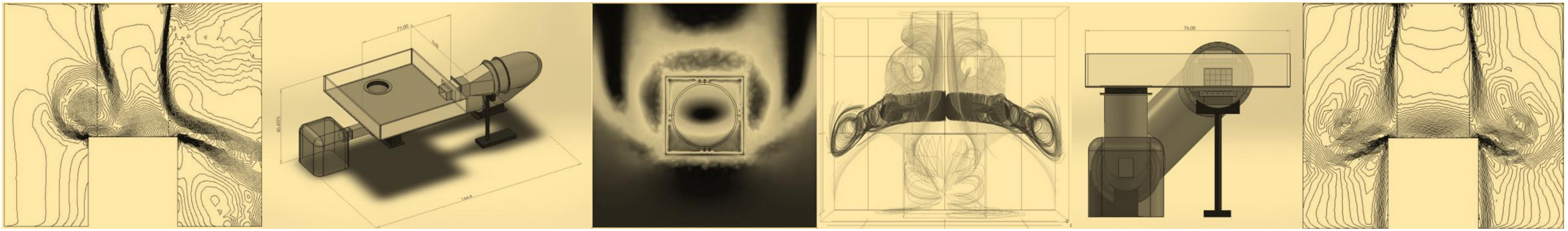


Vortical Structures of an Impinging Jet in Cross-flow

KEITH KUCINSKAS
UNIVERSITY OF HARTFORD
COMSOL CONFERENCE IN BOSTON
OCTOBER 2013

COMSOL
CONFERENCE
BOSTON2013

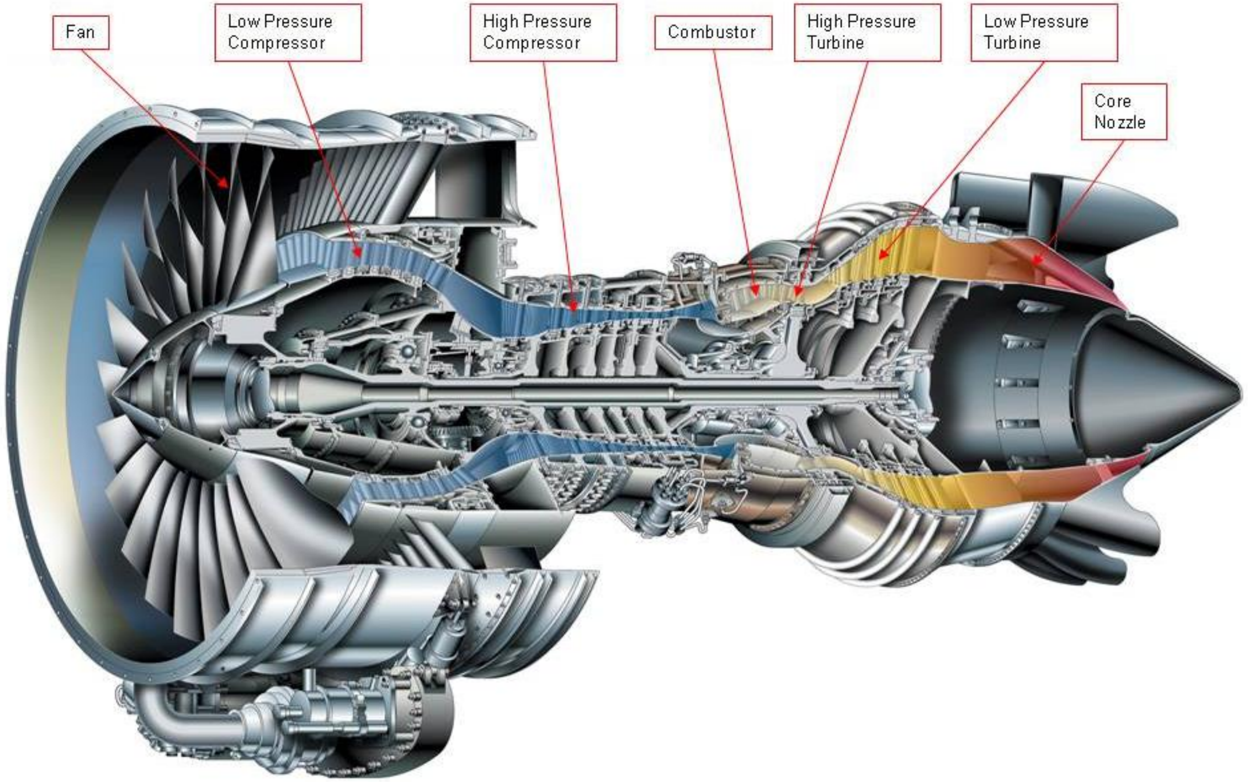
Introduction

Turbofans are the prevalent engine architecture in modern day avionics

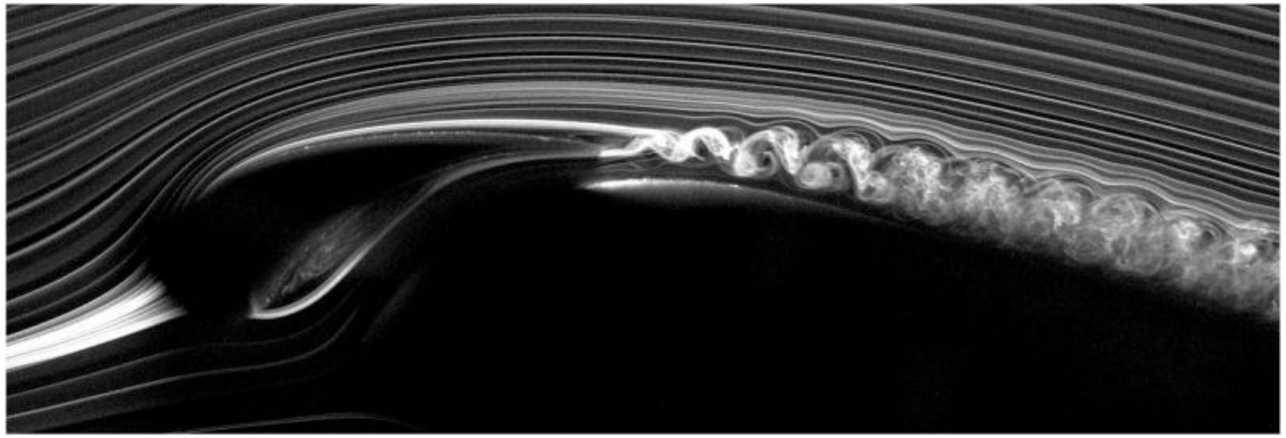
Secondary flows exist in every module causing debits in isentropic efficiency

Understanding the formation of vortical structures is essential to reduce the thermal specific fuel consumption

This computational study analyzes the similar vortices of an impinging in a cross-flow to maximize flow visualization in a water tunnel



Source: Pratt and Whitney, PW6000 Cutaway http://www.pw.utc.com/Content/PW6000_Engine/img/B-1-6_pw6000_cutaway_high.jpg (accessed July 3, 2013)



Source: Tokyo Metropolitan University. Vortex Shedding and Noise Radiation from a Slat Trailing Edge http://aero-fluid.sd.tmu.ac.jp/en/research/acoustics.html (accessed July 3, 2013)

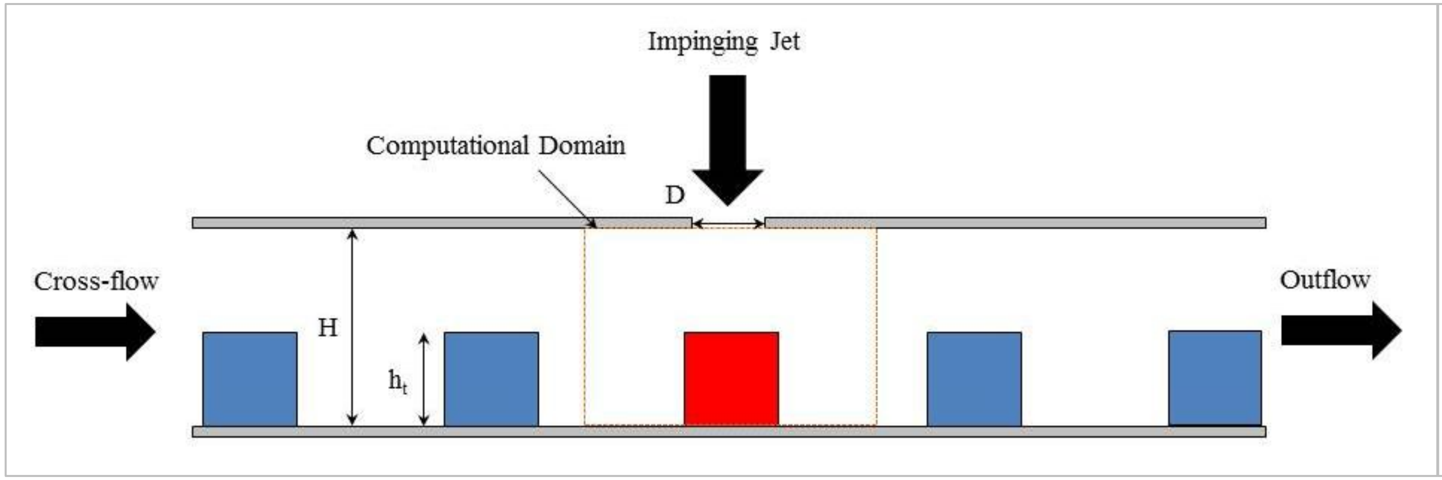
Validation CFD Modeling

- Previous studies
  - Airflow of an impinging jet in cross-flow
  - Particle Image Velocimetry (PIV) in an experiment
  - Single cube CFD studies using Reynolds Stress Model (RSM) and $\overline{v^2} - f$
- Current Study
  - Water flow of an impinging jet in cross-flow
  - k-ε turbulence model using COMSOL
- Validation study
  - Airflow
  - k-ε turbulence model

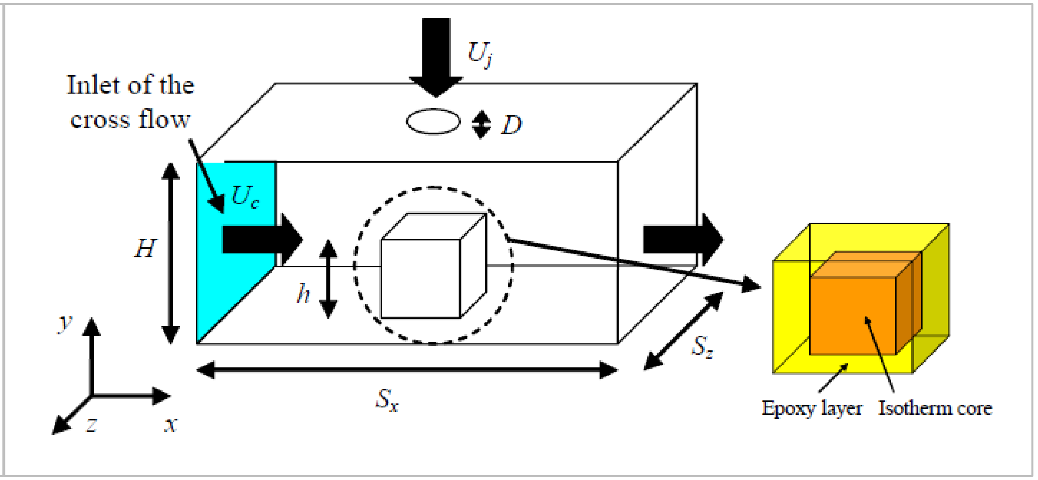
Table of Variables

Variable	Value	Units	Description
D	12	mm	Diameter of hole
h_t	15	mm	Cube side length
H	30	mm	Box height
S_x	60	mm	Box length
S_z	60	mm	Box width
δ_c	1.5	mm	Epoxy thickness
U_c	1.73	m/s	Cross-flow velocity
U_j	10	m/s	Jet flow velocity
$U_{j/}U_c$	5.78	N/A	Velocity ratio

