Effect of turning angle on performance of 2-D turning diffuser via Asymptotic Computational Fluid Dynamics

Y T Khong\textsuperscript{1, a}, N Nordin\textsuperscript{1, b}, S M Seri\textsuperscript{1}, A N Mohammed\textsuperscript{1}, A Sapit\textsuperscript{1}, I Taib\textsuperscript{1}, K Abdullah\textsuperscript{1}, A Sadikin\textsuperscript{1} and M A Razali\textsuperscript{1}

\textsuperscript{1} Flow Analysis, Simulation and Turbulence Research Group, Center for Energy and Industrial Environment Studies, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, Johor, Malaysia

E-mail: yanteng.0714@gmail.com\textsuperscript{a}, mayati@uthm.edu.my\textsuperscript{b}

Abstract. The present work aims to numerically investigate the effect of varying turning angle, $\phi = 30^\circ - 90^\circ$ on the performance of 2-D turning diffuser and to develop the performance correlations via integrating the turning angle using Asymptotic Computational Fluid Dynamics (ACFD) technique. Standard k-\textit{\v{e}} adopting enhanced wall treatment of $y^+ \approx 1.1$ appeared as the best validated model to represent the actual cases with deviation of $\pm4.7\%$. Results show that the pressure recovery, $C_p$ and flow uniformity, $\sigma_{\text{out}}$ are distorted of respectively $37\%$ and $28\%$ with the increment of turning angle from $30^\circ$ to $90^\circ$. The flow separation starts to emerge within the inner wall, $S=0.91L_p/W_1$ when $45^\circ$ turning diffuser is applied and its scale is enlarged by further increasing the turning angle. The performance correlations of 2-D turning diffuser are successfully developed with deviation to the full CFD solution approximately of $\pm7.1\%$.

1. Introduction

Turning diffuser is often introduced in applications such as HVAC [1 - 3], wind-tunnel [4 - 6], gas turbine cycle [7] and aircraft engine [8, 9] as an adapter to join the conduits of different cross-sectional areas or an ejector to decelerate the flow and raise the static pressure before discharging to the atmosphere. The performances of diffuser are measured primarily using outlet pressure recovery coefficient ($C_p$) and flow uniformity index ($\sigma_{\text{out}}$) that are so much affected due to the nature of its geometry by the existence of flow separation and dispersion of core and secondary flow.

Study of the geometrical effect on diffuser performance has been of fundamental interest to researchers in the area of fluid mechanics since decades and it continues to grow [1-12]. Turning angle ($\phi$) is one of important geometrical parameters that is deemed to influence significantly the performance of turning diffuser. Sullerey et al. [13] found that the $C_p$ of $55^\circ$ turning diffuser was slightly lower than a straight diffuser. It may however improve comparable to the performance of straight diffuser by applying the turbulent intensity of minimum $3.4\%$. The $\phi = 90^\circ$ [6, 14] and $180^\circ$ [15] were found to affect notably the flow performance and often flow control devices such as guide vanes, screens, honeycomb and vortex generators were introduced to improve the flow.

Nordin [1] has recently established empirical correlations integrating the effects of both geometrical and operating parameters on the performance of $90^\circ$ 2-D turning diffuser using Asymptotic Computational Fluid Dynamics (ACFD) technique. However, these correlations, i.e. Equations (1) and (2) have been developed not to integrate the effect of $\Delta\phi$. 

A CFD is a relatively new analytical technique established by Herwig et al. that applies Taylor Series expansion to produce correlations between all the relevant non-dimensional variables of the problem analysed by CFD [16-19]. This technique requires less number of solutions \( n + 1 \) than the usual-used linear regression and curve fitting techniques, \( 4^n \) number of solutions to develop correlations from the CFD simulations, where \( n \) represents non-dimensional variables.

The present work aims to numerically investigate the effects of varying turning angle (\( \phi = 30^\circ, 45^\circ, 55^\circ, 75^\circ \) and \( 90^\circ \)) as shown in Figure 1 on the performance of 2-D turning diffuser. The ACFD technique is then applied to develop performance correlations via integrating the effect of turning angle, i.e. \( C_p = f(L_{in}/W_1, W_2/W_1, \phi, Re_{in}) \) and \( \sigma_{out} = f(L_{in}/W_1, W_2/W_1, \phi, Re_{in}) \).

