LM-03K110
October 20, 2003

Experimental and Numerical Investigation of Impinging Jet Flow in Square Ducts Intersecting at 90°

David Corson & Peter Vassallo

NOTICE

This report was prepared as an account of work sponsored by the United States Government. Neither the United States, nor the United States Department of Energy, nor any of their employees, nor any of their contractors, subcontractors, or their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness or usefulness of any information, apparatus, product or process disclosed, or represents that its use would not infringe privately owned rights.

Experimental and Numerical Investigation of Impinging Jet Flow in Square Ducts Intersecting at 90°

David Corson
e-mail: corsoda@kapl.gov

Peter Vassallo
e-mail: vassallo@kapl.gov

Lockheed Martin Corporation
P.O. Box 1072
Schenectady, NY 12301-1072

Abstract

An experimental and numerical investigation has been conducted on flow through two square ducts with a 2:1 hydraulic diameter ratio joined at a right angle. Measurements of the velocity field were acquired using a laser Doppler velocimeter at various planar locations throughout the ducts at a nominal Reynolds number of 68,000. Pressure drop measurements were taken for 3 Reynolds numbers between 46,000 and 93,000. Computational fluid dynamics (CFD) analyses were performed using STAR-CD to determine how well the experimental data could be predicted using the k - ϵ , k - ϵ RNG, k - ϵ Chen, k - ϵ quadratic, k - ω , and Spalart-Allmaras models. The results show that there are distinct differences in the CFD results. The standard k - ϵ model overpredicted the loss coefficient by 4% and underpredicted the exit swirl magnitude by 43%. The best predictor of the swirl decay was found to be the k - ω model, which adequately followed the data throughout the entire geometry and underpredicted the exit swirl by 16%. The

best overall model was found to be Spalart-Allmaras, which overpredicted the loss coefficient by 2% and underpredicted the exit swirl magnitude by 40%.

1.0 Introduction

Complex hydraulic systems generally contain a variety of turbulent flow features including stagnation regions, accelerating flow, separated flow, swirling flow and jet/wall interactions. All of these features affect the overall system pressure drop, which is one of the primary concerns of a design engineer. Other concerns may include noise generation, vibration effects and rates of heat transfer. In order to optimize the system design or understand how a given system will behave, it is necessary to develop tools that can adequately predict the turbulent flow in the system. CFD methods are increasingly being used to accomplish this goal.

Over the last several decades, CFD techniques and capabilities have grown to the point where extensive hydraulic design is first accomplished via code/modeling approaches in order to reduce dependence on expensive testing. Selected and more limited testing may be performed during later stages to qualify key results. In many previous studies, particular turbulent flow features have been tested and compared to CFD results to better understand code limitations and preferred modeling strategies. Some examples of these studies include Iaccarino [1], Guo, *et al.* [2], Mentor [3] and Freitas [4]. In addition, Wilcox [5] has assembled a thorough compilation of turbulence models and compared CFD results incorporating these models with pertinent data. Some examples of this data include free shear flows (i.e., wakes, mixing layers, jets) and wall bounded flows (i.e.,

backward facing steps, channel and pipe flow, boundary layers). In most cases, the qualification studies have focused their attention on a single dominant flow feature so that subsequent relationships between models and data can be more easily inferred. In addition, these studies have mainly been concerned with the behavior of the flow in two dimensions.

The purpose of this work was to provide a qualification database in a three dimensional flow where multiple flow features are occurring - features that are generally not predicted well by commonly used turbulence models. For this investigation, the flow consisted of an impinging jet contained within a square duct. As the jet flows through the inlet duct, it impinges against the wall of the second duct, acting as a swirl generator and creating numerous secondary cells that dissipate down the length of the second duct. To characterize the flow field, detailed 3-D laser Doppler measurements and pressure drop data were obtained at various locations within the ducts. The data was then used to assess a variety of turbulence models available (and functional) in the finite volume code known as STAR-CD, distributed by Computational Dynamics. The models consist of four two-equation high Reynolds Number closures utilizing the standard Boussinesq approximation, a nonlinear two-equation model, and a single equation closure model.

2.0 Experimental Details

The test section consisted of 3.81 cm and 7.62 cm square ducts fabricated from optically clear 12.7 mm thick acrylic sheet. The test section was set up in a water flow loop

containing a pump, pressurizer, heater, chiller, flow meters, air extractor, and seed filter. The working fluid was deionized water, preheated to remove air from solution. Any residual air was removed by passing the flow through an air extractor before data acquisition. To enable LDV measurements, 10 μm diameter latex seed was added with a syringe through a seed injection port located at the inlet of the 3.81 cm duct. The water was filtered before the addition of the seed particles to ensure that it was free of contaminants.

A diagram of the test setup is shown in Fig. 1. The experiment was designed such that flow had sufficient time to develop in the small side duct before entering the main duct and impinging against the back wall. This was accomplished by building the assembly with sufficient side duct length to achieve fully developed flow at its outlet. A Vortab flow straightener installed at the entrance of the inlet duct effectively removed any skew or rotation present in the flow resulting from the 90° elbows present in the upstream piping system.

The acrylic walls of the test section were glued together such that the width of the channel fell within a tolerance of ± 1.27 mm. The pressure taps were positioned along the centerline of the ducts on the top wall in the main duct, and along the side wall in the inlet duct. The taps were drilled with a hole diameter of 0.8 mm. The centerline spacing for the pressure taps were 30.48 cm in the side duct, and 34.29 cm in the main duct. Care was taken to remove any burrs from the taps. The integrity of the holes was confirmed

by observing the quality of the small fluid jets produced by forcing water backwards through the pressure taps into the drained test section.

The fluid properties used in all of the analyses were based on a nominal loop temperature of 26.7° C and pressure of 137.9 kPa. The loop temperature was normally maintained within 1° C, and the pressure was maintained within 6 kPa.