Schematic of Experimental Set-up



Schematic of Computational Domain

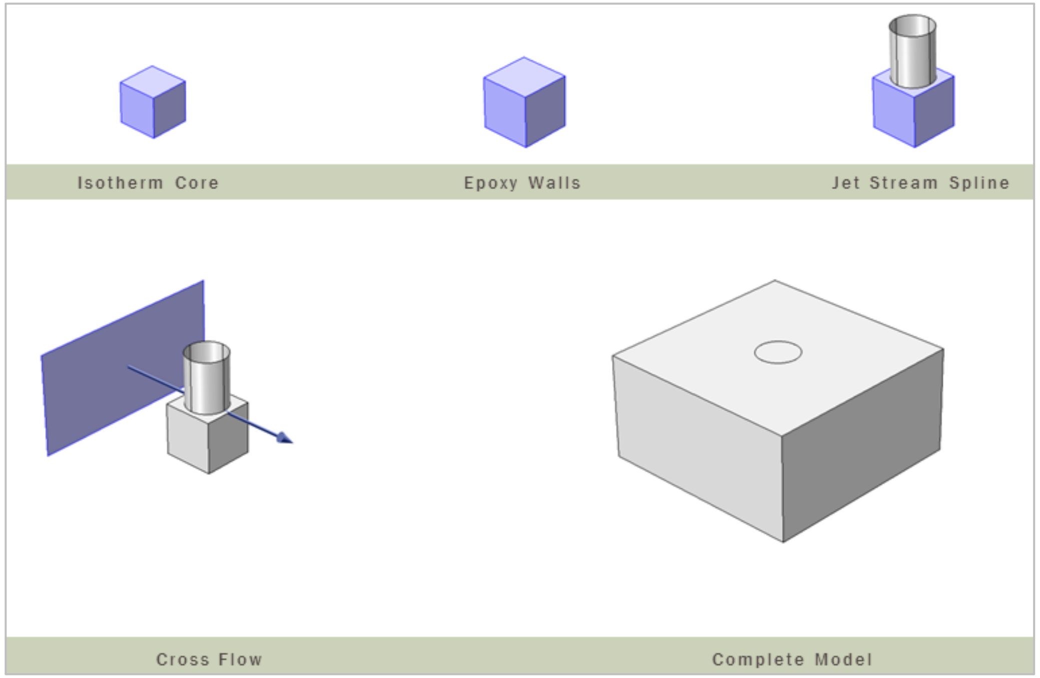


Source: Rundstrom, D., B. Moshfegh, and A. Ooi. 2007. "RSM and V2-f Predictions of an Impinging Jet in a Cross Flow on a Heated Surface and on a Pedestal." *16th Australasian Fluid Mechanics Conference*: 317

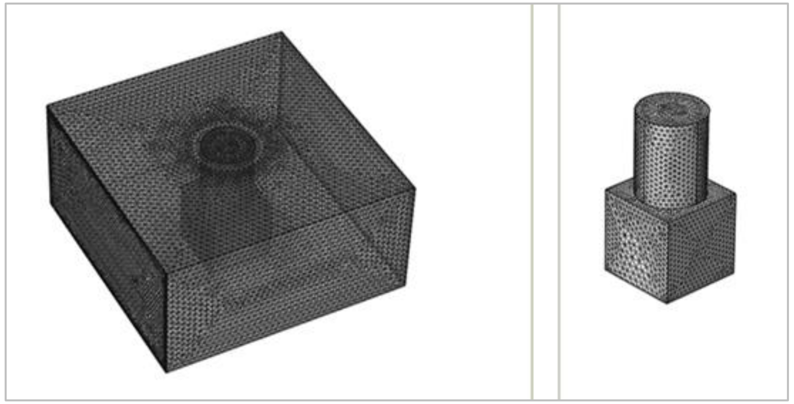
Geometric Modeling and Mesh

- Geometric modeling generated with a circular spline
    - Expected boundary of the jet
    - No physical boundaries assigned
    - Utilized for mesh refinement
- Meshing
    - Initially a normal physically controlled mesh per the default settings of COMSOL
    - Mesh refined at the jet spline and cube surfaces through manual manipulation
    - Manual coarse mesh applied to core

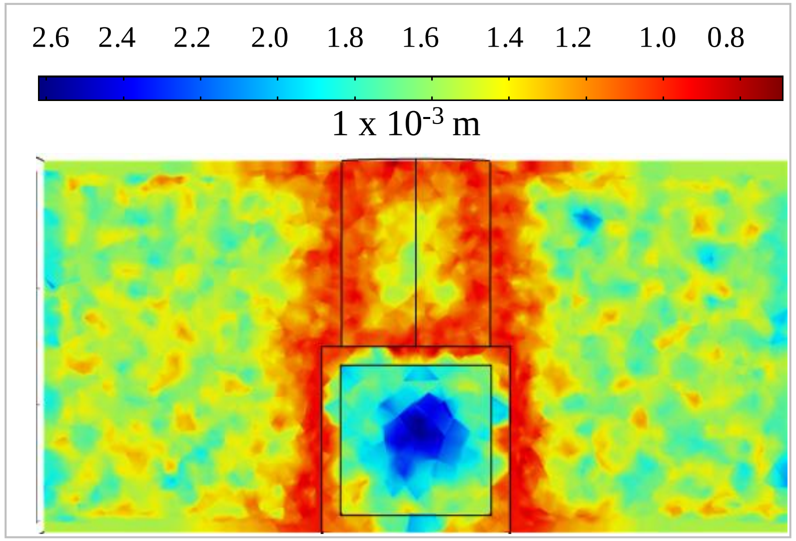
Model Geometry



Mesh Visualization

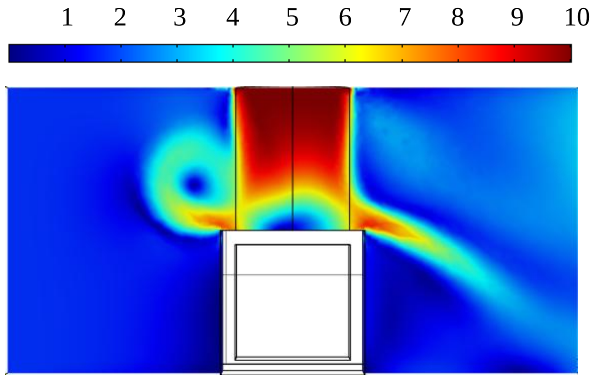


Mesh Size Graph

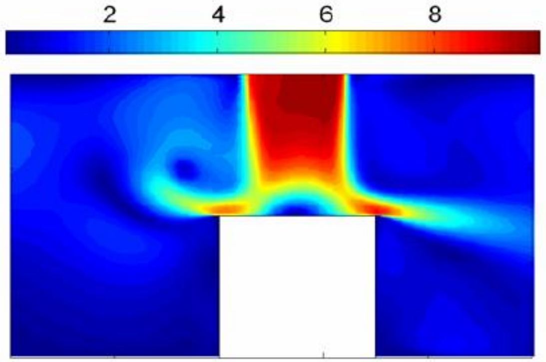


Validation Model – Velocity Contours

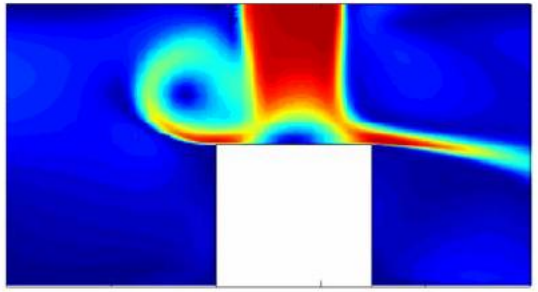
- Velocity magnitude contours in (m/s)
- Horseshoe vortex size in all plots are roughly 80% of the cube side length
- k-ε validation model
  - Comparable results to the $\overline{v^2} - f$
  - Comparable results to the PIV measurements except it overestimates the velocity magnitude at the top of the vortex
- Previous studies
  - RSM seems to be the least like the PIV data
  - The $\overline{v^2} - f$ matches the PIV data better



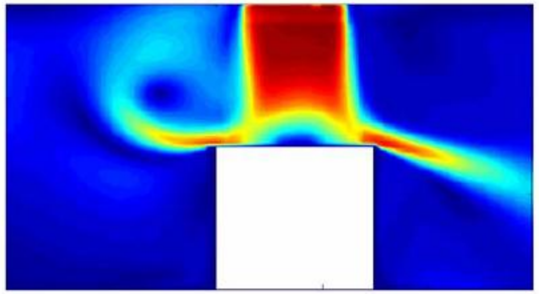
Current Study: k-ε model



Previous Study: RSM



Previous Study: $\overline{v^2} - f$ model



Experiment: PIV data

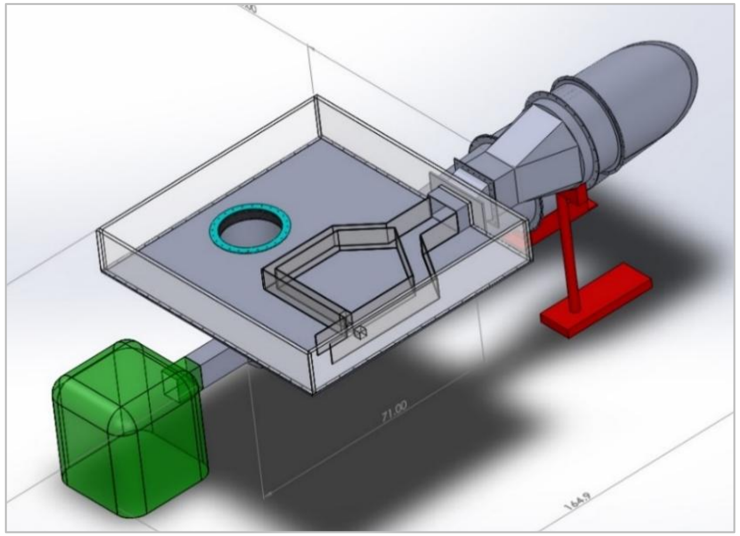
Impinging Jet in Cross-flow within a Water Tunnel

