REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. REPORT DATE *(DD-MM-YYYY)*	**2. REPORT TYPE**	**3. DATES COVERED** *(From - To)*
June 2014	Technical Paper	June 2014- July 2014

4. TITLE AND SUBTITLE

Numerical and experimental investigation of confined turbulent multiple transverse jets

5a. CONTRACT NUMBER
IN-HOUSE

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)

F. Davoudzadeh, D. Forliti

5d. PROJECT NUMBER

5e. TASK NUMBER

5f. WORK UNIT NUMBER
Q09Z

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

Air Force Research Laboratory (AFMC)
AFRL/RQRC
10 E. Saturn Blvd.
Edwards AFB CA 93524-7680

8. PERFORMING ORGANIZATION REPORT NO.

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)
Air Force Research Laboratory (AFMC)
AFRL/RQR
5 Pollux Drive.
Edwards AFB CA 93524-7048

10. SPONSOR/MONITOR'S ACRONYM(S)

11. SPONSOR/MONITOR'S REPORT NUMBER(S)
AFRL-RQ-ED-TP-2014-015

12. DISTRIBUTION / AVAILABILITY STATEMENT
Distribution A: Approved for Public Release; Distribution Unlimited

13. SUPPLEMENTARY NOTES
Technical paper presented at 50th AIAA/ASME/SAE/ASEE Joint Propulsion Conference, Cleveland, OH, 28-30 July, 2014.
PA#14048

14. ABSTRACT

The flow and mixing properties of confined transverse jets are relevant to a myriad of combustion devices ranging from propulsion to energy generation and chemical processing. The current effort focuses on understanding the mixing process between a transverse jet mixing in a confined system. The current study involves the simulation of a single confined transverse jet configuration under matched conditions of a companion experiment. The main flow Reynolds number considered is in the range of 25000 - 53000 and the jet-to-main flow momentum flux ratio is varied from 3.2 -14.3. The momentum and scalar mixing is investigated through the solution of the Reynolds-Averaged Navier Stokes (RANS) equations. The mean scalar mixing characteristics are compared to experimental data. The turbulence model that is used is the low Reynolds number k-ϵ model. Due to demonstrated symmetry, only a one-half section of the geometry is considered. All numerical simulations capture salient flow structures such as the counter-rotating vortex pair (CRVP). The current investigation shows the numerical simulations predict the experimental data with a good degree of accuracy.

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			**17. LIMITATION OF ABSTRACT**	**18. NUMBER OF PAGES**	**19a. NAME OF RESPONSIBLE PERSON**
					Nils Sedano
a. REPORT	**b. ABSTRACT**	**c. THIS PAGE**	SAR	10	**19b. TELEPHONE NO** *(include area code)*
Unclassified	**Unclassified**	**Unclassified**			661-275-5972

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std. 239.18

NUMERICAL AND EXPERIMENTAL INVESTIGATION OF CONFINED TURBULENT MULTIPLE TRANSVERSE JETS

Farhad Davoudzadeh[1]
U.S. Air Force Research Laboratory, Edwards AFB, California 93524

David Forliti[2]
Sierra Lobo, Inc., Edwards AFB, CA, 93524

ABSTRACT

The flow and mixing properties of confined transverse jets are relevant to a myriad of combustion devices ranging from propulsion to energy generation and chemical processing. The current effort focuses on understanding the mixing process between a transverse jet mixing in a confined system. The current study involves the simulation of a single confined transverse jet configuration under matched conditions of a companion experiment. The main flow Reynolds number considered is in the range of 25000 - 53000 and the jet-to-main flow momentum flux ratio is varied from 3.2 -14.3. The momentum and scalar mixing is investigated through the solution of the Reynolds-Averaged Navier Stokes (RANS) equations. The mean scalar mixing characteristics are compared to experimental data. The turbulence model that is used is the low Reynolds number k-ϵ model. Due to demonstrated symmetry, only a one-half section of the geometry is considered. All numerical simulations capture salient flow structures such as the counter-rotating vortex pair (CRVP). The current investigation shows the numerical simulations predict the experimental data with a good degree of accuracy.

