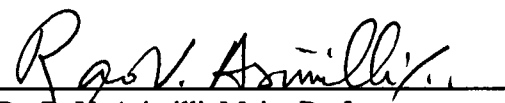
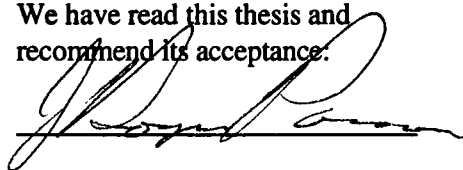
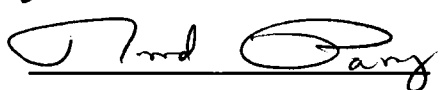


To the Graduate Council:

I am submitting herewith a thesis written by Colby Russell Evans entitled "An Investigation of the Turbulent Flow and Heat Transfer Predictions of FIDAP". I have examined the final copy of this thesis for form and content and recommend that it be accepted in partial fulfillment of the requirements for the degree of Master of Science, with a major in Mechanical Engineering.


Dr. R. V. Arimilli, Major Professor

We have read this thesis and
recommend its acceptance.

Accepted for the Council:


Associate Vice Chancellor and
Dean of The Graduate School

**AN INVESTIGATION OF THE
TURBULENT FLOW AND HEAT TRANSFER
PREDICTIONS OF FIDAP**

A Thesis Presented for the
Master of Science Degree
The University of Tennessee, Knoxville

Colby Russell Evans
August 1996

ACKNOWLEDGMENTS

I am grateful to my family and my major professor, Dr. Rao V. Arimilli, for their patience, guidance and encouragement. I would also like to thank my committee members, Drs. M. Parang and J. R. Parsons, and others for their time and support. Thanks to their help, the difficulties I encountered and overcame during the last two years have given me confidence and wisdom.

ABSTRACT

In this study, turbulent flow and heat transfer characteristics of three two-dimensional and one axisymmetric geometries are modeled using a Finite-Element k - ϵ technique provided in the form of the computational fluid dynamics package FIDAP. The first case studied is a backward-facing step downstream of a developing channel flow with a Reynolds number of 38,800 based on the 3.3 millimeter step height. The surfaces are held at a constant temperature. The time-averaged Navier-Stokes equations are solved in a two-dimensional form and reveal close agreement with experimental measurements of flow and heat transfer characteristics for this case. The second case is a converging duct with low Reynolds number accelerating turbulent flow conditions. A constant heat flux is applied to the lower wall. The experimental data shows a relaminarization of the turbulent boundary layers. The simulation results for this case do not correlate well with the data, due to the low Reynolds number used in the experiment and the overprediction of the turbulent structure. The third case is a fully-developed flow over a two-dimensional ribbed geometry. The periodic nature of the fully-developed regime is identified far downstream of the inlet by the repetition of the velocity and turbulence components' profiles and the temperature profile. The surface temperature is linearly increased in the flow direction as in a counter-flow heat exchanger. Two-dimensional simulations are performed, and friction coefficients and Nusselt numbers are compared to experimental data. The results for this case show the same trends as the data, although the values are underpredicted. The fourth case is an axisymmetric developing flow within concentric cylinders with a small rib located downstream. The measured wall temperatures from the experiment are applied as the boundary conditions in the models. The heat transfer trends for this case follow the experimental data with some overprediction. The predicted heat transfer and flow structure results from the models of the different geometries generally follow the trends of the

experimental data. The models are limited by the availability of good measurements of the flow and turbulence structure.

TABLE OF CONTENTS

	PAGE
1. INTRODUCTION	1
1.1 Literature Review	1
2. PROBLEM STATEMENT	4
2.1 Problem Description	4
2.2 Governing Equations	5
2.3 Boundary Conditions	6
2.3.1 Hydrodynamic boundary conditions (Backward-facing step symmetric top model)	6
2.3.2 Thermal boundary conditions (Backward-facing step symmetric top model)	7
2.3.3 Hydrodynamic boundary conditions (Backward-facing step, full-size channel model)	7
2.3.4 Thermal boundary conditions (Backward-facing step, full-size channel model)	8
2.3.5 Fully-developed flow hydrodynamic boundary conditions (Ribbed geometry)	8
2.3.6 Fully-developed flow thermal boundary conditions (Ribbed geometry)	9
2.3.7 Hydrodynamic boundary conditions (Converging duct geometry)	9
2.3.8 Thermal boundary conditions (Converging duct geometry)	10
2.3.9 Hydrodynamic boundary conditions (Axisymmetric duct geometry)	10
2.3.10 Thermal boundary conditions (Axisymmetric duct geometry)	11
2.3.11 Turbulent flow boundary conditions (All geometries)	11
2.3.12 Universal near-wall approximations of velocity components (All models)	12
2.3.13 Universal near-wall approximations of temperature (All models)	13
3. DEVELOPMENT OF TURBULENT EQUATIONS OF MOTION	14
3.1 The governing equations of flow	14
3.2 Modeling the near-wall viscosity-affected layers	18
3.2.1 Theoretical Development	18
3.2.2 Problems with Low Reynolds Number Simulations	20
3.3 Discussion of the implementation of the finite element model and solution procedure	22
4. DISCUSSION OF RESULTS	26
4.1 Results and discussion of Backward-facing step geometry	26
4.1.1 Mesh sensitivity study	28
4.1.2 Results	31
4.1.3 Conclusions	39
4.2 Two-dimensional converging channel	40
4.2.1 Mesh sensitivity study	41
4.2.2 Results	45
4.2.3 Conclusions	50
4.3 Two-dimensional ribbed geometry	51

4.3.1 Results from the $Re = 103,775.5$ simulation.....	54
4.3.2 Simulation with Reynolds number = 51,887.75	58
4.3.3 Conclusions	61
4.4 Axisymmetric ribbed geometry	62
4.4.1 Results	65
4.4.2 Conclusions	70
5. CONCLUSIONS	71
LIST OF REFERENCES	74
VITA	77

LIST OF TABLES

TABLE	PAGE
1. Empirical Turbulent Constants.....	24
2. Relaxation Factors.....	25
3. Mesh sensitivity study of the full-size channel model (Case B).....	29
4. Results of the simulation with turbulence intensity of 1.0%.....	30
5. Properties and Parameters used for cases A and B of this configuration.....	31
6. Steady-state instantaneous velocities at $x/H = 0.75$ downstream of the step.....	31
7. Steady-state turbulence intensities at $x/H = 0.75$ downstream of the step.....	32
8. Steady-state boundary layer thickness δ_s/H at the step location.....	33
9. Steady-state reattachment lengths X_r/H	33
10. Steady-state heat flux measurements at three positions.....	33
11. Nusselt numbers (Nu_s) at the step location.....	34
12. Peak values of Nu/Nu_{step} near the reattachment point and locations.....	35
13. Turbulent properties range.....	38
14. y^+ ranges from inlet to outlet corresponding to wall element thicknesses.....	44
15. Parameters used consistently with this configuration.....	44
16. Friction coefficient at $X/H = 16.3$ and $Re = 21,000$	47
17. Friction coefficients for fully-developed laminar and turbulent channel flows.....	47
18. Stanton number at $X/H = 16.3$ and $Re = 21,000$	48
19. Turbulent flow structure characteristics.....	49
20. Properties and Parameters used in this configuration.....	53
21. Properties and parameters used for $Re = 103,775.5$ simulations.....	54
22. Mesh sensitivity study.....	54
23. Steady-state fully-developed reattachment lengths X_r/H	55
24. Structure upstream of the step at the module exit.....	55
25. Average Nusselt number for a smooth duct.....	55
26. Comparison of simulation results to experimental data.....	56
27. Smooth duct friction factor from Blasius Equation.....	56
28. Heat transfer and friction factor trends with Reynolds number.....	56
29. Turbulent characteristics of the flow field.....	58
30. Properties and boundary conditions for lower Reynolds number simulation.....	59
31. Average Nusselt number for a smooth duct.....	59
32. Heat transfer and friction factor trends with Reynolds number.....	59
33. Smooth duct friction factor from Blasius Equation.....	60
34. Input parameters used in the simulation and measured values.....	64
35. Properties used in the simulation.....	64
36. Comparison of simulation and experimental results.....	66
37. Location of circulation cells upstream and downstream of the rib.....	66
38. Turbulence properties range for the present study.....	66

LIST OF FIGURES

FIGURE	PAGE
1. Geometry used by Tsou, Chen and Aung (1991) (Case B).....	26
2. Symmetric channel geometry (Case A).....	28
3. Mesh sensitivity study.....	29
4. Velocity profile at $x/H = 0.75$	35
5. Turbulence intensity profile at $x/H = 0.75$	36
6. Nusselt number profile downstream of the step.....	37
7. Nusselt number profile downstream of the step.....	37
8. Streamlines in the vicinity of the backward-facing step.....	38
9. Isotherms in the vicinity of the backward-facing step.....	38
10. Geometry used by Mutama and Iacovides (1993).....	40
11. Variation in velocity distribution with wall element thickness at $X/H = 16.3$	42
12. Variation in turbulence intensity with wall element thickness at $X/H = 16.3$	43
13. Variation in temperature profile with wall element thickness at $X/H = 16.3$	43
14. Mean velocity field at $X/H = 16.3$ and $Re = 21,000$	45
15. Normalized turbulence intensity profile at $X/H = 16.3$ and $Re = 21,000$	46
16. Temperature profile at $X/H = 16.3$ and $Re = 21,000$	46
17. Centerline turbulence distribution from inlet to outlet.....	49
18. Streamlines at $X/H = 16.3$	49
19. Isotherms at $X/H = 16.3$	50
20. Geometry used by Liou and Hwang (1992).....	51
21. Centerline pressure coefficient from simulation with 16,000 nodes and Reynolds number = 103,775.5.....	57
22. Average Nusselt number distribution within one module.....	57
23. Average Nusselt numbers along the walls for Reynolds number = 51,887.7.....	60
24. Streamlines in the fully-developed module.....	60
25. Isotherms in the fully-developed module.....	61
26. Geometry used by Vilemas, Cesna and Survila (1987).....	62
27. Predicted and measured heat transfer comparison for $I = 1.0\%$	67
28. Predicted and measured heat transfer comparison for $I = 1.5\%$	68
29. Predicted and measured heat transfer comparison for $I = 2.0\%$	68
30. Model boundary conditions and measured wall temperatures.....	69
31. Streamlines in the vicinity of the rib.....	69
32. Isotherms in the vicinity of the rib.....	70

LIST OF SYMBOLS AND NOMENCLATURE

SYMBOLS

U_{inlet}	uniform inlet velocity
u_*	friction velocity
u_i	instantaneous velocity
u'_i	fluctuating component of the instantaneous velocity
$\bar{u}_i$	mean component of the instantaneous velocity
u_∞	average inlet velocity
u^+	near-wall velocity profile
U_B	bulk velocity
T	temperature
T_{wall}	wall temperature
$T_{\text{wall,inlet}}$	wall temperature at the inlet
T^+	near-wall temperature function
ΔT	temperature difference
T'	fluctuating component of temperature
$\bar{T}$	mean component of temperature
T_m	mean temperature
T_B	bulk temperature
ρ	density
μ	laminar viscosity
p	pressure
c_p	constant pressure specific heat
λ	fluid thermal conductivity
k	fluid thermal conductivity
Re	Reynolds number
Pr	Prandtl number

St	Stanton number
C_f	friction coefficient
$\overline{Nu_p}$	average periodic Nusselt number
$\overline{Nu_s}$	average periodic step Nusselt number
Nu_x	local Nusselt number
Nu_s	step height Nusselt number
$\dot{q}_w''$	local wall heat flux
q_i	thermal energy flux
H	heat generation term
μ_t	eddy-viscosity
ϵ	turbulent dissipation rate
k	turbulent kinetic energy
I	free-stream turbulence intensity
y^+	non-dimensional wall distance
ℓ_m	mixing length
c_1	empirical constant
c_2	empirical constant
c_3	empirical constant
c_μ	empirical constant
σ_k	diffusion constant
σ_ϵ	diffusion constant
κ	von Karman constant
A	van Driest's constant
σ_t	turbulent Prandtl number
E	empirical constant
R_μ	ratio of eddy-viscosity to laminar viscosity
D_T	variable in the near-wall temperature calculation

P_T	variable in the near-wall temperature calculation
R_T	variable in the near-wall temperature calculation
σ_{ij}	stress tensor
f_i	body force
δ_{ij}	Kronecker delta
τ_{ij}	deviatoric stress tensor
ϵ_{ij}	shear rate tensor
k_{ij}	thermal conductivity tensor
H	step height
δ_s	boundary layer thickness at the step
X_r	reattachment length
η	normal direction
k	acceleration factor
α	convergence angle

SUBSCRIPTS

avg	average	r	reattachment
x	local value	∞	free-stream
inlet	inlet value	x	x-direction
y	y-direction	z	z-direction
θ	circumferential direction	m	mean
B	bulk	wall	value at the wall
w	value at the wall	s	value at the step
s	smooth duct	p	periodicity
infinity	free-stream	t	turbulent
0	standard reference state	μ	relating to viscosity

1. Introduction

The fluid flow and heat transfer within rectangular and annular geometries is important in heat exchanger design. The most common example is parallel or concentric pipes with two fluids of different temperature in counter-flow. The process is dependent on many parameters such as surface area, pipe material and fluid Reynolds numbers. For the purpose of designing compact heat exchangers, many of these parameters are varied to obtain the maximum heat transfer in the minimum space necessary. An effective method for increasing the overall heat transfer coefficient associated with this geometry is to add ribs to the surface with low heat transfer. This increases surface area and turbulence in the mixing process.

For low Reynolds number laminar flow, the heat transfer in a smooth pipe can be poor. The addition of axial fins can cause significant changes in the heat transfer and flow characteristics. In certain cases the presence of a certain geometry of fins can decrease the heat transfer below that of a smooth pipe while increasing pumping power, as shown by Rowley and Patankar (1984). Other fin geometries have been shown to effectively increase heat transfer by causing small recirculation cells and reattachment. Further, for high Reynolds number turbulent flow, axial fins increase the mixing process by continually tripping the formation of the boundary layer. The maximum heat transfer is recorded near the reattachment point beyond each step, as indicated by Aung and Goldstein (1971) and Vogel and Eaton (1985).

1.1 Literature Review

The developing and fully-developed laminar flow characteristics and heat transfer for ribbed concentric annuli have been studied extensively, both experimentally and numerically. Many researchers including Nguyen, Khodadadi, and Vlachos (1989) experimentally investigated heat transfer from ribbed tubes for fully-developed flow at low Reynolds numbers. Rowley and

Patankar (1984) numerically investigated two-dimensional fully-developed laminar flow. By taking advantage of the periodic nature of the flow, they were able to model a single rib-pitch with symmetrical velocity boundaries and an applied pressure gradient. They discovered that for low Prandtl number fluids and a Reynolds number of 100, the heat transfer for the ribbed geometry was less than that of an unfinned tube. Stevenson and Arimilli (1994) numerically investigated low Reynolds number developing flow and heat transfer from finned tubes using an axisymmetric analysis. They modeled an annulus with sufficient length to capture both the hydraulically and thermally developing regions as well as the fully-developed zone. Their findings showed that, relative to a smooth tube, the presence of ribs decreases the heat transfer performance for low Prandtl number fluids and significantly improves it for high Prandtl numbers. They also discovered that placement of ribs in the developing region decreased the heat transfer below that of the unfinned tube for all Prandtl numbers.

Turbulent flow characteristics and heat transfer have been studied experimentally for many geometries. Durst and Armaly (1983) measured fully-developed laminar, transitional and turbulent flow over a backward-facing step. Their Reynolds number range varied from 70 to 8,000. They found that several recirculation cells existed in addition to the primary cell. These cells were located at the top of the channel (i.e., the wall facing the step) and downstream of the reattachment point. They also reported the spatial variation of these cells and the reattachment point within the Reynolds number range. Chen and Aung (1991) performed experiments with turbulent flow over a backward-facing step. Their experiments involved the transient flow development characteristics and heat transfer. They used a high Reynolds number of 38,800 based on the step height and constant wall temperature. Steady-state conditions occurred after 7 seconds, and their measurements showed peak turbulence intensities after the step at one step height above the wall. These results agreed with the findings of Moss and Baker (1980) and Vogel and Eaton (1985).

The peak heat transfer occurred near the reattachment point after the step, as pointed out by Aung and Goldstein (1971) and Vogel and Eaton (1985). Liou and Hwang (1992) experimentally studied developing flow in a two-dimensional rectangular duct with baffles. The Reynolds number range they studied was 5,000 to 50,000 based on hydraulic diameter. Their duct was long enough to obtain fully-developed conditions. The centerline pressure distribution was measured to show a periodically repeating flow structure in the fully-developed regime. Their measurements of heat transfer revealed a peak after each rib, indicating a recirculation zone. They also studied the effects of Nusselt number and friction coefficient with increasing Reynolds numbers which they adapted into correlations. Fully-developed turbulent flow characteristics and heat transfer for an axisymmetric annular geometry was studied experimentally by Vilemas, Cesna, and Survila (1987). They measured the effects on the heat transfer from both the inner and outer tube with the addition of a sharp ring on the inner tube. Their experiments covered a variation of the applied temperature distribution on the tubes, Reynolds numbers and tube geometries. Experimental results showed the presence of recirculation zones and peak heat transfers.

There is a balance between laminar flow experimentation and numerical simulation in the literature. However, the bulk of turbulent flow research is strictly experimental. This study is directed toward using the new developments in turbulent flow models, numerical methods and computing resources to fill the absence of simulations in the literature. The numerical simulations can augment the effectiveness and decrease the cost of experiments. In addition, the use of modeling enlightens the experimentalist as to what measurements are important to acquire.

2. Problem Statement

2.1 Problem Description

In this investigation four configurations are modeled, and they include a diverse range of geometry and flow structure. The details of the geometry are shown in Chapter 4. Developing and fully-developed flow conditions with different levels of turbulence intensity are studied. Constant and variable temperature wall conditions and constant heat flux conditions are examined. The effects on the predictions of constant and variable properties is observed. Low and high Reynolds number flows are used and the relaminarization phenomenon is studied. The geometry and the coordinate system for each model are presented in the discussion of results for each case. Two-dimensional and axisymmetric forms of the governing equations are used in the different cases. The choice of configurations to model are diverse in geometry and flow structure and are limited by the availability of good measurements.

The following general assumptions and justifications are made:

1. Turbulent flow (Characteristic of high Reynolds numbers)
2. Newtonian and incompressible fluid (The fluid is air, and the Mach number is less than 0.3)
3. Constant fluid properties are used when the temperature differences are small
4. Negligible viscous dissipation (Characteristic of small Brinkman numbers)
5. Steady-state conditions
6. Zero-gradient thermal and hydraulic conditions at outlets (Characteristic of high Peclet numbers)
7. Buoyancy terms are eliminated due to small Grashof numbers
8. No-slip velocity at the walls

Variable properties with temperature are also incorporated into the models as a comparison to the constant property assumption. The inlet turbulent kinetic energy and dissipation

rates are calculated based on measured inlet turbulence intensity, using Equations (2.7) and (2.8).

Non-dimensional distance from the walls (y^+) is maintained between 30 and 100, as suggested by FIDAP for accurate predictions.

2.2 Governing Equations

The time-averaged mean flow equations are derived in detail in the next chapter in their two-dimensional form. The final approximate equations are presented here in the form from which the discretized versions are derived. As presented in the FIDAP Theory Manual with buoyancy and viscous dissipation terms discarded:

Conservation of Mass:

$$\bar{u}_{j,j} = 0 \quad (2.1)$$

Conservation of Linear Momentum:

$$\rho_0 \left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \bar{u}_{i,j} \right) = -\bar{p}_{,i} + \rho_0 f_i + \left[\mu (\bar{u}_{i,j} + \bar{u}_{j,i}) \right]_{,j} \quad (2.2)$$

Conservation of Energy:

$$\rho_0 c_p \left(\frac{\partial \bar{T}}{\partial t} + \bar{u}_j \bar{T}_{,j} \right) = (\lambda \bar{T}_{,j})_{,j} \quad (2.3)$$

Cartesian form of Turbulent Kinetic Energy:

$$\rho_0 \left(\frac{\partial k}{\partial t} + \bar{u}_j k_{,j} \right) = \left(\mu_0 + \frac{\mu_t}{\sigma_k} k_{,j} \right)_{,j} - \rho_0 \epsilon \quad (2.4)$$

Cartesian form of Turbulent Kinetic Energy Dissipation Rate:

$$\rho_0 \left(\frac{\partial \epsilon}{\partial t} + \bar{u}_j \epsilon_{,j} \right) = \left(\mu_0 + \frac{\mu_t}{\sigma_\epsilon} \epsilon_{,j} \right)_{,j} - \rho_0 c_2 \frac{\epsilon^2}{k} \quad (2.5)$$

Where k is the value of the turbulent kinetic energy in Equations (2.4) and (2.5), and λ represents the thermal conductivity in Equation (2.3).

