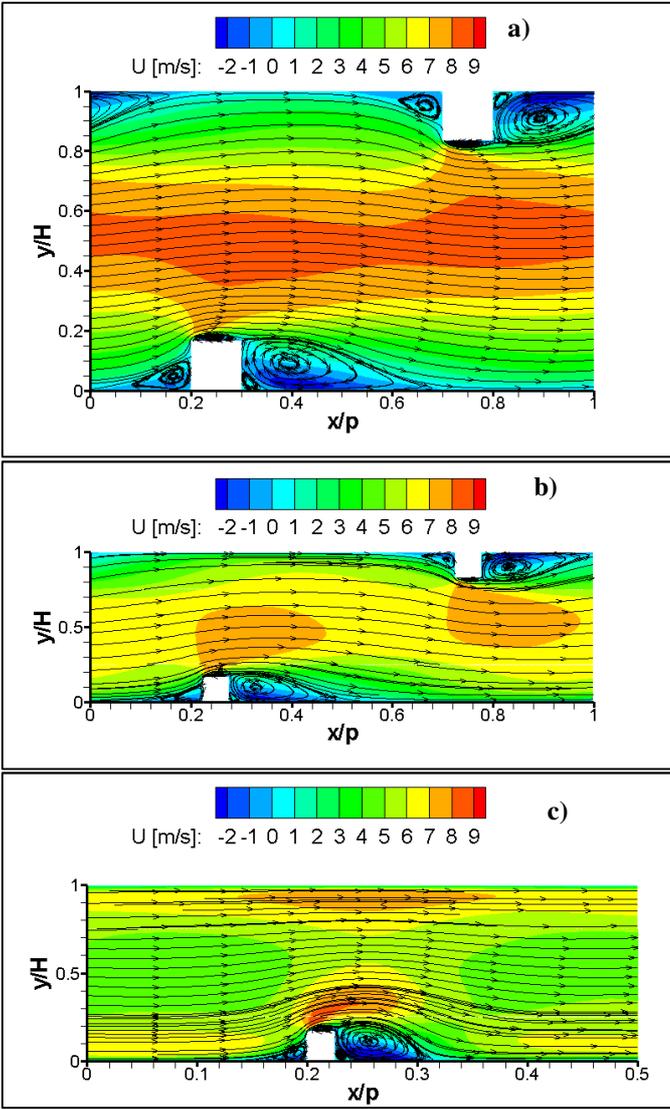
p/e	Re = 1171			Re = 2942			Re = 4119		
	s 10	s 20	s 40	s 10	s 20	s 40	s 10	s 20	s 40
2	0.375	0.209	0.075	0.370	0.210	0.113	0.403	0.227	0.121
4	0.288	0.214	0.117	0.320	0.248	0.131	0.359	0.278	0.156

## HEAT TRANSFER

The average heat transfer coefficient over a module is defined as:

$$h_{av} = \frac{\dot{m}c_p}{2(p-s)} \ln \left( \frac{T_w - T_{b,in}}{T_w - T_{b,out}} \right) \quad (3)$$

Where  $T_{b,in}$  and  $T_{b,out}$  are the bulk temperatures at the inlet and at the outlet of the module, respectively,  $p$  is the pitch length and  $s$  is the rib height.



**Figure 3:** Contour plots of x-velocity and streamline for  $Re=4119$ ,  $e=2mm$  and a)  $p/e=10$ ; b)  $p/e=20$ ; c)  $p/e=40$

The corresponding average Nusselt number over a module is:

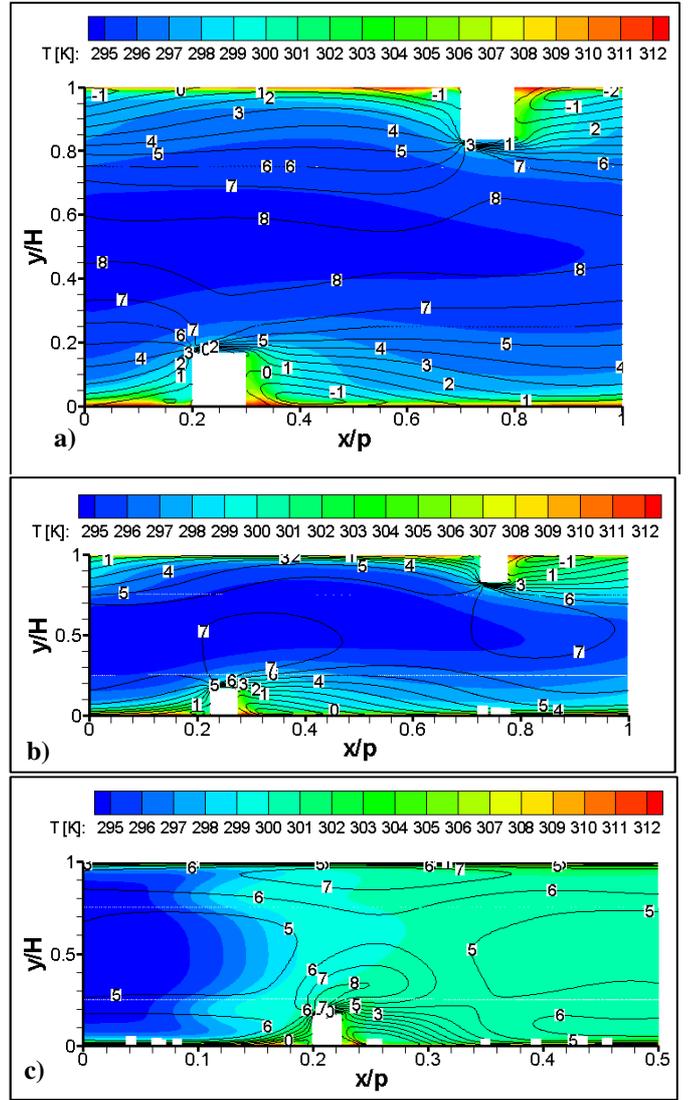
$$Nu = \frac{hH}{k_f} \quad (4)$$

Figure 4 shows the temperature contours for  $Re=4119$ ,  $e=2mm$  and pitch-to-height ratios of 10, 20 and 40, together with the constant velocity lines.

Hot spots exist close to the rear side of the ribs, attributed to the poor heat transfer from the wall to the fluid due to the recirculation zone. These spots decrease with increasing  $p/e$  (at the same  $Re$  number) because of the reduced vortex size.

## RESULTS AND DISCUSSION

The influence of wall roughness, i.e. the ribs, on friction is evaluated through the numerically calculated total dissipation function, where the turbulent dissipation is basically contained in the turbulent energy dissipation rate  $\varepsilon$ . The method is a post-processing step, imposing no additional cost to the calculation itself. The analysis of the dissipation distribution in the flow field close to the roughness elements allows gaining a deep insight into the physics.



**Figure 4:** Contour plots of temperature and constant x-velocity lines (m/s) for  $Re=4119$ ,  $e=2mm$  and a)  $p/e=10$ ; b)  $p/e=20$ ; c)  $p/e=40$

When using a RANS approach for a 2D turbulent flow simulation, the specific dissipation of mechanical energy can be divided in a part due to the time averaged velocity and the other due to the velocity fluctuations [4]:

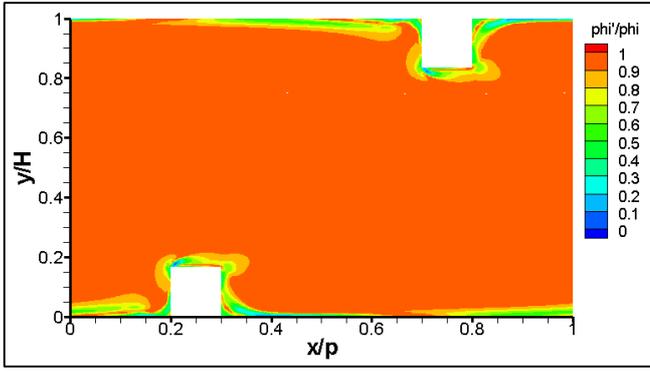
$$\bar{\varphi} = \frac{\mu}{\dot{m}} \left( 2 \left[ \left( \frac{\partial \bar{u}}{\partial x} \right)^2 + \left( \frac{\partial \bar{v}}{\partial y} \right)^2 \right] + \left( \frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x} \right)^2 \right) \quad (3)$$