Nomenclature

A, b, c	=	empirical constants
C	=	local mean mass fraction of the jet
$\overline{C}$	=	average mass fraction of the jet over the cross sectional area A
D	=	diameter
J	=	momentum flux ratio
$\dot{m}$	=	mass flow rate
n	=	number of jets
Q	=	volume flow rate
r	=	velocity ratio
x	=	stream-wise coordinate
y	=	cross stream coordinate
$_j$	=	subscript for jet flow
Us	=	Unmixedness

INTRODUCTION

Transverse jets, or jets in cross flow (JICF) provide an effective mixing mechanism, via vortical structures, that can be utilized in a number of applications. Examples of these can be found in propulsion combustion devices, chemical scrubbers, snow makers and air conditioning

[1] Senior Scientist AFRL/RQRC, 10 East Saturn Blvd., Bldg. 8424, Edwards, CA. 93524
[2] Research Scientist, AFRL/RQRC, 10 East Saturn Blvd., Bldg. 8451, Edwards, CA. 93524

units. Generally the intent in all of these applications is to expedite the uniform distribution of the jet's fluid within the crossflow.

This type of configuration offers a number of technical challenges for design tools such as CFD depending on the application; including turbulent combustion, complex geometry with strong streamline curvature, very large density ratios between the mixing fluids, and transcritical phenomena due to the high operating pressure. A recent study summarized by Davoudzadeh et al. explored the CFD validation to an existing eight jet configuration documented in the literature [1]. The current validation effort builds upon this previous study through the consideration of fewer jets and different measurement locations. The purpose of this phase of the validation effort is to establish a foundational methodology that is able to accurately simulate the complex geometry in a higher fluid density configuration. This will essentially establish a departure point as we incrementally add salient features, including density ratio, high pressure, and chemistry.

The experimental configuration was numerically modeled using the commercial Fluent software program. The computational work was performed to identify the current capabilities of Fluent in modeling the complex flow structures encountered in JICF, and to identify the appropriate CFD model elements (solver, turbulence model, computational extend and grid, etc.) related to the flow regime under consideration. Current simulations show a good comparison between the computational results obtained by Fluent and the experimental results. The effort also has identified relevant CFD parameters (turbulence model, geometry and grid distribution, solver type) of importance in this flow regime. Effort is continuing to perform more detailed calculations looking into grid resolution, boundary conditions, and solver models.

Demuren categorized four different models, in terms of accuracy, that has been used in investigations of the JICF: empirical, integral, perturbation and numerical [2]. The empirical method is the simplest model that relies on the correlation of experimental data, and the accuracy of an empirical model will in general depend on whether a given point of interest lies within the cloud of data used for construction of the correlation; the empirical model is essentially a curve fit. Due to their low cost and ease of use, empirical models are most useful for first-order estimates and as qualitative checks for results produced by other methods.

One of the most common JICF attributes treated using empirical models is the jet trajectory. For a single circular turbulent jet injected normally into a cross flow, the trajectory has the form [2]:

$$\frac{y}{D} = aJ^{b}\left(\frac{x}{D}\right)^{c} \tag{1}$$

where D is the jet diameter, J is the momentum flux ratio, x and y are the cross-stream and stream-wise coordinates of the jet trajectory, and a, b, and c are empirical constants. In the range of J between 2 and 2,000, the constant a has a value between 0.7 and 1.3, b has a value between 0.36 and 0.52, and c takes a value between 0.28 and 0.40, depending on experimental conditions [2].

On the other end of the spectrum, Demuren pointed out that "numerical models have the most potential for wide generality and can, in principle, be applied to the whole range of jet in cross flow situations, confined or unconfined, low, medium or high J, single or multiple jets, impinging on a wall or on other jets, swirling, homogeneous or heterogeneous cross flow, compressible or incompressible, etc [2]."

