Verification and Validation of CFD Simulations involving Twin Jets using Steady RANS in Star-CCM+

L. B. Carasik & Y. A. Hassan

Texas A&M University: 3133 TAMU, College Station, TX 77843-3133

Abstract – *Component/system design and consequential operational and safety analysis can be conducted through careful application of computational fluid dynamics (CFD) codes. CFD codes can be powerful tools to determine flow and heat transfer behavior within a system if they are used appropriately. Before these codes can be considered ready for specific analyses, benchmarking and/or appropriate verification and validation practices should be established for that type of analysis. Solution verification and validation for within a geometry with a high aspect ratio change are investigated for a turbulent flow situation. The injection of two turbulent planar-like jets into a significantly large volume are simulated. The combining of the two jets into a single self-similar jet is appropriately simulated and characteristic parameters of the behavior is used for analysis. An appropriate meshing strategy based on specific volumetric refinement regions was determined appropriate to simulate this behavior with a reasonable computational demand. Solution verification using velocity and gradients of velocity data in conjunction with available analytical solutions of single self-similar planar jets was found sufficient. The behavior of the jet for the merge point, combined point, centerline velocity, and velocity profiles at different vertical heights is shown to compare favorably to available experimental data and twin jet literature.*

I. INTRODUCTION

CFD tools have to undergo verification and validation (V&V) for specific flow situations before investigative or design studies can occur. There are two forms of V&V, solution and code. These two forms are readily discussed by Oberkampf and Trucano [1] and the different definitions they have in fields that utilized CFD. Solution V&V is done by the user of the codes. Whereas code V&V is done by the code developers and maintainers. In this work, the definitions of solution V&V that follow will be utilized. Solution verification is defined as effectively building a case of evidence to show the solution is "verified." Whereas solution validation is defined as the accuracy of a set of simulation results compared to an appropriate experiment. The solution V&V for CFD for twin turbulent planar-like jets will be the focus of this work.

Twin turbulent planar-like jets are a semi-classical problem investigated throughout the last 40 years. The work by Tanaka [2] provides an early account and discussion of the physics dominating why and how the twin jet merges. A schematic by Anderson et. al. [3] of the twin jet with the regions annotated and key geometric features are shown in Fig 1. The twin jets have the same bulk velocity before injection into the large domain. Once the jets have reached the large domain, the jets encounter the converging region where two distinct jet profiles can be observed. The jets begin to entrain fluid and the region between the jets develops significant recirculation. This causing the pressure of this region to become negative which pulls the two jets together. The time averaged centerline velocity in the streamwise direction is negative until reaching the merge point of the jets. The merge point is where the two jets being touching and exchanging momentum. After the merge point, the twin jets encounter the merging region where the jets continue to exchange momentum. Towards the end of this region, the two jet profiles cannot be distinctly seen. In place of the two profiles is a single jet profile where maximum centerline streamwise velocity is observed. This is referred to as the combined point and indicates that the two jets have completely merged into one jet. Above this point in the combined region of flow, the single jet develops into a single self-similar jet.

The twin jet water facility (TJWF) developed originally at the University of Tennessee, Knoxville has been purposed for aiding in V&V work based on the twin turbulent planar jet problem [4]. The TJWF allows for large data sets to be created using state of the art non-invasive measurement techniques within a mostly transparent testing volume. Techniques such as Laser Doppler Velocimetry (LDV) and Particle Image Velocimetry (PIV) are utilized within the TJWF to develop these data sets. Two data sets collected within the TJWF for LDV [5] and PIV [6] are used for comparison in this study.

II. MODELING APPROACH

The CFD modeling was performed using CD-Adapco's Star-CCM+ v11.04.012 [7]. The following subsections outline the geometry and simplifications, mesh strategies, and modeling setup.

1. Geometry and Simplifications

The CAD geometry was created using Autodesk Inventor 2016 and based off the as-build dimensions of the TJWF [1], The TJWF is shown in Fig. 2 with twin jets, weir overflows indicated. The total height of the computational domain is 1122.2 mm and 965.2 mm in width. The length of the computational domain is 711.2 mm. The jet width (a) and length (l) is 5.8 mm and 87.63 mm. The jets separation (S) is 17.8 mm. The jet height (h) is 279.4 mm from the point where the jets are attached to the TJWF enclosure volume or jet head top surface. The jet heat top surface is 766.6 mm from the top surface of the computational domain. The other relevant physical dimensions of the TJWF will be presented during the presentation of this summary. The width, length, and height are defined as the x, y, and z directions and shown in Fig. 2.