$$\varphi' = \frac{\rho \varepsilon}{\dot{m}} \quad (4)$$

where the overbar denotes the temporal average of the quantity. The overall dissipation,  $\varphi_{tot}$ , between two cross sections in a fully developed or periodic horizontal channel flow is equivalent to  $\varphi_{tot} = -\Delta P / \rho$ . The Darcy-Weisbach friction factor is then calculated as:

$$f \equiv \frac{\varphi_{tot}}{p} \frac{2H}{u_b^2} \quad (5)$$

where  $\varphi_{tot}$  is the volume integral of  $\bar{\varphi} + \varphi'$ .



**Figure 5:** Contour plots of  $\phi'/\phi_{tot}$   $Re=4119$ ,  $e=2mm$  and  $p/e=10$

**Table 4:** Nu for the various configurations and Re

p/e	Re = 1171			Re = 2942			Re = 4119		
	10	20	40	10	20	40	10	20	40
2	23.30	19.00	15.62	31.23	26.20	21.66	36.77	31.65	26.53
4	26.34	19.96	16.56	35.27	27.15	22.52	45.66	33.56	25.10

**Table 5:** f for the various configurations and Re

p/e	Re = 1171			Re = 2942			Re = 4119		
	10	20	40	10	20	40	10	20	40
2	0.327	0.140	0.081	0.308	0.141	0.079	0.282	0.134	0.074
4	0.637	0.282	0.150	0.664	0.276	0.141	0.820	0.276	0.143

The dissipation field of mechanical energy gives detailed information on the background physics. As shown in Figure 5, the dissipation of mechanical energy can be mostly attributed to the velocity fluctuations and therefore to the Reynolds stresses. As it can be seen, only in the region close to the ribs, where flow is separating and mean velocity gradients are high, dissipation due to the mean flow becomes important.

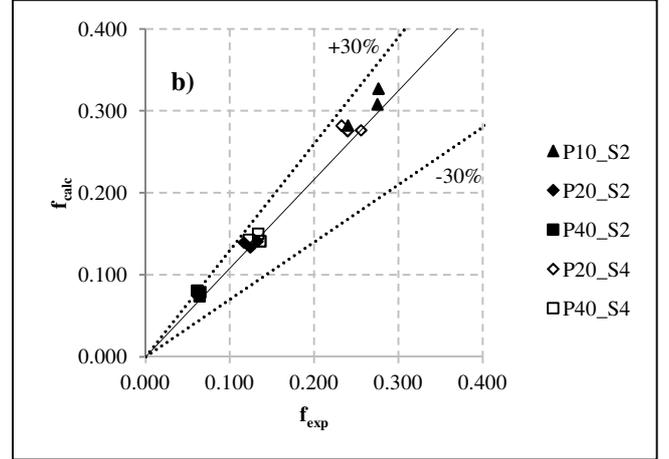
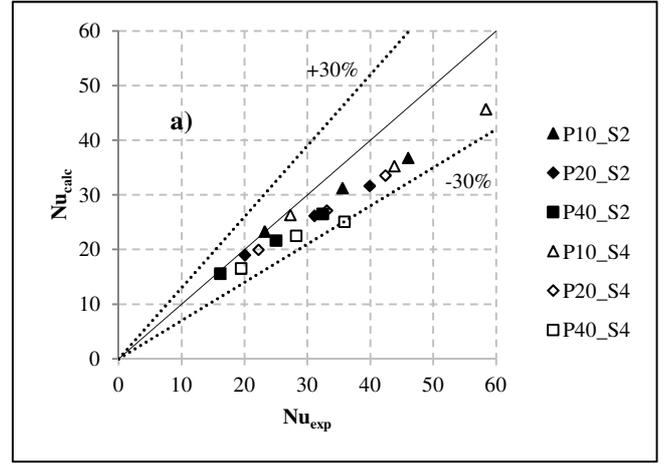
In Table 4 and 5 the calculated values of friction factor and Nusselt number for the different configurations and Reynolds numbers are compared.