The research results presented in this paper numerically investigates the various flow characteristics experienced when a single and cross flow interact in a confined configuration, and studies the induced resultant flow structures and their contribution to the mixing of the two constituents.

COMPUTATIONAL DOMAIN AND BOUNDARY CONDITIONS

The geometry and flow conditions investigated in this computational study match a companion experiment. A few brief details of the experiment are given here, while further information regarding the experimental facility and measurements is given by Forliti and Salazar[3]. The experiment utilizes hardware that is capable of injecting from one to six jets into a fully-developed cross flow. The computational model of the experiment is shown in Fig. 1. It should be noted that this geometry is constructed for the CFD analysis and is considered too closely represent the experimental configuration. The use of an outlet off the axis of the main flow as shown in the figure is employed in the experiment to allow optical access through a window on the downstream end of the exhaust plenum. Only the top jet is used in the current study.

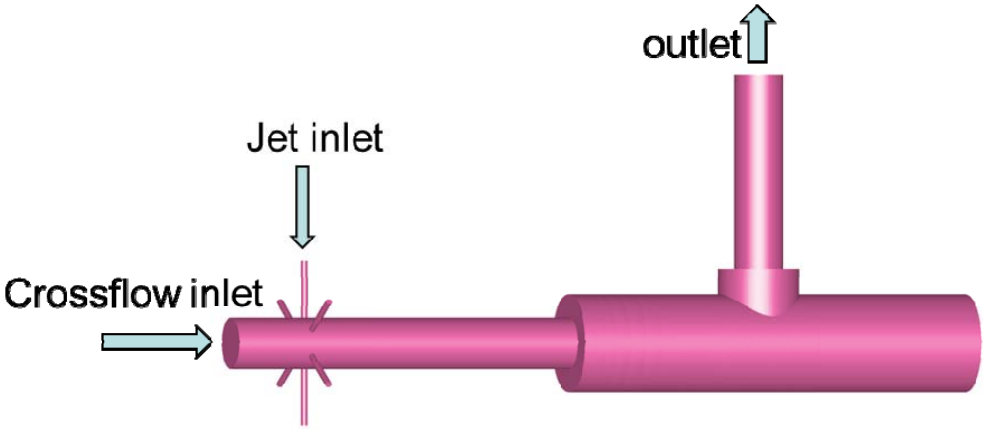

Figure 1. Geometry in the vicinity of the jet injection system.

Figure 2 shows the grid distribution on and around an injection port, and at the exit of the mixed flow. A total of approximately 3M mixed grid elements are used. As shown in the inlet/exit grid distribution, no singularity axis is used in the construction of the grid. Wall grid distribution corresponds to a $y+ < 1$.

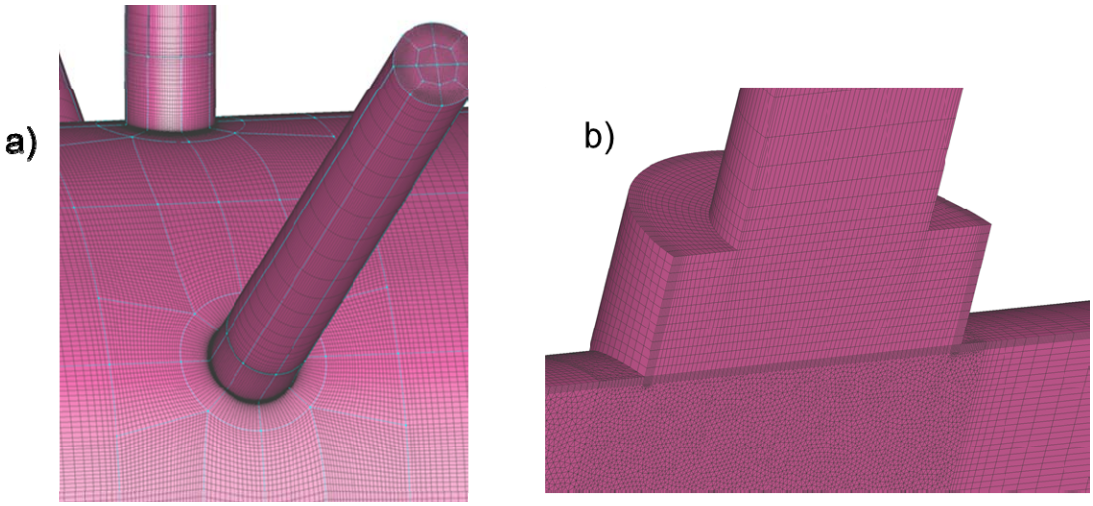

Figure 2. Grid distribution near a) the injection port and b) outlet port.