![Figure 1. 2-D turning diffuser of (a) \( \phi = 30^\circ \), (b) \( \phi = 45^\circ \), (c) \( \phi = 55^\circ \), (d) \( \phi = 75^\circ \) and (e) \( \phi = 90^\circ \)](image-url)
2. Methodology

2.1. CFD Method

The geometrical domains as shown in Figure 1 were modelled using Solidworks. Three types of boundary conditions were imposed as depicted in Table 1. At the solid wall, the velocity was zero due to the no-slip condition. The inlet velocity, \( V_{in} \) respective to \( \text{Re}_{in} = 6.382 \times 10^4 \) was specified at 14.25 m/s. This corresponded to the turbulent intensity, \( I_{in} \) of 4.0%. At the outlet boundary, the pressure was set at the atmospheric pressure (0 gage pressure). The working fluid was air at 30°C with \( \rho = 1.164 \text{ kg/m}^3 \) and \( \mu = 1.872 \times 10^{-5} \text{ kg/m.s} \).

| Table 1. Boundary operating conditions |
|----------------------------------------|
| Inlet Type of boundary                  |
| Velocity magnitude, \( V_{in} \) (m/s) |
| Pressure (Pa)                           |
| Temperature (°C)                        |
| Density, \( \rho \) (kg/m³)             |
| Dynamic viscosity, \( \mu \) (kg/m.s)   |
| Working fluid                          |
| Mesh Nodes                              |
| Pressure Recovery, \( C_p \)            |
| Deviation, %                           |
| 1                                       |
| 2                                       |
| 3                                       |
| 4                                       |
| 5                                       |

The grid was generated using ANSYS ICEM CFD by adopting enhanced wall treatment with the size of wall-adjacent cell, \( y^+ \approx 1.0 \). The grid independency test was carried out as shown in Table 2 with Mesh 4 appeared as the optimum to produce accurate result with reasonable CPU time.

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0
\]

(3)

x- momentum equation:

\[
\frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = -\frac{\partial P}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) - \frac{\partial (\bar{u}' \bar{v}')}{\partial y} - \frac{\partial (\bar{u}' \bar{w}')}{\partial z} + S_i
\]

(4)
The performance correlations of 2-D turning diffuser were developed using ACFD technique. This technique involved several steps as follows [1, 16 - 19]:

1) Identifying the dependent and independent variables
2) Linearizing the relationship between the dependent and independent variables
3) Applying the Taylor’s series expansion
4) Determining the convergence point and gradients
5) Substituting all the constants to complete the correlations

Firstly, the dependent and independent variables were identified. The $C_p$ and $\sigma_{out}$ were dependent on $Re_{in}$, $W_2/W_1$, $L_{in}/W_1$ and $\phi$. The preliminary relationships between the dependent and independent variables were written as follows:

$$\begin{align}
C_{p, ACFD} &= f(Re_{in}, W_2/W_1, L_{in}/W_1, \phi) \\
\sigma_{out, ACFD} &= f(Re_{in}, W_2/W_1, L_{in}/W_1, \phi)
\end{align}$$

Five set of solutions were needed for each independent variable $(Re_{in}, W_2/W_1, L_{in}/W_1, \phi)$. The reference values were $Re_{in, ref} = 6.382 \times 10^4$, $W_2/W_1, ref = 2.16$, $L_{in}/W_1, ref = 4.37$ and $\phi_{ref} = 90^\circ$. The relationship of dependent and independent variable was linearized in which appropriate chosen reference values may ease the linearization procedure otherwise the modification to the term for instance by adding an exponent should be considered.

The performance correlations were developed by applying Taylor series based expansion:

$$\begin{align}
\eta(\phi_1, \phi_2, \phi_3, ..., \phi_n) &= \eta_{ref} + \left( \phi_1 - \phi_{1, ref} \right) \frac{\partial \eta}{\partial \phi_1} + \left( \phi_2 - \phi_{2, ref} \right) \frac{\partial \eta}{\partial \phi_2} + \left( \phi_3 - \phi_{3, ref} \right) \frac{\partial \eta}{\partial \phi_3} + \cdots + \\
&\left( \phi_n - \phi_{n, ref} \right) \frac{\eta}{\partial \phi_n}
\end{align}$$

where,

$\eta = \text{dependent variables (} C_p, \sigma_{out})$

$$\begin{align}
\phi_1 &= \left( \frac{Re_{in}}{Re_{in, ref}} \right)^\xi \\
\phi_2 &= \left( \frac{L_{in}/W_1}{L_{in, ref}/W_1} \right)^\xi
\end{align}$$