2.3 Boundary Conditions

In addition to the general assumptions listed in section 2.1 that are applied to all of the simulations, the following boundary conditions measured in the experiment are imposed on the specific configurations.

2.3.1 Hydrodynamic boundary conditions (Backward-facing step symmetric top model)

1. The two-dimensional developing flow regime is modeled with a supplied constant inlet x-component of velocity (Dirichlet boundary condition) and a zero y-component

$$(u_y = 0, u_z = 0, u_x = U_{inlet}).$$

2. A zero-gradient (Neumann boundary condition) outlet is assumed because the fully-developed velocity profile should have recovered by the outlet location. This is done in the case of the backward-facing step because the outlet is 40 step heights downstream, and there is an absence of experimental data at the outlet ($u_y = 0, u_z = 0, \frac{\partial u_x}{\partial x} = 0$).

3. The velocity in the z-direction is assumed to be zero.

4. The walls are impermeable and have no-slip conditions ($u_y = 0, u_z = 0, u_x = 0$).

5. The symmetric surface is located three step heights above the flat plate and has a zero y-component of velocity and a zero-gradient x-component ($u_y = 0, u_z = 0, \frac{\partial u_x}{\partial y} = 0$). This is done

because the velocity boundary layer does not grow beyond 3.5 step heights above the plate.

6. Zero-gradient turbulence parameters are placed on the symmetric surface.

7. Constant inlet turbulence parameters are imposed on the model.

8. Universal near-wall profiles for flow and turbulence represent the viscous sublayer and buffer transition layer, as discussed in 2.3.11.

2.3.2 Thermal boundary conditions (Backward-facing step symmetric top model)

1. The inlet and wall temperatures are specified with a 17 degree Celsius difference $(T_{inlet} - T_{walls}) = 17^\circ$. This is done by Chen and Aung (1991) to minimize the variable property effect in their experiment.
2. The lower wall has a constant temperature.
3. The inlet fluid temperature is constant at room temperature.
4. The outlet has a zero-gradient temperature condition because it is far enough downstream from the step, and measurements are not available $\frac{\partial T}{\partial x} = 0$. This boundary condition is shown by Patankar (1980) to be valid for flows with high Peclet numbers.
5. The symmetric surface has a zero-gradient temperature condition because it is outside of the thermal boundary layer $\frac{\partial T}{\partial y} = 0$.

2.3.3 Hydrodynamic boundary conditions (Backward-facing step, full-size channel model)

1. The two-dimensional developing flow regime is modeled with a supplied constant inlet x-component of velocity (Dirichlet boundary condition) and a zero y-component $(u_y = 0, u_z = 0, u_x = U_{inlet})$.
2. A zero-gradient (Neumann boundary condition) outlet is assumed because the fully-developed velocity profile should have recovered by the outlet location. This is done in the case of the backward-facing step because the outlet is 135 step heights downstream, and no data is available for the outlet $(u_y = 0, u_z = 0, \frac{\partial u_x}{\partial x} = 0)$.

3. The velocity in the z-direction is assumed to be zero.
4. The walls are impermeable and have no-slip conditions ($u_y = 0, u_z = 0, u_x = 0$).
5. Constant inlet turbulence parameters are imposed on the model.
6. Universal near-wall profiles for flow and turbulence represent the viscous sublayer and buffer transition layer as discussed in 2.3.11.

2.3.4 Thermal boundary conditions (Backward-facing step, full-size channel model)

1. The inlet and wall temperatures are specified with a 17 degree Celsius difference ($T_{inlet} - T_{walls} = 17^\circ$). This is done to minimize the variable property effect.
2. The lower wall has a constant temperature.
3. The inlet fluid temperature is constant at room temperature.
4. The outlet has a zero-gradient temperature condition because it is far enough downstream from the step, and there is no data for the outlet $\frac{\partial T}{\partial x} = 0$. This boundary condition is shown by Patankar (1980) to be valid for flows with high Peclet numbers.

2.3.5 Fully-developed hydrodynamic boundary conditions (Ribbed geometry)

1. The fully-developed profiles of k , ϵ , pressure, and the x and y components of velocity are found by placing constant values of these variables ($u_y = 0, u_z = 0, u_x = U_{inlet}$) on a beginning module and allowing the flow to develop. This is implemented by modeling only one module with the constant values at the inlet and Neumann boundary conditions for the turbulence variables and velocity at the outlet ($u_y = 0, u_z = 0, \frac{\partial u_x}{\partial x} = 0$). The resulting solution for the outlet flow structure is used as inlet boundary conditions for the next module to be modeled. Following this procedure for several modules, the inviscid core disappears, and the solution for the

outlet profiles become identical to the imposed inlet conditions. A periodically decreasing pressure profile along the centerline of the channel is recorded, as pointed out by Liou and Hwang (1992).

2. The walls are impermeable and have no-slip conditions applied to them

$$(u_y = 0, u_z = 0, u_x = 0).$$

3. In the case of the two-dimensional simulations, the z-component of velocity is assumed to be zero.

4. Universal near-wall profiles for flow and turbulence represent the viscous sublayer and buffer transition layer, as discussed in 2.3.11.

2.3.6 Fully-developed flow thermal boundary conditions (Ribbed geometry)

1. The inlet temperature profile for fully-developed modules is determined in the same way as the flow variables.

2. The outlet of each module is given a Neumann boundary condition for temperature

which is consistent with high Peclet number flows ($\frac{\partial T}{\partial x} = 0$).

3. The wall temperature must vary linearly within the rib-pitch in the flow direction

($T_{walls}(x) = ax + T_{wall,inlet}$). This temperature boundary condition follows a periodic nature from rib module to module identified by Rowley and Patankar (1984).

2.3.7 Hydrodynamic boundary conditions (Converging duct geometry)

1. The two-dimensional developing flow regime is modeled with a supplied constant inlet x-component of velocity (Dirichlet boundary condition) and a zero y-component

$$(u_y = 0, u_z = 0, u_x = U_{inlet}).$$

2. A zero-gradient (Neumann boundary condition) outlet is assumed because the test duct

exits to the atmosphere, and there is no experimental data available ($u_y = 0, u_z = 0, \frac{\partial u_x}{\partial x} = 0$).

3. The velocity in the z-direction is assumed to be zero.
4. The walls are impermeable and have no-slip conditions ($u_y = 0, u_z = 0, u_x = 0$).
5. Constant inlet turbulence parameters are imposed on the model.
6. Universal near-wall profiles for flow and turbulence represent the viscous sublayer and buffer transition layer as discussed in 2.3.11.

2.3.8 Thermal boundary conditions (Converging duct geometry)

1. The inlet fluid temperature is supplied at room temperature ($T_{inlet} = Constant$).
2. The bottom wall is supplied with a constant heat flux ($Q''_{wall} = Constant$).
3. The top wall is insulated.
4. The outlet has a zero-gradient temperature condition, due to the absence of data, but is consistent with high Peclet number flows ($\frac{\partial T}{\partial x} = 0$).

2.3.9 Hydrodynamic boundary conditions (Axisymmetric geometry)

1. The axisymmetric developing flow regime is modeled with a supplied constant inlet z-component of velocity (Dirichlet boundary condition) and a zero r-component ($u_r = 0, u_\theta = 0, u_z = U_{inlet}$) far upstream from the beginning of heating.
2. A zero-gradient (Neumann boundary condition) outlet is assumed because the test duct outlet is many rib heights downstream, and there is no outlet data ($u_r = 0, u_\theta = 0, \frac{\partial u_z}{\partial z} = 0$).
3. The velocity in the θ -direction is assumed to be zero.
4. The walls are impermeable and have no-slip conditions ($u_r = 0, u_\theta = 0, u_z = 0$).

5. Universal near-wall profiles for flow and turbulence represent the viscous sublayer and buffer transition layer, as discussed in 2.3.11.

2.3.10 Thermal boundary conditions (Axisymmetric geometry)

1. The inlet fluid temperature is supplied at room temperature ($T_{inlet} = Constant$).
2. The inner tube wall has the measured temperature distribution from the experiment.
3. The outlet has a zero-gradient boundary condition due to the absence of outlet data, but

is consistent with high Peclet number flows ($\frac{\partial T}{\partial z} = 0$).

4. The outer tube wall is held constant at room temperature.

2.3.11 Turbulent flow boundary conditions (All geometries considered)

The inlet boundary conditions must include the turbulent kinetic energy (k) and dissipation rate (ϵ) profiles. These profiles are determined by experimental measurements of the turbulent intensity in the inviscid core entering the channel. This is accomplished by hot-wire measurements of the fluctuations in the flow. The free-stream fluctuations are very low but are of most importance because that value of the intensity is representative of the inlet intensity, since the inlet boundary layers are thin. From the measured fluctuations, the turbulent kinetic energy is calculated from Equation (2.7), as described in the FIDAP Tutorial Manual:

$$I = \left(\overline{u'^2} \right)^{1/2} / u_{\infty} \quad (2.6)$$

$$k = 1.5(Iu_{\infty})^2 \quad (2.7)$$

The value of u_{∞} used as the inlet average velocity. This analysis is valid for shear-free situations occurring only in the free-stream inviscid core of the inlet. The characteristic value of dissipation rate is found from the kinetic energy as shown:

$$\varepsilon = \rho c_\mu \frac{k^2}{R_\mu \mu} \quad (2.8)$$

$R_\mu = \mu_t / \mu$ is the ratio of eddy-viscosity to laminar viscosity and is assumed as 10 for most low turbulence flows. c_μ is an empirical constant determined from shear layer flow experiments and has the value of 0.09. The values of kinetic energy and dissipation rate for the viscosity-affected near-wall areas are not calculated within the viscous sublayer and transitional layer. Instead, Van Driest's mixing length approach is used to model the variation in turbulent diffusivities of momentum and heat. This is done by the approximation of the turbulent viscosity, as shown by Equations (3.39) and (3.40). The boundary conditions for the k and ε equations are given in Equations (2.9) and (2.10) and are imposed at the top of the wall elements.

$$\varepsilon = \frac{\left(c_\mu^{1/2} k \right)^{1.5}}{\kappa \delta} \quad (2.9)$$

$$\frac{\partial k}{\partial n} = 0 \quad (2.10)$$

Neumann boundary conditions are applied to the outlet for both of these values. Non-zero initial conditions for both the kinetic energy and dissipation rate must be specified for the entire flow field. This value is the same as the inlet boundary conditions. This is done for the steady-state problem to improve the convergence characteristics of the simulation.

2.3.12 Universal near-wall approximations of velocity components (All models)

Reichardt's law is used to calculate the velocity profile within the wall elements and is shown in Equation (3.33). As shown, the velocity is a function of non-dimensional wall distance (y^+). The y^+ is also a local Reynolds number. A value of y^+ less than 5 indicates laminar flow conditions and a linear velocity distribution. For a y^+ range of 5 to 30, the flow is in transition

and has a logarithmic velocity distribution. A y^+ greater than 30 indicates the fully turbulent regime where the time-averaged Navier-Stokes and k - ϵ equations are solved. Reichardt's law adequately represents the velocity distributions in the sublayer and buffer layer and into the turbulent regime. The wall elements must enclose these layers, and the resulting y^+ should be between 30 and 100.

2.3.13 Universal near-wall approximations of temperature (All models)

The temperature distribution in the sublayers is a function of y^+ , the turbulent Prandtl number and the Schmidt number, as shown in Equation (3.34). Within the wall elements, the temperature profile is calculated from Equations (3.35) - (3.38).

3. The Development of the Turbulent Equations of Motion

3.1 The governing equations of flow

The general form of the partial differential equations that govern fluid flow and heat transfer are shown in unsteady Cartesian form. The development of the time-averaged form of these equations is performed due to the impracticality of their direct numerical simulation. These equations reflect the assumption of constant fluid properties with temperature and pressure variations. This facilitates the uncoupling of their interdependence. As described in the FIDAP Theory Manual, the governing equations in tensor notation are shown in Equations (3.1) to (3.3).

$$\text{Conservation of Mass:} \quad \frac{\partial \rho}{\partial t} + (\rho u_j)_{,j} = 0 \quad (3.1)$$

$$\text{Conservation of Linear Momentum:} \quad \rho \left(\frac{\partial u_i}{\partial t} + u_j u_{i,j} \right) = \sigma_{ij,j} + \rho f_i \quad (3.2)$$

$$\text{Conservation of Energy:} \quad \rho c_p \left(\frac{\partial T}{\partial t} + u_i T_{,i} \right) = -q_{i,i} + H \quad (3.3)$$

The instantaneous velocity is represented by u_i where $i, j = 1, 2$ for two-dimensional and axisymmetric flows. The gravitational body force is given by ρf_i . The stress tensor is defined as:

$$\sigma_{ij} = -p\delta_{ij} + \tau_{ij} \quad (3.4)$$

For fluids with Newtonian and incompressible properties, the shear rate is related to the stress tensor as follows:

$$\epsilon_{ij} = \frac{1}{2} (u_{i,j} + u_{j,i}) \quad (3.5)$$

$$\tau_{ij} = 2\mu\epsilon_{ij} \quad (3.6)$$

As mentioned above, the general form of the governing equations does predict the motion of turbulence and can be used to model turbulent flow. The problem is that the scale of turbulent

spatial and temporal variations is so small that excessive meshing would be necessary, especially in the viscous wall areas. This would lead to unacceptably large computational time. By the Reynolds decomposition, the instantaneous fluid properties are expressed as an averaged and fluctuating value.

$$\eta = \bar{\eta} + \eta' \quad (3.7)$$

$$\text{where:} \quad \bar{\eta} = \frac{1}{\Delta t} \int_t^{t+\Delta t} \eta dt \quad (3.8)$$

The average value involves the instantaneous property divided by a large time scale to eliminate the turbulent variations. The time-averaged form of the governing equations becomes:

$$\text{Conservation of Mass:} \quad \bar{u}_{j,j} = 0 \quad (3.9)$$

Conservation of Linear Momentum:

$$\rho_0 \left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \bar{u}_{i,j} \right) = -\bar{p}_{,i} + \rho_0 f_i + \left[\mu (\bar{u}_{i,j} + \bar{u}_{j,i}) - \rho_0 \overline{u'_i u'_j} \right]_{,j} \quad (3.10)$$

Conservation of Energy:

$$\rho_0 c_p \left(\frac{\partial \bar{T}}{\partial t} + \bar{u}_j \bar{T}_{,j} \right) = \left(\lambda \bar{T}_{,j} - \rho_0 c_p \overline{u'_j T'} \right)_{,j} + H \quad (3.11)$$

These equations represent the mean flow variables $(\bar{u}_i, \bar{p}, \bar{T})$ with no approximations.

The averaging process introduces new unknowns into the equations. $\rho_0 \overline{u'_i u'_j}$ in the momentum equations are called the Reynolds stresses and physically represent turbulent transport by eddy behavior. $\rho_0 c_p \overline{u'_j T'}$ in the energy equation is the turbulent heat flux caused by turbulent eddy fluctuations. The lack of theoretical knowledge of these terms represents the problem with closure of the exact equations. The approximate correlation of these values used by FIDAP is based on the Boussinesq eddy-viscosity concept.

$$-\rho_0 \overline{u'_i u'_j} = \mu_t (\overline{u_{i,j}} + \overline{u_{j,i}}) \quad (3.12)$$

Boussinesq assumed that the Reynolds stresses are proportional to the mean velocity gradients as in laminar flow. μ_t is the eddy-viscosity and it is dependent on the turbulence of the flow. This is not a property of the fluid but of the entire flow field. Its determination comes from relating it to Prandtl's mixing length and eddy velocity.

$$\mu_t \propto \rho_0 u_t l_t \propto \frac{\rho_0 k^2}{\varepsilon} \quad (3.13)$$

$$\mu_t = c_\mu \frac{\rho_0 k^2}{\varepsilon} \quad (3.14)$$

It is more useful to express the eddy-viscosity as measured properties than with the mixing length. k is the turbulent kinetic energy contained in the fluid eddies and comes from measurements of turbulence intensity. ε is the dissipation rate.

$$I = \left(\overline{u'^2} \right)^{1/2} / u_\infty \quad (3.15)$$

$$k = 1.5 (I u_\infty)^2 \quad (3.16)$$

The turbulence intensity (I) is measured in the inviscid core of a confined flow and is used to define the kinetic energy profile for the inlet. Once the kinetic energy at the inlet plane is determined from measurement using (3.16), the viscous dissipation rate of turbulent kinetic energy can be determined from (3.17).

$$\varepsilon = \rho c_\mu \frac{k^2}{R_\mu \mu} \quad (3.17)$$

$R_\mu = \mu_t / \mu$ is the ratio of eddy-viscosity to laminar viscosity and is assumed to be 10 for most low turbulence flows. c_μ is an empirical constant determined from shear layer flow

experiments and has the value of 0.09. The resulting fluid viscosity is based on the sum of the constant laminar viscosity and the eddy-viscosity.

$$\mu = \mu_0 + \mu_t \quad (3.18)$$

The turbulent heat flux introduced by the time-averaging process is eliminated by the formation of a turbulent thermal conductivity.

$$\rho C_p \overline{u'_j T'} = \lambda_t \overline{T_{,j}} \quad (3.19)$$

$$\lambda_t = \frac{c_p \mu_t}{\sigma_t} \quad (3.20)$$

The value of the turbulent Prandtl number σ_t can be found either by empirical data or numerical experimentation and in this case is 0.9. As in the case of viscosity, the resulting fluid thermal conductivity is the combination of the laminar and turbulent values.

$$\lambda = \lambda_0 + \lambda_t \quad (3.21)$$

λ is used to represent the thermal conductivity because k represents the turbulent kinetic energy in the following equations.

Two equations based on k and ϵ can be determined from the Navier-Stokes equations. The introduction of some simplifying assumptions and empirical constants results in the two-equation k - ϵ model developed by Jones and Launder, as shown in (3.22) and (3.23).

$$\rho_0 \left(\frac{\partial k}{\partial t} + \bar{u}_j k_{,j} \right) = \left(\mu_0 + \frac{\mu_t}{\sigma_k} k_{,j} \right)_{,j} + \mu_t \Phi - \rho_0 \epsilon \quad (3.22)$$

$$\rho_0 \left(\frac{\partial \epsilon}{\partial t} + \bar{u}_j \epsilon_{,j} \right) = \left(\mu_0 + \frac{\mu_t}{\sigma_\epsilon} \epsilon_{,j} \right)_{,j} + c_1 \frac{\epsilon}{k} \mu_t \Phi - \rho_0 c_2 \frac{\epsilon^2}{k} \quad (3.23)$$

For this model the empirical constants are $\sigma_k = 1.00$, $\sigma_\epsilon = 1.30$, $c_1 = 1.44$ and $c_2 =$

1.92. The final form of the governing equations for turbulent transport take the form:

Conservation of Mass: $\bar{u}_{j,j} = 0$ (3.24)

Conservation of Linear Momentum:

$$\rho_0 \left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \bar{u}_{i,j} \right) = -\bar{p}_{,i} + \rho_0 f_i + \left[\mu (\bar{u}_{i,j} + \bar{u}_{j,i}) \right]_{,j} \quad (3.25)$$

Conservation of Energy:

$$\rho_0 c_p \left(\frac{\partial \bar{T}}{\partial t} + \bar{u}_j \bar{T}_{,j} \right) = (\lambda \bar{T}_{,j})_{,j} \quad (3.26)$$

Turbulent Kinetic Energy:

$$\rho_0 \left(\frac{\partial k}{\partial t} + \bar{u}_j k_{,j} \right) = \left(\mu_0 + \frac{\mu_t}{\sigma_k} k_{,j} \right)_{,j} - \rho_0 \epsilon \quad (3.27)$$

Turbulent Kinetic Energy Dissipation Rate:

$$\rho_0 \left(\frac{\partial \epsilon}{\partial t} + \bar{u}_j \epsilon_{,j} \right) = \left(\mu_0 + \frac{\mu_t}{\sigma_\epsilon} \epsilon_{,j} \right)_{,j} - \rho_0 c_2 \frac{\epsilon^2}{k} \quad (3.28)$$

3.2 Modeling the near-wall viscosity-affected layers

3.2.1 Theoretical Development

Modeling the viscosity-affected near-wall regions of turbulent flow requires special attention. The viscous sublayer and buffer layer, as observed from experiments, have extremely sharp variations in flow variables. To capture these variations properly, an extremely fine grid would be required at the solid boundaries. Another concern is that the k - ϵ model is only valid for high Reynolds number flows and cannot be used at the wall where low Reynolds numbers occur. Both of these considerations are taken into account by FIDAP. The k - ϵ equations, the mean flow equations and the energy equation are not solved in these two layers. Instead, some universal near-wall profiles based on empirical data are used to acquire the flow structure and temperature profiles. The velocity wall law states that the fully turbulent region of the fluid begins where the

cell Reynolds number (y^+) becomes greater than 30. Between 5 and 30 is the transition layer, and under 5 is the viscous sublayer. As shown in the FIDAP Theory Manual:

$$y^+ = \frac{\rho u_* \delta}{\mu} \quad (3.29)$$

$$u^+ = \frac{u - u_w}{u_*} = f_u(y^+) \quad (3.30)$$

u^+ is the non-dimensional velocity profile within the wall elements, $u_* = (\tau_w / \rho)^{1/2}$ is the friction velocity, and δ is the normal distance from the wall. Within the viscous sublayer, the linear velocity wall law prevails.

$$u^+ = y^+ ; y^+ < 5 \quad (3.31)$$

Within the fully turbulent region, the logarithmic velocity wall law prevails.

$$u^+ = \frac{1}{\kappa} \ln(Ey^+) ; y^+ > 30 \quad (3.32)$$

$\kappa = 0.41$ is the von Karman constant, and $E = 9.0$ for aerodynamically smooth walls.