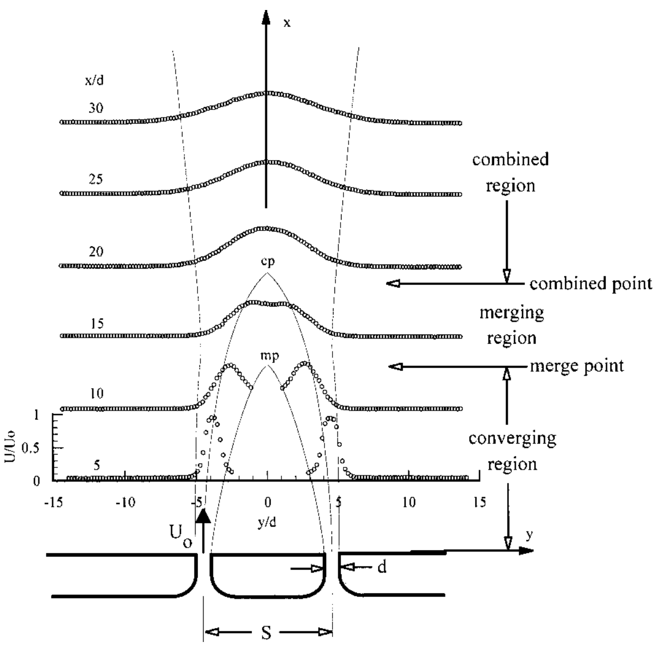

Fig 1: Schematic of Twin Planar-Like Jets Merging into One Jet [3] where d = a and x = z and y = x Directions for this Study

2. Mesh Strategies

The mesh was created using the built-in Star-CCM+ meshing tools. The trimmer mesher was selected to mesh the mean flow while the prism layer mesh was used to develop near wall cells. The trimmer mesher is a hexahedral mesher that trims cells near wall/surface regions [7]. The extruder mesher was used to create the inlet jet and outlet extension regions found in Fig. 2.

The inlet jets and outlet extensions are being held constant for both refinement strategies discussed below. Both inlet jets are created using a target surface minimum size of 0.15 mm and target maximum size of 0.25 mm. The inlet jets are 50 meshing layers stretched by 1.3 ratio for a total distance of 279.4 mm. The outlet extensions are created by using the surface mesh where the original "outlet" surface is defined. The mesh is extruded with 25 meshing layers being stretched using a 1.4 ratio for a total distance of 600 mm.

The near wall mesh created using the prism layer mesher on all surfaces using a total thickness of 2 mm. There are four prism layers with a stretching factor of 1.1 to smooth the transition for each layer.

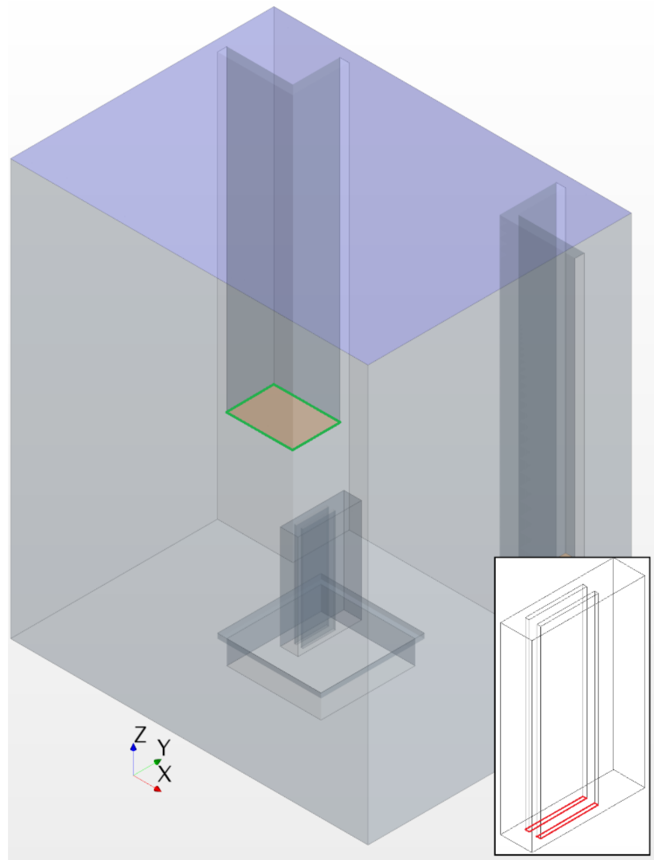

Fig 2: Geometric View of the Computational Domain used. Inlet Surfaces are Outlined in Red, Top Surface is shown as Purple, Outlet Surface is Outlined in Green. Note: The Second Outlet is not Shown.

A. "Bulk" Refinement Meshing

The "bulk" refinement meshing strategy is being used to investigate the meshing requirements for non-specific mesh refinement. The entire domain, excluding the inlet jets and the outlet extensions, are being meshed using the same overall requirements. The bulk meshing base size and maximum size and the resulting cell counts are shown in Table I. Fig. 3 provides a comparison of the "bulk" and "spot" meshes used.

B. "Spot" Refinement Meshing

The "spot" refinement meshing strategy is being used to investigate the reduced meshing requirements for when specific regions are being refined. There are three spot refinement regions utilized and labeled, jet inlet, core, and expansion. Each of the refinement regions were defined with in an isotropic manner such that x, y, and z directions with have the same size. The base and maximum cell sizes defined for the mesh overall are the same as seen in Table I. The specific sizes for each spot region and the cell count for each mesh is found in Table II.