Figure 3 shows the computational domain and boundary conditions used in the simulations. To take advantage of symmetry in the geometry, a 180° section of the geometry is simulated, with a plane of symmetry boundary condition. The downstream boundary is located 8D downstream of the diluent injection port, where D is the main pipe diameter. Note that for this study the details of the exhaust section including the sudden expansion and side outlet were removed; the influence of the downstream geometry will be explored in the near future. This subdomain uses 2.2M hexahedral only grid elements. The upstream boundary is a fully-developed pipe flow boundary condition that is calculated on a separate domain that uses periodic boundary conditions to efficiently produce a fully-developed state. The exit BC uses the

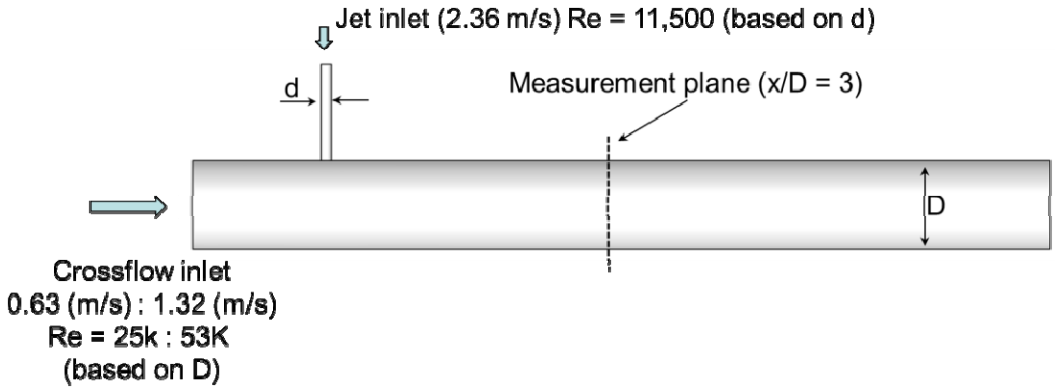

Figure 3. Computational domain and boundary conditions.

static pressure and is set to standard atmospheric pressure. The jet to main flow diameter ratio (d/D) is 0.12 and matches the experiment. Water is used for both flows. To reflect the operating space of the experiment, the main flow rate was varied while the jet flow is held constant. The experimental measurements were made at a distance of three main pipe diameters D downstream of the injection location.

FLOW SOLVER AND OPERATING CONDITIONS

All simulations are three dimensional, second-order accurate, and uses Ansys-Fluent (pressure-velocity coupled) flow solver Release 14.5. Equations solved are x-, y-, and z-momentum, energy, turbulent kinetic energy (TKE), turbulent dissipation rate (TDR), species mass fraction, and continuity. The second-order spatial discretization solution method was used in all calculations. The low Reynolds number k-ε model was used, although other models were considered which in general degraded results compared to the experimental data. The default value of the turbulent Schmidt number of 0.7 was used.

All the results presented are after the numerical iteration has converged. Convergence is considered achieved when the residuals of the equations being solved have dropped by approximately four orders of magnitude, the residual gradients have approached zero and have plateaued, there are negligible changes in the flow variables, and mass is conserved throughout the computational domain. All calculations are performed on a one CPU processor with 4 cores workstation. Simulations using Fluent typically took approximately 10,000 iterations to achieve a fully-converged solution. This typically took about 24 hours, dependent on how many grid points were used. On the quad-core Intel Zeon processor the clock time used was 25 x 10e-3 microseconds per iteration per grid point.