2.1 Flow Rate Measurements

At the beginning of testing, the loop flow rate was set using an in-line Venturi with a large uncertainty over the flow rates of interest (i.e., $189.3 < Q < 757.2$ lpm). Midway through testing, an orifice plate was installed which allowed lower uncertainties in the flow rate measurements; however, this required an increased pump speed to maintain the desired flow due to the increased resistance of the orifice plate in the loop. The Venturi was used to set the equivalent flow rate, and ongoing LDV data confirmed the similarity of the results. Subsequent LDV measurements of the velocity profiles in the fully developed entrance duct revealed a 3.5% difference in flow rate between the two pump speeds. It was decided that the best estimate of the loop flow rate would be obtained from an average of the orifice plate data and the entrance duct LDV data at the higher pump speed, with an uncertainty encompassing the variability in the LDV data for both pump speeds.

Table 1 summarizes the flow conditions studied. Extensive LDV data was taken at the middle flow rate (277.1 lpm) to provide a database for subsequent CFD comparisons. The

Reynolds number associated with this flow rate falls well into the turbulent regime for duct flow, and provides a desirable velocity field for evaluating the effectiveness of turbulence models. Detailed pressure drop data was taken at all three flow rates to confirm the relative insensitivity to Reynolds number over a Reynolds number range.

Table 1: Flow rate measurements for each test condition (in lpm). The averaged flow rates were calculated based on the flow measured with the orifice and the flow determined from LDV data taken in the entrance duct at the end of testing.

Orifice Measurement	LDV Measurement (start of test)	LDV Measurement (end of test)	Averaged Flow Rate	Reynolds #	Data Taken
274.9 (± 7.5 lpm)	269.9 (± 8.3 lpm)	279.4 (± 8.3 lpm)	277.1 ($\pm 2.5\%$)	~ 68000	LDV, ΔP
184.8 (± 7.5 lpm)		189.3 (± 5.7 lpm)	187.0 ($\pm 2.5\%$)	~ 46000	ΔP
377.1 (± 7.5 lpm)		383.1 (± 11.3 lpm)	380.1 ($\pm 2.5\%$)	~ 93000	ΔP

2.2 Pressure Drop Measurements

The static pressure within the two ducts was measured at various tap locations along the ducts (see Fig. 1) using a Rosemont 1151 transducer (0 to 7.45 kPa range). The transducer was calibrated using a dead weight reference to achieve an accuracy of $\pm 0.5\%$ of reading. Flexible tubes of 3.175 mm outer diameter were connected from the test section pressure taps to the Rosemont transducer via a scanning valve mechanism. The desired pressure drop in the test section was measured by selecting the appropriate taps using the scanning valve, and then acquiring and averaging individual pressure

measurements at 50 Hz over a 60 second time period. Zero flow measurements confirmed the transducer zero point to within 3.45 Pa.

The pressure drop data for all 3 flow conditions were correlated by a loss factor, K , as follows:

$$K = \frac{\left[(P_i - P_{ref}) + \frac{1}{2} \rho (V_i^2 - V_{ref}^2) \right]}{\frac{1}{2} \rho V_{ref}^2} \quad (1)$$

where: P_{ref} = pressure at reference tap #2,

V_{ref} = average velocity at reference tap #2,

ρ = fluid density,

P_i and V_i = pressure and velocity at the given downstream tap.

The loss factor above corrects for the change in static pressure due to the change in flow area between the two ducts and includes the irrecoverable losses due to the intersection as well as frictional losses along the duct. The uncertainty in K was determined using an error propagation of Eq. (1), and knowledge of the individual uncertainties of the pressure and velocity measurements. The overall uncertainty in pressure drop measurements, including transducer and data acquisition uncertainties, is estimated to be ± 15.17 Pa. The uncertainty in velocity is taken to be the same as the uncertainty in flow rate, or $\pm 2.5\%$. This yields a typical uncertainty for $Q=277.1$ lpm of ± 0.035 for K in

the large duct and ± 0.032 for K in the small duct. The extent of the symbols in Fig. 2 reflects these uncertainty magnitudes.

Fig. 2 shows the loss factor plotted as a function of downstream distance from the reference tap. The distance was calculated as the projected distance in the streamwise direction in the X-Z plane along the test section centerline (i.e. the change in Y coordinate between the two ducts was neglected). Two sets of data were taken about a month apart; the agreement between both data sets is good. All the data follows a similar trend: a frictional loss in the entrance duct, followed by an irrecoverable loss at the intersection, and finally a frictional loss in the large duct. The Blasius correlation for smooth walled ducts, taken from Shames [6] is used to confirm the slope of the frictional losses in the small and large ducts (for $Q=277.1$ lpm). The Blasius correlation slightly overpredicts the slope of the entrance duct loss and underpredicts the slope of the main duct loss. The latter is likely due to the presence of swirl in the main duct, which would increase the pressure drop somewhat. The loss coefficient at the duct intersection is seen to be independent of Reynolds number; only in the straight ducts, especially in the smaller duct, does the loss coefficient increase slightly with decreasing Reynolds number. This is due to the higher friction factors at the smaller Reynolds numbers.

2.3 Laser Doppler Velocimetry (LDV) Measurements

The velocity field within the impinging jet test section was acquired at numerous planes surrounding the region of intersection as well as at a number of downstream locations.

The velocity profiles were measured using a 2-D Dantec LDV system and a TSI fiber optic probe. Two pairs of laser beams of wavelength 488 nm (blue) and 514 nm (green) from a 2 W Argon laser were coupled through the fiber optic probe and focused using two different lenses. Table 2 shows the length and diameter of the measurement volume for the different lenses.

Table 2: Lens specifications for LDV measurements.

Focal Length (mm)	Beam	# of Fringes	Diameter (mm)	Length (mm)
350	Green	11	0.053	0.58
	Blue	11	0.051	0.54
250	Green	11	0.039	0.30
	Blue	11	0.036	0.28

The 250 mm focal length lens was used in the smaller diameter entrance duct while the 350 mm lens was used for all other data. A TSI calibration wheel rotating at known constant speed was used to determine the fringe spacing for each probe volume to within $\pm 0.5\%$. The power of the blue and green beams exiting the probe was 20-50 mW for each beam.