- Hydraulic analogy -- use water as the flow medium instead of air
- Maintain the Reynolds number and cross-flow to jet velocity ratio
- Low speed flow for enhanced flow visualization that has the same vortical structures as the airflow models

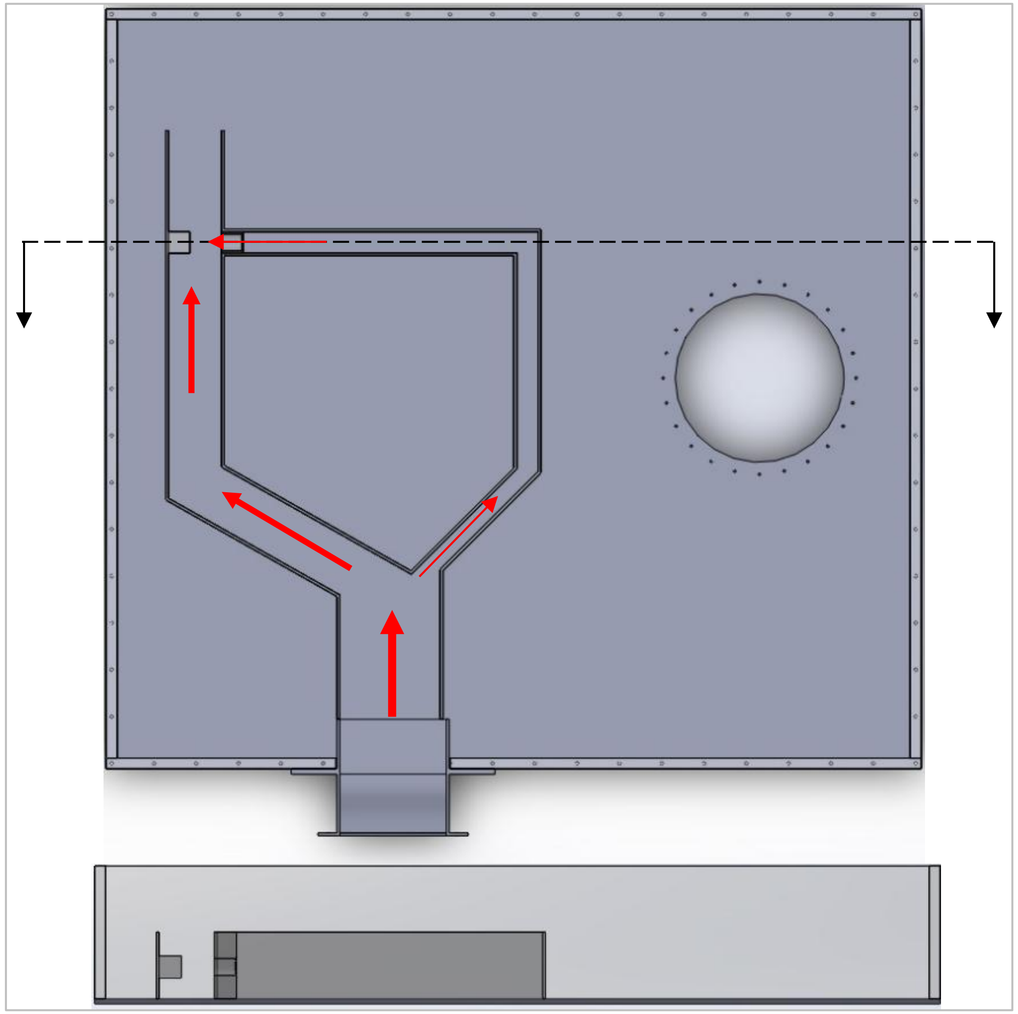
$$\mathrm{Re} = \frac{\rho_a \cdot U_{air} \cdot L_{char}}{\mu_a} = \frac{\rho_w \cdot U_{water} \cdot L_{char}}{\mu_w} \longrightarrow \frac{U_{air}}{U_{water}} = \frac{\rho_w \cdot \mu_a}{\rho_a \cdot \mu_w}$$

- Less expensive equipment
- Less expensive models – aerodynamic bodies do not need to withstand the high drag and lift forces
- Same method used by NASA’s flow visualization facility (FVF) established in 1983 for studying secondary flows

Overall Water Table Setup and Test Section

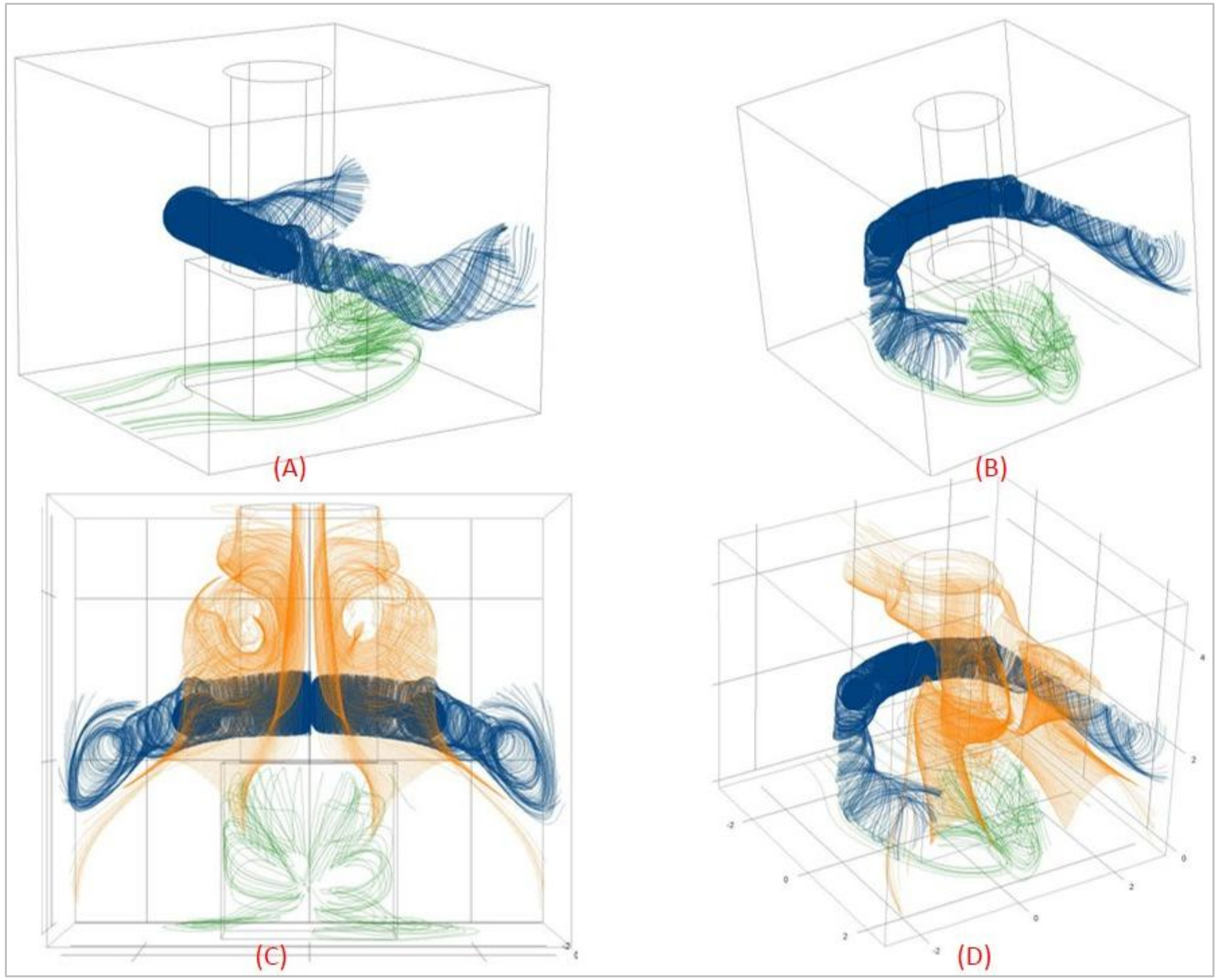


Top and Section View of Test Cell

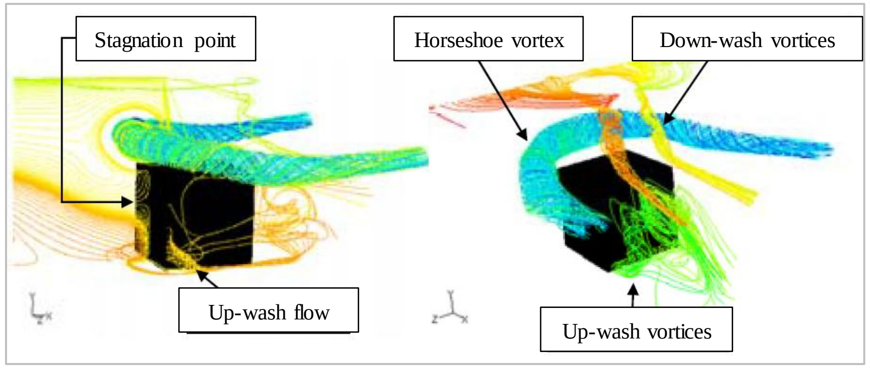


Streamlines of Water Model

Streamline Plots of the Steady State Water Model



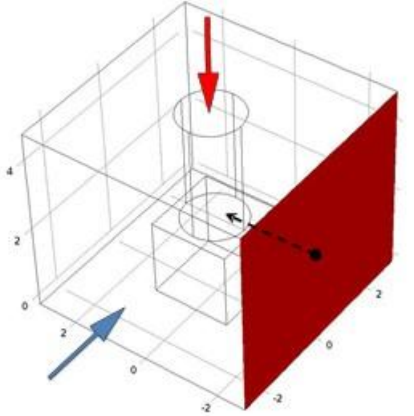
Streamline Plots of the Previous Study Air Model through RSM

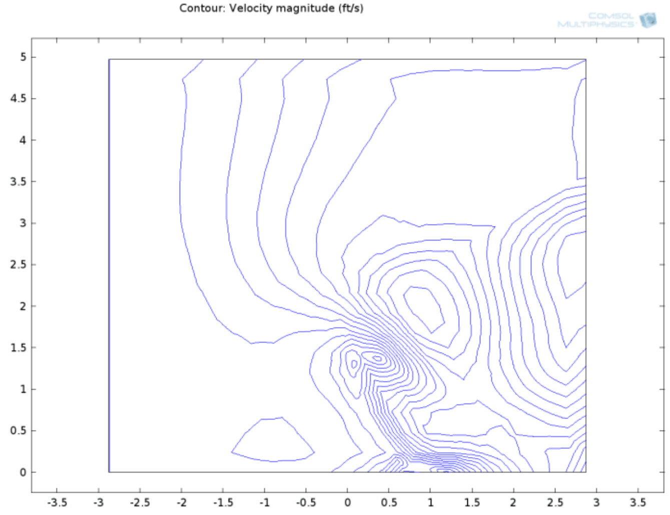


Source: Rundstrom, D., B. Moshfegh, and A. Ooi. 2007. "RSM and V2-f Predictions of an Impinging Jet in a Cross Flow on a Heated Surface and on a Pedestal." *16th Australasian Fluid Mechanics Conference*: 319