As shown in Figure 6, where the numerical results for Nu and f are compared to the corresponding experimental value obtained by the authors with the same operating condition [2], the error band is  $\pm 30\%$ .

More precisely, calculated values of Nu are lower than the measured ones; indeed it has already been shown in [5] that the two-layer k- $\epsilon$  model underpredicts the heat transfer rate because of its inability to reproduce the secondary flow structures experimentally observed. Therefore the values of the Nu numbers obtained with the 2D simulations in the present work can only qualitatively indicate the most thermally efficient configuration among the considered ones.

## CONCLUSIONS

In the present work turbulent forced convective heat transfer in a ribbed channel is numerically simulated using a RANS approach. Although the inability of turbulence models in predicting heat transfer with this configurations is well known, they can give valuable qualitative information. These are used to identify the most promising configurations that are subsequently further analyzed both experimentally and numerically with LES or DNS techniques.



**Figure 6:** Comparison between numerical and experimental values of a) Nusselt number and b) friction factor

## NOMENCLATURE

Symbol	Quantity	SI Unit
$c_p$	specific heat at constant pressure	J/kgK
f	Darcy-Weisbach friction factor	-
$h_{av}$	module average heat transfer coefficient, Eq. (3)	W/m <sup>2</sup> K
H	channel height	m
k	turbulent kinetic energy	m <sup>2</sup> /s <sup>2</sup>
$k_f$	thermal conductivity	W/mK
$\dot{m}$	mass flow rate	kg/s
Nu	Nusselt number	-
p	pitch length	m
P	pressure	Pa
Re	Reynolds number, $\rho u_b H / \mu$	-
s	rib height	m
T	local temperature	K
$T_b$	bulk temperature	K
$T_{b,in}$	bulk temperature at module inlet	K
$T_{b,out}$	bulk temperature at module outlet	K
$T_w$	wall temperature	K
u	velocity component in x-direction	m/s
$u_b$	bulk velocity	m/s
v	velocity component in y-direction	m/s
$\epsilon$	turbulent kinetic energy dissipation	m <sup>2</sup> /s <sup>3</sup>
$\theta$	dimensionless temperature, Eq. (2)	-
$\mu$	dynamic viscosity	Pa s
$\rho$	density	kg/m <sup>3</sup>

$\bar{\phi}$	mechanical energy dissipation due to mean flow	$m^2/s^2$
$\phi'$	mechanical energy dissipation due to flow fluctuations	$m^2/s^2$

## REFERENCES

- [1] L. Marocco, D. Fustinoni, P. Gramazio, A. Niro, First numerical results on forced convection heat transfer inside a rectangular channel with straight ribs on lower and upper walls, *UIT Conference*, Torino, 2011.
- [2] D. Fustinoni, P. Gramazio, A. Niro, Heat transfer characteristics in forced convection through a rectangular channel with ribbed surfaces, *UIT conference*, Torino, 2011.
- [3] D.D. Luo, C.W. Leung, T.L. Chan, W.O. Wong, Flow and forced-convection characteristics of turbulent flow through parallel plates with periodic transverse ribs, *Numerical Heat Transfer, Part A*, vol. 48, pp. 43-58, 2005.
- [4] H. Herwig, D. Gloss, T. Wenterodt, A new approach to understanding and modelling the influence of wall roughness on friction factors for pipe and channel flows, *Journal of fluid mechanics*, vol. 613, pp. 35-53, 2008.
- [5] A. Ooi, G. Iaccarino, P.A. Durbin, M. Behnia, Reynolds averaged simulation of flow and heat transfer in ribbed ducts, *International Journal of Heat and Fluid Flow*, vol. 23, pp. 750-757, 2002.