The probe was mounted and leveled (within 1°) onto a 3 axis traverse which was controlled through a software program that interfaced directly with the LDV data acquisition system. Care was taken to align the traverse axes with the test section axes to within 1° . A square measurement grid was input into the LDV controller software having 529 points (23 x 23), with an equal spacing of 3.0711 mm between points in both the

horizontal and vertical directions. All points in the grid were acquired with the LDV probe mounted horizontally, such that it measured the axial (W) velocity and the vertical (V) velocity components. The same grid was also acquired with the probe mounted vertically, with the beams entering the test section from above. This measurement yields the axial (W) and horizontal (U) components of the velocity. The probe was realigned to each time it was moved such that it was centered in the square duct before beginning to acquire each plane. Centering in the direction parallel to the beams was accomplished by measuring the distance between beams as they entered the test section and as they exited on the other side. Assuming that the test section is symmetric within reasonable tolerances, the probe is centered when the beam separation on both sides of the test section is equal. The probe alignment in the direction normal to the beams was accomplished by simply centering the measurement volume between the edges of the test section. In most cases, it was possible to align the probe in the same position when acquiring the cross-section from both orientations using the procedure described above. However, it was occasionally necessary to make minor adjustments by shifting the beam slightly in the X or Y direction until the axial velocity measured from both orientations matched within approximately 5 percent of the mean velocity at that location. In highly turbulent regions with large local gradients, the tolerance could not always be held to this level.

Two Dantec Burst Spectrum Analyzers (BSA) were used to process the signals returned by the fiber optic probe. The processors were run in coincidence mode (with a 30 μ s coincidence window) to acquire the Reynolds shear stresses. A 40 MHz Bragg shift was

employed to measure reverse velocities. Filters of ± 15 m/s were used in the processors to improve signal quality without excluding any potential data. The velocity statistics were obtained by acquiring data over a time of 240 seconds at each point, from each probe orientation. A typical sample contained between 2,000 and 5,000 points.

2.3.1 Uncertainties in Velocity Measurements

The main source of random uncertainty in each individual LDV measurement is associated with statistical uncertainty. From Benedict and Gould [7], for a 95% confidence interval of uncorrelated samples with a normal distribution, the uncertainties in velocity statistics are given as:

$$\epsilon_u = \pm 1.96 \left(\frac{u'}{\sqrt{N}} \right) \quad (2)$$

$$\epsilon_w = \pm 1.96 \left(\frac{u' \sqrt{2N}}{u'} \right) \quad (3)$$

$$\epsilon_{uv} = \pm 1.96 \{ [u'^2 v'^2 + uv'^2] / N \}^{1/2} \quad (4)$$

where N is the number of samples in the average, U is the mean velocity, u' is the fluctuating or RMS velocity, and uv is the Reynolds shear stress. The validity of Eqs. (2)-(4) for the current data was confirmed by acquiring data for numerous repeat

measurements at various locations in the flow field and then analyzing for statistical comparison. A total of 34-40 repeat points at a given position were taken to obtain average statistics, then the furthest outliers in both directions (+/- %) were removed to obtain an overall uncertainty for 95% confidence. In general, it was seen that the random uncertainties in the LDV measurement at a single point were reasonably predicted by Eqs. (2)-(4). Therefore, a reasonable estimate of the random uncertainty at any given point in the flow field may be obtained by inserting the values of mean velocity, fluctuating velocity and shear stress measured at that point along with a nominal sample number (taken as 2000) into Eqs. (2)-(4). Typical uncertainties at $X=11.53\text{mm}$ (near the impingement point) normalized by the bulk mean velocity ($U_B=0.78\text{ m/s}$) are: +/- 5.3% for the mean, +/- 4.2% for fluctuating velocity and +/- 26.3% for the square root of the shear stress. Typical normalized uncertainties at $X=37.04\text{mm}$ (in the downstream swirl region) are: +/- 0.7% for the mean, +/- 0.5% for fluctuating velocity and +/- 11.6% for the square root of the shear stress.

One potentially large source of bias error, especially in highly turbulent flows, is velocity bias error. This error is based on the premise that, for uniform seed density, more particles will be measured when the local velocity is high compared to when it is low, and a bias toward the higher velocities will result when an average is taken. Several methods may be used to eliminate this bias, including correction schemes to post-process the data or equal-time sampling during data acquisition. The Dantec processor and software offer both a transit time correction scheme and dead-time acquisition option. In evaluating these methods, the dead-time single-burst mode was first used to obtain a

baseline measurement for 2000 samples taken at a rate of 4 per minute. According to Erdmann and Tropica [8] as well as TSI [9], this approach will yield unbiased results if the dead-time interval is sufficiently long with respect to the validated particle interarrival time. In this case, the sampling rate was 0.067 Hz and the average particle data rate was 10-50 Hz, implying a ratio of at least 150 between dead-time interval and particle interarrival time (TSI recommends at least a ratio of 10). This data was then compared to single-burst random data taken at higher data rates (between 10 and 50 Hz), with and without transit time correction. The comparisons showed that the uncorrected velocity statistics using single-burst mode was in better agreement with the statistics calculated using the dead-time mode, especially at higher turbulence intensities (the correction scheme decreases both the mean and fluctuating velocities by amounts proportional to the turbulence intensity). According to Edwards [10], the proper use of residence time correction should yield accurate results; however, he points out that obtaining accurate measures of the transit time is difficult. Based on the data comparisons, the results reported herein were calculated using the single-burst mode processing scheme with no transit time correction employed.