Between the wall and the turbulent regime ($0 \leq y^+ \leq 30$), Reichardt's Law is used to approximate the velocity distribution.

$$u^+ = f_R(y^+) = \frac{1}{\kappa} \ln(1 + \kappa y^+) + 7.8 \left[1 - \exp\left(-\frac{y^+}{11}\right) - \frac{y^+}{11} \exp(-0.33y^+) \right] \quad (3.33)$$

Reichardt's Law is shown by FIDAP to closely match Equations (3.31) and (3.32) in the sublayer and buffer layer and extend into the turbulent region. The temperature variation is also a function of the cell Reynolds number, turbulent Prandtl number and Schmidt number. The universal temperature profile for the wall elements is calculated from Equations (3.34) through (3.38).

$$T^+ = \frac{\rho c_p (T_w - T) \mu_w}{q_w} = f_T(y^+, Pr, \sigma_t) \quad (3.34)$$

$$T^+ = f_T(y^+, Pr, \sigma_t) = Pr u^+ D_T + \sigma_t (1 - D_T) (u^+ + P_T) \quad (3.35)$$

$$D_T = \exp\left(-\frac{y^+}{11} R_T^{0.333}\right) \quad (3.36)$$

$$R_T = \frac{Pr}{\sigma_t} \quad (3.37)$$

$$P_T = 9.24(R_T^{0.75} - 1) \left[1 + 0.28 \exp(-0.007 R_T)\right] \quad (3.38)$$

The above equations model the variation of the velocity and temperature within the wall region. A method of determining the turbulent viscosity in this region must be used because the k - ϵ equations are not solved inside the wall elements. Inside the wall elements, this variation is modeled using van Driest's mixing length approach for smooth walls, as shown in Equations (3.39) and (3.40).

$$\mu_t = \rho l_m^2 \left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \right]^{1/2} \quad (3.39)$$

$$l_m = \kappa \delta \left(1 - \exp(-y^+ / A) \right) \quad (3.40)$$

where $A = 26$ is van Driest's constant. The empirical turbulent constants used with the k - ϵ model are listed in section 3.3.

3.2.2 Problems with Low Reynolds Number Modeling

A fundamental problem with turbulence modeling is the low Reynolds number effect on high Reynolds number k - ϵ models. A special test case in the 1981 Stanford Conference was a turbulent flow which undergoes a reverse transition to laminar flow, as discussed by White (1991).

Many experiments have been performed to study this flow structure and the corresponding friction coefficients and heat transfer rates. Two flow situations are known to cause relaminarization. One is the convective acceleration of the turbulent boundary layer, and the other is an expansion of the turbulent flow. The convective acceleration of turbulent flow in a converging channel may not totally relaminarize the flow, but the friction coefficient and heat transfer rates approach that of a laminar flow. Launder (1964) suggested a dimensionless parameter for this phenomenon which is known as the acceleration parameter k and is defined as:

$$k = \frac{\tan\alpha}{Re} \quad (3.41)$$

This represents a two-dimensional converging duct of angle α . Launder (1964) stated that when the acceleration parameter is larger than 2.3×10^{-6} a two-dimensional turbulent flow will laminarize. This phenomenon was never modeled because of the low values of k and ϵ caused by the low Reynolds number transition.

In the simulations, a "Clipping Procedure" described in the FIDAP manuals is used in the calculation of the turbulent kinetic energy and dissipation. This method eliminates zero or negative values which could cause divergence because of their unrealistic physical nature. Low Reynolds number flow phenomena is susceptible to these unrealistically low values of k and ϵ because of the relaminarization effect. The procedure involves checking all values of turbulent kinetic energy, and if any fall below a minimum value they are set to that minimum value. The formula from the FIDAP manual defines that value as:

$$\text{lower bound} = \frac{\text{maximum } k \text{ in the flow domain}}{CLIP} \quad (3.42)$$

$$CLIP = 1.0 \times 10^7 \quad (3.43)$$

The second step is that all values of dissipation rate are checked, and if any are less than the computed minimum for that node, they are clipped and the value of ϵ set to:

$$\epsilon = \frac{\rho c_{\mu} k^2}{20\mu} \quad (3.44)$$

The lower bound for ϵ is defined as:

$$\text{lower bound} = \frac{\text{maximum } \epsilon \text{ in the flow domain}}{CLIP} \quad (3.45)$$

$$CLIP = 1.0 \times 10^7 \quad (3.46)$$

This procedure sets the effective viscosity to the value of the laminar viscosity which is very beneficial for near laminar simulations. This eliminates the turbulent viscosity where turbulence is low.

3.3 Discussion of the implementation of the finite element model and solution procedure

The Finite Element Method reduces the continuous equations of the flow domain into a finite number of elements defined by the meshing process. The elements used in all models in this study are quadrilaterals with a node defining each of the four corners. This type of element is chosen because of the ease in controlling the height of the layers and y^+ at the walls. An Eulerian inertial reference frame is used for each element. Each element has its own velocity and turbulence components, pressure and temperature profiles. The approximations within the element involve a bilinear interpolation function for the variables. The pressure degree of freedom is also located at the center of the element, indicating a discontinuous approximation across the element boundaries. Streamwise oscillations and flow noise are eliminated by the addition of Tensor viscosity/Streamline upwinding terms to the governing equations. The discretization and Galerkin approximation methods are described in detail in Chapter 3 of the FIDAP Theory Manual.

This simulation involves the steady-state form of the governing equations. The small temperature differences cause negligible changes in the fluid properties. This means that the momentum and turbulence equations are independent of temperature and can be solved for the pressure, velocity and turbulence fields without temperature effects being introduced. The energy equation can then be solved directly from the velocity and turbulence fields. This uncoupled solution procedure also eliminates the temperature dependent terms from the k and ϵ equations. These equations describing the physics of the flow are highly nonlinear and result in sparsely populated matrices that must be solved iteratively. The implicit solution algorithm, named the Segregated Solver, is used in these simulations.

The segregated solution method solves the momentum, pressure and energy equations in a sequential manner. After the components of velocity are found from the momentum equations, the velocity field is mass adjusted and the pressure field calculated. The energy equation is then solved for the temperature field. The convergence check is then performed. The convergence criterion involves the change in the solution vector from one iteration to the next for each degree of freedom.

$$\left\| \frac{\mathbf{U}_i - \mathbf{U}_{i-1}}{\mathbf{U}_i} \right\| \leq \text{Tolerance} \quad (3.47)$$

The tolerance used in these simulations is $\text{Tolerance} = 0.0001$.

The segregated algorithm is an implicit formulation specifically made to handle large problems. It does not formulate a global matrix containing all unknowns and degrees of freedom. Instead, many linear segregated matrices are formed, each containing only the nodal unknowns for one equation. These linear systems all involve non-symmetric coefficient matrices. The iterative methods are used for the small non-symmetric matrices, and the Conjugate Gradient Squared method is default. They store the global coefficient matrix in a sparse format. Preconditioning of

the matrices is also used to lower their condition number. This is done because the number of iterations varies with the square root of the condition number.

The method of calculating the pressure field is called the Pressure Projection Version. This procedure has seven steps. At the beginning of an iteration, the latest values of the variables are used to calculate an approximation to the pressure equation. Secondly, the pressure field is relaxed. Next, the other discretized equations are solved sequentially, and a pressure correction equation solved. Finally, the velocity field is mass-adjusted with the pressure correction to satisfy continuity at the end of each iteration.

The segregated algorithm initially applies heavy relaxation to the k and ϵ equations in order to stabilize the flow field before introducing the turbulent effects. This damping is applied for the first five iterations, and then the full effects of turbulence are introduced to the system. A spatially reduced integration method for the turbulent viscosity and source and sink terms in the equations is used. This method improves stability and convergence behavior by smoothing the turbulence field. The turbulent empirical constants used in the models is shown in Table 1. The relaxation factors used are presented in Table 2.

Table 1. Empirical Turbulent Constants

Empirical Turbulent Constant	Value used
C_μ	0.09
C_1	1.44
C_2	1.92
R_μ	10.0
σ_k	1.00
σ_ϵ	1.30
σ_t	0.90
A	26.0
E	9.00
κ	0.41

Table 2. Relaxation Factors

Degree of Freedom	Relaxation Factor
U_x	0.20
U_y	0.20
P	0.00
T	0.10
k	0.15
ϵ	0.15

4. Discussion of Results

4.1 Results and Discussion of Backward-Facing Step Simulation

Simulations of the backward-facing step geometry of Tsou, Chen and Aung (1991), as shown in Figure 1, are performed in this investigation. Their detailed measurements include the transient development of turbulent flow over the step caused by a sudden burst of high pressure air at the inlet of the channel. The simulations performed in this study only encompass the steady-state turbulent flow which they found to occur between 7 and 19 milliseconds. The initial air pressure they used to start the flow is 2 atmospheres, and a choked flow condition at the exit caused a shock wave which lowered the air temperature 16-18 degrees Celsius below the wall temperature. They found that the wall temperature did not vary more than 0.5 degrees Celsius. This is adequate for a constant wall temperature assumption. The step height is 3.3 millimeters, and the length of the flat plate before the step is 254 millimeters. This length is used to ensure that there would be a turbulent boundary layer at the step. The Reynolds number based on step height (Equation 4.6) is 38,800, and the measured free-stream turbulence intensity is 0.4%.

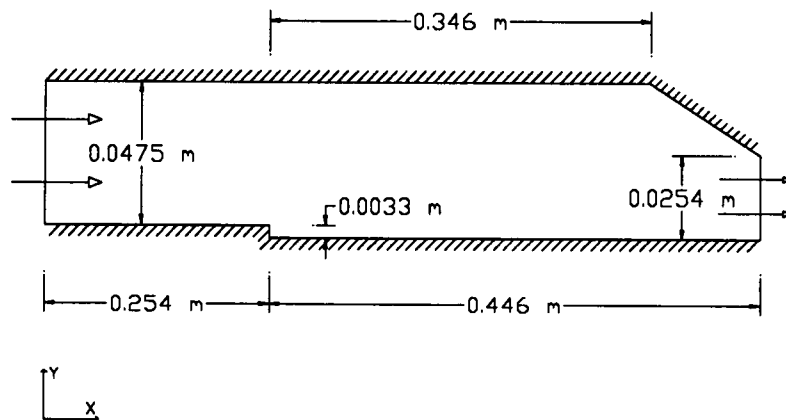


Figure 1. Geometry used by Tsou, Chen and Aung (1991) (Case B)

In addition to the general assumptions applied to all of the simulations, the following experimentally measured conditions are applied to this configuration:

1. High Reynolds number of 38,800 based on step height
2. Constant wall temperature
3. 17° Celsius maximum temperature difference
4. Constant fluid properties with temperature
5. The density of air corresponds to the measured pressure of two atmospheres

The conditions applied to the model include that the high pressure air 7 milliseconds after its release has a density corresponding to approximately 2 atmospheres. Also, the initial temperature of the air and the walls is assumed to be at room temperature. The air properties are evaluated at the average of the wall and free-stream temperatures at 1 atmosphere. The tunnel is modeled as two-dimensional, although the width-to-height ratio is 2.5. The zero-gradient outlet is located 140 step heights downstream of the step. The Mach number of air for this experiment is 0.3 which is appropriate for applying an incompressible assumption. For all simulations, the momentum y^+ values are kept between 30 and 50 for the majority of the wall elements. The flow field is insensitive to y^+ values between 30 and 300, but the heat flux must have y^+ between 30 and 100 for accuracy.

Simulations involving a similar channel without an outlet nozzle and a symmetric top, as shown in Figure 2, are also performed. According to Vilemas, Cesna and Survila (1987), the reattachment length is unaffected by the channel height above the step. Also, the thermal and velocity boundary layers are observed not to penetrate the free-stream above the step more than 3.5 step heights. The turbulent kinetic energy and dissipation rate contours show their containment within 2 to 3 step heights. According to the findings of Armaly and Durst et. al. (1983), a recirculation cell exists on the top wall of the channel almost directly above the flow reattachment point after the step on the lower wall. Aside from this fact, symmetric conditions are assumed because the channel height is much larger than the step height. Tsou, Chen and Aung (1991) also

confirm that the lower wall of the channel can be approximated as external flow. The outlet is located 41 step heights downstream to ensure that reattachment has occurred and the zero-gradient condition can be applied.

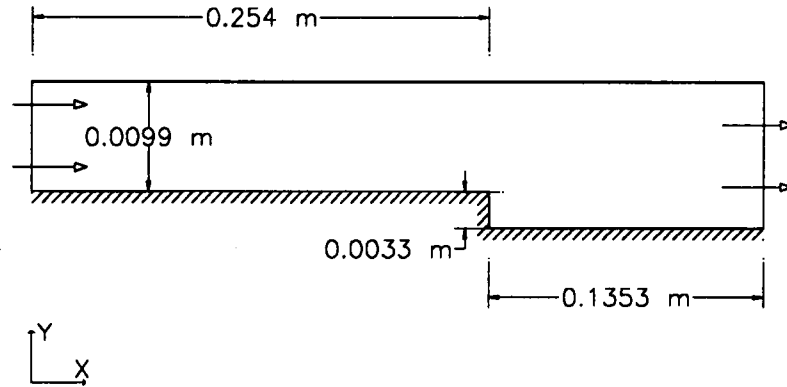


Figure 2. Symmetric channel geometry (Case A)

4.1.1 Mesh sensitivity study

The initial step in this study is to determine the sensitivity of the solution to mesh density. Figure 3 and Table 3 illustrate the effect of increasing the number of nodes from 6,000 to 24,000 on the heat transfer beyond the step. Included in the figure is the heat transfer beyond the step from the symmetric channel simulation. Also presented is a 24,000 node simulation with variable properties. The heat transfer beyond the step is presented as the local Nusselt number based on step height and 17 degree Celsius temperature difference (Equation 4.5) normalized with the steady-state boundary layer solution over a flat plate (Holman 1976, Equations 4.1 to 4.4). As shown in Figure 3, the peak heat transfer near reattachment is constant at 1.21 as the number of nodes is increased from 6,000 to 24,000 for the constant property full-size channel simulations (Case B). The dimensionless location of the peak changed by 6% in that node range. The peak value for the symmetric channel (Case A) is 1.16 and is 1.23 for the 24,000 node variable property model (Case C).

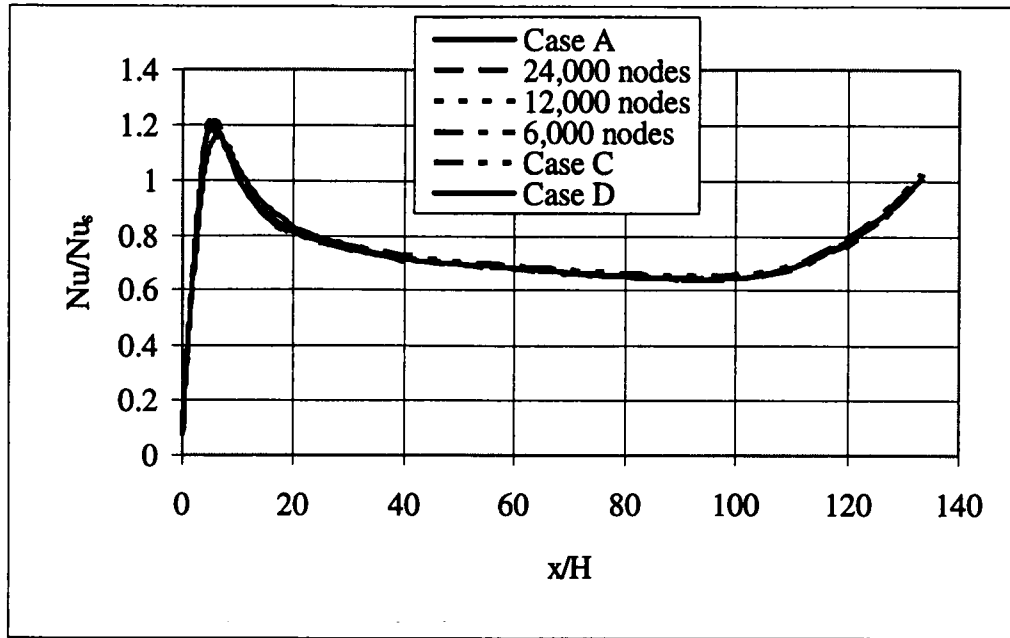


Figure 3. Mesh sensitivity study

Table 3. Mesh sensitivity study of full-size channel model (Case B).

Number of nodes in the model	6,000	12,000	24,000
Reattachment length (m)	5.385	5.285	5.284
Instantaneous velocity at 11.0 mm (m/s)	99.31	99.5	99.307
Instantaneous velocity at 3.8 mm (m/s)	78.1	78.75	78.39
Instantaneous velocity at 2.2 mm (m/s)	-0.081	-0.135	-0.11539
Turbulence intensity at 11.0 mm (%)	0.016	0.016	0.01639
Turbulence intensity at 3.8 mm (%)	5.28	5.34	5.614
Turbulence intensity at 2.2 mm (%)	4.65	4.48	4.483
Boundary layer thickness (mm)	3.8	3.86	4.158
q_w (W/m^2) at $x/H = -20.3$	4809	4845	4870.5
q_w (W/m^2) at $x/H = 2.13$	4567	4656	4712.9
q_w (W/m^2) at $x/H = 40.9$	4799	4762	4810.5
q_w (W/m^2) at the step	5143	5201	5227.2
Nusselt number at the step	39.84	40.28	40.5
Peak value of Nu/Nu_s near the reattachment point.	1.195	1.208	1.22

Case D is the 24,000 node constant property simulation with a turbulence intensity of 1.0%. As shown in Figure 3 and Table 4, the heat transfer and flow structure of Case D are in good agreement with the results of the $I = 0.4\%$ simulations. From these results, it can be

concluded that the models are insensitive to a variation of turbulence intensity at the inlet in the range of 0.4% to 1.0%.

The mesh sensitivity study shown in Table 3 reveals that the flow structure is relatively unaffected by the increase in nodes from 6,000 to 24,000 for case B. The heat transfer data, however, is somewhat dependent on the mesh density showing a 1.6% increase with the increase in the number of nodes in that range. The conclusions from this study are that the flow structure and heat transfer characteristics of this model are independent of mesh density and property variation. Table 4 shows that an increase in turbulence intensity to 1.0% has negligible effects on flow structure and heat transfer. The FIDAP manuals recommend a y^+ value between 30 and 100 for the most accurate predictions. The flow structure and heat transfer characteristics in this configuration are found to be unaffected by wall element thicknesses and y^+ values in that range. All results for this configuration are presented for a y^+ range of 30 to 50.

Table 4. Results of the simulation with turbulence intensity of 1.0%.

Quantity of nodes in the model (Case D)	24,000
Reattachment length (m)	5.284
Instantaneous velocity at 11.0 mm (m/s)	99.309
Instantaneous velocity at 3.8 mm (m/s)	78.36
Instantaneous velocity at 2.2 mm (m/s)	-0.07033
Turbulence intensity at 11.0 mm (%)	0.01638
Turbulence intensity at 3.8 mm (%)	5.6187
Turbulence intensity at 2.2 mm (%)	4.486
Boundary layer thickness (mm)	4.17
q_w (W/m^2) at $x/H = -20.3$	4873
q_w (W/m^2) at $x/H = 2.13$	4721
q_w (W/m^2) at $x/H = 40.9$	4803
q_w (W/m^2) at the step	5229
Nusselt number at the step	40.51
Peak value of Nu/Nu_s near the reattachment point.	1.212

4.1.2 Results

Table 5 shows the fluid properties evaluated at the film temperature, and the boundary conditions applied to the full-size channel constant property models (Case B). The turbulence parameters calculated from (3.16) and (3.17) are also shown. The intermediate steps in calculating the flat plate turbulent Nusselt number from the Colburn analogy and the fluid-friction-heat-transfer analogy are also shown in Equations (4.1) to (4.5).

Table 5. Properties and Parameters used for cases A and B of this configuration.