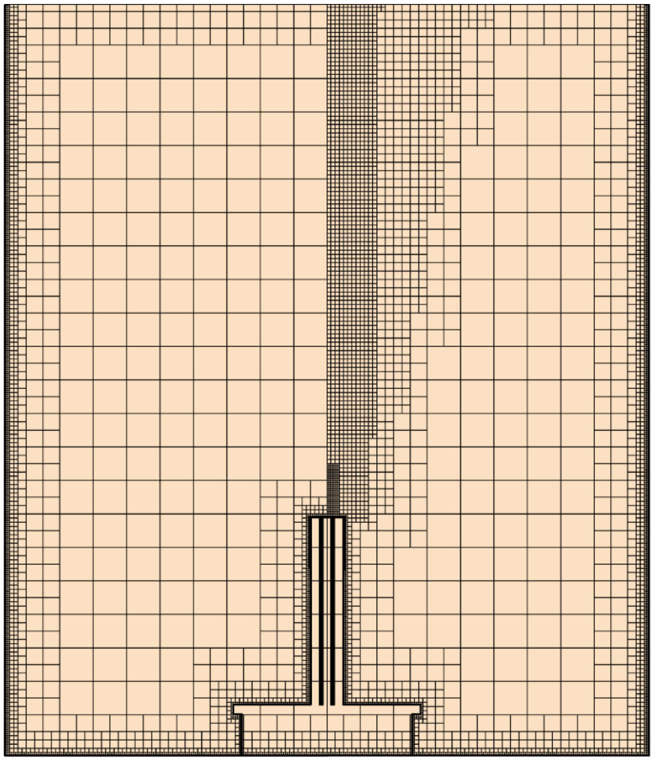

Fig 3: Meshing for Bulk (left) and Spot (Right) Refinement Strategies – Coarsest Mesh for Each.

Table I. "Bulk" Mesh Sizes and Cell Counts

Mesh Identifier	Base Size (mm)	Maximum Cell Size (mm)	Cell Count
M1	25	50	~4.8x10^6
M2	12.5	25	~14.3x10^6
M3	6.25	12.5	~14.5x10^6
M4	3.125	6.25	~16.9x10^6
M5	1.5625	3.13	~38.2x10^6
M6	0.78125	1.56	~236x10^6

3. Modeling Setup

A. Modeling Physics

The twin jet simulations were approached using an incompressible RANS framework with isothermal conditions. The simulations were conducted as steady state where the time dependent times were set to zero. The RANS equations were discretized using the finite volume method and solved using the SIMPLE algorithm with Rhoe-Chow interpolation. This is referred to as the “segregated” solver in Star-CCM+ v11.04.012 [7].

The turbulence model used in these simulations is the standard κ-ε turbulence model [8], [9]. This model was used in conjunction with the two-layer wall treatment model [10]. The standard κ-ε turbulence model was selected to reduce the complexity of the analysis and limit the computational resources needed. The underlying assumptions of this two-equation turbulence model are discussed in greater detail in the provided references.

Table II. "Spot" Refinement Region Sizes and Cell Counts

Mesh Identifier	Jet Inlet Size (mm)	Core Size (mm)	Expansion Size (mm)	Cell Count
M1	5	10	20	4.9x10^6
M2	2.5	5	10	14.9x10^6
M3	1.25	2.5	5	19.2x10^6
M4	0.625	1.25	2.5	53.2x10^6
M5	0.3125	0.625	1.25	322x10^6

B. Boundary and Initial Conditions

The boundary surfaces where the boundary conditions are applied are shown in Fig 2. The inlet surfaces have a velocity inlet defined with a uniform constant profile. The inlet turbulent specification is done using the turbulence intensity of 0.053 and a length scale of 0.762 mm. The top surface is defined where a water surface with an average constant water height is. To provide a simple modeling of the water surface without adding complexity, a slip wall condition is applied (symmetry boundary condition). The outlets are defined as static-pressure outlets where the velocity gradients are set to zero. The boundary conditions and the corresponding surface color and value is shown in Table III.

Table III. Boundary and Initial Conditions for the Numerical Simulations

Boundary Type	Value	Surface
Velocity Inlet	0.75 m/s	Red
Static Pressure Outlet	0 Pa	Green
Symmetry (Slip-Wall)	-	Purple
Non-Slip	-	-

The Reynolds number of the flow was calculated to ~9100 based off hydraulic diameter of a single jet. The velocity profile where the inlet jet “exhausts” into the larger volume was found to be appropriately developed as compared to experimental profiles (not shown).

III. RESULTS

1. Bulk vs. Spot Refinement Strategies

A. Centerline Streamwise Velocity

The centerline streamwise velocity profiles along the z axis with the starting point defined at 0.5*S and 0.5*l are shown in Fig 4 and 5 for bulk and spot refinement simulations. The centerline streamwise velocity is considered a “global” quantity allowing for mesh “convergence” comparison for both meshing strategies. The merge and combined points are determined from these profiles.