Because all three components of velocity were not measured simultaneously, there will be additional random uncertainty in "matching" the velocity data taken with the LDV probe in one configuration with the data taken with the probe in its orthogonal configuration. This uncertainty results primarily from uncertainties in absolute positioning of the measurement volume (± 0.5 mm) as well as the change in orientation of the measurement volume. Since the streamwise component was acquired for both

LDV probe configurations, one way to assess this potential error is to examine the relative difference between these measurements. An example of this data near the impingement point, normalized by the bulk mean velocity, is shown in Fig. 3. In general, the relative difference is within $\pm 5\%$, which is the approximate random uncertainty of a single measurement. There are some regions where the relative difference is larger, about $\pm 10\%$, and this may be attributed to residual positioning errors or perhaps spatial averaging errors due to the change in probe volume orientation. The positioning uncertainty ($\pm 0.5\text{mm}$) and probe volume length (0.6mm) may be used with the mean velocity gradients at any given position to more accurately determine this random uncertainty if the application demands it.

A total of 21 cross-sections (X-Y plane) were acquired in the main duct of the test section. The bulk of the measurements were taken in the region extending up to $Z=381\text{mm}$ from the back wall of the duct. In this region, the inlet flow impinges against the side wall of the main duct, and generates a large amount of swirl. In addition to this, multiple small recirculation regions are created. These regions exist on both sides of the centerline of the inlet duct. As the flow moves away from the inlet duct, these regions begin to collapse, and the flow converges into a single large swirling region. Measurements were also taken well downstream of this mixing region to reflect the rate of swirl decay, and the motion of the vortex center as the distance from the back wall increased.

3.0 CFD MODELING

All of the turbulence models studied are available in STAR-CD, a commercial CFD solver. Computations were performed on the same computational domain using the different turbulence models available within the code. The computational domain consisted of 343 mm (9 inlet hydraulic diameters) of the inlet duct, and 1524 mm (20 main duct hydraulic diameters) of the main duct. Preliminary measurements and calculations showed that this length of domain was sufficient to capture all of the recirculation regions that were present as a result of the impinging jet region. The flow was seen to evolve into a single vortex by approximately $3D_M$ in the main duct. The remainder of the domain in the streamwise direction was sufficient to provide a good determination of the rate of swirl decay predicted by the turbulence models.

The turbulence models evaluated in STAR-CD consisted of the high Reynolds Number versions of the Launder-Sharma k-e model (also known as the "standard" k-e model), Wilcox's k- ω model, the k-e RNG model, Chen's k-e model, the Spalart-Allmaras model, and the nonlinear, quadratic k-e model. With the exception of the quadratic model, all of these models utilize the Boussinesq approximation to model the Reynolds stresses. A summary of the turbulence models used is provided in Table 3.

Table 3: Summary of turbulence models used. All models are commercially available in STAR-CD.

The computational mesh consists of a hexahedral mesh generated with GridPro. The mesh was created with stretched elements in the streamwise direction sufficiently far downstream of the impinging region to keep the mesh size reasonable. A mesh sensitivity study was performed to determine the mesh density required to produce a grid converged solution. A solution was obtained on four different meshes of decreasing mesh spacing using the standard high Reynolds number turbulence model. The results were then analyzed by making line plots of all 3 velocity components and non-dimensional swirl number and loss coefficient at different locations in the flow to determine whether grid convergence had been achieved. Examination of the line plots between the last two grids tested revealed comparable results. Fig. 4 shows the detail of the final mesh in the region of intersection of the two ducts.

3.1 Computational Domain

Turbulence Model	References	Notes
Standard k- ϵ	Lauder and Sharma [11]	Most commonly used 2-equation high Reynolds Number model.
k- ϵ RNG	Yakhot and Orszag [12]	May perform better than k- ϵ in flows exhibiting rapid distortions and highly strained regions.
k- ϵ Chen	Chen and Kim [13]	Similar to RNG, containing additional strain-dependant source terms in the ϵ equation.
Quadratic k- ϵ	STAR-CD Methodology [14]	Only nonlinear model evaluated in this analysis. Accounts for anisotropy of Reynolds Stresses.
k- ω	Wilcox [5]	2-equation model known to perform better in swirling flows than k- ϵ .
Spalart-Spallart and Allmaras [15]		Complete single equation eddy viscosity transport model.

Due to the fact that the near wall region was not clustered for the high Reynolds number turbulence models, the near wall spacing was set based on the grid density. The ideal near wall spacing corresponds to a y^+ value falling in the range of 30-300. If the spacing is such that the y^+ is out of that range, the traditional "Law of the Wall" may not apply, potentially making the solution in that area incorrect (most notably the wall shear). The large range of velocities present within the impinging jet model made it difficult to satisfy the requirements on y^+ throughout the entire domain. The mixing region of the model poses a problem due to the high velocity inlet flow impinging against the side wall of the main duct. This region requires very close grid spacing to keep y^+ within the desired range. However, well downstream of this area, the flow has evolved into a rotating vortex centered roughly in the middle of the duct. The velocities near the wall

3.1.1 Near Wall Spacing

As can be seen in Fig. 4, the mesh near the intersection of the ducts is quite dense. The spacing in the J direction is approximately 45 wall units, while the spacing in the other two directions is approximately 70 wall units. The mesh begins to stretch axially at approximately 5 D_M from the back wall of the main duct, then again at 10 D_M to reach the element geometry at the outlet boundary. Through these regions, the aspect ratio of the mesh transitions smoothly from ~1 to ~5, then again from ~5 to approximately 18.5, which represents the peak aspect ratio within the model. The final mesh contains 1,661,040 cells and 1,720,848 vertices.

3.2 Boundary Conditions

are a fraction of what they are in the entrance region, however the near wall spacing is the same due to the topology used to build the mesh. This effect caused the y^+ values within the solutions to vary between a lower bound of ~ 12 and an upper bound of ~ 250 . The upper bound value falls within the acceptable range for near wall spacing; however, the lower value does not. The effect of this was investigated by building two more periodic duct models. One of the models had a near wall spacing corresponding to a y^+ of ~ 60 , and the other with a y^+ of ~ 10 . The only differences in the mesh is that the one with tighter near wall spacing was clustered. Both models were run with a fixed mass flow rate, and the resulting pressure drop between the periodic faces was examined. The results showed that the pressure drop between the two models was within $\sim 5\%$, with the lower y^+ yielding the slightly higher pressure drop.