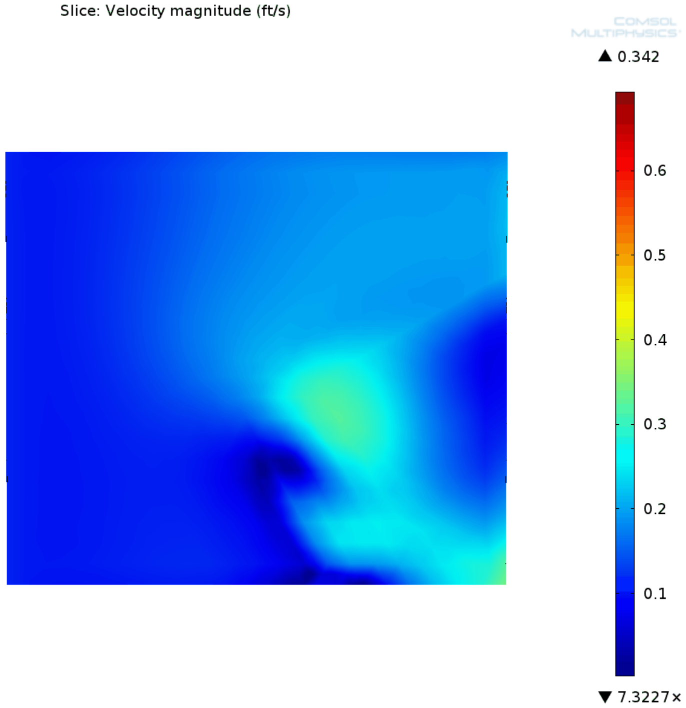
- CFD water tunnel model generated streamline plots
  - Horseshoe vortex
    - Induced by cross-flow and impinged jet colliding
    - Counter-rotating vortex pair (CVP)
    - Diverging from the center
    - Vortex diameter increasing
  - Up-wash vortices in the wake of the cube
    - Cross-flow induced
    - Low velocity pocket
  - Down-wash vortices
    - A pair of vortical structures
    - Induced by a normal cross-flow at the top
    - Inconsistent diameter that dissipates
- Compared to RSM of previous literature
  - Does not accurately depict increasing diameter of CVP
  - Down-wash vortex is depicted with a constant diameter