RESULTS AND DISCUSSION

Forliti [4] and Forliti and Salazar [3] proposed that a new scaling parameter for transverse jets is the parameter

$$B\left(\frac{d}{D}\right) = \frac{J}{\frac{3.4}{\pi} + 0.32J^{1/2}}\left(\frac{d}{D}\right), \quad (2)$$

where the term B, shown as a complex expression of J, correlates the jet trajectories of transverse jets, and (d/D) is the jet to main pipe diameter ratio. Forliti and Salazar show that the mixing properties of a single confined transverse jet scales with B(d/D), therefore this parameter is considered a candidate for defining flow regimes. Table 1 shows the operating conditions in terms of B(d/D) and J for the present study. Forliti and Salazar found that the locally optimum mixing point is associated with a B(d/D) of 0.75 over a range of diameter ratios from 0.12 to 0.21. Strong impaction occurs for B(d/D) > 1.

Target B(d/D)-experimental	Actual B(d/D)-experimental	Actual J experimental	Computational J
0.25	0.232	3.2	3.2
0.50	0.49	8.18	8.2
0.75	0.747	14.27	14.3
			20.3
			28.3

Table 1: Summary of operating conditions.

Figure 4 shows the RANS simulation results in the symmetry plane for a momentum flux ratio of 14.3. The figure shows the distributions of the jet mass fraction, mean stream-wise velocity (x velocity), and mean normal velocity (y velocity). For this particular case, the jet penetrates well into the pipe flow leading to a slight bias of jet flow along the lower side of the pipe. The strong velocity gradients are seen to smooth out with downstream distance due to turbulent diffusion. Although the jet penetrates well into the pipe flow, the interaction with the other wall appears to be relatively weak. One of the primary considerations of the validation study is to determine how the CFD performs through the operating space of this flow, including conditions where the jet impacts on the wall. Jet-wall interactions may lead to complications for the CFD if large-scale unsteadiness develops that is not accurately captured by the turbulence model.

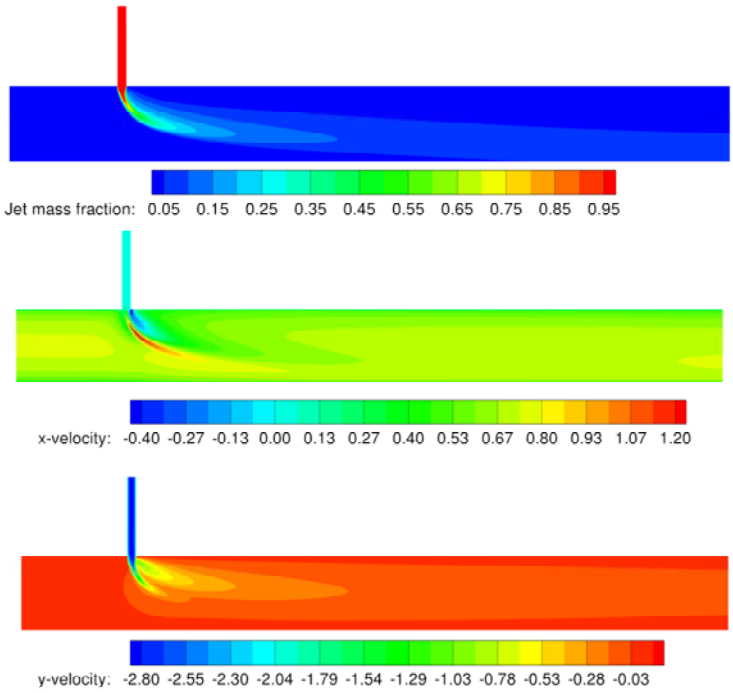

Figure 4. Flow field solution for J = 14.3.