Although the effect of the wall functions on the pressure drop is noticeable for the range of y^+ present in the models, it is not significant. The majority of the pressure drop occurs in the main duct at the inlet region, and the requirements for near wall spacing are satisfied in this area. The discrepancies in the wall spacing do not appear until approximately $6 D_M$ downstream of the inlet. Therefore, the pressure drop due to the intersection of the ducts is accurate (in terms of wall function validity), but the pressure drop in the region of swirl dissipation may be slightly high. However, the extent of this effect is judged to be negligible.

The length of the inlet duct in the CFD model is not sufficiently long to develop into a fully developed flow profile before encountering the intersection region (given a flat profile on the inlet boundary). However, test data shows that the inlet duct flow is fully developed as it reaches the intersection. The effect of this on the solution was investigated by performing two different simulations. One simulation utilized an inlet boundary with a planar profile for the velocity and turbulence quantities. The other consisted of a prescribed inlet boundary corresponding to a fully developed flow in the inlet duct. The results of this exercise were compared and it was found that the two different profiles on the inlet did cause some change in the results. Although the flow maintained the same general patterns within the duct, the values of the velocity in local regions within the main duct near the intersection varied. The turbulent quantities showed a similar behavior, similar trends, but the values changed more drastically. Additionally, the differences in the turbulent quantities (eddy viscosity, and kinetic energy) were visible further downstream in the domain. This is obviously an undesired consequence of using a flat inlet profile. It is suspected that the differences in results are primarily due to the prescription of k and ϵ at the inlet. The eddy viscosity is proportional to the quotient of k^2 and ϵ . Small variations in either can result in values of turbulence that are inaccurate.

To ensure that this effect is minimized, the inlet boundary conditions for each turbulence model were taken from a fully developed channel flow profile predicted by that model. This was accomplished by running a periodic channel flow with the desired turbulence model, and a prescribed mass flow rate matching that of the test. The results of the

3.4 Convergence Criteria

To accelerate the convergence of the model, the Algebraic Multigrid Solver (AMG) was utilized on the pressure correction equation. Under-relaxation factors for momentum and pressure were also employed to facilitate convergence.

acceptably small.

dissipation present as a result of the first order differencing scheme is expected to be converged solution. Due to the fine mesh present in this model, the amount of numerical higher order accurate, it proved to be the only differencing scheme that would provide a to compute cell face fluxes based on cell centered values. Although this method is not correction equations were solved in a segregated manner. Upwind Differencing was used (Patankar and Spalding [16]) was used in which the momentum, turbulence, and pressure A steady state solution was sought for this analysis. The SIMPLE solution scheme

3.3 Solution Procedure

sampling point.

due to the interpolation), but diminished before reaching the location of the first pressure duct. In some cases, small variations were present within the first two cells (most likely using user-coding. The flow maintained essentially the same profile throughout the inlet channel model were then interpolated onto the inlet boundary of the impinging jet model

4.1 Inlet Channel Flow Predictions

The results of the CFD models were compared to the experimental data to determine which turbulence model predicted the flow most accurately. The velocity profiles, pressure field, turbulent kinetic energy, and amount of swirl present were all compared.

4.0 RESULTS

Convergence was achieved when the global sum of the cell residuals of the Pressure, Momentum, and Turbulent Kinetic energy equations had fallen approximately 6 orders of magnitude, and the field values at a monitoring location within the mixing region were no longer changing. Some models converged very nicely with the Upwind Differencing scheme, but others were very difficult. The quadratic turbulence model, for example, only converged about four orders of magnitude on the momentum equations, and 5 on the turbulence and pressure equations. Although this does not meet the convergence criteria stated above, the field values at the monitoring location were stationary, and the solution appeared to be accurate based on physical reasoning and comparison to the other models. For these reasons, the solution from this model is included in this paper.

The first model to converge was the standard k- ϵ model. Therefore, the majority of the other simulations were restarted from the results of this model to assist convergence by providing a good initial field to start the computations from.

Fig. 5 contains values that have been normalized by the inlet diameter and velocity. Although most of the models do predict the peak value of the velocity in the duct within 3%, the profile is not predicted accurately by any of the models. All of the turbulence models predict a flow that has a more gradual slope from the centerline of the duct

models. However, some discrepancies exist in the velocity profile predicted across the bisector of the duct. Fig. 5 shows the ratio of the peak to mean velocity taken across the horizontal bisector of the inlet duct. The scale of the plot is adjusted to show the differences in the All of the turbulence models evaluated predict the peak velocity well in the duct flow.

Test	Data	U_{peak}/U_{Mean}					
		1.19	1.17	1.17	1.16	1.17	1.13
k-ε			RNG	k-ε	Chen	k-ω	Spalart-Allmaras
k-ε	Quadratic						

Table 4: Peak velocity predictions.

The first comparison among the turbulence models measures their ability to accurately predict a fully developed velocity profile within a square channel. As mentioned before, periodic channel flow problems were run for each model to provide an inlet boundary condition for the impinging jet model. Gessner [17] reports that the ratio of the peak velocity to the bulk velocity for fully developed channel flow at a Reynolds number of 136,000 is 1.18. The calculated values of peak velocity determined using the turbulence models were normalized by the bulk velocity to obtain the results shown in Table 5.