Properties and Parameters Used	Value in the Present Studies
Wall temperature (K)	293
Fluid inlet temperature (K)	276
Fluid inlet velocity (m/s)	95.5
Inlet turbulence intensity (%)	0.4
Step height (m)	0.0033
Length of flat plate before the step (m)	0.254
Viscosity (kg/s·m)	$1.7685 \cdot 10^{-5}$
Density (kg/m ³)	2.1773
Specific Heat (kJ/kg·K)	1006.69
Thermal Conductivity (W/m·K)	0.02506
Inlet turbulence kinetic energy (m ² /s ²)	0.2188886
Inlet turbulence dissipation rate (m ² /s ³)	53.087428
Reynolds Number based on step height	38800
Prandtl Number	0.710427
Reynolds Number based on flat plate length	2986415.95
Stanton Number at the step	0.001884727
Local heat transfer coefficient at the step (W/m ² ·K)	394.517
Local Nusselt Number at the step	51.95158
Fluid medium	high pressure air

Table 6. Steady-state instantaneous velocities at $x/H = 0.75$ downstream of the step.

Position (mm)	Tsou, Chen and Aung (1991) u (m/s)	Present Study (A) u (m/s)	Present Study (B) u (m/s)
11.0	100	99.97	99.3
3.8	72	74.251	78.39
2.2 x-component	4.7	-0.5145	-0.11539
2.2 Speed	4.7	3.0573	3.0836

The values of instantaneous flow speed for the present study shown in Table 6 are the solved quantities for the x-component of velocity for the models. The 2.2 millimeter measurement

is taken in the recirculating zone by Tsou, Chen and Aung (1991) by a single hot-wire approach. This measurement technique only reveals the magnitude of the flow speed and not the direction. The models clearly show the recirculating nature of the flow at that location in Table 6. The 3.8 millimeter measurement is in the shear-affected layer above the step. Table 6 shows that the models and the experiment are in good agreement there. The 11.0 millimeter measurement is in the inviscid core regime, and the predictions and measurement are in good agreement. The speed measurement is the square root of the squares of the x and y components of velocity. This absolute velocity is reported as a direction-independent comparison with the measurements.

Table 7. Steady-state turbulence intensities at $x/H = 0.75$ downstream of the step.

Position (mm)	Tsou, Chen and Aung (1991) I %	Present Study (A) I %	Present Study (B) I %
11.0	0.04	0.01625	0.01639
3.8	11.3	5.91	5.614
2.2	3.3	4.32	4.483

The measured values of Tsou, Chen and Aung (1991) shown in Table 7 for the turbulence intensity are obtained from hot-wire anemometry, while the values for this study are calculated from the solution of the turbulent kinetic energy equation. The calculation of turbulence intensity from turbulent kinetic energy involves the use of the shear-free wind tunnel equation (3.16) discussed previously. Both models reveal a significant variation from the measured values. The similarity between cases indicates the insensitivity of turbulent kinetic energy values to mesh density and outlet effects. The major influence on measured results is the inlet turbulence intensity. In the models, an increase in the inlet intensity above that which is measured would increase the values at the probe location. This would result in a better agreement with the measured values at the probe shown in Table 7. Another known effect of increasing the intensity is an increase in measured heat transfer. The boundary layer thickness at the step was determined as 99% of the distance from the wall to the inviscid core in the present study. Table 8 shows that the

approximate boundary layer thicknesses from the models are thinner than the experimental measurements.

Table 8. Steady-state boundary layer thickness δ_s/H at the step.

Tsou, Chen and Aung (1991)	Present Study (A)	Present Study (B)
1.4	1.3	1.26

Table 9. Steady-state reattachment lengths X_r/H .

	19,000 node symmetric channel	24,000 full-size channel
Tsou, Chen and Aung (1991)	Present Study (A)	Present Study (B)
7.6	6.29	5.284

As previously mentioned, the reattachment length is insensitive to channel height in the experiments. The simulations, however, show that the prediction of the reattachment lengths are highly dependent on channel height. As shown in Table 9, the length predicted by case A is within 17% which is better than the 20% underprediction by other models. The total channel height in case A is only 3 step heights which means that the step is a significant portion of the model. Other simulations of symmetric top boundary channels with only slightly different total heights have similar predictions to case A. Case B had a total height of 15 step heights and a 30% underprediction. The measured reattachment length is known to be very sensitive to Reynolds number, so velocities and fluid properties are kept consistent within all models.

Table 10. Steady-state heat flux measurements at three positions.

Position x/H	Tsou, Chen and Aung (1991) $q_w (W/m^2)$	Present Study (A) $q_w (W/m^2)$	Present Study (B) $q_w (W/m^2)$
-20.3	6900	5030.9	4870.5
2.13	2500	4463.7	4712.9
40.9	6250	4752.3	4810.5

The heat fluxes of Tsou, Chen and Aung (1991) shown in Table 10 are measured with fast response gages. The heat fluxes for the models are calculated based on the solved temperature field. As mentioned above, the heat fluxes in this study are insensitive to y^+ changes between 30 and 100 over the surfaces. Also, small changes in velocity or properties have negligible effects. A

negligible increase in the calculated heat flux is usually observed with increasing mesh density, as shown in Section 4.1.1. The differences shown in study A and B in Table 10 are based on channel height.

Table 11. Nusselt numbers (Nu_s) at the step.

Flat Plate Value	Present Study (A)	Present Study (B)
51.95	40.98	40.491

The flat plate value shown in Table 11 comes from the steady-state boundary layer solution over a flat plate (Holman 1976). The Colburn analogy and the fluid-friction-heat-transfer analogy are used to obtain that value, as shown in equations (4.1) - (4.4). The models underpredict the flat plate value at the step by 21%.

$$St_x Pr^{2/3} = \frac{Cf_x}{2} \quad (4.1)$$

$$Cf_x = 0.0592 Re_x^{-1/5} \quad (4.2)$$

$$h_x = St_x \rho C_p U_\infty \quad (4.3)$$

$$Nu_x = \frac{h_x x}{k} \quad (4.4)$$

The local Nusselt number from the models arises from the local heat transfer at the step, the step height, and the 17 degree temperature difference, as shown in equation (4.5).

$$Nu_x = \frac{\dot{q}_x'' H}{k \Delta T} \quad (4.5)$$

$$Re = \frac{\rho U_i H}{\mu} \quad (4.6)$$

The data presented in Table 12 reflect the maximum augmentation of the heat transfer near the reattachment point. The normalization is performed with the heat transfer from the steady-state

boundary layer solution over a flat plate (Holman 1976), as shown in Table 11. The peak values approach 1.6 when the local Nusselt numbers at the step from the simulations are used. Vogel and Eaton (1985) report that this ratio should be closer to 2.0.

Table 12. Peak values of Nu/Nu_{step} near the reattachment point and locations.

Tsou, Chen and Aung (1991)	Present Study (A)	Present Study (B)
1.10	1.16	1.22
$x/H = 7.6$	6.47	5.68

The next 4 figures are flow structure and heat transfer results obtained from the full-size channel model (Case B) with 24,000 nodes. Similar trends with the simulations and the experimental data are revealed, although some underprediction of turbulence intensity and heat transfer exists.

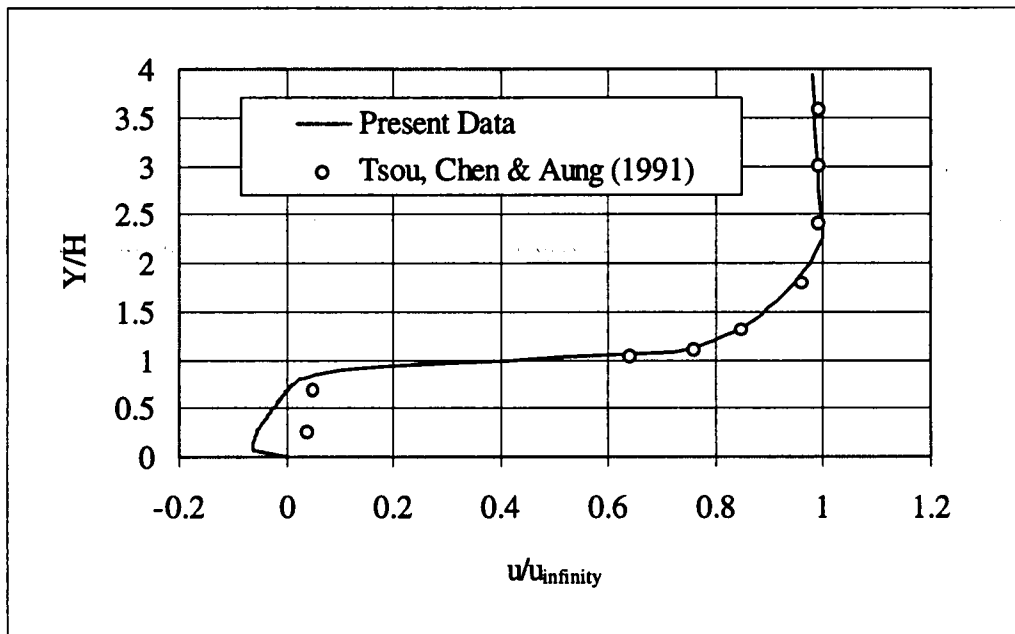


Figure 4. Velocity profile at $x/H=0.75$.

Figure 4 represents the velocity profile and channel height normalized with the maximum velocity and step height, respectively. The experimental data does not show the recirculation zone (negative velocity) at the measurement location beyond the step. The simulations, however, reveal the recirculation and closely match the measured velocity profile above the step.

Figure 5 is the turbulence intensity profile calculated from the inviscid flow wind tunnel equation and the calculated values of turbulent kinetic energy. The channel height is normalized with the step height. The predicted turbulence intensity follows the pattern of the measured values, although considerably different in magnitude. An increase in the inlet intensity would increase the model predictions and more closely match the data.

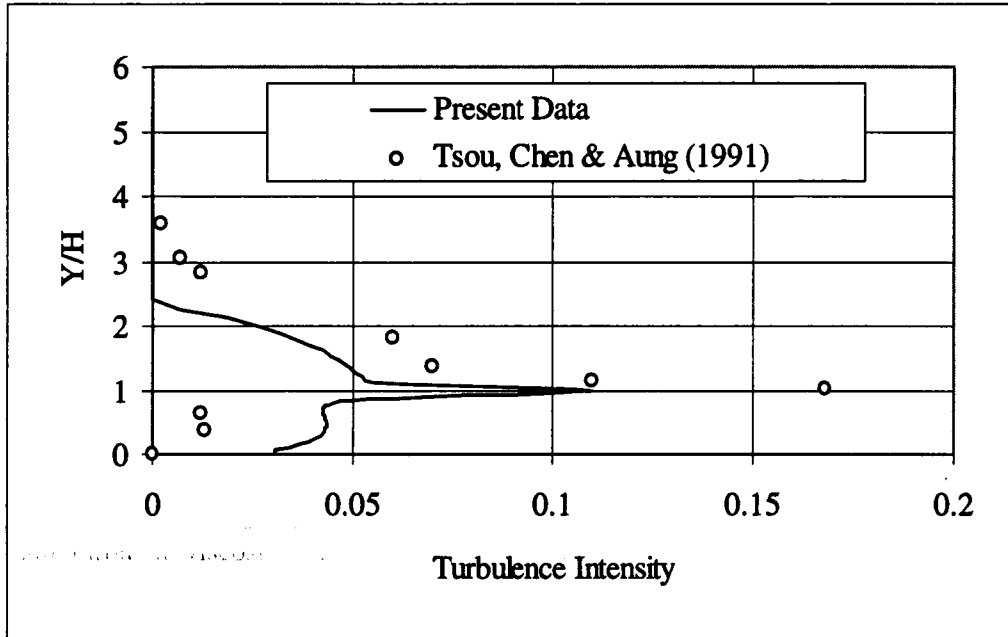


Figure 5. Turbulence intensity profile at $x/H = 0.75$.

Figure 6 shows the experimental local Nusselt numbers beyond the step normalized with the value from the flat-plate turbulent boundary layer solution (Holman 1976). The predicted Nusselt numbers are normalized with the predicted value at the step. The steady-state Nusselt numbers shown by both simulation and experiment reveal the presence of the peak values near the reattachment points. According to Aung and Goldstein (1971) and Vogel and Eaton (1985), these peak heat transfer regions can be used to estimate the reattachment lengths. The boundary layers begin to redevelop at these points and grow downstream. Boundary layer recovery and growth is evident from the decrease in heat transfer with channel length. The simulation results show the

peak value 10% overpredicted from experimental results and a much more quickly developing boundary layer.

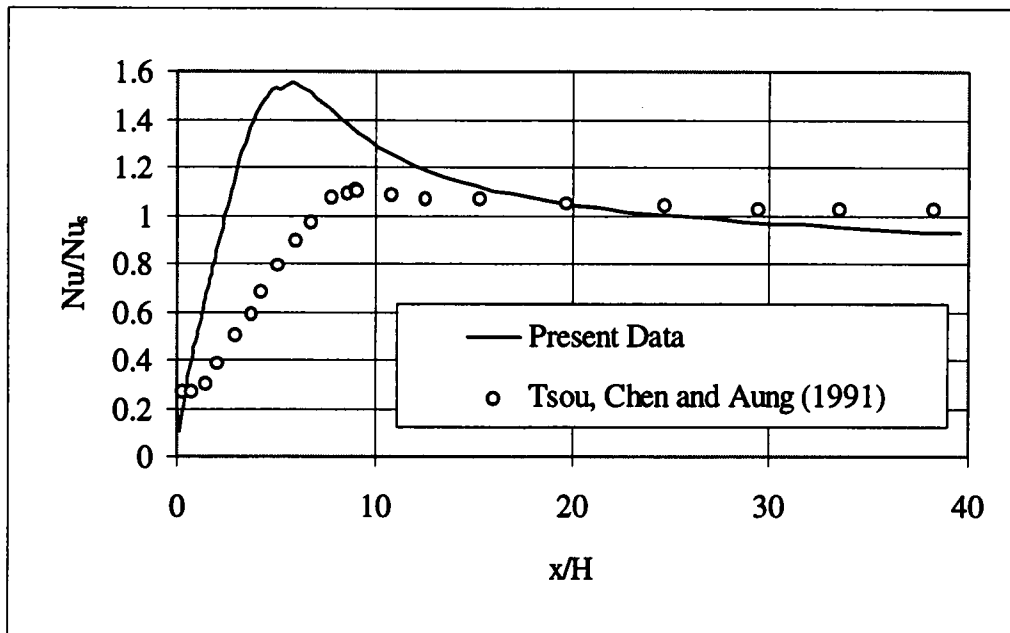


Figure 6. Nusselt number profile downstream of the step.

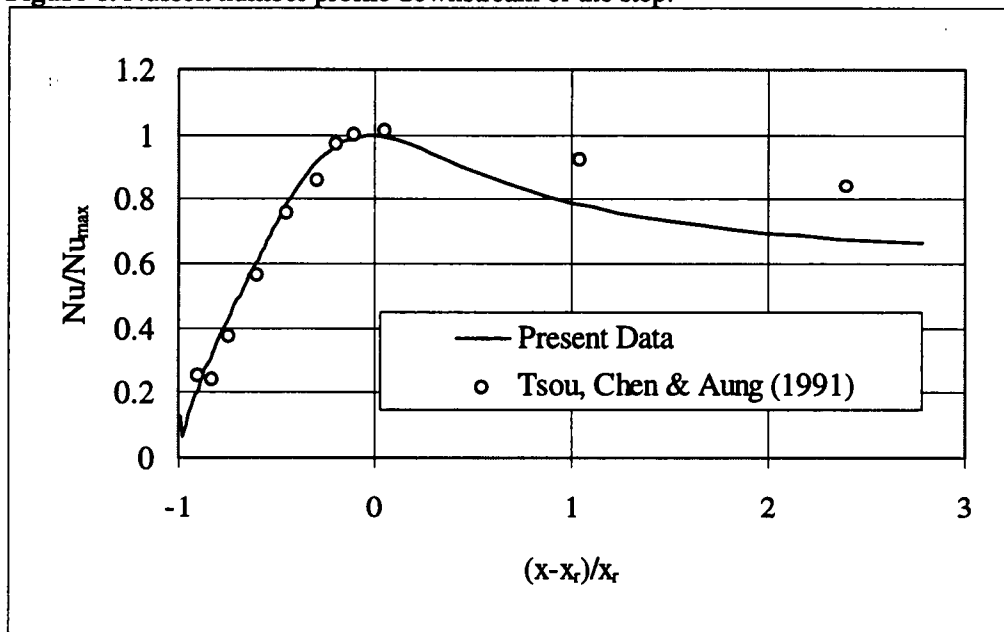


Figure 7. Nusselt number profile downstream of the step.

Figure 7 shows a plot of the local Nusselt number beyond the step normalized with the maximum value at the reattachment point. Also, the distance down the channel is normalized with

the reattachment length (X_r). The more quickly decreasing heat transfer after the reattachment point is in better agreement with experimental results of Seban (1964) and Vogel and Eaton (1985) than with results of Tsou, Chen and Aung (1991). The turbulence kinetic energy, as shown in Table 13, is the maximum at the top corner of the step and recovers to its pre-step value approximately 30 step heights downstream of the step. As pointed out by Vogel and Eaton (1985), the maximum occurs at the point of maximum heat transfer before reattachment. The results presented here are consistent with their findings. The dissipation rate follows the same patterns in the same areas as the kinetic energy and reaches its maximum also at the top corner of the step. It recovers about 16 step heights downstream of the step. The turbulent viscosity has its highest values directly beyond the step and steadily decreases toward the outlet.

Table 13. Turbulent properties range.

Maximum turbulence kinetic energy (m^2/s^2)	367.56958
Minimum turbulence kinetic energy (m^2/s^2)	$0.367569 \cdot 10^{-3}$
Maximum dissipation rate (m^2/s^3)	$34.196 \cdot 10^6$
Maximum turbulent viscosity ($\text{kg/s} \cdot \text{m}$)	$0.1043078 \cdot 10^{-1}$

Figure 8 shows the streamlines in the immediate vicinity of the step. The streamfunction values shown range from $-0.02 \text{ m}^2/\text{s}$ to $0.14 \text{ m}^2/\text{s}$ in 18 equal increments. Figure 9 shows the isotherms from 278° Kelvin to 292° Kelvin in one degree increments.



Figure 8. Streamlines in the vicinity of the backward-facing step.

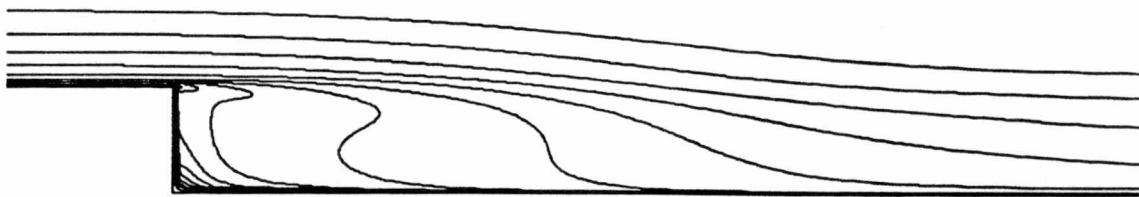


Figure 9. Isotherms in the vicinity of the backward-facing step.

4.1.3 Conclusions

The results of this study are in general agreement with previous experimental work. Flow structure and heat transfer trends follow experimental data. The velocity profile downstream of the step closely matches the data and also reveals the recirculation zones which cannot be measured easily. Turbulent intensity downstream of the step shows the proper trends but is different in magnitude than the data. Heat transfer results show the proper form with low values at the step, a peak value near reattachment, and a decrease down the channel as the boundary layer grows. The position of reattachment and the peak heat transfer numbers reported by the model vary from the experimental measurements by 25% and 10%, respectively. As mentioned by Vogel and Eaton (1985), the heat transfer in the reattachment region is governed primarily by turbulent fluctuations. However, the predictions of the models show that the heat transfer and flow structure are independent of turbulent intensity in the range 0.4% to 1.0%.

The constant property assumption is valid due to the small 17 degree Celsius temperature difference, and greatly simplifies the convergence process. The models are generally independent of mesh density above 8,000 nodes. The high speed flows are insensitive to wall element thicknesses provided y^+ remains between 30 and 100. Experimental data on the outlet boundary conditions is not available, and the effects of the outlet are assumed to not affect the upstream phenomena. Also, it is difficult to achieve a good prediction of small phenomena within a large channel. From the simulation of this geometry, it is further concluded that the experimenters must measure and record the turbulent flow structure and the test conditions with high accuracy. Inlet measurements of turbulent intensity and velocity profile are important for the development and success of the model.

4.2 Two-Dimensional Converging Channel

Simulations of the converging duct geometry of Mutama and Iacovides (1993) shown in Figure 10 are performed in this investigation. The duct has a 5.1 cm inlet a 1.7 cm outlet and a Reynolds number of approximately 21,000. The Reynolds number is based on the bulk velocity and channel inlet height as shown in Equation (4.7).