Steady State XY Cut Planes

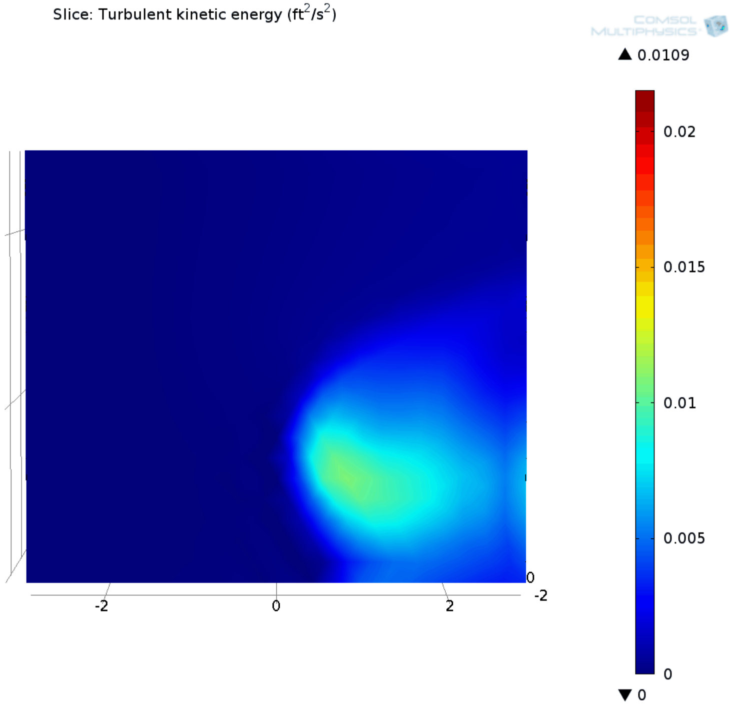




Velocity Contours

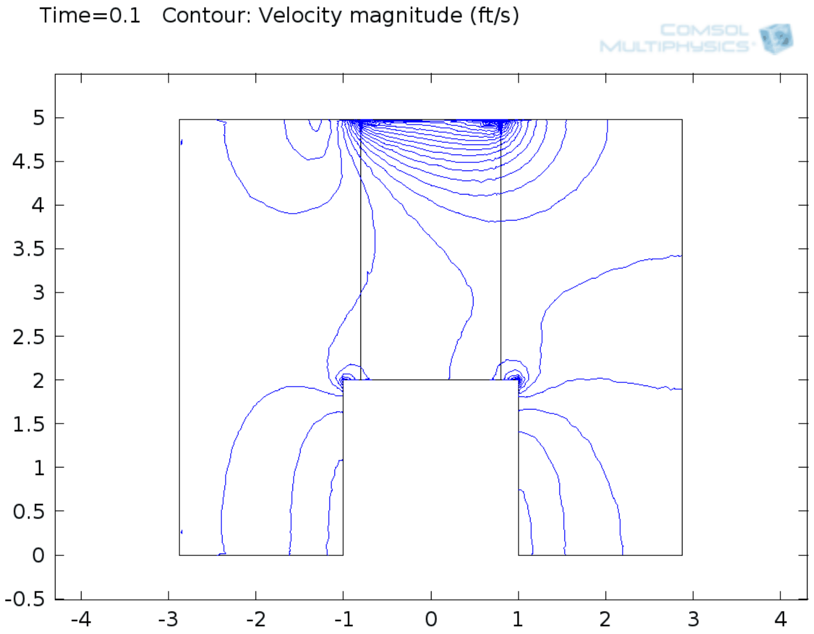


Velocity Magnitude Contours

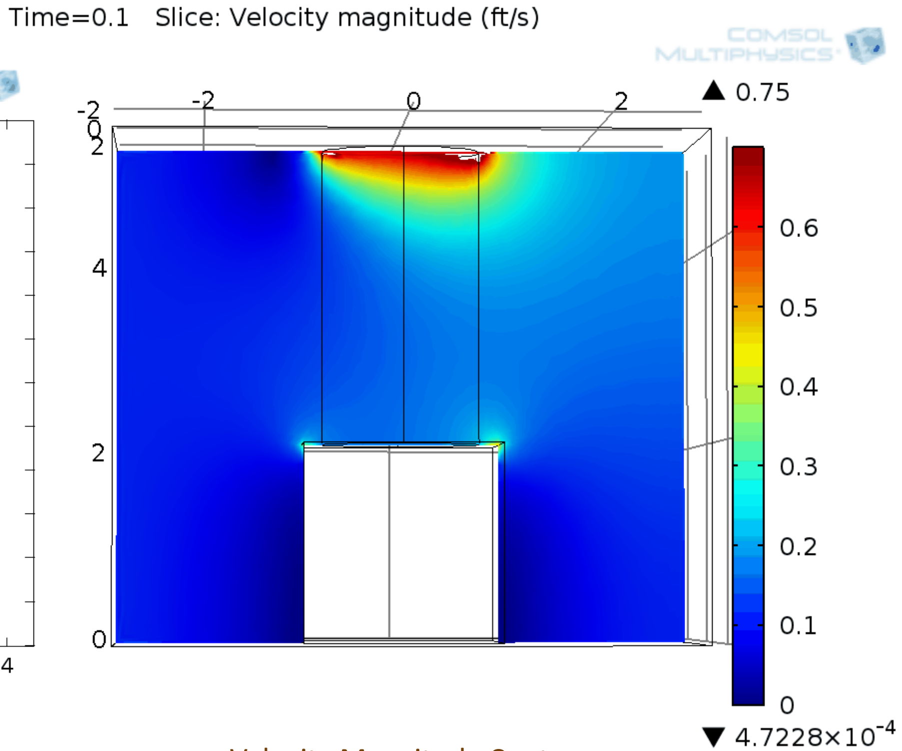


Turbulent Kinetic Energy Magnitude Contours

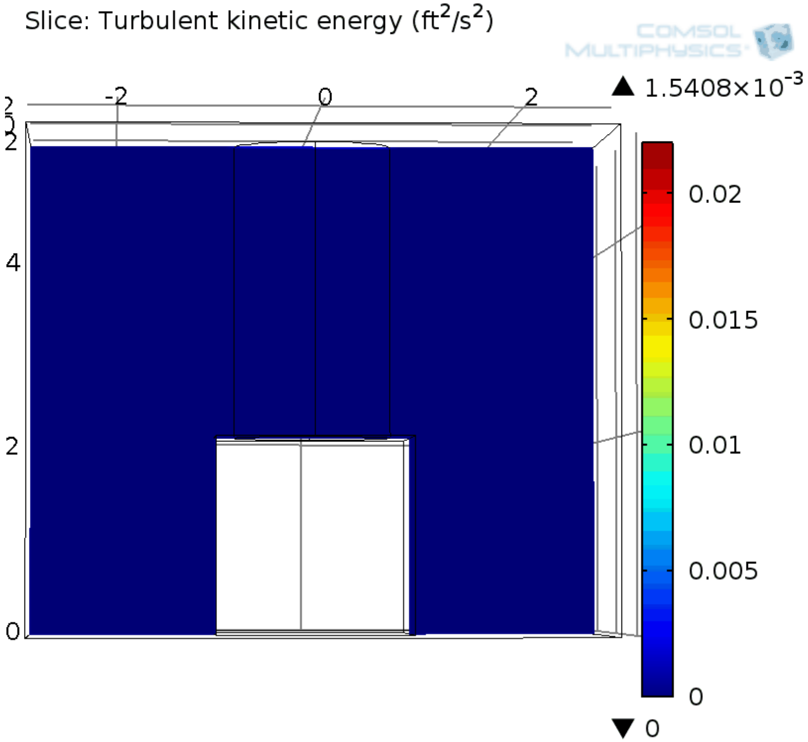
Time Dependent – XY Plane



Velocity Contours



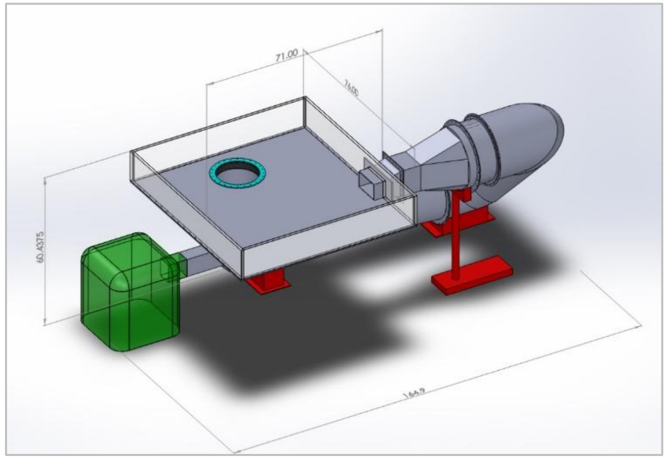
Velocity Magnitude Contours

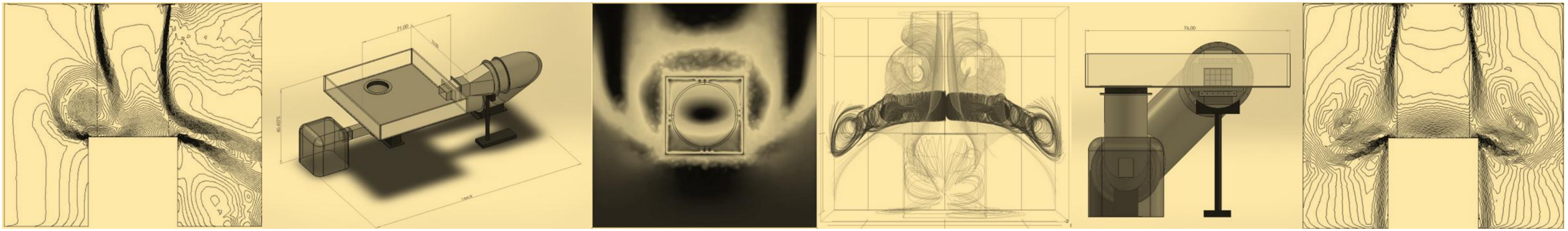


Turbulent Kinetic Energy Magnitude Contours

Conclusion

- Impinging jet in cross-flow
  - Secondary flow structures
  - Validated CFD modeling
  - Utilized hydraulic analogy
  - Detailed steady state and time dependent analysis of the flow
- Study continuation
  - Refurbishment and assembly of a water tunnel donated to UHART by UTRC
  - Experimentation to confirm findings found with COMSOL





Auxiliary Slides for Specific Questions

- Acknowledgements
- References
- Secondary Flow Development in Turbines
- Validation Model Inputs
- Physics Background
- Governing Equations
- Validation Model – Turbulent KE Contours
- Hydraulic Analogy Variable Determination
- Steady State Non-dimensionilized Comparison Forward of the Cube
- Steady State Non-dimensionilized Comparison Aft of the Cube
- Steady State YZ Cut Planes → Movie
- Steady State XZ Cut Planes → Movie
- Time Dependent YZ Cut Planes → Movie
- Laminar CFD Model Results
- Comparison of Flow without Jet

HOME

Acknowledgements

- Dr. Ivana Milanovic – Graduate project advisor
- Dr. William Cousins – UTRC lead for the water tunnel install
- Dr. Joel Wagner – P&W water tunnel specialist
- Alexander Nelson – Colleague graduate student supporting water tunnel install
- Anton Banks – Graduate student completing the pending tasks
- Katrina Kucinskas – My very patient wife and editor

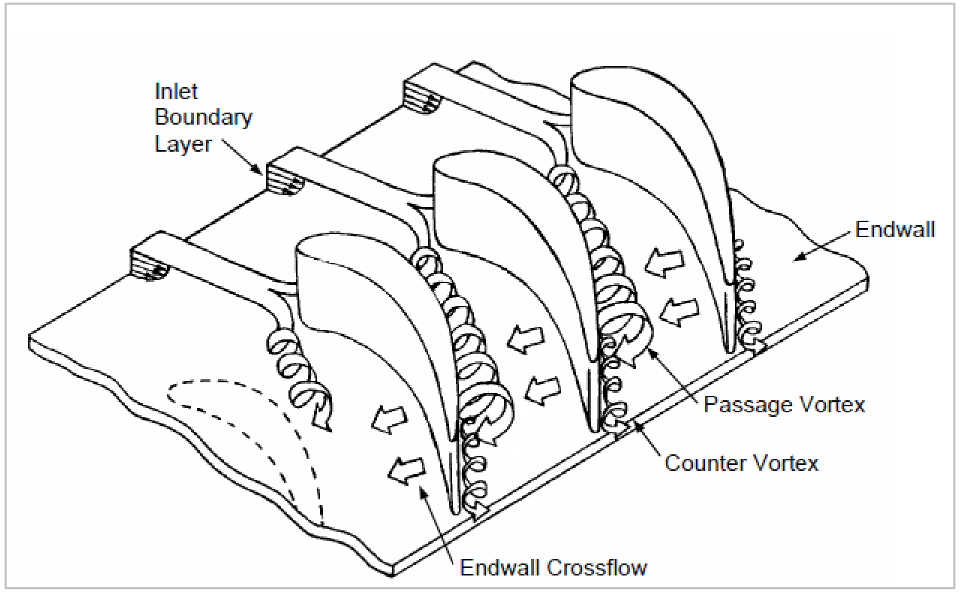
References

- Davidson, P. A. 2004. *Turbulence: An Introduction for Scientists and Engineers*. Oxford: Oxford UP
- Holley, Brian Matthew. 2008. Surface Measurements of Flow in a Plane Turbine Cascade. Ph. D. diss. University of Connecticut
- Kumar. V., I. Ng., G.J. Sheard, K. Hourigan, A. Fouras. 2009. “Hydraulic Analogy Examination of Underexpanded Jet Shock Cells using Reference Image Topography.” 8th International Symposium on Particle Image Velocimetry – PIV09, August 25-28 in Melbourne Australia
- NASA. Flow Visualization Facility. http://www.nasa.gov/centers/dryden/history/pastprojects/FVF/index_prt.htm (accessed July 15, 2013)
- Rundstrom, D., B. Moshfegh, and A. Ooi. 2007. "RSM and V2-f Predictions of an Impinging Jet in a Cross Flow on a Heated Surface and on a Pedestal." *16th Australasian Fluid Mechanics Conference*: 316-323.
- Rundstrom, D., and B. Moshfegh. 2009. "Large-eddy Simulation of an Impinging Jet in a Cross-flow on a Heated Wall-mounted Cube*."* *International Journal of Heat and Mass Transf*er 52.3-4: 921-31.
- Tummers, M. J., M. A. Flikweert, K. Hanjalic, R. Rodink, and B. Moshfesh. 2005. "Impinging Jet Cooling of Wall-mounted Cubes." *Proceedings of ERCOFTAC International Symposium on Engineering Turbulence Modeling and Measurements - ETMM6*: 773-782

Secondary Flows

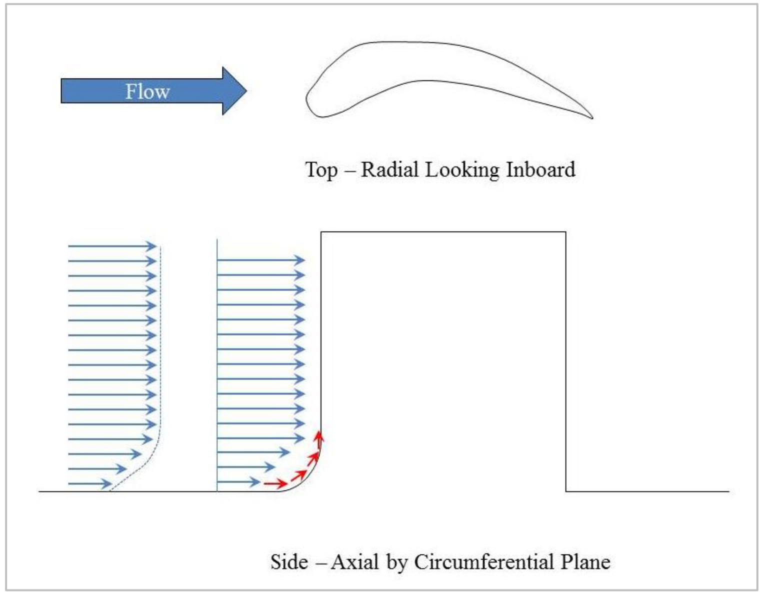
- In a turbine, the flow approaches leading edge of the airfoil
- Boundary layer on end wall causes a low speed cross-flow
- Horseshoe vortex forms at the leading edge close to the root
- Two legs of the vortex have an opposite sense of rotation and increase in diameter as they progress through the passage
- Visualization is difficult using airfoils due to the curved surfaces and multiple passages
- The impinging jet in cross-flow can also be created using a jet against a cube and results in better flow visualization

Secondary Flow Model through a Turbine Cascade

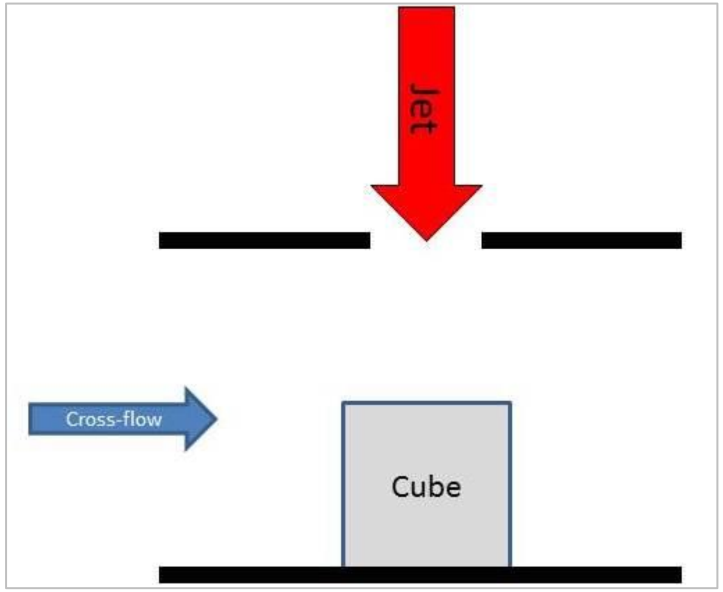


Source: Holley, Brian Matthew. 2008. Surface Measurements of Flow in a Plane Turbine Cascade. Ph. D. diss. University of Connecticut, pg. 1

Impinging Jet in Cross-flow of a Turbine Stage



Impinging Jet in Cross-flow using a Jet and Cube



HOME

Validation Model Inputs

Variable	Value	Units	Description
$c_{p,a}$	1,006.4	J/(kg-K)	Heat capacity constant pressure, air
$c_{p,e}$	1,668.5	J/(kg-K)	Heat capacity constant pressure, epoxy
k_a	0.0257	W/(m-K)	Thermal Conductivity, air
k_e	0.236	W/(m-K)	Thermal Conductivity, epoxy
p	1	atm	Pressure, air
R	287	J/(kg-K)	Gas Constant, air
T_c	20	°C	Static temperature of cross flow
T_i	70	°C	Temperature of isothermal core
T_j	20	°C	Static temperature of jet flow
U_c	1.73	m/s	Velocity of cross flow
U_j	10	m/s	Velocity of jet
ε_e	0.89	--	Surface emissivity, epoxy
μ_a	1.789E-05	kg/(m-s)	Dynamic viscosity, air
ρ_a	1.204	kg/m^3	Density, air
ρ_e	1,150.0	kg/m^3	Density, epoxy
Υ	1.4	--	Ratio of specific heat, air