Before considering further comparisons between model calculations and data, it is useful to discuss the major characteristics of the flow field. Three primary flow features are formed as the flow passes through the inlet duct and impinges against the wall of the main duct. The most dominant feature is a large three-dimensional separation region. The flow within this region is characterized by reversing axial velocities (flow rotating about the Y-axis), caused by expansion and turning as the flow transitions from the inlet

4.4 Flow Field Characteristics

The turbulent kinetic energy between the CFD predictions and the test data for the 2-equation models can also be compared, as seen in Fig. 6. Although it is possible to estimate the kinetic energy from the Spalart-Allmaras model, it is not displayed here. The turbulent kinetic energy also shows some differences between measured and predicted values. Again, it should be noted that the scale of the plots is adjusted such that differences are magnified.

the profiles may be attributable to the turbulence model. Gessner's data at locations along the bisector of the duct. Therefore, the discrepancies in the flow was developed. Additionally, the measured velocity profile compared well with change in the velocity profiles between these locations, which led to the conclusion that was measured at 5 diameters, 2 diameters, and 0.5 diameters. There was no noticeable development of the inlet flow in the test, the velocity profile along the horizontal bisector toward the wall, then a very steep gradient near the wall. As an independent check on the

The red streamline in Fig. 7 represents a particle that has become trapped in the primary separation region. The coiled path outlines the region of reversing axial flow, which causes the particle to traverse the region twice in the Z direction before escaping the

above.

convey the complexity of the flow, and illustrate the three primary features described moves downstream. Fig. 7 shows streamlines from the CFD results that are intended to is a single rotating vortex that traces a helical pattern about the centerline of the duct as it downstream of the inlet, and give rise to the third major feature of the flow. This feature The primary separation region and the counter rotating vortex diffuse away as they move

dimensional effect, and only displays rotation about the Z-axis.

separation zone against the impinged wall of the duct. This feature is more of a two-flow turns upward as it expands into the main duct. This turning is sufficient to create a small counter rotating vortex that attaches to the bottom corner of the main duct. The Another prominent feature that develops as a result of the intersection of the ducts is a

concept is similar.

Obviously, the interactions are much more complex in three dimensions, but the general the flow features that result from flow over a two-dimensional backward facing step. flow area of the inlet to the main duct. Individually, these rotations can be compared to about the Z-axis as a result of the expansion in the Y direction as it transitions from the duct to the main duct. The flow within this primary separation region is also rotating

4.2 Velocity Results and Comparisons

gradually dissipating with axial distance. exit of the duct, the primary vortex establishes itself across the entire duct cross section, This region effectively disappears near $Z/D_H=2.49$ as well. From $Z/D_H=2.49$ until the separation region is seen directly above the entrance jet in the form of reverse axial flow. stagnation region until near $Z/D_H=2.49$, where the vortex collapses. The primary rotating vortex is observed in the bottom left corner, growing in size away from the near wall velocity is negative at $Z/D_H=0.26$ and positive at $Z/D_H=1.0$. The small counter-velocity is nearly zero. However, on either side of this plane, the jet spreads so that the the stagnation point of the impinging jet; at the far wall in the direct path of the jet, the measurement planes near the intersection of the ducts. The data at $Z/D_H=0.42$ is close to Additional details are seen in Fig. 8, which shows a series of vector plots at 4 LDV evident in this streamline trace.

vortex is roughly given by the red streamline. The helical pattern of the center line is main vortex. After the collapse of the two recirculation regions, the center of the main particle that avoids both of the recirculation regions, then traces out the exterior of the that found in the primary separation region. The green streamline shows the path of a of the main duct. This path also includes some axial flow reversal, but not as severe as path of a particle that travels through the counter-rotating vortex near the bottom corner adverse pressure gradient and continuing downstream. The blue streamline displays the

The swirl number involves an integration over the area of the 2-dimensional plane on which it is being evaluated. To perform this integration, the results of the CFD models were linearly interpolated onto the same grid used for the LDV measurements. The swirl number for a given plane was computed by summing the average face value multiplied by the area of the face for all elements cut by the plane. Even if the cross-sectional flow contains multiple vortices, or a complicated secondary flow distribution, this method is

the cross-stream direction compared to that in the axial direction. transport of *axial* momentum. This gives a reasonable measure of the amount of flow in Eq. (5) represents the net axial transport of *tangential* momentum divided by the net axial

$$U_{\text{tangential}} = \frac{1}{r} (V_x - U_y) \quad (6)$$

The tangential velocity is defined by:

$$S = \frac{\iint r U_{\text{tangential}} |U_{\text{axial}}| dA}{\iint r U_{\text{axial}}^2 dA} \quad (5)$$

do this, we consider a non-dimensional swirl number, commonly defined as: any cross section and use this parameter to evaluate the effectiveness of the models. To it is convenient to define a global parameter that can be derived from the velocity data at comparing each calculated velocity field to the data at every measurement plane. Instead, It is difficult to consider each turbulence model and assess its predictive ability by

still a useful way of comparing the calculated velocity distribution to the measured data. Fig. 9 shows the result of this calculation compared to the test data.

As seen in Fig. 9, all of the models over predict the swirl at the inlet by as much as a factor of 2.5. This is caused by the over prediction of the secondary recirculation cell against the back wall of the main duct, and differences in axial velocity predictions. All of the models are significantly inaccurate with the prediction of the size and shape of the secondary recirculation cell that appears until a distance of approximately $0.75 D_m$. The main cause for the swirl number discrepancies is the difference in the axial velocity predictions, as will be shown.

The two-equation models that performed the best in the inlet region are the ones that contain the additional strain dependency terms in the source of the ϵ equation (RNG, Chen). These models show inaccuracy in the prediction of the size and shape of the recirculation region, like the others do. However, they also show regions of higher axial velocity at large distances from the centerline of the duct, which is consistent with what is found in the data. These models predict higher levels of dissipation, which would act to decrease the eddy viscosity, and permit the higher velocities in regions of high shear. The swirl number calculation near the inlet is dominated by the denominator, which contains the square of the axial velocity multiplied by distance from the centerline. For the RNG and Chen models, the higher axial velocities reduce the swirl number significantly and cause it to be closer to the test data. Although the swirl number is primarily reflecting the net axial transport of momentum in these regions, it still provides

a good metric for comparing with test data. The test data shows even higher axial velocities than any of the 2-equation turbulence models predict, which causes the swirl number for the test to be significantly lower. The point where the CFD transitions from over predicting to under predicting the swirl number (i.e., $Z/D_h \sim 4$) represents the end of any reversing flow in the axial direction. At this point the denominator is approximately equal for all of the models and the test data (basically representing the mass flux). Differences in the swirl number from this point forward in the down stream direction are due almost entirely to differences in tangential momentum.