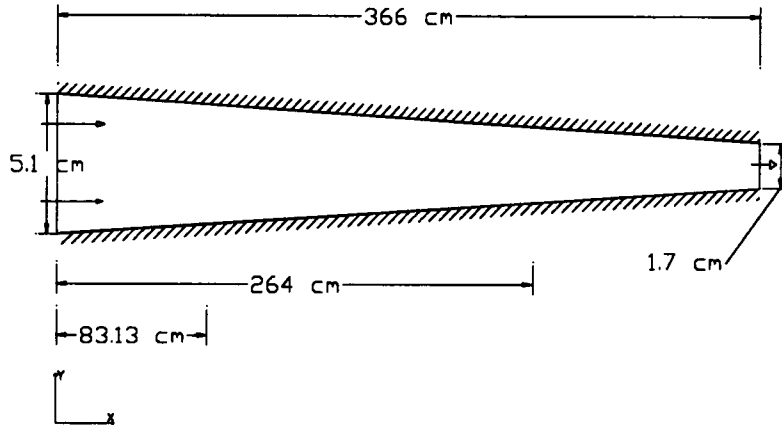


Figure 10. Geometry used by Mutama and Iacovides (1993).

These parameters correspond to an acceleration parameter of 4.47×10^{-7} given by Equation (4.9). According to Launder (1964), turbulent flow in a channel with an acceleration parameter greater than 2.3×10^{-6} will relaminarize. The experimental data of Mutama and Iacovides (1993) for this geometry and flow conditions show the heat transfer and fluid friction levels approach that of laminar flow. They also show that for flow with lower levels of Reynolds number a strong convective acceleration causes a complete reverse transition. Their experiments also reveal that the duct has two-dimensional symmetry because of the 1:16 entry aspect ratio. The Mach number for air in the experiment was 0.02, allowing an incompressible assumption. Also the maximum temperature variation is 5 degrees Celsius which made all variables considered constant in the model.

In addition to the general assumptions applied to all of the simulations, the following experimentally measured conditions are applied to this configuration:

1. Low Reynolds number turbulent flow
2. Constant heat flux on the lower wall
3. Uniform inlet velocity, temperature and turbulent parameters
4. Constant fluid properties with temperature
5. A measured inlet turbulence intensity of 0.4%

$$Re = \frac{\rho u_b H}{\mu} \quad (4.7)$$

$$u_b = \frac{\int_0^H u(y) dy}{H} \quad (4.8)$$

$$k = \frac{\tan \alpha}{Re} \quad (4.9)$$

The experimenters measured uniform inlet conditions, including a turbulence intensity of 0.4%. They measured negligible spanwise velocity gradients in the third direction to insure two-dimensionality. They also confirmed that their copper heater, located only on the lower wall, had a constant heat flux input to the fluid. They used hot-wire probes to measure the mean and fluctuating velocity components and the temperature variations are measured by a thermocouple probe.

4.2.1 Mesh sensitivity study

As shown in the backward-facing step geometry section 4.1.1, the models are insensitive to mesh density, small variations in the properties and outlet boundary conditions. For high Reynolds number models with inlet air velocities of 90 m/s or greater, the flow structure and heat transfer

characteristics are also insensitive to y^+ values at the walls between 30 and 100. That lower limit of flow speed allows the use of a wall element layer of size 0.3 mm or smaller to contain the viscous sublayer and buffer layer. Small changes in that element layer thickness, and consequently y^+ , have negligible effects on the results.

The geometry under consideration here has a very low inlet velocity of 6.35 m/s, and the results shown in the Figures 11-13 can be seen to be very sensitive to wall element thickness and y^+ . The region of the duct considered in this investigation and compared with the experimental data is located $X/H = 16.3$ downstream from the inlet (83.13 cm). Figure 11 shows the velocity distributions associated with different wall element layer thicknesses prescribed at the inlet. The 0.3 mm thickness model clearly shows realistic results for the velocity profile in Figure 11. However, the temperature distribution in Figure 13 reveals nonphysical phenomena for the 0.3 mm wall element thickness.

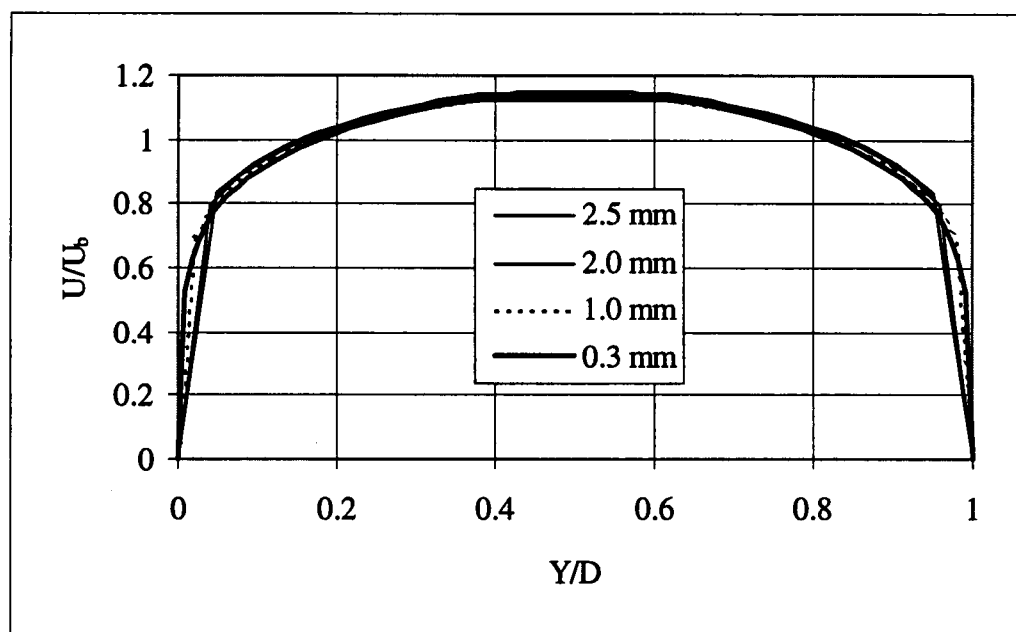


Figure 11. Variation in velocity distribution with wall element thickness at $X/H = 16.3$.

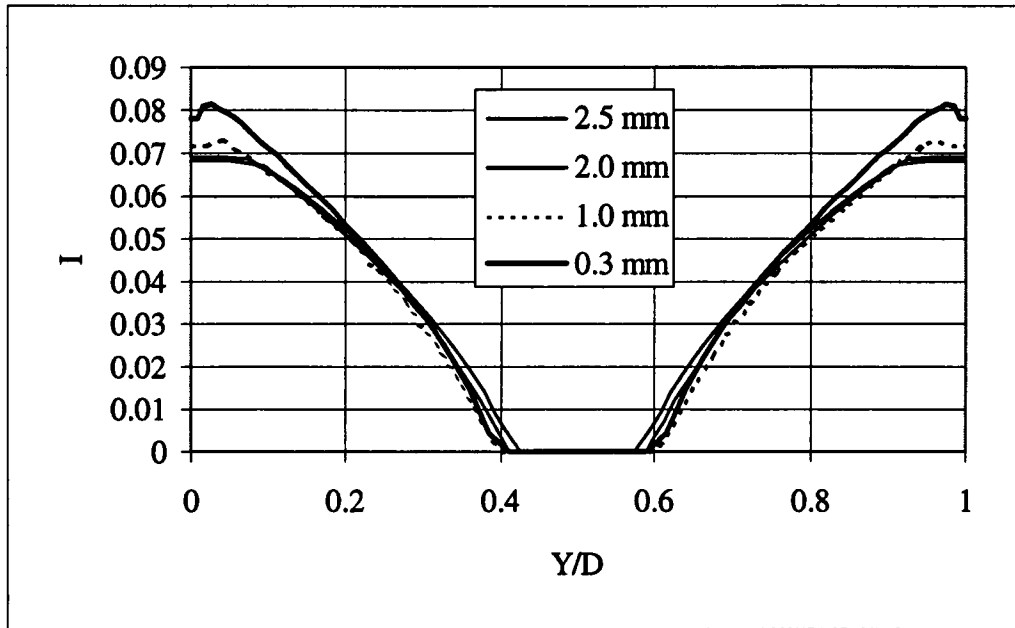


Figure 12. Variation in turbulence intensity with wall element thickness at $X/H = 16.3$.

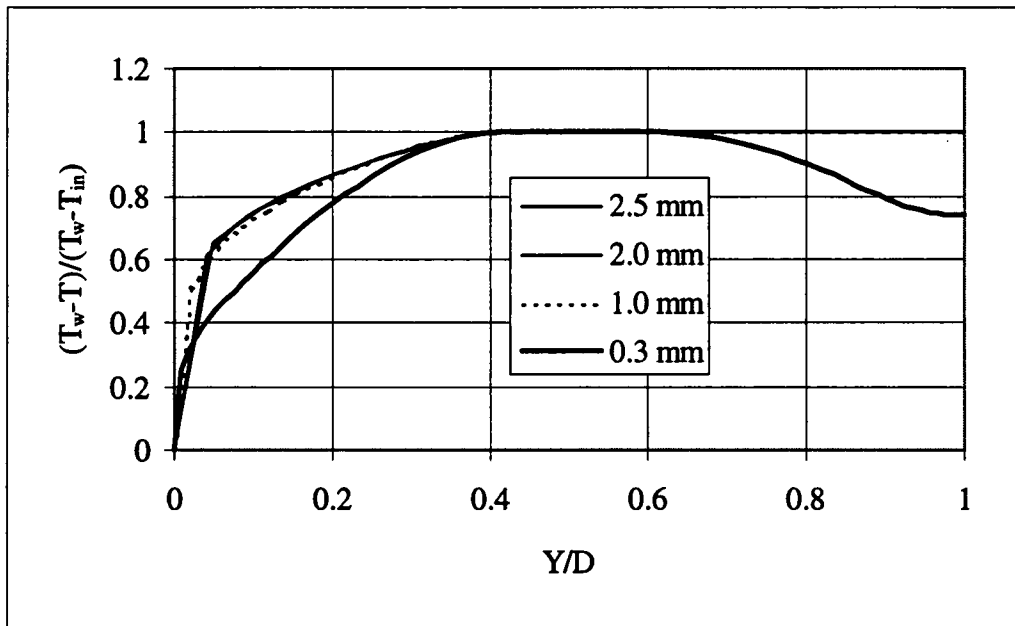


Figure 13. Variation in temperature profile with wall element thickness at $X/H = 16.3$.

Table 15 shows the boundary conditions used in this configuration. As shown in Table 14, the y^+ values for the 0.3 mm thickness model range from 8 to 21, which is well below the accepted minimum of 30. This means that the governing equations are solved in the viscous

sublayer. The 1.0, 2.0 and 2.5 mm models show realistic results with a convergence of the distributions above 2.5 mm.

Table 14. y^+ ranges from inlet to outlet corresponding to wall element thicknesses.

Thickness	Maximum y^+	Minimum y^+
0.3 mm	21	8
1.0 mm	29	20
2.0 mm	41	36
2.5 mm	48	41

Table 14 shows that the y^+ range for the 2.5 mm model is acceptable, although nearly 10% of the channel height is occupied by the wall elements. This means that a significant portion of the flow and heat transfer characteristics in the channel are approximated by the universal near-wall profiles. Also, it is observed from the figures that this large of a wall element layer prevents the capturing of the flow and temperature definitions near the walls. The following flow structure and heat transfer results are presented for the 2.0 mm thickness model.

Table 15. Parameters used consistently with this configuration.

Parameters used in the Simulations	Value
Fluid medium	air
Imposed heat flux on the lower wall (W/m^2)	151.208
Measured inlet Stanton number	0.004
Fluid density (kg/m^3)	1.1824
Fluid Specific heat ($J/kg \cdot K$)	1006.91
Fluid viscosity ($N \cdot s/m^2$)	$1.8235 \cdot 10^{-5}$
Fluid thermal conductivity ($W/m \cdot K$)	0.02594
Inlet turbulent kinetic energy (m^2/s^2)	$9.6782 \cdot 10^{-4}$
Inlet dissipation rate (m^2/s^3)	$5.4662 \cdot 10^{-4}$
Inlet x-component of velocity (m/s)	6.35
Reynolds number	21,000
Duct location under consideration (X/H)	16.3
Inlet duct height (H) (m)	0.051
Local channel height at $X/H = 16.3$ (D) (m)	0.04327887
Inlet turbulence intensity (%)	0.4
Inlet air temperature (K)	293
Inlet wall temperature(K)	298
Number of nodes for each model	12,000
Mach number	0.02
Acceleration parameter	$4.47 \cdot 10^{-7}$

4.2.2 Results

The results are presented in the next four figures, at a location $X/H = 16.3$ from the inlet of the channel. Figure 14 shows the velocity profile with a Reynolds number of 21,000 based on the experimental measurements and the simulation. The simulation shows that the inviscid core is still present, but the flow is much more developed than the measurements show.

Figure 15 shows the turbulence intensity profiles at $X/H = 16.3$. The simulation profile is calculated from the turbulent kinetic energy variable using equation (2.7) and the bulk velocity at this location. The measured value comes from hot-wire analysis of the flow. The predicted profile is in poor agreement with the measured profile.

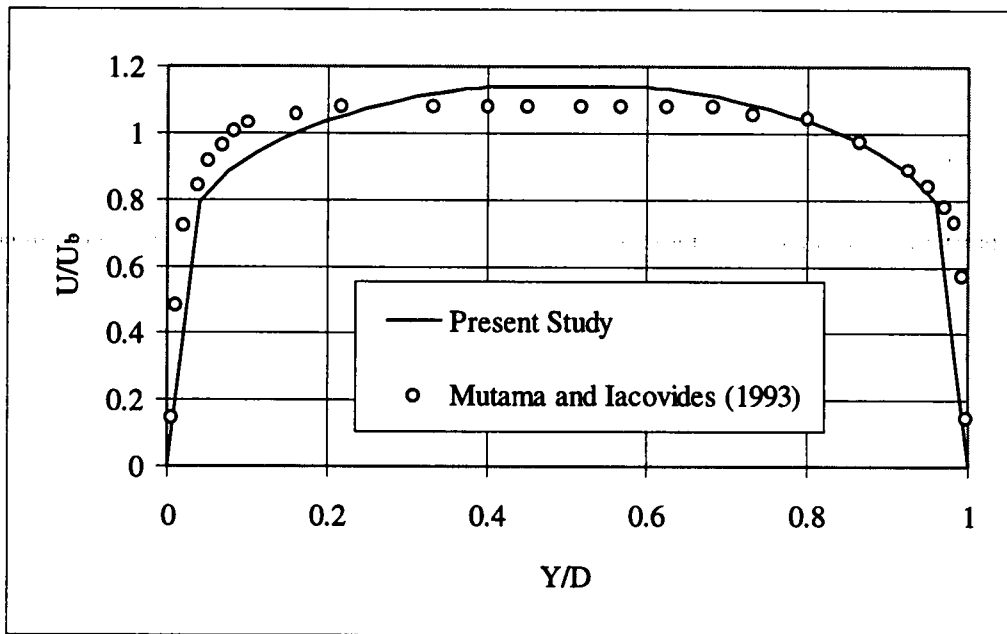


Figure 14. Mean velocity field at $X/H = 16.3$ and $Re = 21,000$.

The simulated distribution has a smaller intensity unaffected core and appears to resemble the more developed measured profiles closer to the outlet of the channel. The intensity-unaffected core shown by the simulation at this location has values smaller than the measurements due to the “Clipping Procedure” discussed in Section 3.2.2. Near the outlet, the experiments show that the

core completely disappears and the measured profile at $X/H = 51.9$ shows a constant intensity equal to 0.010. The simulations predicted much higher levels. Figure 15 also shows that the high turbulence intensity near the walls reported by the simulated results is shown to exist in the measured profiles but with larger magnitude. Figure 16 shows the predicted heat transfer results.

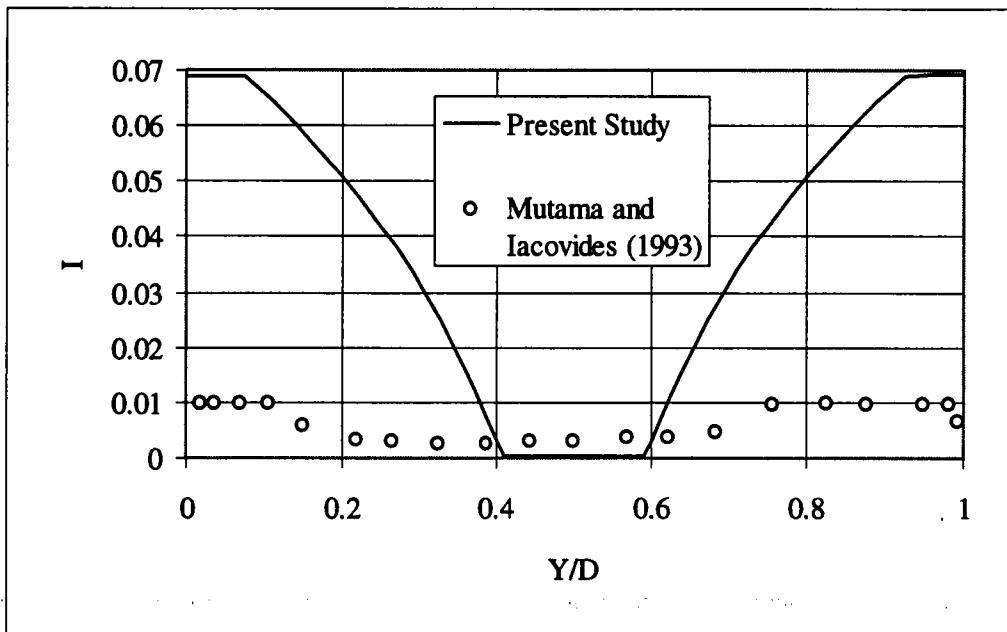


Figure 15. Normalized turbulence intensity profile at $X/H = 16.3$ and $Re = 21,000$.

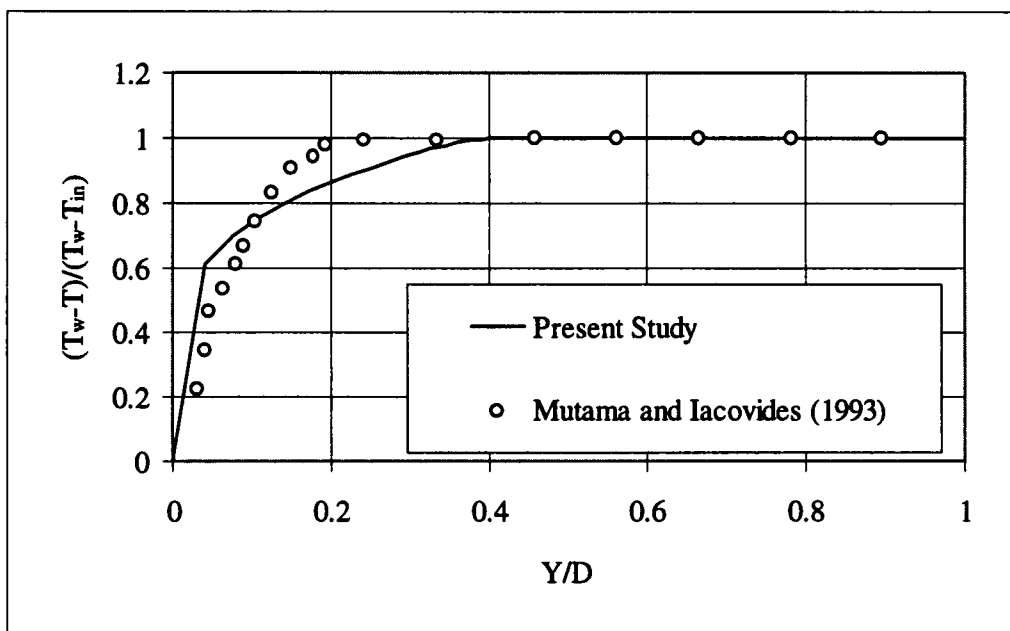


Figure 16. Temperature profile at $X/H = 16.3$ and $Re = 21,000$.

Figure 16 shows the non-dimensionalized temperature profile at $X/H = 16.3$ at a Reynolds number of 21,000. The simulated results indicate that the thermal boundary layer is thicker than the measured value at this station. The further penetration of the thermal effects into the fluid is consistent with the overprediction of the turbulent structure. However, the predicted temperature gradients near the wall are in good agreement with the measured values.

Table 16. Friction coefficient at $X/H = 16.3$ and $Re = 21,000$.

	Mutama and Iacovides (1993)	Present Study
$C_f/2$	0.0015	0.00257

Table 17. Friction coefficients for fully-developed laminar and turbulent channel flows.