Physics Background

- Flow Regime
  - To set the proper physics in the model, the flow regime must be determined
  - Reynolds number → Ratio of inertia to viscous forces (eq. 1)
  - Cross-flow
    - Characteristic length is the hydraulic diameter (eq. 2)
    - Solving yields a Reynolds number of 4,657 (eq. 3)
    - Flow is turbulent
  - Jet Flow
    - Characteristic length is the jet diameter
    - Solving yields a Reynolds number of 8,076 (eq. 4)
    - Flow is turbulent
- Compressibility
  - Air's density cannot be considered constant at a threshold
  - Mach number < 0.2 is considered incompressible
  - Speed of sound at room temperature and atmospheric pressure (eq. 5)
  - Mach number calcualtions
    - Cross-flow → M = 0.005 → Incompressible (eq. 6)
    - Jet Flow → M= 0.019 → Incompressible (eq. 7)

Eq #	Equations
1	$\mathrm{Re} = \frac{\rho \cdot U \cdot L_{char}}{\mu}$
2	$D_{H,c} = \frac{4A}{P} = \frac{2(S_z \cdot H)}{(S_z + H)} = 0.04m$
3	$\mathrm{Re}_c = \frac{\rho_a \cdot U_c \cdot D_{H,c}}{\mu_a}$
4	$\mathrm{Re}_j = \frac{\rho_a \cdot U_j \cdot D}{\mu_a}$
5	$a = \sqrt{\gamma RT} = \sqrt{1.4 \cdot (287) \cdot 293.15} = 343.202 \left[\frac{m}{s}\right]$
6	$M_c = \frac{U_c}{a} = 0.005$
7	$M_j = \frac{U_j}{a} = 0.029$

Governing Equations

- Reynolds Average Navier Stokes (RANS) equations
  - Derived based on Newton’s 2nd law of motion regarding momentum
  - For laminar flows, the equations are capable of converging
  - The flow in the experiment is however turbulent

RANS

$$\rho\left(\frac{\partial u_x}{\partial t}+u_x\frac{\partial u_x}{dx}+u_y\frac{\partial u_x}{dy}+u_z\frac{\partial u_x}{dz}\right)=\rho g_x-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u_x}{\partial x^2}+\frac{\partial^2 u_x}{\partial y^2}+\frac{\partial^2 u_x}{\partial z^2}\right)$$

$$\rho\left(\frac{\partial u_y}{\partial t}+u_x\frac{\partial u_y}{dx}+u_y\frac{\partial u_y}{dy}+u_z\frac{\partial u_y}{dz}\right)=\rho g_y-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 u_y}{\partial x^2}+\frac{\partial^2 u_y}{\partial y^2}+\frac{\partial^2 u_y}{\partial z^2}\right)$$

$$\rho\left(\frac{\partial u_z}{\partial t}+u_x\frac{\partial u_z}{dx}+u_y\frac{\partial u_z}{dy}+u_z\frac{\partial u_z}{dz}\right)=\rho g_z-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 u_z}{\partial x^2}+\frac{\partial^2 u_z}{\partial y^2}+\frac{\partial^2 u_z}{\partial z^2}\right)$$

- k-ε turbulence modeling
  - RANS does not have closure due to non-linear stress tensors in turbulent flows
  - There are not enough equations for the unknowns
  - k-ε turbulence modeling
    - Solves turbulence by calculating k, turbulent energy, and ε energy dissipation rate
    - Commonly used method to solve closure problem

k-ε turbulence modeling

$$\rho(u_i\cdot\nabla)k=\nabla\cdot\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\nabla k\right]+P_k-\rho\varepsilon$$

$$\rho(u_i\cdot\nabla)\varepsilon=\nabla\cdot\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\nabla\varepsilon\right]+C_{\varepsilon 1}\frac{\varepsilon}{k}P_k-C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k}$$

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon}$$

$$P_k=\mu_t\left[\nabla u_i:\left(\nabla u_i+(\nabla u_i)^T\right)-\frac{2}{3}(\nabla u_i)^2\right]-\frac{2}{3}\rho k\nabla\cdot u_i$$

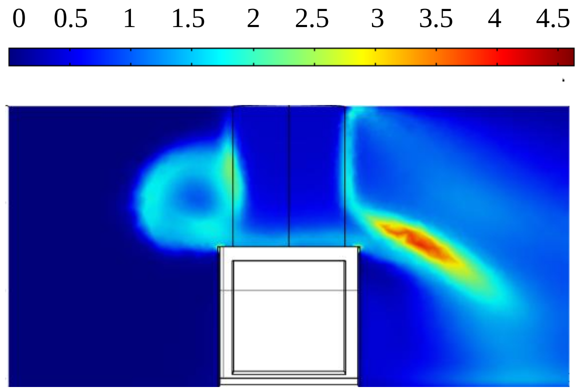
$$\varepsilon=\rho\frac{C_\mu k^2}{\kappa_V\delta_w\mu}$$

$$C_{\varepsilon 1}=1.44,\ C_{\varepsilon 2}=1.92,\ C_\mu=0.09,\ \sigma_k=1.0,\ \sigma_\varepsilon=1.3$$

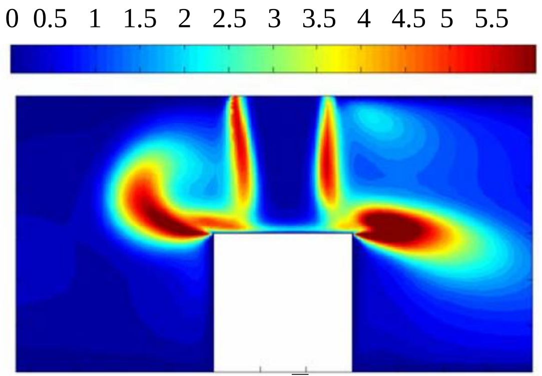
HOME

Validation Model – Turbulent KE Contours

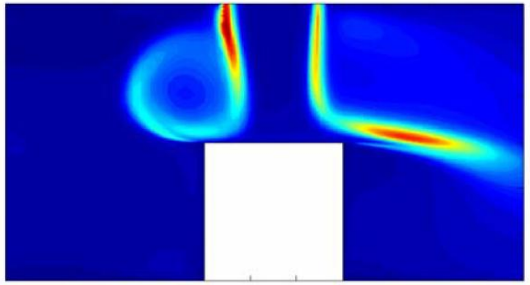
- Turbulent kinetic energy magnitude contours in (m^2/s^2)
- The previous study $\overline{v^2} - f$ calculates excessive KE in comparison to the PIV data
- The current study k-ε validation model calculates 4.5 (m^2/s^2) maximum turbulent kinetic energy
  - Calculates lower than PIV measured data
  - Shape however better matches in comparison to $\overline{v^2} - f$
- The k-ε validation model is the superior method in modeling the flow of this experiment



Current Study: k-ε model



Previous Study: $\overline{v^2} - f$ model



Experiment: PIV data

Hydraulic Analogy Variable Determination

HOME

Variable	Description	AIR - VALIDATION CASE				WATER		Reason for geometry in water
		Value	Units	Value	Units	Value	Units	
h_t	Cube Side Length	15	mm	0.591	in	2	in	Cube is made larger in water for flow visualization
D	Diameter of jet	12	mm	0.472	in	1.6	in	Same h_t:D ratio as air experiment
S_z	Cross flow width	60	mm	2.362	in	5.75	in	Max water depth is 6 inches
r	Density	1.204	kg/m^3	0.075	lbm/ft^3	62.2	lbm/ft^3	Density of water
m	Dynamic viscosity	1.789E-05	kg/m-s	1.202E-05	lbm/ft-s	6.580E-04	lbm/ft-s	Value of water at room temperature
Re_j	Reynolds Number Jet	8,076	--	--	--	8,076	--	Reynolds Number kept the same
Re_c	Reynolds Number Crossflow	4,657	--	--	--	4,657	--	Reynolds Number kept the same
U_j	Jet Velocity	10	m/s	32.808	ft/s	0.641	ft/s	$U_j=(Re_j m)/(r\ D)$
A_j	Area of jet	1.131E-04	m^2	1.217E-03	ft^2	1.396E-02	ft^2	$A_j=(pD)/4$
m_j	Jet mas flow rate	1.362E-03	kg/s	3.002E-03	lbm/s	0.556	lbm/s	$m_j=r\ U_j A_j$
U_j/U_c	Velocity ratio	5.78	--	--	--	5.78	--	Velocity ratio kept the same
U_c	Cross flow velocity	1.73	m/s	5.676	ft/s	0.111	ft/s	$U_c=U_j/(U_j/U_c)$
$D_{h,c}$	Hydraulic Diameter Crossflow	40	mm	1.575	in	5.333	in	$D_{h,c}=(Re_c m)/(r\ U_c)$
H	Crossflow height	30	mm	1.181	in	4.973	in	$H=(D_{h,c}S_z)/(2S_z-D_{h,c})$
A_c	Area of Cross Flow	0.0018	m^2	0.019	ft^2	0.199	ft^2	$A_c=S_z H$
m_c	Cross flow mass flow rate	3.749E-03	kg/s	0.008	lbm/s	1.369	lbm/s	$m_c=r\ U_c A_c$
J_h	Jet Length	15	mm	0.591	in	2.973	in	$J_h=H-h_t$
J_h/H	Jet length per total height	0.5	--	--	--	0.598	--	J_h/H
m_i	Inlet mass flow rate	NA	NA	NA	NA	1.926	lbm/s	$m_i=m_c+m_j$
A_i	Inlet Area	NA	NA	NA	NA	0.419	ft^2	Per water table
U_i	Inlet Velocity	NA	NA	NA	NA	0.074	ft/s	$U_i=m_i/(r\ A_i)$

