ISROMAC 2017
International Symposium on Transport Phenomena and Dynamics of Rotating Machinery
Maui, Hawaii
December 16-21, 2017

[Extended Abstract]

Direct numerical simulation of heat and fluid flow around pin fin arrays and its experimental validation

Yutaka Fukuda, Institute of Industrial Science, The University of Tokyo, Tokyo, Japan
Yukinori Kametani, Institute of Industrial Science, The University of Tokyo, Tokyo, Japan
Yosuke Hasegawa, Institute of Industrial Science, The University of Tokyo, Tokyo, Japan

1. Introduction

The efficiency of a modern gas turbine has been improved primarily through the increase of turbine inlet temperature. Since the service temperature of materials is limited, the efficient cooling of an internal blade is one of key technologies. In the present study, we focus on a pin-fin array typically employed around the tailing edge of a turbine blade. The purpose of the present study is to achieve a higher heat transfer rate with a minimum pressure drop. To this end, we measure the pressure drop and the heat transfer rate of a channel between parallel walls with different pin-fin arrays fabricated by a 3D printer, and compare them with the results obtained from direct numerical simulation (DNS).

2. Methodology

We consider three configurations of staggered pin-fins between two parallel plates as summarized in Fig 1 and Table1. The governing equations for the velocity and thermal fields are the incompressible Navier-Stokes, continuity and energy transport equations. The finite volume method is used for spatial discretization, and the numbers of grid points employed are $(N_x, N_y, N_z) = (128, 128, 65)$, respectively. At the interface between the fluid and solid phases, no-slip and iso-thermal conditions are applied and they are realized by using the volume penalization method [1]. For all the cases, the Reynolds number based on the bulk mean velocity, the channel height is set to be $Re_H = 1000$.
To verify the results obtained by the numerical simulation described above, we also measured a pressure drop and a heat transfer ratio. The experimental apparatus is composed of a wind tunnel, a heater, and a test section. The pressure drop of the test section is evaluated from the pressure difference between the inlet and the outlet, where the latter is the same as an environmental pressure. The overall heat transfer rate inside the test section is evaluated by a transient single blow method[2], where the inlet temperature is increased stepwise by a heater, and then the heat transfer rate is estimated from the transient behavior of the outlet temperature.

3. Results

The present numerical and experimental results are summarized in Table 1. The predictions based on the existing correlations by Jacob[3] and Metzger et al.[4] are also listed for comparison.
Figure 3 shows the friction factor as a function of Reynolds number. In the cases 2 and 3, the measured pressure drops are slightly larger than the numerical results. This might be attributed to the surface

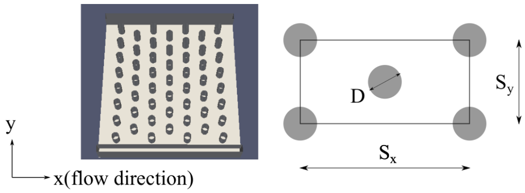

Figure 1. schematic view of experimental set up

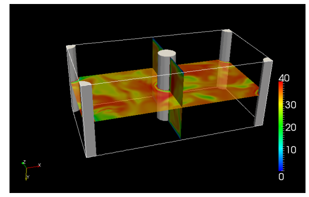

Figure 2. Instantaneous thermal field obtained from DNS for Case 2

Table 1. Comparison between numerical and experimental results at $Re = 1000$

case	h	S_x	S_y	D	friction factor Sim.	friction factor Exp.	friction factor Jacob[3]	Nu number Sim.	Nu number Exp.	Nu number Metzger et al.[4]
case1	10.0	12.5	25.0	5.0	0.306	0.262	0.216	11.92	10	10.2
case2	10.0	12.5	25.0	2.5	0.107	0.129	0.132	10.01	7	8.23
case3	10.0	20.0	20.0	5.0	0.186	0.209	0.162	10.28	9	9.47

roughness of the test samples employed int the present experiments.

Figure 4 shows transient temperature behavior in case 1. The measurement and numerical data are denoted by solid and broken lines, respectively. As shown in Table 1, the present experiments agree fairly well with both numerical and literature data, although there exists some discrepancy in the initial period.

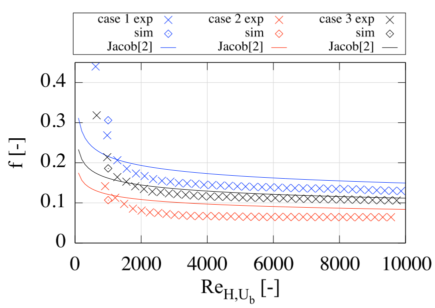

Figure 3. Friction factor versus Re number

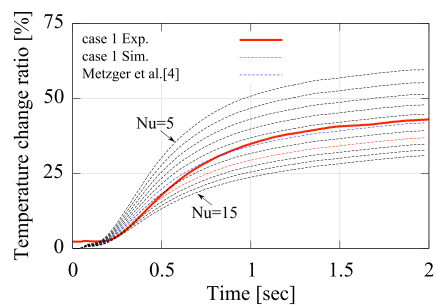

Figure 4. Transient temperature behavior for Case 1 compared with corresponding results at $Re = 1000$

References

[1] Kai Schneider. Numerical simulation of the transient flow behaviour in chemical reactors using a penalisation method. *Computers & Fluids*, 34(10):1223–1238, dec 2005.

[2] CY Liang and WJ Yang. Modified Single-Blow Technique for Performance Evaluation on Heat Transfer Surfaces. *Jounal of Heat Transfer-Transactions of the ASME*, (97):16–21, 1975.

[3] M. Jacob. Heat Transfer and Flow Resistance in Cross Flow of Gases Over Tube Banks. *Trans. ASME*, 59:384–386, 1938.

[4] D. E. Metzger, W. B. Shepard, and S. W. Haley. Row Resolved Heat Transfer Variations in Pin-Fin Arrays Including Effects of Non-Uniform Arrays and Flow Convergence. *ASME: International Gas Turbine Conference and Exhibit*, page V004T09A015, jun 1986.