	Laminar value ($16/Re$)	Patel and Head ($0.0376 \cdot Re^{-1/6}$)
$C_f/2$	0.00076	0.00358

The friction coefficient found in the present study is higher than the measured value as shown in Table 16. The higher value from the simulation is more consistent with the fully-developed friction coefficient of Patel and Head (1968) for turbulent channel flows shown in Table 17. This result shows that the simulation predicts a turbulent flow structure that is bracketed between that of a non-accelerating turbulent channel flow and that found by the experiment and that of laminar flow. The method by which Mutama and Iacovides (1993) calculated the friction coefficient shown in Table 16 is based on differentiating the velocity distribution at the wall. It is calculated in the present study by the use of the viscous shear stress at the wall found by FIDAP and the bulk velocity at $X/H = 16.3$ as shown by Equation 4.10.

$$C_f = \frac{\tau_w(x)}{1/2 \rho u_b(x)} \quad (4.10)$$

The predicted heat transfer to the fluid from the heated wall is shown to be higher from the simulation results than from the measurements as shown in Table 18. The Stanton number for the

present study is calculated from the local heat flux from the wall found by FIDAP and the bulk velocity and temperature at this location (Equations 4.11 and 4.12).

Table 18. Stanton number at $X/H = 16.3$ and $Re = 21,000$.

	Mutama and Iacovides (1993)	Present Study
St	0.0013	0.00219

$$St = \frac{\dot{q}_{wall}''}{(T_b - T_w)\mu_b \rho C_p} \quad (4.11)$$

$$T_b = \frac{\int_0^H T(y) dy}{D} \quad (4.12)$$

Figure 17 shows that the centerline turbulence intensity of the experiment and the simulation increases toward the outlet, indicating that the turbulence effects have grown from the walls into the center of the channel. The simulation shows that the intensity in the core is very low until location $X/H = 22$ where the intensity boundary layers merge. The simulation also shows that the magnitude of the predicted turbulence intensity is much larger than that of the experimental measurements. The simulations show that between normalized lengths of 35 and 70 the core turbulence intensity decreases, indicating a reduction in overall turbulence structure not present in a standard pipe flow.

Table 19 shows that the turbulent kinetic energy is the largest at the walls near the outlet. The dissipation rate is also at a maximum in the same areas. This confirms that the flow is becoming more turbulent as it travels down the channel which is opposite of the experimental data. The turbulent viscosity acquires its maximum values half way down the channel and then slowly decreases toward the outlet.

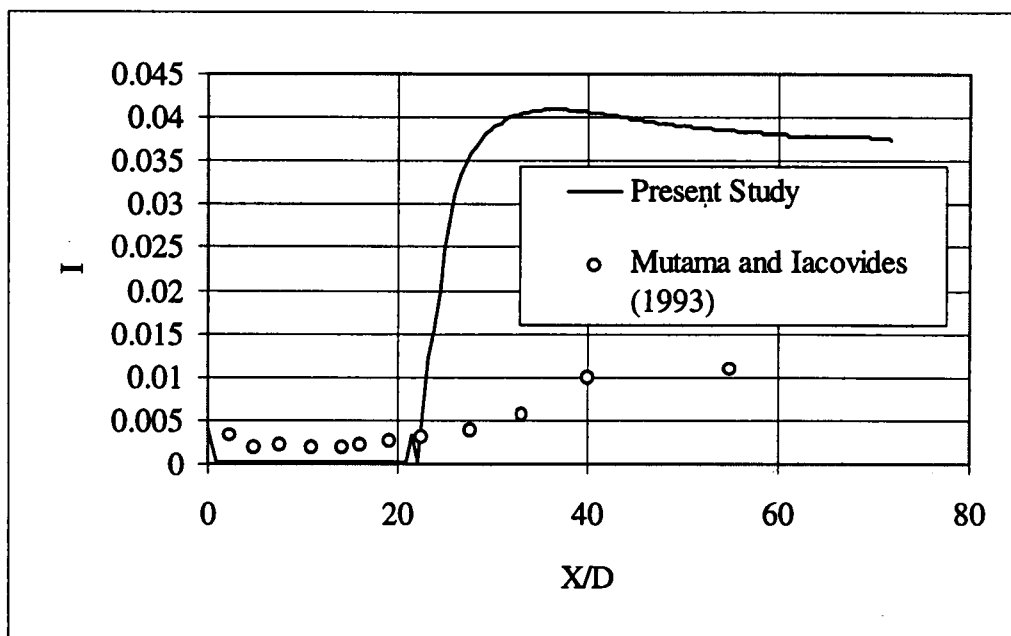


Figure 17. Centerline turbulence distribution from inlet to outlet.

Table 19. Turbulent flow structure characteristics.

Maximum turbulence kinetic energy (m^2/s^2)	2.62678
Minimum turbulence kinetic energy (m^2/s^2)	$0.26268 \cdot 10^{-5}$
Maximum dissipation rate (m^2/s^3)	2079.85
Minimum dissipation rate (m^2/s^3)	$0.68711 \cdot 10^{-6}$
Maximum turbulent viscosity ($\text{kg}/\text{s} \cdot \text{m}$)	$0.13485 \cdot 10^{-2}$
Wall temperature at the inlet (K)	293
Wall temperature at the outlet (K)	301.879

Figure 18 shows the streamlines in the channel at $X/H = 16.3$. The values range from $0.0154 \text{ m}^2/\text{s}$ to $0.2926 \text{ m}^2/\text{s}$ in 10 increments. Figure 19 shows the isotherms at $X/H = 16.3$ ranging from 293° Kelvin to 299° Kelvin in one degree increments.

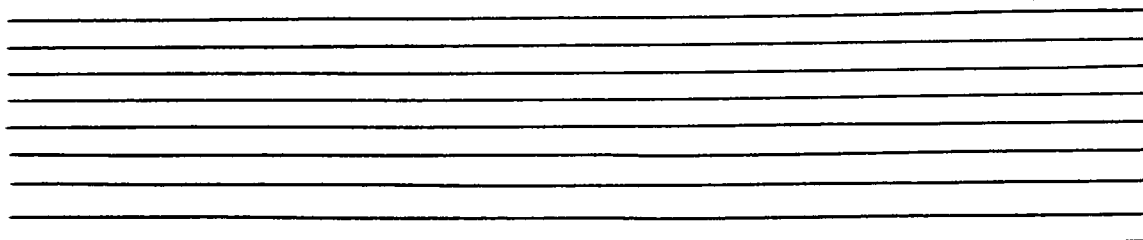


Figure 18. Streamlines at $X/H = 16.3$

Figure 19. Isotherms at $X/H = 16.3$

4.2.3 Conclusions

The conclusions that can be drawn from this case study are that a low Reynolds number flow requires a significant portion of the channel to be occupied by wall elements. Therefore, a large portion of the flow regime is approximated by the universal near-wall profiles. Also, the definition of the variables near the walls is lost due to the linear approximations used in the wall elements. The predictions from models of low speed flows are sensitive to the wall element thicknesses. A balance between wall element thickness, y^+ and the model's sensitivity to these factors must be achieved.

Another conclusion that can be drawn is that this high Reynolds number k - ϵ model does not accurately predict the relaminarization phenomena. The low Reynolds number simulation results show higher levels of turbulence intensity than the measured data and higher Stanton numbers and friction coefficients. This indicates a more turbulent flow structure. However, the predicted heat transfer and friction results are lower than that of standard turbulent pipe flows. The more quickly developing boundary layers are also a result of the higher intensity. The reduction in predicted centerline turbulence intensity along the length of the channel reveals a turbulence reduction phenomenon not present in standard flows. The predicted results from this simulation are bracketed between the standard turbulent flow values and the experimentally measured near laminar values.

4.3 Discussion of Two-Dimensional Ribbed Channel Results

The experiment conducted by Liou and Hwang (1992) included a rectangular duct with opposite walls covered with square ribs. Figure 20 shows one rib pitch module of the duct. They studied the developing and fully-developed region of the duct using laser holographic interferometry (LHI). In addition, thermocouples and pressure transducers were used to measure the flow field. From their experimental results they developed some data correlations, including the fully-developed friction factor and average Nusselt number with respect to Reynolds number and pitch-to-height ratio of the ribs. The maximum uncertainties for their data correlations are 4.2% and 5.5%, respectively. The Reynolds number is based on the hydraulic diameter of the duct and defined in Equation (4.13). The Reynolds number range of the experiment was from 5,000 to 50,000. The simulations of this geometry are performed at $Re = 50,000$ and $Re = 100,000$ in order to satisfy the high Reynolds number requirements of the $k - \epsilon$ model.

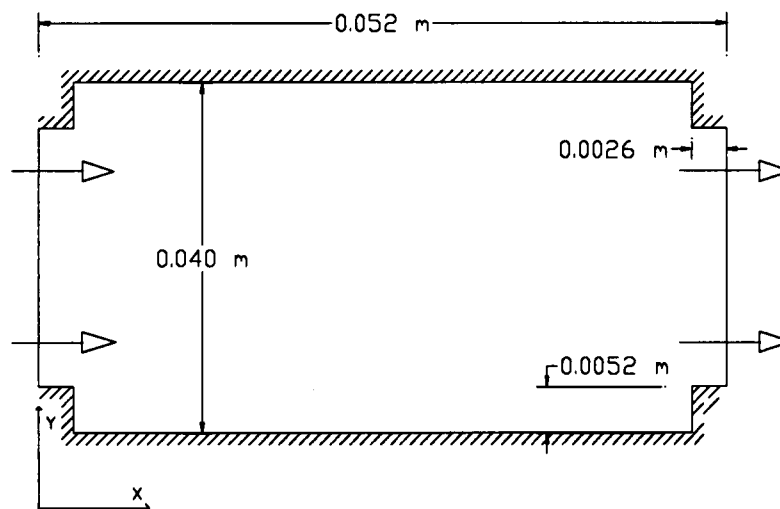


Figure 20. Fully-developed module used by Liou and Hwang (1992).

The general assumptions are applied to this configuration, as well as the measured geometry and boundary conditions used in the experiment. The width of the channel is one-fourth of the height, although the experimental measurements show negligible temperature and pressure

variation in the cross-channel direction. The assumption of a two-dimensional symmetry about the channel midplane is valid for the models. The Mach number for air in the simulation is 0.3 which allows for an incompressible flow assumption. The properties are assumed to be constant due to a very small temperature variation present between the walls and the air.

The method by which they obtained the local Nusselt number is shown in Equation (4.15) and is estimated to have an uncertainty less than 6.5 %. Also, the average Nusselt number shown in Equation (4.14) has an uncertainty less than 10%. The simulation predictions are also presented in terms of these equations. The experiments of Liou and Hwang (1992) reveal that the average Nusselt numbers obtained from this geometry are insensitive to wall temperature and heat flux in the flow direction. For this reason, neither the wall heat flux nor the wall temperature distributions measured in the experiment are reported. Although Liou and Hwang (1992) state that the thermal boundary conditions can be arbitrary in this case, the simulations are assigned a linearly increasing wall temperature in order to satisfy the fully-developed modular symmetry condition, as described by Rowley and Patankar (1984).

This study involves only the periodic fully-developed portion of the duct nine hydraulic diameters downstream from the inlet. The fully-developed profiles for x and y components of velocity, turbulent kinetic energy, turbulent dissipation rate, pressure and temperature are determined by modeling individual rib pitch modules beginning at the inlet. Constant profiles are specified at the inlet of the first module and a zero-gradient outlet is used. The solution for the profiles at the outlet of the first module are placed as inlet conditions for the next module. Following this procedure, it is observed that the profiles are periodic at the inlet and outlet of the module after 7 to 8 rib pitch modules downstream. These fully-developed profiles are then used consistently as the inlet boundary conditions for this configuration. The fully-developed experimental results reported by Liou and Hwang (1992) are measured after 9 rib pitch modules.

Table 20 shows the geometric and flow properties used consistently in the modeling of this configuration.

Table 20. Properties and Parameters used in this configuration .

Properties and Parameters Used	Value in the Present Studies
Hydraulic diameter (D_h) (m)	0.016508
Rib height (H) (m)	0.0052
Rib height to channel height ratio (H/2B)	0.13
Half width of channel to rib height ratio (W/H)	1
Rib pitch-to-rib height ratio (PR)	10
Wetted length in one rib pitch (L_h) (m)	0.0156
Channel width (W) (m)	0.0104
Fluid medium	air
Viscosity (kg/s·m)	$1.846 \cdot 10^{-5}$
Specific Heat (kJ/kg·K)	1007
Thermal Conductivity (W/m·K)	0.0263
Prandtl Number	0.7
Inlet turbulence intensity (%)	0.4
Inlet velocity (m/s)	100
Inlet turbulent kinetic energy (m^2/s^2)	0.24
Inlet dissipation rate (m^2/s^3)	32.7159

The local Nusselt number, average Nusselt number and Reynolds number are calculated from the heat flux quantities from the simulation and the experiment by Equations (4.13) to (4.17).

The calculation of the friction factor and centerline pressure drop are shown in Equations (4.18) and (4.19). These equations have uncertainties of 6.2% and 7.1%, respectively.

$$Re = \frac{\rho U_{avg} D_h}{\mu} \quad (4.13)$$

$$\overline{Nu_p} = \int_0^{L_h} Nu dx / L_h \quad (4.14)$$

$$Nu = q''(x) D_h / k_f (T_w - T_m) \quad (4.15)$$

$$T_m = T_{inlet} + Q(x) / (\dot{m} C_p) \quad (4.16)$$

$$Q(x) = \int_0^x [k_f (dT/dy) \cdot 2 \cdot W] dx \quad (4.17)$$

$$C_f = (P_x - P_o) / (\rho U^2 / 2) \quad (4.18)$$

$$f = (-dP/dx) \cdot D_h / (\rho U^2 / 2) \quad (4.19)$$

The quantity $Q(x)$ represents the total heat transfer to the air from the inlet of the duct to the cross section under study in the experiment. The simulation only encompasses one rib pitch so the total heat added to the air includes only the module being modeled.

4.3.1 Results from the $Re = 103,775.5$ simulation:

Table 21 shows the room temperature density of air and the corresponding Reynolds number used in this study. Heat transfer and flow predictions for this high Reynolds number are compared to the correlations found in the experiment. Table 22 reveals that the predicted flow structure and heat transfer vary negligibly with the doubling of the mesh density. High Reynolds number simulations are insensitive to y^+ variations within the 30 to 100 range. The majority of the wall elements fall within this range. The predicted results are presented for the 16,000 node simulations when compared with the measured data.

Table 21. Properties and parameters used for $Re = 103,775.5$ simulations.

Density (kg/m^3)	1.165
Reynolds Number	103,775.5

Table 22. Mesh sensitivity study.

	8,000 nodes	16,000 nodes
Axial pressure gradient (Pa/m)	42475.77	46184.04
Friction coefficient (f)	0.121193	0.131909
Average Nusselt number ($\overline{Nu_p}$)	320.91	326.93
Mass flow rate (kg/s)	0.035742	0.035724
Average inlet velocity (m/s)	99.66	99.61
Reynolds number	103828.6	103775.5

The reattachment length found experimentally by Liou and Hwang (1992), shown in Table 23, is estimated based on the peak heat transfer within the rib pitch. Their reported profile of the measured heat transfer along the rib pitch clearly shows a peak value at that location. The table

shows that the model underpredicts the reattachment lengths. The much higher Reynolds number in the present study is expected to produce a larger recirculation cell and a longer reattachment length than the measured experimental value. This error is larger than the results from the previous modeled geometries which show a typical 12% underprediction of reattachment length by this simulation method. Table 24 reveals the recirculation cell upstream of the module's outlet rib.

Table 23. Steady-state, fully-developed reattachment lengths X_r/H .

	Present study	Liou and Hwang (1992)
Reynolds Number	103,775.5	12,600
Reattachment length (X_r/H)	3.75	4

Table 24. Structure upstream of the step at the module exit.

	X_r/H
Beginning location of circulation cell upstream of the step	-0.6

Table 25. Average Nusselt number for a smooth duct.

$\overline{Nu_s}$ from the simulation	181.3
$\overline{Nu_s}$ from the Dittus-Boelter Equation	205.42

The average Nusselt number for a smooth duct shown in Table 25 is determined by modeling a unribbed channel with the same hydraulic diameter as the ribbed geometry. The heat transfer analysis and averaging process follow the same procedure as the ribbed Nusselt number shown by equations (4.14) - (4.17). The results show 12% agreement with the Dittus-Boelter Equation. Table 26 shows that the results of the simulation are within 16% agreement with the average Nusselt number correlation. The results also show that the friction factor is well underpredicted but much larger than the smooth duct value shown in Table 27. The trend of the results is encouraging because the friction factor is shown to decrease with increasing Reynolds

number and the heat transfer is shown to increase. This fact is shown by Liou and Hwang (1992) for the Reynolds number range of 5,000 to 50,000 as presented in Table 28.

Table 26. Comparison of simulation results to experimental data.

	Present Study	Experimental
		Correlations
Average Nusselt number ($\overline{Nu_p}$)	326.9335	378.4534
Friction coefficient (f)	0.131909	0.318905

Table 27. Smooth duct friction factor from Blasius Equation.

Smooth duct Friction coefficient (f)	0.01761
--	---------

Table 28. Heat transfer and friction factor trends with Reynolds number.

	Experiment Re = 5,000	Experiment Re = 50,000	Simulation Re = 103,775.5
Average Nusselt Number ($\overline{Nu_p}$)	60	200	327
Friction factor (f)	0.55	0.35	0.13

Figure 21 shows the simulated pressure variation along the centerline of one fully-developed module for one rib pitch. The centerline pressure distribution calculated from the simulation (Equation 4.18) is very similar in form to the profiles provided by experiment. The data from Liou and Hwang (1992) shows this periodicity in pressure coefficient from module to module. The value at the inlet of the module is nearly identical to the peak located 6.3 rib heights down the module. The reported experimental data shows only the centerline periodicity of pressure. The results of this study reveal that the centerline distribution of x-component of velocity, turbulent kinetic energy, turbulent dissipation rate and pressure periodically repeat themselves from module to module.

Figure 22 shows the comparison of experimental to simulated heat transfer patterns in a fully-developed module. Although the experimental results are found for a Reynolds number of 12,600 and the simulation is for 103,775.5, certain patterns are similar. It should be noted that a

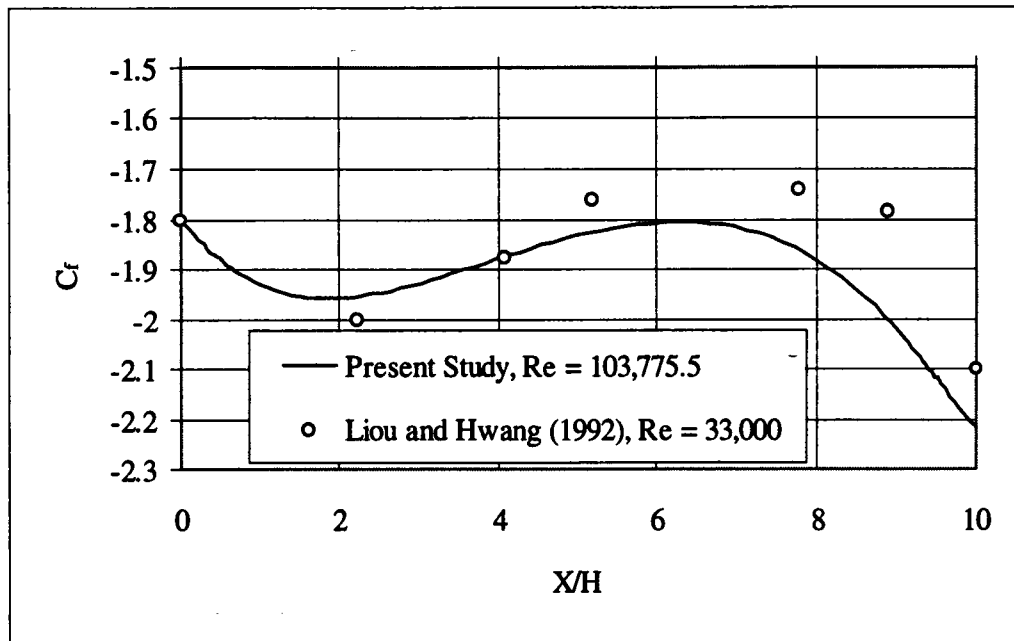


Figure 21. Centerline pressure coefficient from simulation with 16,000 nodes and Reynolds number = 103,775.5.