Figure 5 shows the mixture fraction distributions for the three J values considered in this study. As expected, an increase in J results in a jet that penetrates further into the pipe flow. The lowest J value of 3.2 leads to a jet that quickly deflects and the jet fluid is concentrated on the top half of the pipe flow for the whole stream-wise domain. Figure 6 shows the cross plane mean mixture fraction distribution comparisons between experiment and CFD. The comparisons indicate that the CFD performs well compared to the experiment in that the jet core movement across the pipe is accurately predicted. The structure of the mixture fraction distribution does show some differences, as the CFD tends to have a more complex distribution exhibiting a strong 'kidney' shape. In general, the diffusion of the jet appears to be slightly reduced compared to the experiment, as indicated by the presence of nominally unmixed core fluid along the periphery of the pipe wall. The plots on the right side of Fig. 6 compare the experimental and computational mean mixture fraction profiles along the plane of symmetry. Again the computational solutions perform well, matching the general trend of the experiment. The peak mixture fraction distributions are slightly higher for the CFD, indicating that mixing is slightly under-predicted.

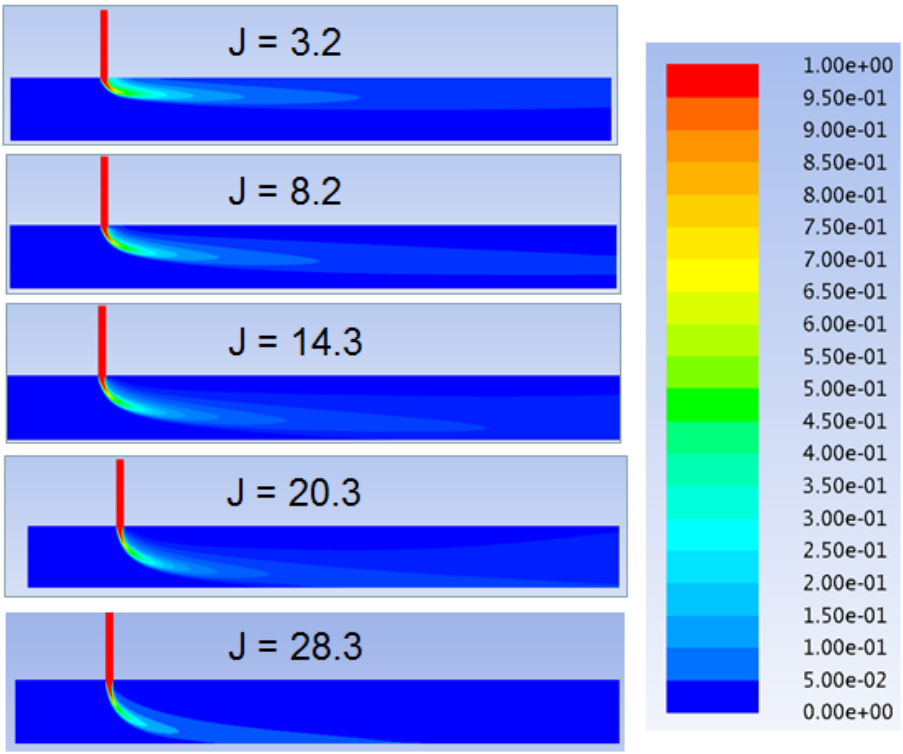

Figure 5. Mean mixture fraction distributions for different momentum flux ratios.