Increased the size of the domain

Used properties of water

Retained the Reynolds number of the previous experiment

Determined jet flow

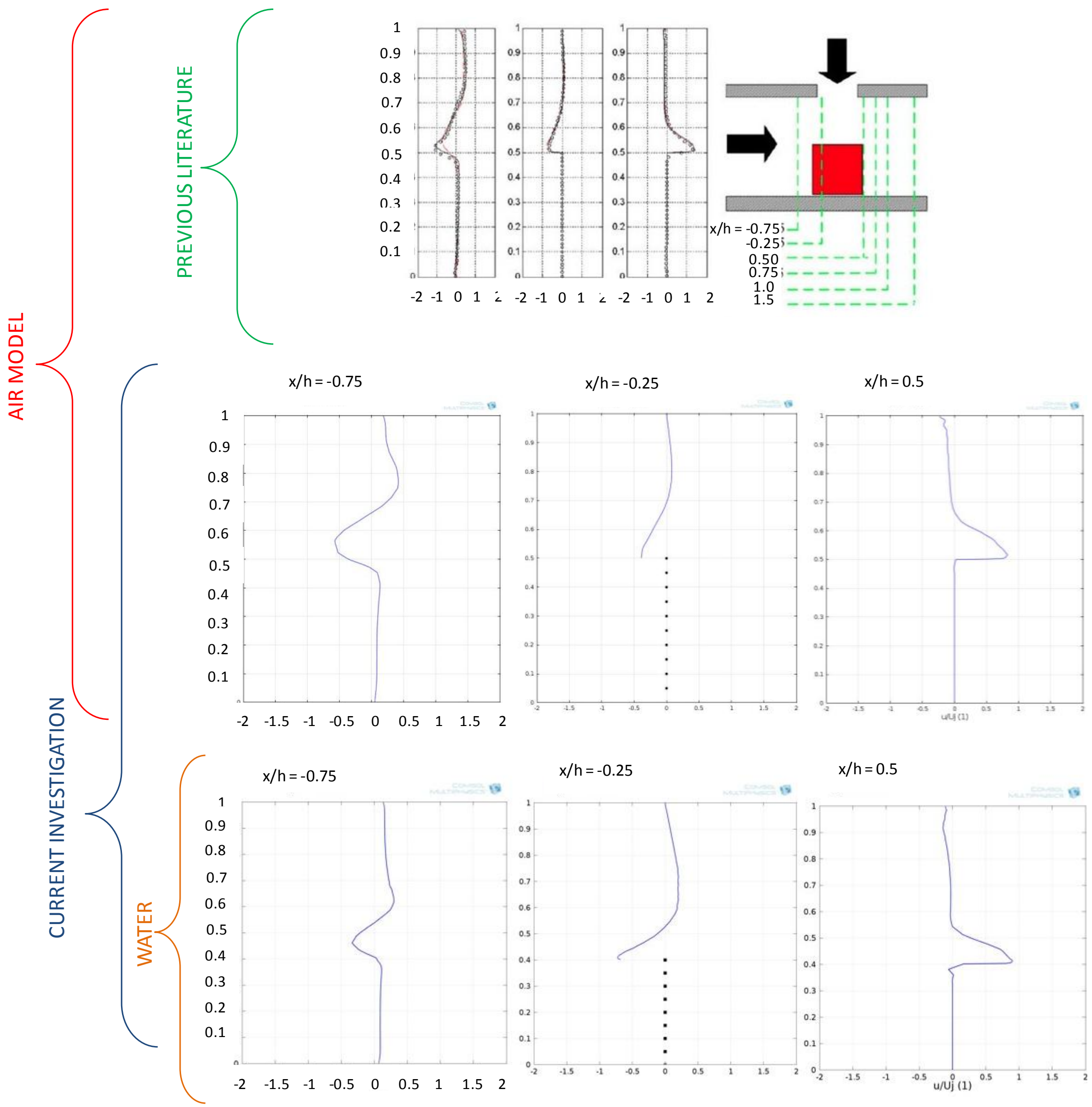
Kept velocity ratio constant and calculated cross-flow variables

Established length of jet

Determined required inlet parameters

Steady State Non-Dimensionalized Comparisons (1/2)

- Non-dimensionalized comparison
    - Axial velocity divided by jet velocity - u/U_j
    - Vertical height divided by total height - y/H
    - At various cut lines - x/h_t
- Cut lines x/h_t = -0.75, -0.25, & 0.5
    - Trend is the same between all models
    - k-ε models show lower velocity magnitudes than previous literature
    - Impingement happens at lower y/H in water model

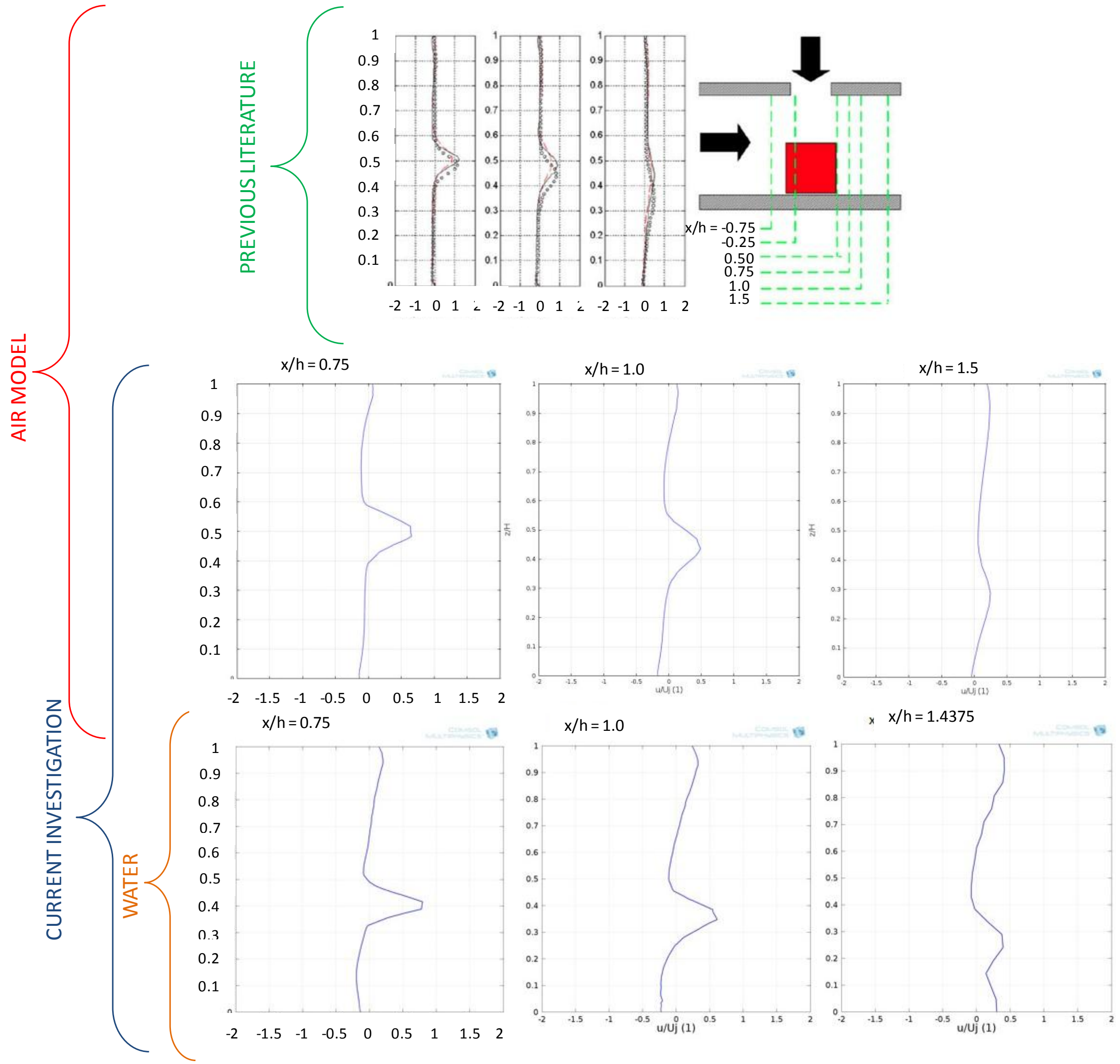


Steady State Non-Dimensionalized Comparisons (2/2)

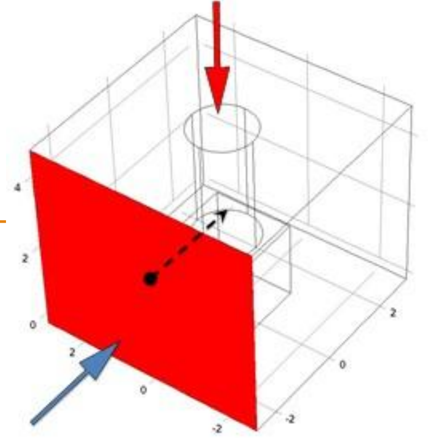
HOME

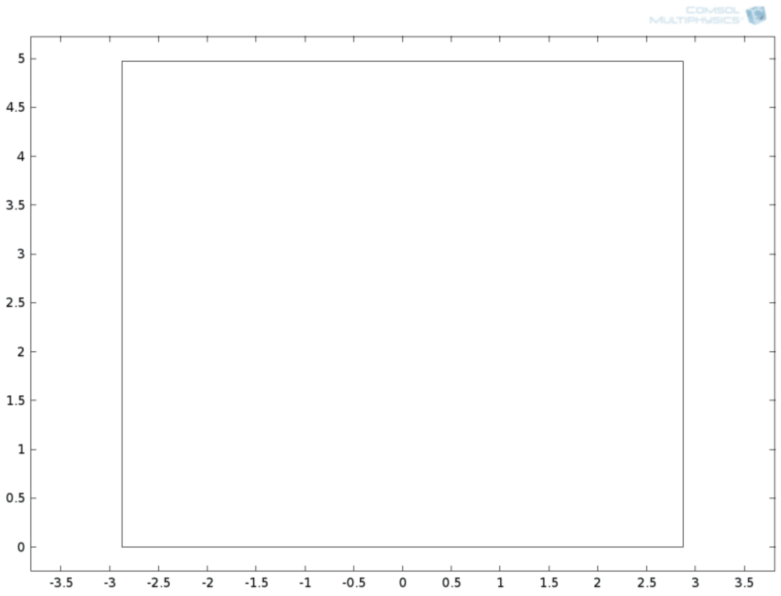
- Non-dimensionalized comparison
  - Axial velocity divided by jet velocity - u/U_j
  - Vertical height divided by total height - y/H
  - At various cut lines - x/h_t

- Cut lines $x/h_t = 0.75, 1.0, \& \sim 1.5$
  - Trend is similar between all models
  - k-ε models show lower velocity magnitudes than previous literature
  - k-ε models show more negative x-velocity components than previous literature
  - Water model final cut line is at 1.4375 due to smaller domain