As the secondary recirculation region collapses downstream (at about $2.5 D_h$), the benefits of the RNG and Chen models disappear, and their results are actually less accurate than the standard $k-\epsilon$ model. As the recirculation regions collapse, the flow forms a single rotating vortex, with the center of the core forming a helical trace about the centerline of the duct. The models do predict some of this behavior, but large differences appear when comparing the strength and core location of the vortex. The test data shows a reasonably strong rotation, while all turbulence closures except $k-\omega$ predict drastically smaller secondary velocity components.

The single vortex established at about $Z/D_h=5$ decreases in strength with increasing axial distance down the duct. Fig. 10 shows a semi-log plot of the normalized swirl number for $5 < Z/D_h < 70$ (duct exit). Also included in the plot is the correlation from Li and Tomita [18] for swirl flow data taken in a 80mm diameter pipe. The Reynolds number for their pipe experiment encompasses the duct flow Reynolds number for the present

The loss coefficient, defined in Eq. (1), was calculated using CFD results and is compared to the test data in Fig. 11. All the CFD results overpredict the pressure drop, with the worst predictions being about 15% high. Observe that, although the RNG and Chen variants of the k- ϵ model proved to give a better approximation of the swirl number in the inlet region, they do not give a good estimate of the pressure drop. In fact, nearly all of the k- ϵ models, with the exception of the standard k- ϵ model, show similarly poor pressure drop predictions. On the other hand, the single equation Spalart-Allmaras model predicts the data well, being within 4% of the test data for all of the points.

Determining the exact cause of the differences in the prediction of the pressure drop is difficult to do. In general, the models predict larger recirculation regions at the inlet, which would produce higher pressure drops than measured in the test. However, the models also predict smaller velocity gradients in the axial direction, which would reduce the pressure drop. The only conclusion that can be made is that the pressure drops are

4.3 Pressure Drop Comparison

most significant.

dissipate the swirl in the duct faster than in the pipe. The latter effect is believed to be the compared to the pipe, and the additional losses in the sharp corners of the duct that would the pipe flow. Possible explanations include the non-centered vortex in the duct experiment. It is seen that decay in swirl for the duct flow is significantly higher than for

inaccurate due to differences in predicted velocity profiles or differences in wall functions, but the exact region causing the error is difficult to isolate.

5.0 Conclusions

Detailed measurements of velocity and pressure drop have been obtained in 2 square ducts intersecting at 90° to provide qualification data for a selection of turbulence models. The geometry studied contains regions of swirling flows, massive separation, turning flow, and jet impingement. The ability of a turbulence model to predict these features is vital to CFD's success in the design and analysis of fluid flow systems.

Six turbulence models were studied: k- ϵ , k- ϵ RNG, k- ϵ Chen, non-linear k- ϵ , k- ω , and

Spalart-Allmaras. The universal trend observed for all of the turbulence models was poor prediction in the inlet mixing region and in the region of swirl decay. The reversing axial velocities calculated in the mixing region were not as high as those measured in the test. This appears to be caused by the incorrect prediction of the axial flow distribution upon impinging against the wall of the main duct (some wrap around against the back wall while some head downstream). An over prediction of eddy viscosity in the downstream region is suspected to cause the under prediction of swirl at the outlet.

The three versions of the k- ϵ model evaluated in this analysis provided surprisingly

different results. Although the two models that contained the additional strain terms in

the ϵ equation were often times very similar (RNG and Chen), they both differed from the standard version. The standard version showed the best pressure drop prediction among them (4% high), but the worst swirl number at the inlet region (45% low). It failed to capture the highly negative values of W velocity as the jet impinged against the back wall of the main duct. It also provided one of the worst estimates of the location of the collapse of the recirculation region. Due to the inadequacies listed above, none of the two-equation k - ϵ models are suggested for use on this type of flow.

The nonlinear k- ϵ model did not improve the results for the impinging jet model. There are two possibilities for this problem. It can be assumed that the degree of anisotropy is not significant enough to cause the Boussinesq approximation to break down, or that the model fails to predict this anisotropy well. It is likely that the model just fails to predict it well. The non-linear constitutive relation used in the quadratic turbulence model is sufficient to produce the correct secondary flow features (secondary flow of the second kind) for flow in a square duct. According to Wilcox, a number of non-linear models exist that improve the prediction of flow within square ducts, but still fail to capture the gradual adjustment of the Reynolds stresses in flows with rapidly changing rates of strain. It is concluded that this is an example of a model where the non-linear relation provides no substantial improvement over the standard Boussinesq relation. The overall performance of the non-linear model was not as good as the standard two-equation models. Factoring in the difficult and slow convergence of the model makes it altogether an unattractive model for this type of flow. Neither the pressure drop nor the swirl number was predicted well.

The results have shown that the k- ω model does the best job of predicting the swirl number after the disappearance of the reversing axial flow near the inlet. The amount of tangential momentum present after the inlet region was predicted very well by the k- ω model, but extremely poorly by all other models. The k- ω model also showed the highest distribution of non-uniform axial flow at the outlet, which is seen in the test data. Although the level of swirl calculated by k- ω was the most accurate, the pressure drop predicted by this model was inaccurate, and thus detracts from its overall usefulness.

However, it has definitely proven useful for rotating flows that contain no other complicating features.