The applicability of standard $k-\varepsilon$ (std $k-\varepsilon$), shear stress transport $k-\omega$ (SST $k-\omega$) and Reynolds stress model (RSM) to close the RANS equations was verified. Pressure based solver with a robust pressure-velocity coupling scheme, SIMPLE was applied. The gradient was discretised by Green-Gauss Cell-based. As it involved high pressure gradients, pressure was discretised by PRESTO scheme. A 3rd order accuracy scheme, QUICK was used to discretise the convection terms, i.e. momentum, turbulent kinetic energy and turbulent dissipation rate owing to its proven capability to solve the flow in diffuser when hybrid mesh was applied. The convergence criterion was set to be $10^{-6}$.
\[ \varphi_1 = \left[ \frac{w_x/w_1}{w_x/w_x \text{ ref}} \right]^c, \quad \varphi_4 = \left[ \frac{\varphi}{\varphi \text{ ref}} \right]^d \]

\( \varphi_1, \varphi_2, \varphi_3, \varphi_4 = \) Dimensionless independent groups.

\( a, b, c, d = \) Exponent values chosen to let all lines intersect and converge at one point in graph.

\( \eta_{\text{ref}} = \) Reference value of dependent variable (intersection/convergence point at y-axis)

\( \varphi_{1 \text{ ref}}, \varphi_{2 \text{ ref}}, \varphi_{3 \text{ ref}}, \varphi_{4 \text{ ref}} = \) Reference values of dimensionless groups (intersection/convergence point at x-axis)

\[ \frac{\partial \eta}{\partial \omega}, \frac{\partial \eta}{\partial \omega_1}, \frac{\partial \eta}{\partial \omega_2}, \frac{\partial \eta}{\partial \omega_3} = \) Gradients / slopes of the corresponding lines

Convergence point and gradients should be determined in order to solve Equation 9. The detail solution is presented in the following section.

3. Results and Discussion

3.1. CFD validation results

CFD validation was carried out by comparing the simulation results with the experimental results by Nordin [1]. Table 3 shows that the std k-\( \varepsilon \) appears as the most optimum model producing the least deviation to the experiment of 4.7%. Furthermore, as shown in Figure 2, comparable flow structure with almost similar onset flow separation between CFD and experiment is obtained by applying std k-\( \varepsilon \) adopted enhanced wall treatment.

| Solver Model   | \( C_p \) | Deviation, % |
|----------------|-----------|--------------|
| Experiment [1] | 0.2090    | -            |
| std k-\( \varepsilon \) | 0.2188   | 4.7          |
| SST k-\( \omega \) | 0.1573   | 24.7         |
| RSM            | 0.1475    | 29.4         |

Table 3. CFD validation

\[ \text{Figure 2. Flow structure (a) Experiment, PIV [1] (b) CFD, std k-\( \varepsilon \) + enhanced wall treatment} \]
3.2. Effect of turning angle (ϕ)
Table 4 shows the effects of varying turning angle (ϕ) on the pressure recovery coefficient (Cₚ), flow uniformity index (σₜₚₑₑ) and onset flow separation (S). The Cₚ is affected by 37% with the increase of ϕ from 30° to 90° due to considerable flow separation occurs particularly within the 90° diffuser, S=0.61Lₑₑₑₑₑₑ/W₁ as shown in Figure 3(c). The flow separation is undesirable as it not only affects the Cₚ but also diminish the core flow area as shown in Figure 4. This then disturbs the flow uniformity of 28% with the increase of ϕ to 90°.