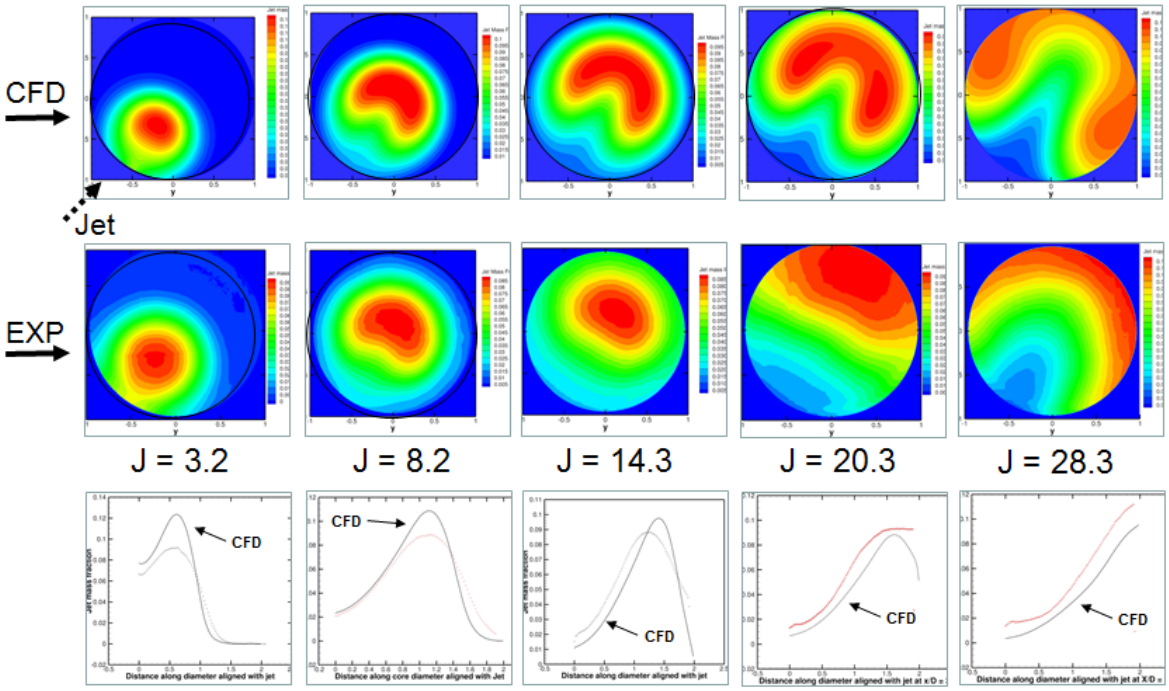

Figure 6. Distribution and profile comparison of the mean mixture fraction for experimental and CFD for J equal to 3.2, 8.2, 14.3, 20.3, and 28.3.

Figure 7 shows comparisons of the J = 8.2 case in the symmetry and normal planes. The figure further supports the trend that the computations reflect the trend quite well, with some differences in the quantitative details.

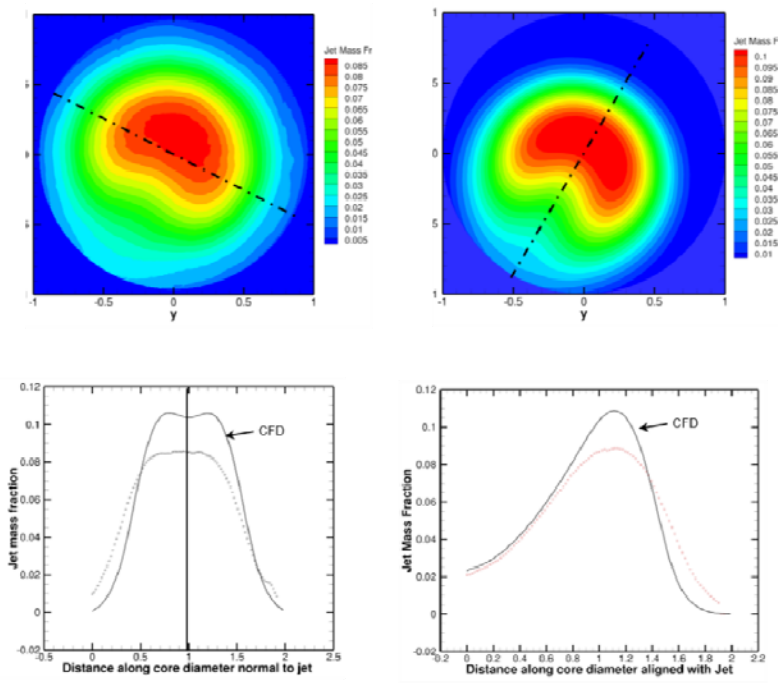

Figure 7. Symmetry and normal plane comparisons between experiment and CFD.