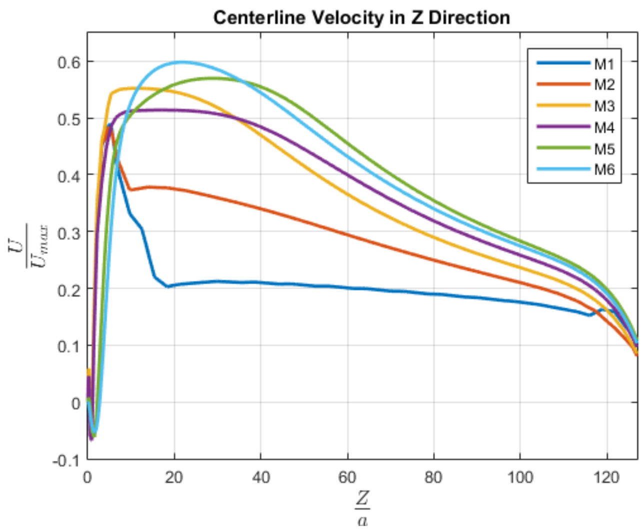

Fig 4: Centerline Streamwise Velocity for the “Bulk” Meshing Strategy Simulations

The “bulk” meshing strategy is observed to have deficiencies where the behavior of the profile does not “converge” between any two meshes. The structure of the velocity profile is vaguely similar but still characteristically different. The “spot” meshing strategy “converges” to approximately the same profile within a few meshing level. Excluding the M1 level, all meshes are observed to have the same trends and roughly the same magnitudes. The behavior of both of these strategies is the same as the profiles observed by Tanaka [2] and Anderson [3].

B. Merge and Combine Points

The merge and combined points for each level of mesh for both meshing strategies are viewed in Table IV. These are provided to show a quantitative measure of mesh “convergence” or lack of “convergence”. The “bulk” strategy is observed again to have a lack of “convergence”. Whereas the “spot” strategy reaches a series of values by the M3 level. The merge point begins to exhibit oscillatory convergence from M3 to M5 for “spot”. While the combined point reaches a somewhat constant value around M4.

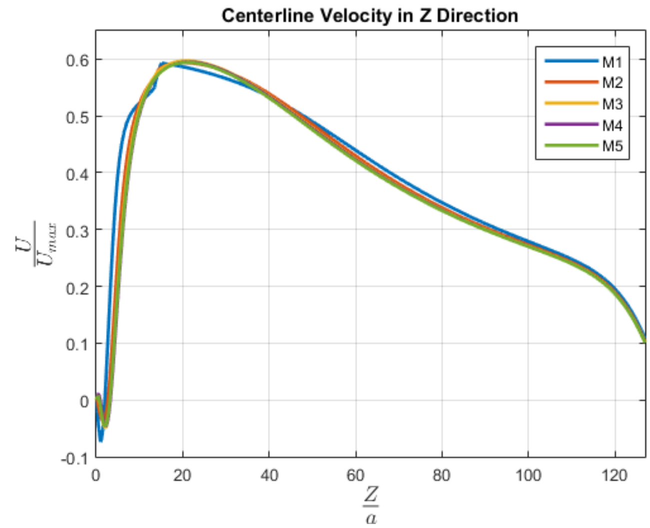

Fig 5: Centerline Streamwise Velocity for the “Spot” Meshing Strategy Simulations

Table IV: Merge and Combine Points - Bulk vs. Spot

Mesh Identifier	MP - Bulk	MP - Spot	CP - Bulk	CP - Spot
M1	1.0166	1.906	5.3369	15.1212
M2	1.0166	2.6684	4.7016	20.4581
M3	0.88948	3.1767	11.8174	20.9664
M4	1.0166	3.3038	17.153	21.3476
M5	2.1602	3.1767	28.9717	21.0934
M6	2.6684	n/a	21.7288	n/a

C. Velocity Profile at Specific Vertical Height

The velocity profile along the x axis at a specified vertical height in the converging region is shown in Fig 6 and 7 for the “bulk” and “spot” strategies. This profile is chosen to show what is the minimal mesh requirement to resolve the highest velocity region. The highest velocity region usually governed the meshing requirements for a simulation. If the highest gradients are not captured within a suitable way, the mesh is likely not sufficient to resolve the behavior of the flow.

The “bulk” meshing is observed to not reach a “converged” result for any of the meshes. The M1-M4 meshes do exhibit what could be considered “convergence” but do not show a physical result seen in literature [2], [3]. The “spot” meshes reach a “converged” result by M3 with slight variations seen for the successive meshes. The different mesh levels show a physical velocity profile with the exception of M1. The meshing levels for the “spot” strategy at even the lower cell counts show an ability to resolve first order flow behavior. Whereas, the meshing requirements for the "bulk" strategy are considered insufficient for even the most computationally intensive mesh level.