Table 4. Effect of turning angle on 2-D turning diffuser performances

| Turning angle, ϕ (°) | Pressure recovery, Cₚ | Flow uniformity, σₑₑₑₑₑₑ | Separation Point, S |
|----------------------|----------------------|-----------------------------|---------------------|
| 30                   | 0.3469               | 2.9477                      | -                   |
| 45                   | 0.3394               | 3.0755                      | 0.91Lₑₑₑₑₑₑ/W₁      |
| 55                   | 0.3045               | 2.9698                      | 0.85Lₑₑₑₑₑₑ/W₁      |
| 75                   | 0.2372               | 3.4278                      | 0.70Lₑₑₑₑₑₑ/W₁      |
| 90                   | 0.2188               | 3.7043                      | 0.61Lₑₑₑₑₑₑ/W₁      |

Figure 3. Flow separation (a) ϕ = 30°, (b) ϕ = 55°, (c) ϕ = 90°

Figure 4. Outlet velocity profiles by varying ϕ = 30°-90°
3.3. Performance correlations via ACFD

Applying Taylor Series expansion to develop outlet pressure recovery ($C_p$) correlation:

$$C_{p,acf} = C_{p,ref} + (\phi_1 - \phi_{1,ref}) \frac{\partial C_p}{\partial \phi_1} + (\phi_2 - \phi_{2,ref}) \frac{\partial C_p}{\partial \phi_2} + (\phi_3 - \phi_{3,ref}) \frac{\partial C_p}{\partial \phi_3} + (\phi_4 - \phi_{4,ref}) \frac{\partial C_p}{\partial \phi_4}$$

(10)

$\phi_1$, $\phi_2$, $\phi_3$ and $\phi_4$ are dimensionless independent groups where the data for $\phi_1$, $\phi_2$, $\phi_3$ are taken from previous work [1]. Table 5 shows the data for $\phi_4$ where in order to fit all lines in a graph as shown in Figure 5 and converge at a point represented as $C_{p,ref} = 0.200$, the exponent values $a = 1$, $b = 1.3$, $c = 3$ and $d = 4.5$ are chosen.

| $\phi$ | $\phi_4 = [\phi/\phi_{ref}]^{4.5}$ | $C_{p, cfd}$ |
|-------|---------------------------------|-------------|
| 30    | 0.0071                          | 0.3469      |
| 45    | 0.0442                          | 0.3394      |
| 55    | 0.1090                          | 0.3045      |
| 75    | 0.4402                          | 0.2372      |
| 90    | 1.0000                          | 0.2188      |

Figure 5. Outlet pressure recovery, $C_p$ with respect to $\phi_1 = \left[\frac{Re_{in}/Re_{in \ ref}}{Re_{in}/Re_{in \ ref}}\right]^{1}$, $\phi_2 = \left[\frac{L_{in}/W_{in}/W_{in \ ref}}{L_{in}/W_{in}/W_{in \ ref}}\right]^{1.3}$, $\phi_3 = \left[\frac{W_2/W_1/W_2/W_1 \ ref}{W_2/W_1/W_2/W_1 \ ref}\right]^{3}$ and $\phi_4 = [\phi/\phi_{ref}]^{4.5}$

Substituting all the constants in Equation (10) yielding,

$$C_{p,acf} = 0.200 + \left(\left[\frac{Re_{in}/Re_{in \ ref}}{Re_{in}/Re_{in \ ref}}\right]^{1} - 1\right) 0.0215 - \left(\left[\frac{L_{in}/W_{in}/W_{in \ ref}}{L_{in}/W_{in}/W_{in \ ref}}\right]^{1.3} - 1\right) 0.4065 - \left(\left[\frac{W_2/W_1/W_2/W_1 \ ref}{W_2/W_1/W_2/W_1 \ ref}\right]^{3} - 1\right) 0.0288 - \left(\left[\frac{\phi}{\phi_{ref}}\right]^{4.5} - 1\right) 0.1284$$

(11)
The efficacy of ACFD correlation to represent the full CFD solution is verified using a parity plot as shown in Figure 6. Satisfied agreement between the ACFD and CFD solutions is achieved within ±5.9%. This basically applies for the range of $5.786 \times 10^4 \leq \text{Re}_m \leq 1.775 \times 10^5$, $4.37 \leq L_w/W_1 \leq 20$, $2.16 \leq W_2/W_1 \leq 4.0$, $30^\circ \leq \phi \leq 90^\circ$.