Figure 8 shows the mean stream-wise vorticity distributions for the different cases. The velocity field was not measured experimentally, thus no validation comparison can be made. The distributions were extracted at an x/D = 3. The distributions, in particular at higher J, are representative of the Deans vortex structure that develops downstream of an elbow in pipe flow. The peak magnitude of the vorticity is nominally constant for all cases.

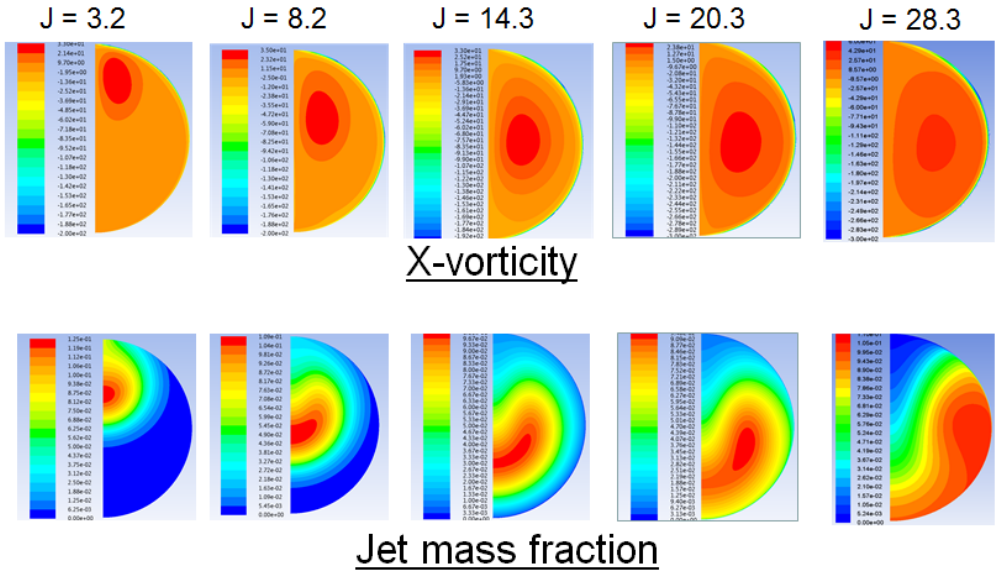

Figure 8. Mean stream-wise vorticity distribution and the corresponding jet mass fractions.

SUMMARY AND CONCLUSIONS

A validation study is currently being conducted on incompressible confined transverse jets. The validation study shows that the adopted CFD approach is able to accurately predict the qualitative aspects of the mixing field. Quantitatively, the mixing appears to be slightly under-predicted.

FUTURE WORK

The present study is part of an overarching effort to validate CFD models for prediction in liquid rocket engine combustion devices, in particular the oxygen-rich preburner. The current state of the effort is focused on the fluid mechanics of confined transverse jets. Future work will consider alternative turbulent Schmidt numbers to see if the quantitative comparisons can be improved. Alternative turbulence models will also be considered. Additional physical aspects will be incrementally introduced to the problem, including large density gradients, real fluid effects, and finally chemical reaction.

ACKNOWLEDGMENTS

The authors would like to thank Mr. David Salazar and Ms. Kayla Kuzmich for their effort in conducting the experiments.

REFERENCES

1. Davoudzadeh, F., Forliti, D. J., Le, A., and Vu, H., "Numerical simulation of confined multiple transverse jets," 42nd AIAA Fluid Dynamics Conference and Exhibit, New Orleans, LA, 25-28 June, 2012.

2. Demuren, A., "Modeling turbulent jets in crossflow," *Handbook of Fluid Dynamics and Fluid Machinery*. Vol. 1, John Wiley and Sons, 1996, pp. 787-807.

3. TBD - needs to be submited for public release approval.

4. Forliti, D. J., "Trajectory and mixing scaling laws for confined and unconfined transverse jets," 42nd AIAA Fluid Dynamics Conference and Exhibit, New Orleans, LA, 25-28 June, 2012.