Overall, the measured flow features were best predicted by the Spalart-Allmaras single equation model. Although the Spalart-Allmaras model was originally formulated for external aerodynamic flow applications, it has proven to be effective for wall bounded flows as well. Its major downfall is the inability to predict the amount of swirl in the duct at the outlet of the CFD model. This is a problem that is shared by most turbulence models, and the Spalart-Allmaras model does no worse than the standard $k-\epsilon$ model. Based on these arguments, it is suggested that the Spalart-Allmaras model be considered for use in wall bounded flows containing stagnation points, rapid expansions, and sharp corners.

This analysis has focused entirely on high Reynolds Number turbulence models due to cost, and their popularity in engineering practice. However, to truly evaluate the accuracy of a turbulence model, and eliminate the effect of wall functions on the predictions, it is necessary to utilize the low Reynolds Number variants of the model. It is quite possible that inadequate wall functions are causing the poor prediction of the velocity profile in the impinging jet region of the ducts where large gradients are present near the wall. Wall functions are typically formulated for equilibrium boundary layers in two-dimensional flows, which is clearly not what is present in this region. It is expected that further investigation of low Reynolds number turbulence closures will improve the

CFD predictions of the velocity profile while retaining the accuracy of the pressure drop prediction.

REFERENCES

- [1] Iaccarino, G., 2001, "Prediction of Turbulent Separated Flow Using Commercial CFD Codes", *Journal of Fluids Engineering*, Vol. 123.
- [2] Guo, B., Langrish, T.A.G., and Fletcher, D.F., 2001, "Numerical Simulation of Unsteady Turbulent Flow in Axisymmetric Sudden Expansions", *Journal of Fluids Engineering*, Vol. 123.
- [3] Mentor, F.R., 1996, "A Comparison of Some Recent Eddy-Viscosity Turbulence Models", *Journal of Fluids Engineering*, Vol. 118.
- [4] Freitas, C.J., 1995, "Perspective: Selected Benchmarks from Commercial CFD Codes", *Journal of Fluids Engineering*, Vol. 117.
- [5] Wilcox, D. C., 2000, Turbulence Modeling for CFD, second edition, DCW Industries, Inc. Methodology, STAR-CD Version 3.10, Computational Dynamics Limited, 1999.
- [6] Shames, I.H., 1982, Mechanics of Fluids, McGraw-Hill Book Company.
- [7] Benedict, L.H. and Gould, R.D., 1996, "Towards Better Uncertainty Estimates for Turbulence Statistics", *Experiments in Fluids*, Vol. 22.
- [8] Erdmann, J.C., and Tropea, C., 1981, "Turbulence Induced Statistical Bias in Laser Anemometry", *Proceedings of the Seventh Symposium on Turbulence*, University of Missouri-Rolla, September.
- [9] TSI Phase Doppler Particle Analyzer / Laser Doppler Velocimeter Operations Manual, 2001.
- [10] Edwards, R.V., 1987, "Report of the Special Panel on Statistical Particle Bias Problems in Laser Anemometry", *Journal of Fluids Engineering*, Vol. 109.
- [11] Launder, B.E., and Sharma, B.I., 1974, "Application of the Energy Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disc", *Letters in Heat and Mass Transfer*, Vol. 1, No. 2.
- [12] Yakhot, V., and Orszag, S.A., 1986, "Renormalization Group Analysis of Turbulence -I: Basic Theory", *Journal of Scientific Computing*, Vol. 1.
- [13] Chen, Y.S., and Kim, S.W., 1987, "Computations of Turbulent Flows Using an Extended k- ϵ Turbulence Closure Model", *NASA CR-179204*.
- [14] STAR-CD Methodology, 1999, STAR-CD Version 3.10, Computational Dynamics Limited.

- [15] Spalart, P.R., and Allmaras, S.R., 1992, "A One-Equation Turbulence Model for Aerodynamic Flows", *AIAA Paper 92-439*, Reno, NV.
- [16] Patankar, S.V., and Spalding, D.B., 1972, "A Computational Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows", *International Journal of Heat and Mass Transfer*, Vol. 15.
- [17] Gessner, F.B., and Emery, A.F., 1981, "The Numerical Prediction of Developing Turbulent Flow in Rectangular Ducts", *Journal of Fluids Engineering*, Vol. 103.
- [18] Li, H., and Tomita, Y., 1994, "Characteristics of Swirling Flow in a Circular Pipe", *Journal of Fluids Engineering*, Vol. 116.

List of Figures

Figure 1. Schematic of impinging jet test assembly.

Figure 2. Loss factors in impinging jet test section

Figure 3. Percent difference in measured axial velocity between vertical and horizontal
LDV probe orientations, $Z=19.05$ mm

Figure 4. Computational mesh detail at the intersection of the two
ducts showing uniform hexahedral elements.

Figure 5. Inlet velocity profiles at $2 D_H$ upstream of the intersection region

Figure 6. Inlet Kinetic Energy profiles at $2D_i$ upstream of the intersection.

Figure 7. Streamlines in impinging jet model.

Figure 8. Vector Plots at Various Locations in Main Duct

Figure 9. Swirl number in main duct

Figure 10. Swirl decay comparison between square duct and pipe geometry.

The pipe data is from Li and Tomita [18].

Figure 11. Loss factors in impinging jet test section compared to CFD results for
 $Q=277.1$ lpm

Figure 1. Schematic of impinging jet test assembly.

Figure 2. Loss factors in impinging jet test section

Figure 3. Percent difference in measured axial velocity between vertical and horizontal LDV probe orientations, $Z=19.05$ mm

Figure 4. Computational mesh detail at the intersection of the two ducts showing uniform hexahedral elements.

Figure 5. Inlet velocity profiles at $2 D_H$ upstream of the intersection region

Figure 6. Inlet Kinetic Energy profiles at $2D_i$ upstream of the intersection.

Figure 7. Streamlines in impinging jet model.

Figure 8. Vector Plots at Various Locations in Main Duct

Figure 9. Swirl number in main duct

Figure 10. Swirl decay comparison between square duct and pipe geometry.
The pipe data is from Li and Tornita [18].

Figure 11. Loss factors in impinging jet test section compared to CFD results for $Q=277.1$ lpm