![Figure 6. Parity plot showing agreement of CFD simulation ($C_{p\, cfd}$) with ACFD correlation ($C_{p\, acfd}$).](image)

Applying Taylor series expansion to develop $\sigma_{out}$ correlation:

$$
\sigma_{out\, acfd} = \sigma_{out\, ref} + (\phi_1 - \phi_{1\, ref}) \frac{\partial \sigma_{out\, ref}}{\partial \phi_1} + (\phi_2 - \phi_{2\, ref}) \frac{\partial \sigma_{out\, ref}}{\partial \phi_2} + (\phi_3 - \phi_{3\, ref}) \frac{\partial \sigma_{out\, ref}}{\partial \phi_3} + (\phi_4 - \phi_{4\, ref}) \frac{\partial \sigma_{out\, ref}}{\partial \phi_4}
$$

$\phi_1, \phi_2, \phi_3$ and $\phi_4$ are dimensionless independent groups where the data for $\phi_1, \phi_2, \phi_3$ are taken from previous work [1]. Table 6 shows the data for $\phi_4$ where in order to fit all lines in a graph as shown in Figure 7 and converge at a point represented as $\sigma_{out\, ref} = 3.45$, the exponent values $a = 2.9, b = 1.6, c = 1$ and $d = 0.1$ are chosen.

| $\phi$ | $\phi_4 = [\phi/\phi_{ref}]^{0.1}$ | $\sigma_{out\, cfd}$ |
|-------|---------------------------------|-------------------|
| 30    | 0.8959                          | 2.9477            |
| 45    | 0.9330                          | 3.0755            |
| 55    | 0.9519                          | 2.9698            |
| 75    | 0.9819                          | 3.4278            |
| 90    | 1.0000                          | 3.7043            |
Figure 7. Flow Uniformity Index, $\sigma_{out}$ with respect to $\phi_1 = \left(Re_{in}/Re_{ref}\right)^2$, $\phi_2 = \left[L_{in}/W_1/W_2/W_{ref}\right]^6$, $\phi_3 = \left[W_2/W_1/W_2/W_{ref}\right]^1$ and $\phi_4 = \left[\phi/\phi_{ref}\right]^{0.1}$, thus are the gradients of the corresponding lines.

Substituting all the constants in Equation 12 yielding,

$$\sigma_{out,acfd} = 3.45 + \left(\left[Re_{in}/Re_{ref}\right]^{2.9} - 1\right)0.2754 - \left(\left[L_{in}/W_1/W_2/W_{ref}\right]^6 - 1\right)0.0711 - \left(\left[W_2/W_1/W_2/W_{ref}\right]^1 - 1\right)0.2509 + \left(\left[\phi/\phi_{ref}\right]^{0.1} - 1\right)7.0854$$

(13)

The efficacy of ACFD correlation to represent the full CFD solution is verified using a parity plot as shown in Figure 8. Satisfied agreement between the ACFD and CFD solutions is achieved within ±8.3%. This basically applies for the range of $5.786 \times 10^4 \leq Re_{in} \leq 1.775 \times 10^5$, $1.50 \leq L_{in}/W_1 \leq 30$, $1.20 \leq W_2/W_1 \leq 4.0$, $30^\circ \leq \phi \leq 90^\circ$.

Figure 8. Parity plot showing agreement of CFD simulation ($\sigma_{out \_cfd}$) with ACFD correlation ($\sigma_{out \_acfd}$)
4. Conclusion
In conclusion, the research to investigate the effect of turning angle and develop performance correlations of 2-D turning diffuser via ACFD has been successfully carried out. The $C_p$ and $\sigma_{out}$ are distorted of respectively 37% and 28% with the increment of $\phi$ from 30° to 90°. The flow separation starts to emerge within the inner wall, $S=0.91L_{in}/W_1$ when 45° turning diffuser is applied and its scale is enlarged by further increasing the $\phi$. The performance correlations by integrating turning angle are successfully developed to meet the full CFD solution within acceptable deviation of ±7.1%. These developed correlations are therefore reliable to be used by one in future to evaluate the performance of 2-D turning diffuser.

Acknowledgements
Immeasurable appreciation is extended to Flow Analysis Simulation and Turbulence Research Group (FAST) for all provided research supports and insight sharing. Also thanks to Mr. Rosman Tukiman (Assistant Engineer of CFD Laboratory, UTHM) for the technical-lab assist.