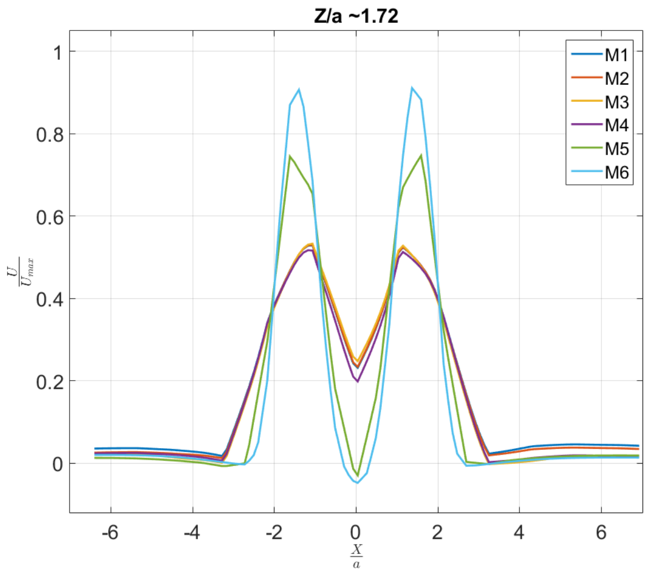

Fig 6: Velocity Profiles at Z/a ~ 1.72 for the "Bulk" Meshes

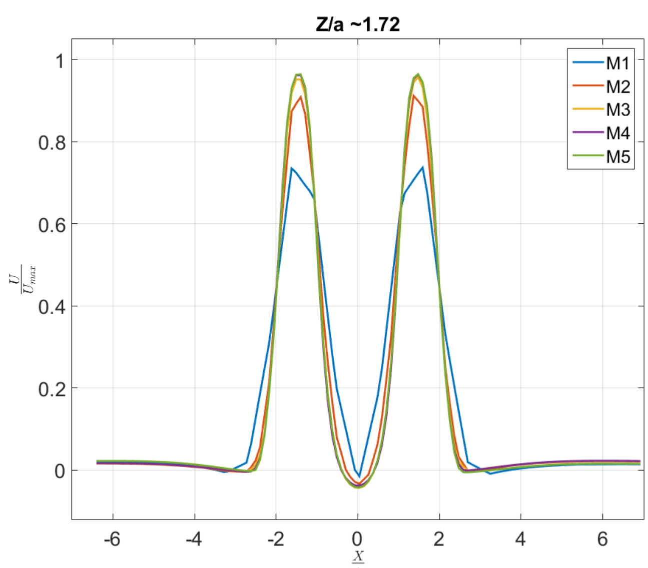

Fig 7: Velocity Profiles at Z/a ~ 1.72 for the "Spot" Meshes

Based on the information provided in the previous results sections, the "spot" meshes will be utilized for the following sections. The "bulk" meshing strategy for high aspect ratio changes in a computational domain (such as the width of the domain over the width of the jet) is considered an unsuitable meshing strategy. Unfortunately, this is one of the approaches recommended by "simpler" uncertainty quantification (mesh "sensitivity") metrics to develop appropriate uncertainty bands of the simulation results [11].

2. Solution Verification of Characteristic Parameters

The solution verification as define by Oberkampf and Trucano [1] is effectively building a case of evidence to show the solution is "verified". Utilizing the previously results shown for the "spot" mesh, it could be concluded the M4 level would be sufficient. In order to provide further evidence not based on the provided simulation results in isolation, analytical solutions from literature can be utilized. Fortunately for the single jet region, there are analytical solutions for self-similar jets [12].

Another means of providing evidence is analyzing higher order statistics and gradients of important parameters. In this case, the y-vorticity of the jets which is based on the gradient of velocity would be useful. In the combining region, the velocity gradients are significantly higher than in higher regions.

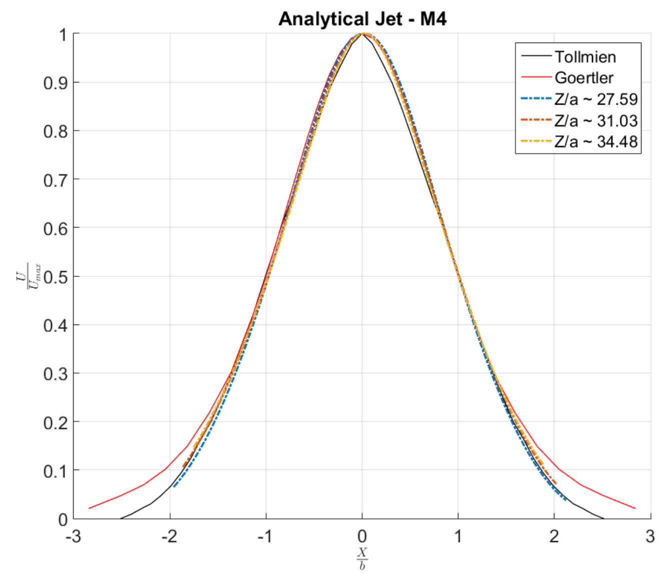

Fig 8: Self-Similar Turbulent Jet Profiles – Lower Heights

A. Comparison of Self-Similar Analytical Jets Profiles

The self-similar turbulent plane jet profiles for Goertler and Tollmien are solved using the Prandtl mixing length and Prandtl turbulent shear stress equations. The velocity profiles are normalized by the maximum velocity of that profile. The b defined as the half height of the jet. It is calculated for each profile by finding the point where the velocity is one half of the maximum velocity. Each of these profiles are plotted with different velocity profiles along the z-axis of domain in Fig 8 and 9.