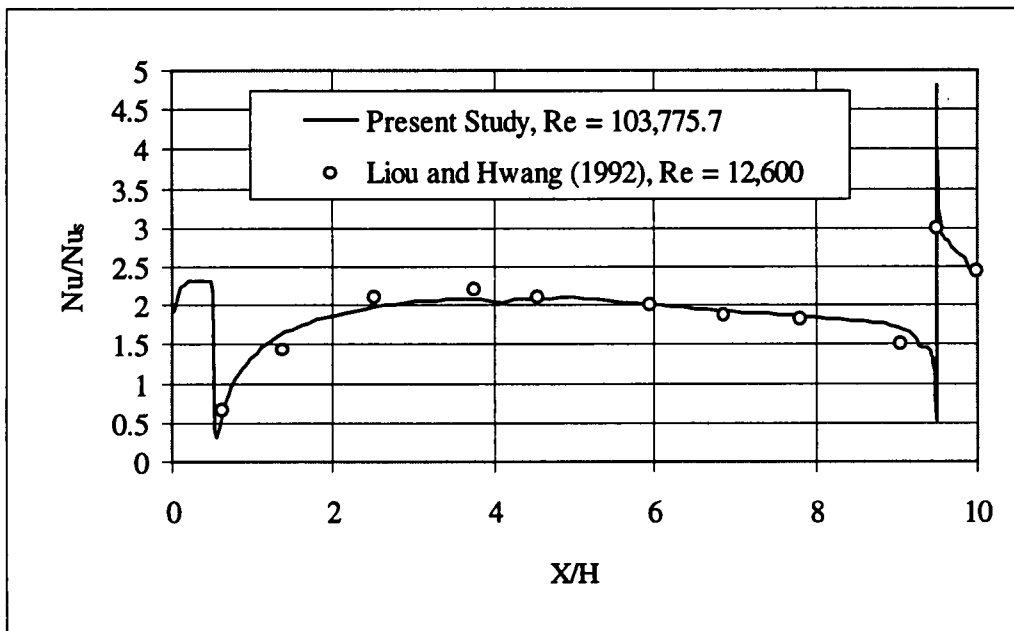


Figure 22. Average Nusselt number distribution within one module.

simulation of the low Reynolds number flow used in the experiment cannot be performed due to limitations of the $k - \epsilon$ model. Both show the sharp decrease in heat transfer at the ending edge of the inlet rib in the module and the sharp increase in heat transfer when the flow encounters the

leading edge of the outlet rib. Also, both show the expected low heat transfer region in the low speed recirculation zones after the inlet rib and before the outlet rib. The streamline plot of the simulated findings show a reattachment zone at $X_r/H = 3.7$, although the heat transfer profile lacks a distinct maximum. The structure implies that the entire region between the ribs is a recirculation cell. This is more consistent with what was expected for this high Reynolds number flow. The hot spots, where $Nu/Nu_s < 1$ as reported by Liou and Hwang (1992), appear to be more prevalent in the simulation results than in the experiments. The similarity of the magnitudes of the experimental data at 12,600 and the simulation results at 103,775.5 suggest that the ratio Nu/Nu_s is independent of Reynolds number.

Table 29. Turbulent characteristics of the flow field.

Pressure before the step (Pa)	$0.525 \cdot 10^4$
Maximum turbulence kinetic energy (m^2/s^2)	$0.2 \cdot 10^4$
Maximum dissipation rate (m^2/s^3)	$0.14 \cdot 10^{10}$
Maximum turbulent viscosity ($\text{kg/s}\cdot\text{m}$)	$0.76 \cdot 10^{-1}$

As shown in Table 29 the pressure is the highest before the flow impacts the outlet rib. The temperature contours after the inlet rib are consistent with the general structure of other models. The turbulent kinetic energy is the highest before the flow impacts the outlet rib. The dissipation rate follows the kinetic energy patterns and reveals its maximum before the outlet rib. The turbulent viscosity is the maximum at the module exit at the centerline.

4.3.2 Simulation with Reynolds number 51,887.75

A simulation of the geometry shown in Figure 20 is also performed with a lower Reynolds number of 51,887.75. The density of the fluid is halved from the room temperature value of air. The velocity, geometry and viscosity remain unchanged. This is done to allow the simulation of the geometry with a lower Reynolds number that represents the lower bound of capability of modeling

with this mesh and model. The result of halving the density decreases y^+ by 30%. Most of the wall elements can withstand this reduction in y^+ without decreasing below 30. The same fully-developed inlet profiles for the flow and turbulence structure degrees of freedom are used for the modules. The average temperature difference between the fluid and the walls is kept to 6 degrees Celsius to allow the use of constant properties with temperature.

Table 30. Properties and boundary conditions for lower Reynolds number simulation.

Wall temperature (K)	303
Inlet fluid temperature (K)	297
Fluid density (kg/m^3)	0.5825
Reynolds number	51887.75
Pressure drop in 1 module (static) (Pa)	23935
Pressure drop in 1 module (centerline) (Pa)	23104

The pressure drop across one module is measured in the experiment by static pressure taps located on the walls. The simulation shows that the pressure drop acquired at the centerline of the channel (dynamic) and the static pressure drop measured along the walls vary by a small percentage as shown in Table 30.

Table 31. Average Nusselt number for a smooth duct.

$\overline{\text{Nu}}_s$ from the simulation	104.13
$\overline{\text{Nu}}_s$ from the Dittus-Boelter Equation	117.98

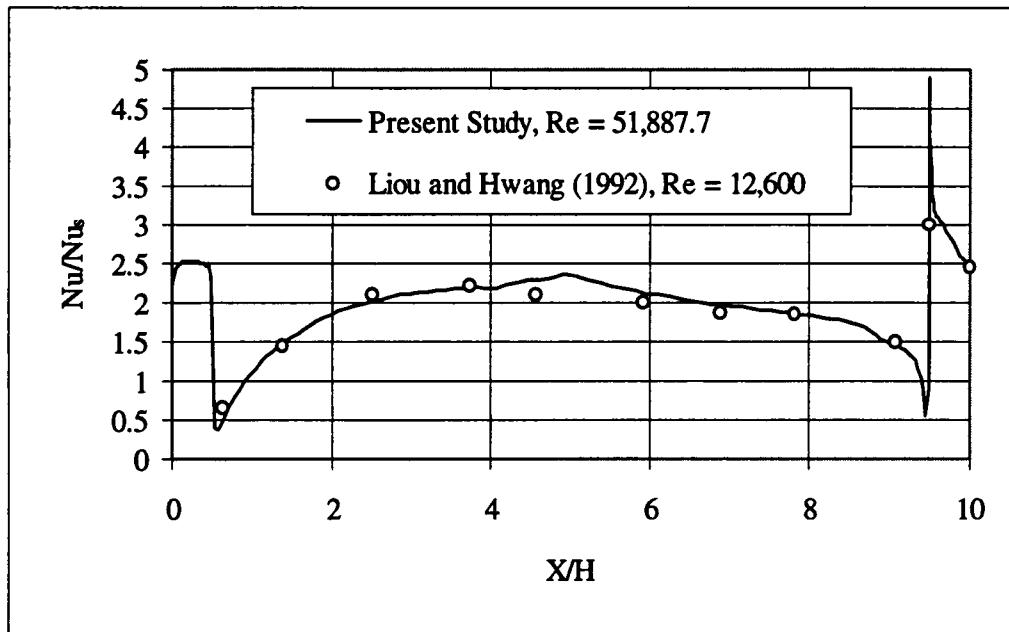
Table 32. Heat transfer and friction factor trends with Reynolds number.

	Experiment	Present study
Average Nusselt Number	239.515	197.7
Friction factor	0.3563	0.1367

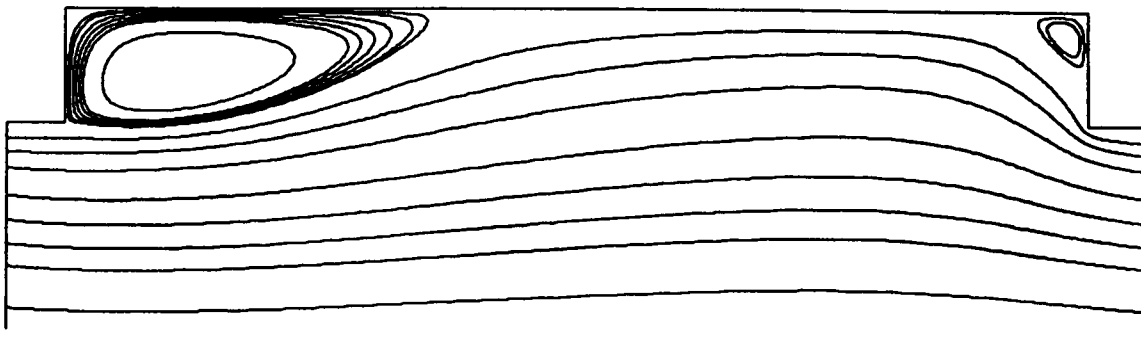
The predicted smooth duct Nusselt numbers are within 12% of the Dittus-Boelter equation as shown in Table 31. Shown in Table 32, the predicted average Nusselt number is within 17% from the experiment. The friction factors are one-third of the experimental values but are larger than that of a smooth duct. Both heat transfer and friction factor values are within the range of the smooth duct values and the experimental values as shown in Table 32 and 33.

Table 33. Smooth duct friction factor from Blasius Equation.

Smooth duct Friction coefficient (f)	0.021
--	-------

**Figure 23.** Average Nusselt numbers along the walls for Reynolds number = 51,887.7.

As mentioned above, the average Nusselt numbers as presented in Figure 23 imply that it is independent of Reynolds number. The experimental data is shown for a Reynolds number of 12,600. The y^+ values are within 30-40 for the majority of the wall areas. Figure 24 shows the streamlines in the fully-developed module. The values range from $-1.0 \text{ m}^2/\text{s}$ to $0.02 \text{ m}^2/\text{s}$ in 16 increments. Figure 25 shows the isotherms ranging from 297° Kelvin to $303.5^\circ \text{ Kelvin}$ in 0.5° Kelvin increments.

**Figure 24.** Streamlines in the fully-developed module.

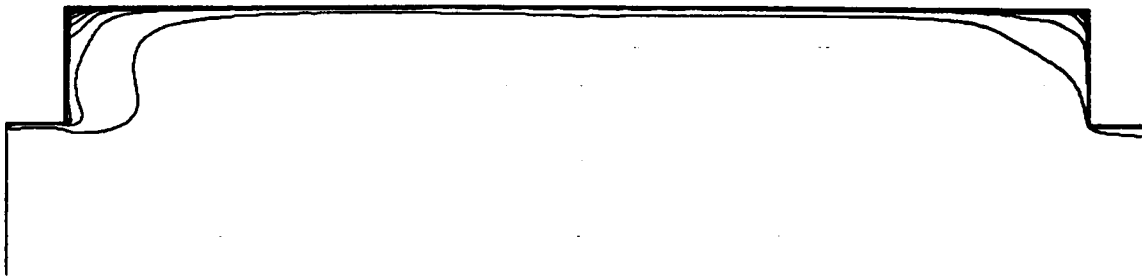


Figure 25. Isotherms in the fully-developed module.

4.3.3 Conclusions

The conclusions that can be drawn from the modeling of this configuration include the fact that the reattachment lengths at the higher Reynolds numbers are underpredicted. The resulting reattachment lengths from the $Re = 103,775.5$ simulations are less than the measured value at $Re = 12,600$. The reattachment lengths are expected to increase with Reynolds number. This result is similar to other findings using this numerical model and other $k-\epsilon$ models which have 20% underpredictions.

The friction factors and heat transfer characteristics of the simulations follow the trends of the experiments, although they are somewhat underpredicted. The predicted average Nusselt numbers for both Reynolds number simulations are within 17% of the experimental data. The friction factors for both simulations are underpredicted by 2/3 but are larger than smooth duct values. The smooth duct heat transfer characteristics are within 12% agreement with the Dittus-Boelter equation for both Reynolds numbers. Both the heat transfer and friction results from the simulations are bracketed within the smooth duct relations and the measured experimental values.

4.4 Discussion of the results for axisymmetric developing flow

The geometry studied experimentally by J. Vilemas, B. Cesna and V. Survila (1987) is modeled in this case and shown in Figure 26. The geometry consists of a concentric annulus with a rectangular ring mounted on the inner tube 41.3 hydraulic radii ($R = r_o - r_i$) downstream from the beginning of heating.

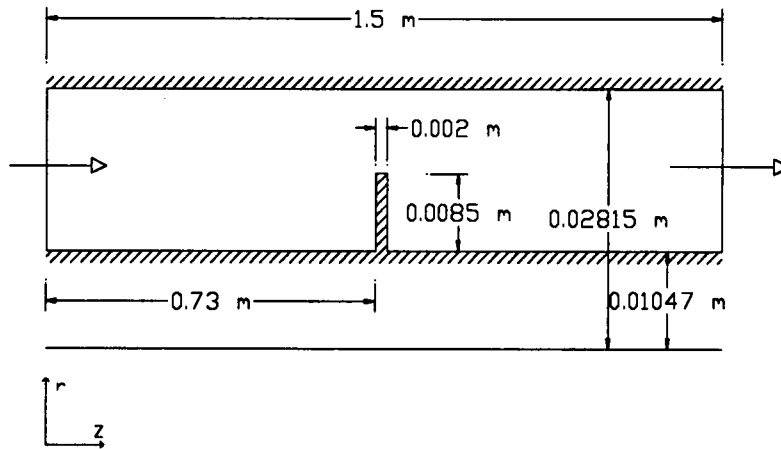


Figure 26. Geometry used by Vilemas, Cesna and Survila (1987)

The general assumptions are applied to this configuration, as well as the measured geometry and boundary conditions in the experiment. The Reynolds number based on the hydraulic diameter (Equation 4.15) is high enough to indicate a fully turbulent flow structure. The axisymmetric form of the governing equations is used to model this geometry. The axisymmetric assumption is used to eliminate the need for a three-dimensional (cylindrical) simulation. Thus the swirling velocity in the tangential direction is assumed to be zero. Tests were performed by Vilemas, et. al.(1987) on the experimental apparatus which confirm this assumption. The Mach number of the air is 0.2 which is small enough to be considered incompressible. Vilemas, et. al.(1987) report that the thermally developing conditions began at 41.3 hydraulic radii before the rib. They also confirm that the hydraulic conditions are fully-developed far upstream from the beginning of heating.

The experimental measurements show a tube-to-fluid temperature factor variation of 1.3 - 1.7 for the entire length of the tube, while the model used the range 1.1 - 1.7. This temperature factor requires a maximum temperature difference of 225 degrees Celsius between the inner tube and the fluid in the model. The use of variable fluid properties with temperature are used in the models due to this large temperature difference.

Since a measurable temperature variation along the length of the inner tube occurs in the experiment, the model uses the measured tube temperature profiles. The zero-gradient outlet of the model is placed 40 hydraulic radii downstream of the rib. The turbulent intensity at the inlet is assumed to have a value ranging from 1.0% to 2.0%. The measurements from the experiment did not include the turbulence parameters at the inlet, although they report this range measured by hot-wire anemometers as typical in their experiments. The outer tube temperature is not reported by the experimenters and is assumed to be constant at room temperature. The air flow is also assumed to be at room temperature before encountering the heated section.

This study includes maintaining the consistency of the geometry and the data measured in the experiment and varying the inlet turbulence intensities from 1.0% to 2.0%. As shown in previous simulations, the variation in intensity from 0.4% to 1.0 % has negligible effect on flow structure and heat transfer. The range of intensity is studied here due to the absence of this measured data. Table 34 shows the geometry and flow structure details. The maximum velocity reported by Vilemas, et. al.(1987) which was used in the experiment was 70 m/s. Using room temperature air properties, this corresponds to a Reynolds number slightly less than their given range. Table 35 shows the properties of room temperature air upstream of the heated section used to evaluate the inlet velocity from the given Reynolds number. As the air approaches the heated section, the thermal conductivity, specific heat and viscosity begin to vary with temperature based on tabulated values. The density always remains constant.

Table 34. Input parameters used in the simulation and measured values.

	Vilemas, et. al. (1987)	Present study(10,000 nodes)
Temperature factor	1.3 - 1.7	1.1 - 1.7
Reynolds Number	180,000 - 160,000	155,725.7
Length to rib (m)	0.73	0.73
Rib height (m)	0.0085	0.0085
Rib length (m)	0.0002	0.002
Inner diameter (m)	0.01047	0.01047
Outer diameter (m)	0.02815	0.02815
Total channel length (m)		1.50
Turbulence Intensity (%)		1.0, 1.5, 2.0
Inlet velocity (m/s)		70
Inlet turbulent kinetic energy (m ² /s ²)		0.735, 1.653, 2.94
Inlet dissipation rate (m ² /s ³)		305.8914, 1548.57, 4894.26

Table 35. Properties used in the simulation.

Fluid medium	air
Density (kg/m ³)	1.1614
Inlet Viscosity (N·s/m ²)	1.846·10 ⁻⁵
Inlet Thermal conductivity (W/m·K)	0.0263
Inlet Specific heat (J/kg·K)	1007
Hydraulic diameter (m)	0.03536
Hydraulic radius (m)	0.0176755

The following equations define some of the properties used in this configuration.

$$\text{Temperature Factor} = \frac{T_{\text{wall}}}{T_{\text{fluid}}} \quad (4.14)$$

$$Re = \frac{\rho U_i d}{\mu} \quad (4.15)$$

Where $d = 2 \cdot (r_o - r_i)$ is the hydraulic diameter and the fluid temperature is a bulk quantity defined by Equation (4.12). A larger thickness for the rib is used in the model due to limitations of the meshing procedure, as shown in Table 34. The reattachment length reported from the experiment is an approximate value determined by the point of maximum heat transfer. The predicted reattachment length from the simulation is also given at the maximum heat transfer position. The Nusselt number from the simulation is calculated based on the local heat flux, bulk

and wall temperature, variable fluid conductivity and hydraulic radius ($R = r_o - r_i$), as shown in Equation (4.16). The experimenters determined heat flux into the fluid from the current flow to the electric heaters, wall temperature from shielded thermocouples, bulk temperature from stagnation enthalpy and Nusselt number from (4.16).

$$Nu = \frac{\dot{q}_w'' R}{(T_w - T_f) k_f} \quad (4.16)$$

The fluid conductivity that the experimenters used in Equation (4.16) was not reported. For this study an average value, based on the varying wall temperature, is used due to the variable property effect ($k_f = 0.0388$ W/m·K). The fluid temperature used in this study is the bulk temperature defined by Equation (4.12).

4.4.1 Results:

It is observed from previous models of other configurations that the flow structure and heat transfer is independent of the number of nodes above 8,000. The predictions from the 10,000 node simulations of this configuration are observed to vary negligibly from 16,000 node simulations with the same criteria. Also, the results of high speed flow simulations are independent of small variations in wall element thicknesses, as long as y^+ remains between 30 and 100. The range for this configuration is between 30 and 100.

The heated section begins 1.0 meter downstream from the inlet of the channel. A constant velocity, temperature, kinetic energy and dissipation rate are used at the inlet. The profiles at 0.9, 1.0 and 1.1 meters are compared to determine if the flow is fully-developed at the beginning of the heated section. The profiles of the 4 degrees of freedom change negligibly in that range. The

presence of the variable properties with temperature causes the flow structure to begin to change beyond that point.

Table 36. Comparison of simulation and experimental results.

	Vilemas, et. al. (1987)	Present study I = 1.0%	I = 1.5%	I = 2.0%
Reattachment length (x/R)	42.4	43.48	43.48	43.48
Maximum Nusselt number	635	536.7	535.03	535.02

Table 37. Location of circulation cells upstream and downstream of the rib.

	Present study I = 1.0%	I = 1.5%	I = 2.0%
Beginning location of the circulation cell before the ring (X_r /rib height)	-0.337	-0.337	-0.337
Reattachment after the ring (X_r /rib height)	2.65	2.67	2.67

Table 38. Turbulence properties range for the present study.

	I = 1.0%	I = 1.5%	I = 2.0%
Minimum value of turbulent kinetic energy (m^2/s^2)	0.00346	0.00346	0.00346
Maximum value of turbulent kinetic energy (m^2/s^2)	3461.615	3457.114	3457.114
Minimum value of dissipation rate (m^2/s^3)	1.357	1.354	1.354
Maximum value of dissipation rate (m^2/s^3)	$8.83 \cdot 10^8$	$8.81 \cdot 10^8$	$8.81 \cdot 10^8$

Table 36 shows that the simulations predict the reattachment lengths within 2% of the experiment. It also indicates that the maximum Nusselt numbers are underpredicted by 15%. Table 37 shows that the recirculation cells exist before and after the rib. The recirculations cells are shown in Table 37 to be independent of turbulent intensity in the 1.0% to 2.0% range . The turbulent kinetic energy maximum levels shown in Table 38 occur at the top of the rib on the inlet side. The kinetic energy returns to its nominal pre-rib levels approximately 45 rib heights downstream of the disturbance. The dissipation rate maximum levels always occur in the same areas of maximum kinetic energy. Therefore, the maximum value occurs at the top of the rib and returns to normal at about 19 rib heights downstream. Table 38 shows the similarity between the 1.5% and 2.0 % turbulence ranges. The turbulent viscosity grows from the walls to occupy the entire channel about 0.433 meters downstream from the inlet. This indicates the beginning of the

fully-developed region. The turbulent viscosity increases significantly directly after the rib and recovers to its pre-rib levels approximately 90 rib heights downstream. The pressure decreases along the channel until the rib location where a large pressure gradient is observed to exist between the front and back of the rib.