Nomenclature

| Symbol | Description |
|--------|-------------|
| 2-D    | Two dimensional |
| $\phi$ | Turning angle (°) |
| $\phi_{ref}$ | Reference turning angle (=90°) |
| $\sigma_{out,ACFD}$ | Flow uniformity index |
| $C_p^{ACFD}$ | Outlet pressure recovery coefficient |
| $L_{in}$ | Inner wall length (m) |
| $L_{in}/W_1^{ref}$ | Reference inner wall length to inlet throat width ratio (=4.37) |
| $Re_{in}$ | Inflow Reynold number |
| $Re_{in,ref}$ | Reference inflow Reynold number (=6.382 x 10⁴) |
| $W_1$ | Inlet width (m) |
| $W_2$ | Outlet width (m) |
| $W_2/W_1^{ref}$ | Reference outlet inlet configuration (=2.16) |

References

[1] Nordin N 2016 Performance investigation of turning diffusers at various geometrical and operating parameters PhD Thesis Department of Mechanical Engineering (Bandar Seri Iskandar: Universiti Teknologi PETRONAS)
[2] El-Askary W A and Nasr M 2009 Performance of a bend diffuser system: Experimental and numerical studies Computer & Fluids 38 160-170
[3] Gan G and Riffat S B 1996 Measurement and computational fluid dynamics prediction of diffuser pressure-loss coefficient Applied Energy 54(2) 181-195
[4] Lindgren B and V Johansson A V, 2002 Design and Evaluation of a Low-Speed Wind-Tunnel with Expanding Corners (Stockholm: Universitetsservice US AB)
[5] Nguyen C K, Ngo T D, Mendis P A and Cheung J C 2006 A flow analysis for a turning rapid diffuser using CFD Comput. Wind Eng
[6] Chong T P, Joseph P F and Davies P O A L 2008 A Parametric Study of Passive Flow Control for a Short, High Area Ratio 90 deg Curved Diffuser J. Fluids Eng. 130(11)
[7] Wang Y C, Hsu J C and Lee Y C 2009 Loss characteristics and flow rectification property of diffuser valves for micropump applications Int. J. of Heat and Mass Transfer 52 328-336
[8] Ibrahim I H, Ng E Y K, Wong K and Gunasekaran R 2008 Effects of centerline curvature and cross-sectional shape transitioning in the subsonic diffuser of the F-5 fighter jet J. Mech. Sci. Technol. 22(10) 1993–1997
[9] Bourgeois J A, Martinuzzi R J, Savory E, Zhang C and Roberts D A 2011 Assessment of Turbulence Model Predictions for an Aero-Engine Centrifugal Compressor J. Turbomach. 133(1) 11025
[10] Nordin N, Karim Z A A, Othman S and Raghavan V R 2013 Design & development of low subsonic wind tunnel for turning diffuser application Advanced Materials Research 614-615 586-591
[11] Nordin N, Karim Z A A, Othman S and Raghavan V R 2014 The performance of turning diffusers at various inlet conditions Applied Mechanics and Materials 465-466 597-602

[12] Nordin N, Raghavan V R, Othman S and Karim Z A A 2012 Compatibility of 3-D turning diffusers by means of varying area ratios and outlet-inlet configurations ARPN Journal of Engineering and Applied Sciences 7(6) 708-713

[13] Sullerey R K, Chandra B and Muralidhar V 1983 Performance comparison of straight and curved diffusers J. of Def. Sci. 33 195-203

[14] Eugene L Z P, Nordin N, Othman S and Raghavan V R 2011 A CFD preliminary study: Pressure losses and flow structure in turning diffuser by means of installing turning baffles 2nd International Conference of Mechanical Engineering in Putrajaya, Malaysia

[15] Nguyen C K, Ngo T D, Mendis P A and Cheung J C K 2006 A flow analysis for a turning rapid diffuser using CFD J. Wind Eng. 108 749-752

[16] Balaji C and Herwig H 2003 The use of ACFD approach in problems involving surface radiation and free convection Int. Comm. Heat and Mass Transfer 30 251-259

[17] Balaji C, Holling M and Herwig H 2007 A general methodology for treating mixed convection problems using asymptotic computational fluid dynamics (ACFD) Int. Comm. In Heat and Mass Transfer 34 682-691

[18] Balaji C, Holling M and Herwig H Determination of temperature wall functions for high Rayleigh Number flows using asymptotics: A systematic approach Int. Journal of Heat and Mass Transfer 50 3820-3831

[19] Premachandaran B and Balaji C 2006 Conjugate mixed convection with surface radiation from a horizontal channel with protruding heat sources Int. Journal of Heat and Mass Transfer 49 3568-3582