The profiles of simulations are observed to compare well with both analytical profiles. This provides confidence in the simulations are at least supported by available theoretical solutions found in literature and not solely the calculated results.

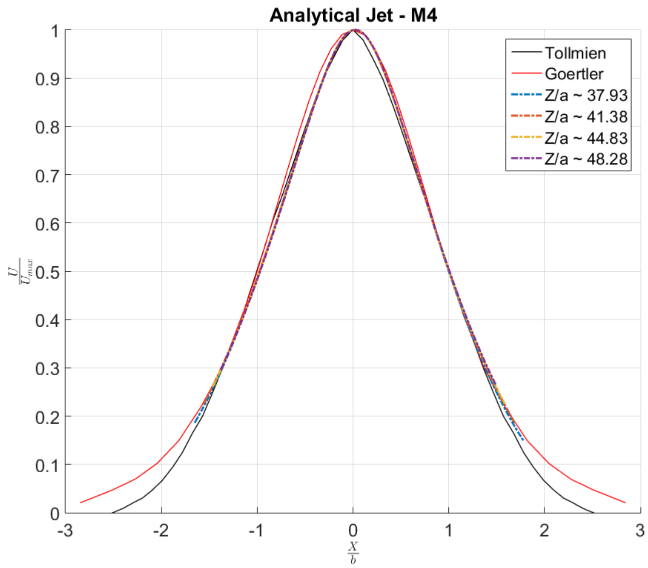

Fig 9: Self-Similar Turbulent Jet Profiles – Upper Heights

B. Comparison of y-Vorticity Profiles

The y-vorticity profiles are calculated using the velocity gradients in the x and z directions. The vorticity profiles for each mesh level of "spot" are shown in Fig 10.

The vorticity results provided show at least M4 needs to be used for comparisons. This supports the previous discussions from velocity profile and MP/CP results. The gradient of velocity is resolved by a reasonable mesh requirement as well.

C. Result of Solution Verification

The "spot" meshes have been shown to be able to be considered mesh "converged" for velocity related data and gradients of velocity related data. The minimum level of mesh required is M4 which corresponds to 53.2x10^6 cells or degrees of freedom in finite volume framework. This is limited in its assertion to the types of data shown and further work would be required for higher order statistics.

3. Solution "Validation" of Characteristic Parameters

The available experimental data for LDV and PIV are presented for comparison and "validation" of the simulation results. The experimental results present do not include confidence intervals and uncertainty bands. These quantities were not available in the provided data sets used for comparison [2], [3]. The results shown are used to discuss if the general trends and flow behavior are captured. As such, the "validation" effort is not a validation effort by the truest definition. Though it does provide insight into the viability of the solution verification efforts shown previously.

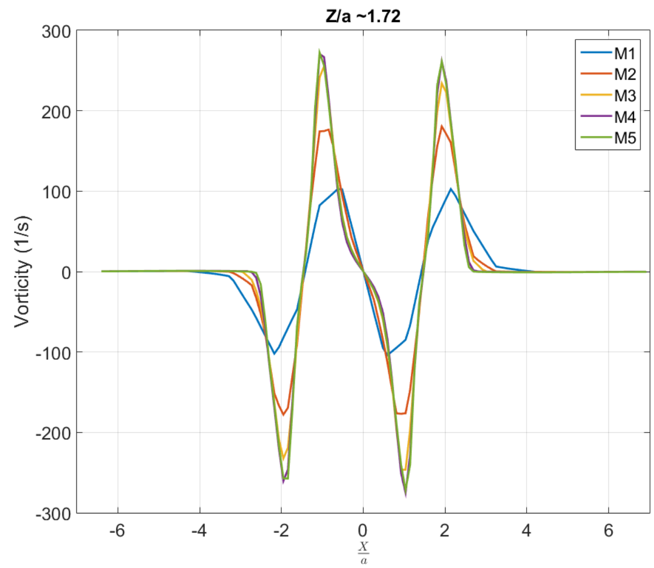

Fig 10: y-Vorticity Profiles at Z/a ~ 1.72 for "Spot" Meshes

A. Merge and Combined Points

The merge and combined points for the LDV and PIV experiments and the "spot" M4 mesh are shown in Table V. The merge point is presented as a range of value for both experimental results. This is due to the difficulty of trying to find the point where the quantity is considered "zero".

Table V. Merge and Combined Points - Solution "Validation"

Identifier	MP	CP
"Spot" - M4	3.30	21.35
Exp - LDV	1.72-3.45	15.52
Exp - PIV	2.66-3.50	16.84

The merge point of the simulation results is within the ranges of the experimental results. It indicates, in additional to the behavior seen in Fig 5, the "spot" mesh is able to predict the behavior in that region. The combined point for the "spot" mesh is approximately 31.62% and 23.62% different than the LDV and PIV results. This is potentially a point of concern and requires the analysis of further quantities shown in the following section.

B. Velocity Profiles at Different Characteristic Heights

The velocity profiles at the different characteristic heights within the domain are shown in Fig 11. These heights are selected to exemplify the predictions in different regions of flow. In particular, the profiles in the converging region before the merge point and at the merge point are presented.