Figures 27, 28 and 29 show that the heat transfer is underpredicted along most of the channel. The figures also show that the heat transfer predictions are insensitive to inlet turbulence intensity in the 1.0% to 2.0% range. As seen in Table 38, the turbulent magnitude range in the flow domains of each of the three models is similar. However, Table 34 shows that the prescribed inlet magnitudes are considerably different. The negative values of Nusselt number shown in Figures 27, 28 and 29 are caused by the heat transfer from the heated air to the cooler surface at those locations.

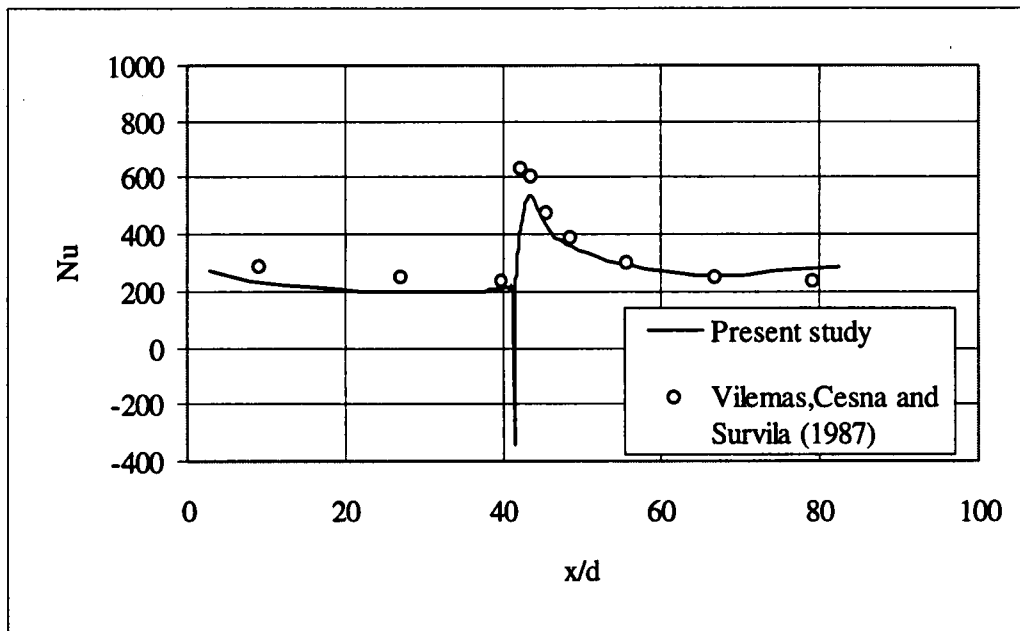


Figure 27. Predicted and measured heat transfer comparison for $I = 1.0\%$.

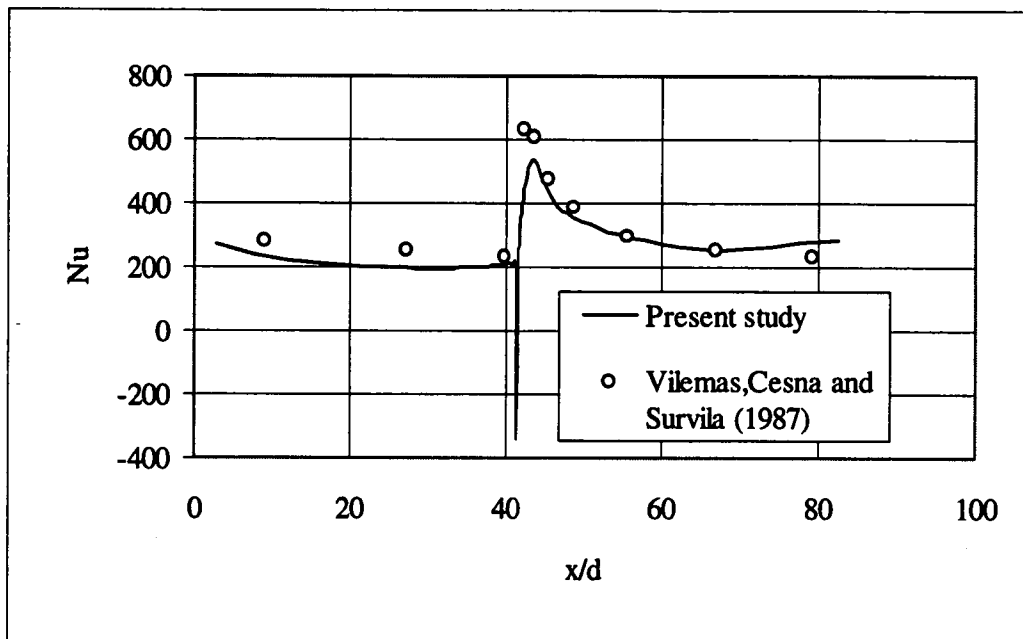


Figure 28. Predicted and measured heat transfer comparison for $I = 1.5\%$.

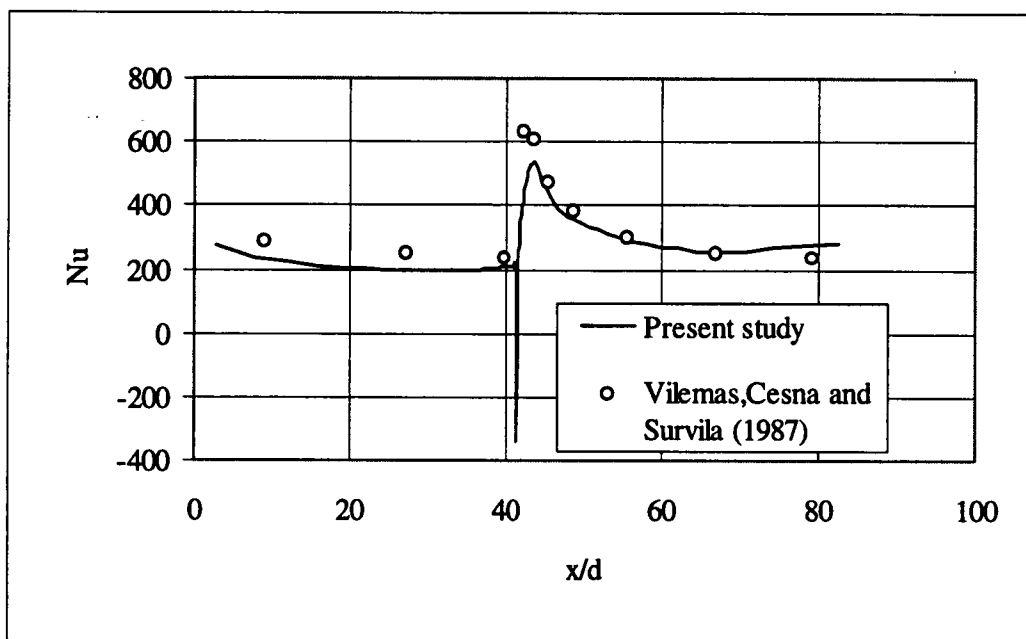


Figure 29. Predicted and measured heat transfer comparison for $I = 2.0\%$.

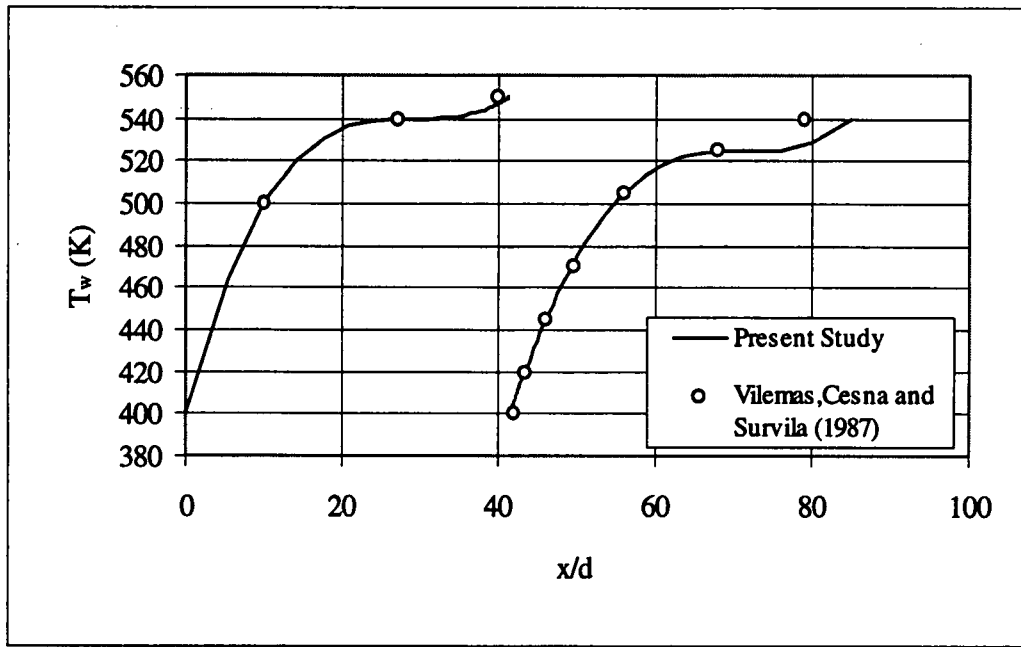


Figure 30. Model boundary conditions and measured wall temperatures.

Figure 30 shows the wall temperatures measured in the experiment and the wall temperatures input as boundary conditions into the model. The discrepancy in the two is due to the inaccuracy in the fitting of the data to third and fourth order polynomials. Figure 31 shows the streamlines in the vicinity of the rib. The values range from $-0.07 \text{ m}^2/\text{s}$ to $-0.05 \text{ m}^2/\text{s}$ in 15 increments. Figure 32 shows the isotherms ranging from 300° Kelvin to 540° Kelvin in 10° Kelvin increments, with the outer tube equal to 300° Kelvin and the inner tube follows Figure 30.

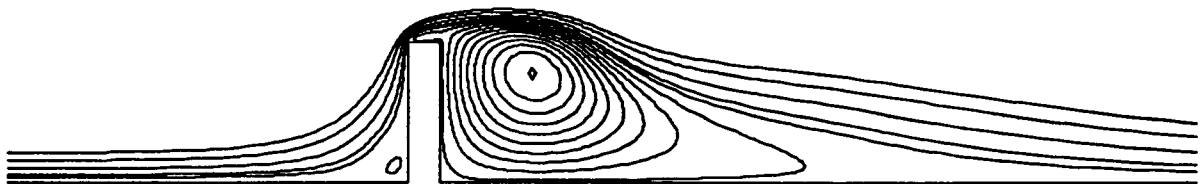


Figure 31. Streamlines in the vicinity of the rib.

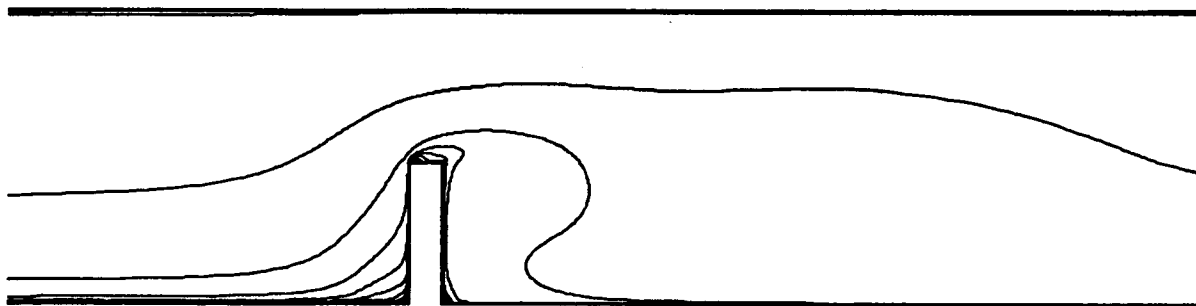


Figure 32. Isotherms in the vicinity of the rib.

4.4.2 Conclusions

The increase in turbulence intensity has been shown by other experiments to increase heat transfer and decrease the effect of variable properties. However, previous model predictions have shown the insignificance of turbulent intensity variation between 0.4% and 1.0% on the heat transfer results. The models of this configuration show a slight decrease in heat transfer as turbulence intensity is increased from 1.0% to 2.0%. The change is small enough to be neglected, and the heat transfer predictions are practically insensitive to turbulence intensities ranging from 0.4% to 2.0%. Overall the heat transfer predictions from this simulation closely approximate the measured values from the experiment. Also, the predicted reattachment length is in good agreement with the measured value.

Other simulations revealed that the predicted reattachment lengths are insensitive to mesh density and weakly dependent on Reynolds number. This configuration shows that the predictions are also independent of the turbulent intensity range from 1.0% to 2.0%. This configuration also shows that the range of turbulence kinetic energy and dissipation rate in the flow domain is nearly independent of the prescribed inlet conditions. Overall, the imposed inlet turbulence conditions within the studied range have little effect on the flow structure and heat transfer predictions of the models.

5. CONCLUSIONS

The many configurations modeled in this investigation include a diverse range of geometry and flow structure. Developing and fully-developed flow conditions with different levels of turbulence intensity are studied. Constant and variable temperature wall conditions and constant heat flux conditions are examined. The effects on the predictions of constant and variable properties is observed. Low and high Reynolds number flows are used and the relaminarization phenomenon is studied. The predicted results are compared to the measured data for each configuration. The following conclusions can be drawn from this investigation.

1. The flow structure and heat transfer predictions of the models are in general agreement with measured trends. Peak heat transfer near flow reattachment and a steady decrease as the boundary layer grows is always predicted by the models.
2. The recirculation cell and reattachment length results are underpredicted in most of the model configurations. The heat transfer predictions are usually larger than the measured data.
3. The predicted results of the simulations are found to be insensitive to mesh density above 8,000 nodes for all cases studied in this investigation.
4. Low speed flows are sensitive to the non-dimensional wall distance between 30 and 100 because the wall element thicknesses occupy a large portion of the channel.
5. High speed flows are insensitive to wall element thicknesses corresponding to non-dimensional wall distances between 30 and 100.
6. Good experimental measurements of geometry and boundary conditions are important for the reliable evaluation of the model predictions.
7. Zero-gradient outlet boundary conditions should always be used because of the absence of experimental measurements. The outlet location must be far downstream from the disturbance under consideration. For high Peclet number flows the outlet has negligible influence upstream.

8. This k - ϵ model has a total of six degrees of freedom. The convergence behavior of all but the dissipation rate is good. Generally the relative errors per iteration of dissipation rate are at least one order of magnitude larger than the others. If a solution method diverges, it usually occurs within the first ten iterations and is observed to have extremely high dissipation rates ($10^9 \text{ m}^2/\text{s}^3$) and turbulent kinetic energy relative errors ($10^6 \text{ m}^2/\text{s}^2$).

9. This high Reynolds number k - ϵ model does not accurately predict the relaminarization phenomenon. The low Reynolds number simulation results show larger levels of turbulence intensity than the measured data and higher Stanton numbers and friction coefficients. However, the predicted heat transfer and friction results are lower than those of standard turbulent pipe flows. The reduction in predicted centerline turbulence intensity along the length of the channel reveals a turbulence reduction phenomenon that is not present in standard flows. The predicted results of the relaminarization process are bracketed between the standard turbulent flow values and the experimentally measured near laminar values.

10. The nine empirical constants used in the model should be left in the standard form shown in Table 1. FIDAP reports that the predictions are very sensitive to the constants and that their recommended values should always be used.

11. The four node quadrilateral type of element should be used in all models because of the ease in controlling the height of the wall layers. The element layer adjacent to the wall has the universal near-wall approximations for velocity and temperature applied. The non-dimensional wall distance must be carefully maintained between 30 and 100.

12. The results from modeling the small backward-facing step in the large channel are different from the small symmetric channel simulation predictions. The mesh density in the region

of the step is similar in both cases. The accumulated errors from the more coarse mesh far from the step in the large channel contribute to the inaccuracy.

13. Experimental measurements show that the heat transfer and flow structure of turbulent flow change with increasing turbulence intensity. The results of this study indicate that the simulations predicted negligible changes in the flow within the prescribed inlet intensity range of 0.4% to 2.0%.

List of References

- Armaly, B. F., F. Durst, C. F. Pereira, and B. Schonung (1983). "Experimental and Theoretical Investigation of Backwards Facing Step Flow." *J. Fluid Mech.* Vol. 127, pp. 473-496.
- Aung, W., S. J. Chen, and F. K. Tsou (1991). "Starting Flow and Heat Transfer Downstream of a Backwards-Facing Step." *J. Heat Transfer.* Vol. 113, pp. 583-589.
- Aung, W., and R. J. Goldstein (1971). "Heat Transfer in Turbulent Separated Flow Downward of a Rearward Facing Step." *Israel J. Technology.* Vol. 10, No. 1-2, pp. 35-41.
- FIDAP Theory Manual (1984). Version 7.0, secs.10.7-10.12, 2.21-2.28.
- FIDAP Tutorial Manual (1984). Version 7.0, secs. 7.1-7.24.
- Holman, J. P. (1976). *Heat Transfer.* McGraw-Hill, New York.
- Jones, W. P., and B. E. Launder (1971). "The Prediction of Laminarization with a Two Equation Model of Turbulence." *Int. J. Heat Mass Transfer.* Vol. 15, pp. 301-314.
- Launder, B. E. (1964). "Laminarization of the Turbulent Boundary in a Severe Pressure Gradient." *ASME Journal of Applied Mechanics.* Vol. 31, pp. 707-708.
- Liou, T., and J. Hwang (1992). "Developing Heat Transfer and Friction in a Ribbed Rectangular Duct with Flow Separation at Inlet." *Journal of Heat Transfer.* Vol. 114, pp. 565-573.
- Moss, W. D., and S. Baker (1980). "Recirculating Flows Associated with Two-Dimensional Steps." *Aeronautical Quarterly.* Aug., pp. 151-171.
- Mutama, K. R. (1989). "The Investigation of Fluid Flow and Heat Transfer in a Converging Duct." M. Sc. Thesis. Faculty of Technology, University of Manchester, United Kingdom.
- Mutama, K. R., and H. Iacovides (1993). "The Investigation of Developing Flow and Heat Transfer in a Long Converging Duct." *ASME Journal of Heat Transfer.* Vol 115, pp. 897-903.
- Nguyen, T. M., J. M. Khodadadi, and N. S Vlachos (1989). "Laminar Flow and Conjugate

- Heat Transfer in Rib Roughened Tubes." *Numerical Heat Transfer*. Part A, Vol. 15, pp. 165-19.
- Patankar, S. V. (1980). *Numerical Heat Transfer and Fluid Flow*. Hemisphere, New York.
- Patel, V. C., and M. R. Head (1968). "Reversion of Turbulent to Laminar Flow." *J. Fluid Mech.* Vol. 34, Pt. 2, pp. 371-392.
- Rowley, G. P., and S. V. Patankar (1984). "Analysis of Laminar Flow and Heat Transfer in Tubes with Internal Circumference Fins." *Int. J. Heat Mass Transfer*. Vol. 27, pp. 553-560.
- Seban, R. A. (1964). "Heat Transfer to the Turbulent Separated Flow of Air Downstream of a Step in the Surface of a Plate." *ASME Journal of Heat Transfer*. Vol. 42, pp. 259-264.
- Stevenson, J. R., and R. V. Arimilli (1994). "Laminar Flow and Heat Transfer in the Entrance Region of Ribbed Concentric Annuli." *HTD*. Vol. 124. ASME. Pp. 59-67.
- Vilemas, J., B. Cesna, and V. Survila (1987). *Heat Transfer in Gas-Cooled Annular Channels*. Eds. A. Zukauskas and J. Karni. Hemisphere, Washington, D. C. Pp. 134-139.
- Vogel, J. C., and J. K. Eaton (1985). "Combined Heat Transfer and Fluid Dynamic Measurements Downstream of a Backward-Facing Step." *Journal of Heat Transfer*. Vol. 107, pp. 922-929.
- White, Frank M. (1991). *Viscous Fluid Flow*. McGraw-Hill, Inc., New York.
- Wilcox, D. C. (1993). "Comparison of Two-Equation Turbulent Models for Boundary Layers with Pressure Gradient." *AIAA Journal*. Vol. 31, No. 8, pp. 1414-1421.
- Yung, Chain Nan (1988). "Numerical Simulation of Axisymmetric Turbulent Flow in Combustors and Diffusers." NASA Contractor Report 4115, pp 1-38.

VITA

Colby Russell Evans was born in Chattanooga, Tennessee on April 4, 1970. He attended Hixson Elementary School in Chattanooga, Tennessee and the Farragut school system in Farragut, Tennessee. He entered the University of Tennessee, Knoxville in 1988. He spent 2 years in the College of Business Administration and 4 years in the College of Engineering. He also worked part-time at Sam's Wholesale Club in Knoxville for 5 years while attending college. He graduated with a Bachelor of Science in Mechanical Engineering in 1994. He then entered graduate school at The University of Tennessee to pursue a Master of Science Degree in Mechanical Engineering. He is a member of Tau Beta Pi and Pi Tau Sigma honor societies.