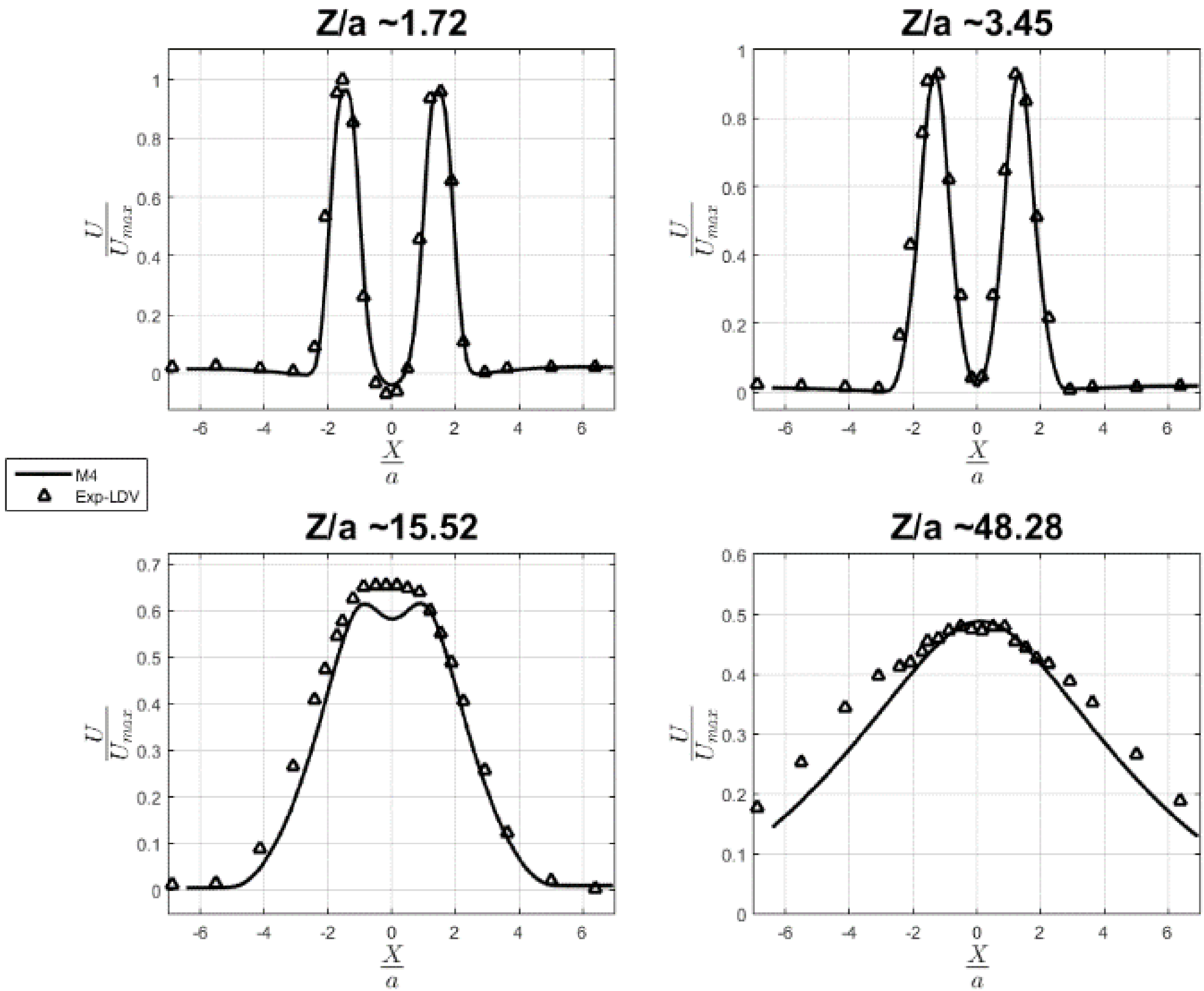

Fig 11: Velocity Profiles at Different Characteristic Heights for "Spot" Mesh and LDV Experimental Results

The profiles of near the combined point and well into the single self-similar jet region are presented as well. Each of these profiles are only compared for the "spot" M4 mesh and the experimental LDV results. The PIV experimental results of this same case were not reported for the same heights as the LDV data.

The "spot" predictions of the converging region and at the merge point are observed to have a strong comparison to the experiment. Some asymmetry of the "left" jet at the Z/a ~ 1.72 height is not captured but are not believed to be significant. The prediction at the merge point has no visible deficiencies observed. The "spot" results near the combined point (Z/a ~ 15.52) confirm what was observed in the previous section. The experiment is showing the two jets have merged into a single jet whereas the "spot" profile is still in the process of merging. It is noted the overall profile is captured excluding in the peak of the profile seen. The single self-similar jet profile at Z/a ~ 48.28 is observed to have a strong comparison in the peak region of the profile. The left and right "legs" are found to be under-predicted by the "spot" results. This has been investigated and the authors suspect the experimental data collected in the upper profile were not collected for a long enough period of time. This area has a much lower velocity than in the converging region. This requires significantly more averaging time than in the converging region to reach a statistically stationary state.

Based off the comparisons of "spot" and LDV experimental results, the simulations were able to capture the majority of the flow behavior. It is suggested that these simulations could be considered "validated" with two strong caveats. First, the simulation results in the merging region of the flow has treated with care. This is due to the significant percent differences of combined point between the simulation and both sets of experimental results. Second, the "validation" is only as strong as the available experimental data. Without additional information regarding the uncertainty bands, the overall "validation" has to be treated with care as well. Further efforts will involve investigating a more appropriate form of validation.

4. CONCLUSION

The simulations of twin turbulent planar-like jets were conducted using a steady RANS formulation. It was found that an appropriate meshing strategy can be determined using standard mesh "convergence" practices found in literature. The mesh "convergence" can serve as a form of solution verification but not as a means of determining uncertainty bands. The current "simple" methods of determining uncertainty in simulations for CFD (namely GCI) have strict requirements which results in enormous computational requirements. For a flow situation where there are large aspect ratio changes in geometry, these requirements are almost implausible of satisfying. As shown in the body of the paper, a "spot" refinement meshing strategy yields results that can be show to be mesh converged. This was observed for qualities such as centerline streamwise velocity, merge and combined points, and y-vorticity (gradient term). Additionally, the velocity profiles in the single self-similar jet region were found to compare favorably to analytical turbulent planar jet analytical solutions. Further work will be undertaken to determine how uncertainty bands can be developed based on the discretization of the problem.

The simulation work was found to compare favorably to the available experimental data with some issues. These issues are likely due to modeling assumptions and identified issues with the experimental data provided. The "validation" does provide confidence in the "spot" simulation compared to the experimental data. The simulations were found to capture and resolve the majority of the flow behavior. This is excluding the combined point prediction and the velocity profiles in that general region which were found to have deficiencies.

5. NOMENCLATURE

Z = Vertical direction of domain
a = Width of a duct
D_h = Hydraulic Diameter based on cross-sectional area of a single duct
S = Separation distance between the centerline of each duct
U_o = Bulk velocity of a duct
U_{max} = Maximum velocity in the computational or experimental domain
U_m = Maximum velocity for a specified profile
b = Half-width height of the velocity profile
M# = Mesh level of a specific mesh strategy
MP = Merge Point
CP = Combined Point
LDV = Laser Doppler Velocimetry
PIV = Particle Image Velocimetry
TJWF = Twin Jet Water Facility
V&V = Verification and Validation

6. ACKNOWLEDGMENTS

This material is based upon work supported under a Department of Energy Nuclear Energy University Programs Graduate Fellowship. Additionally, any opinions, findings, conclusions or recommendations expressed in this publication are those of the authors and do not necessarily reflect the views of the Department of Energy Office of Nuclear Energy.

Portions of this research were conducted with high performance research computing resources provided by Texas A&M University (http://hprc.tamu.edu).

7. REFERENCES

1. W. L. OBERKAMPF, T. G. TRUCANO, "Verification and Validation in Computational Fluid Dynamics," *Prog. Aero. Sci.*, **38**, 209 (2002).
2. E. TANAKA, "The Interference of Two-Dimensional Parallel Jets (2nd report, experiments on the combined flow of dual jet)," JSME, **17**, 920 (1974).
3. E. ANDERSON, R. SPALL, "Experimental and Numerical Investigation of Two-Dimensional Parallel Jets," *Jour. Fluids Eng.*, **123**, 401 (2004).
4. M. CROSSKEY, A. RUGGLES, "UTK Twin jet water facility computational fluid dynamics validation data set," *Proc. ICAPP 2014*, Charlotte, North Carolina, April 6-9, 2014, American Nuclear Society (2014).
5. H. WANG S. LEE, Y. A. HASSAN, A. E. RUGGLES, "Laser-Doppler measurements of the turbulent mixing of two rectangular water jets impinging on a stationary pool," *Int. Jour. Heat and Mass Transfer*, **92**, 206 (2016).
6. H. WANG, S. LEE, Y. A. HASSAN, "Particle Image Velocimetry Measurements of the Flow in the Converging Region of Two Parallel Jets," *Nucl. Eng. Des.*, **306**, 89 (2016).
7. CD-adapco STAR-CCM+ v11.04.012 User's Guide, www.cd-adapco.com
8. W. P. JONES, B. E. LAUNDER, "The Prediction of Laminiarization with a Two-Equation Model of Turbulence," *Int. Jour. Heat and Mass Transf.*, **15**, 301 (1972).
9. B. E. LAUNDER, B. I. SHARMA, "Application of the Energy Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disc," *Lett. Heat Mass Transf.*, **1**, 131 (1974)
10. W. RODI, "Experience with Two-Layer Models Combining the k-e Model with a One-Equation Model Near the Wall," *Proc. 29th Aerospace Science*, Reno, Nevada, January 7-10, 1991, American Institute of Aeronautics and Astronautics (1991).

11. P. J. ROACHE, "Perspective: A Method for Uniform Reporting of Grid Refinement Studies," *Jour. Fluid Eng.*, **116**, 405 (1994).
12. N. RAJARATNAM, "Turbulent Jets", *Elsevier* (1976).