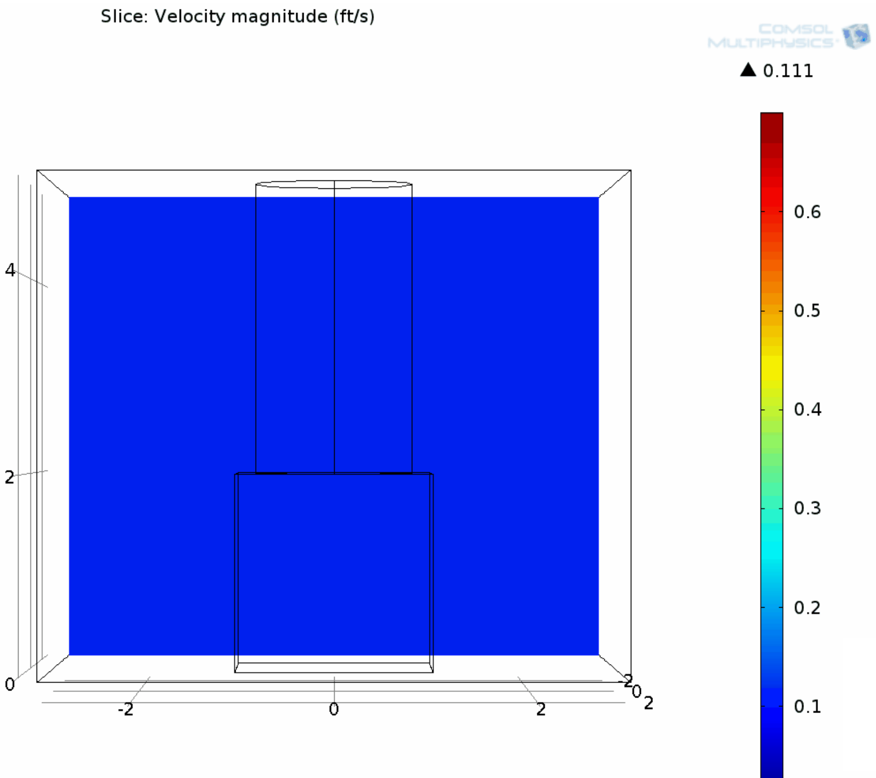


Steady State YZ Cut Planes

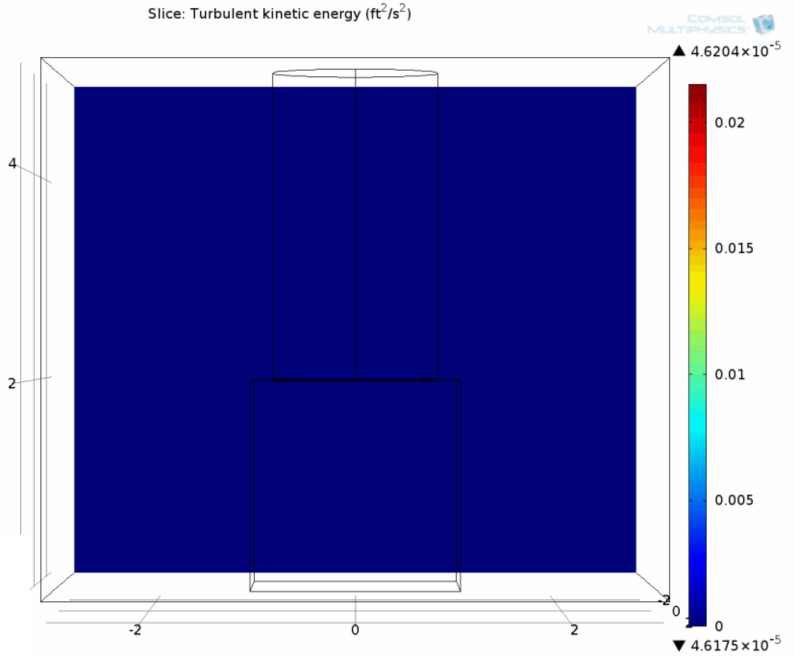




Velocity Contours



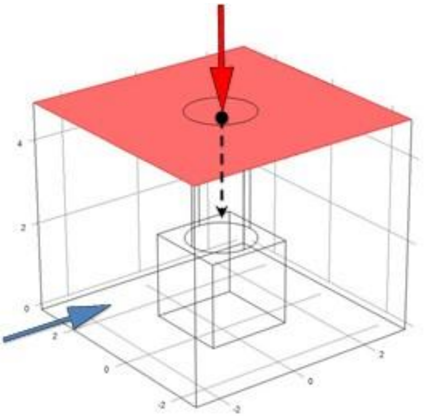
Velocity Magnitude Contours

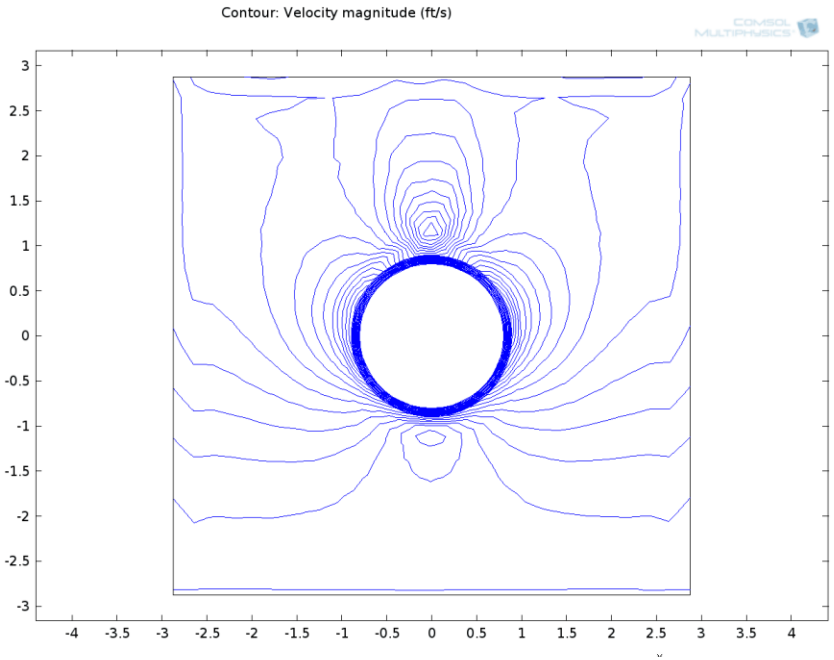


Turbulent Kinetic Energy Magnitude Contours

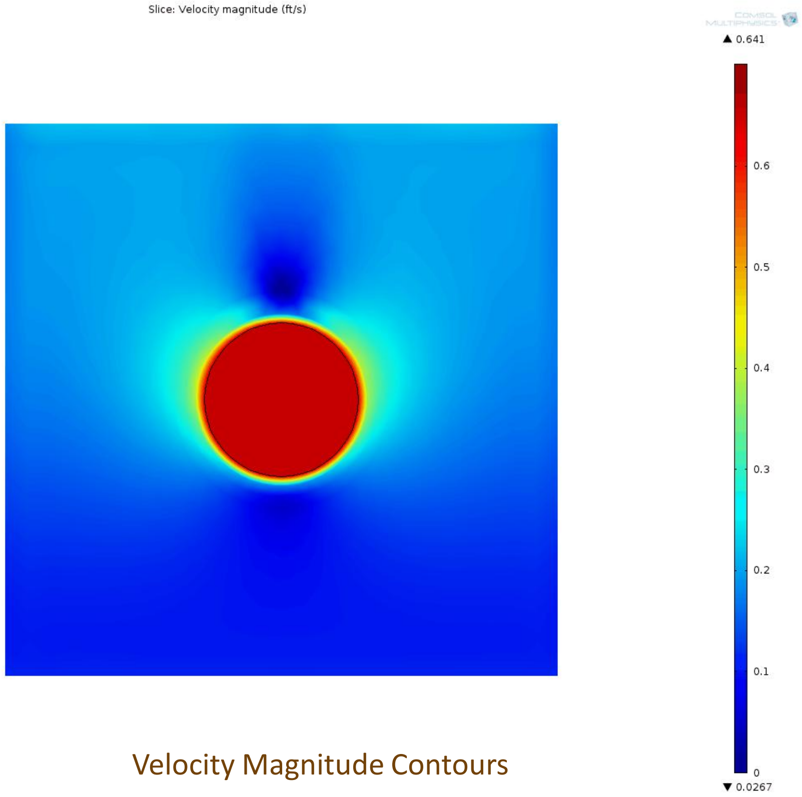
HOME

Steady State XZ Cut Planes

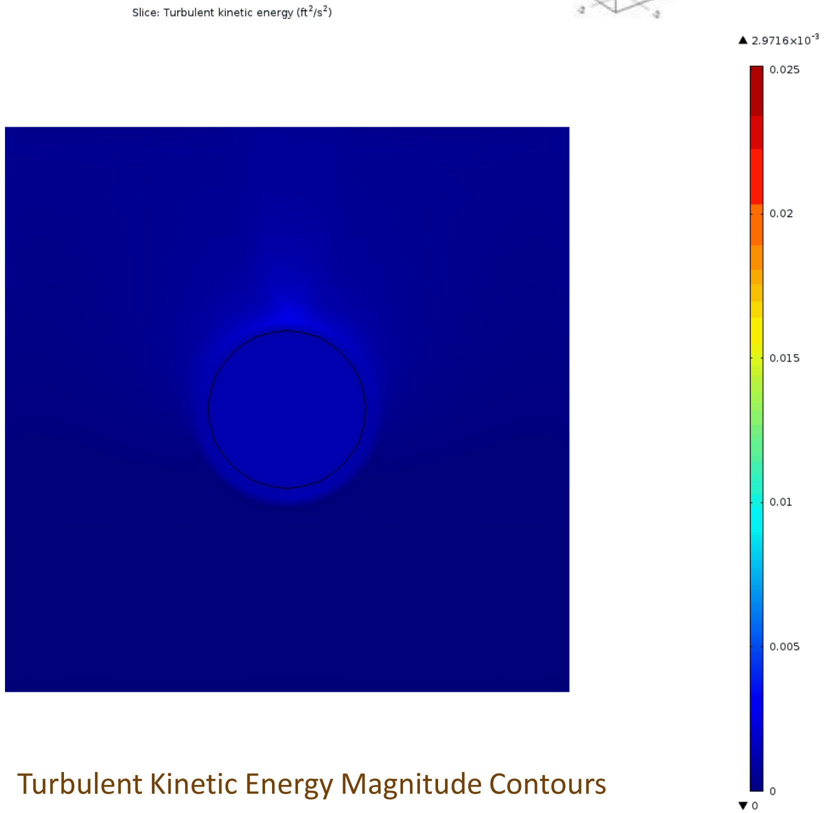




Velocity Contours



Velocity Magnitude Contours

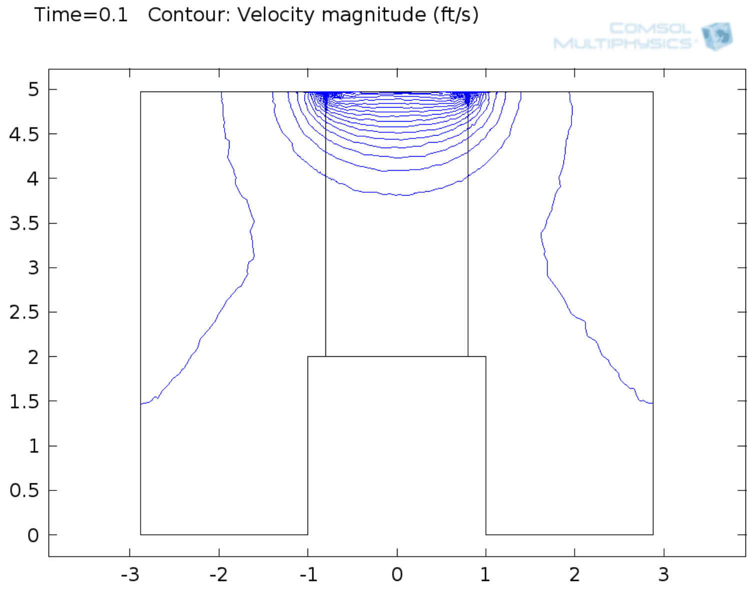


Turbulent Kinetic Energy Magnitude Contours

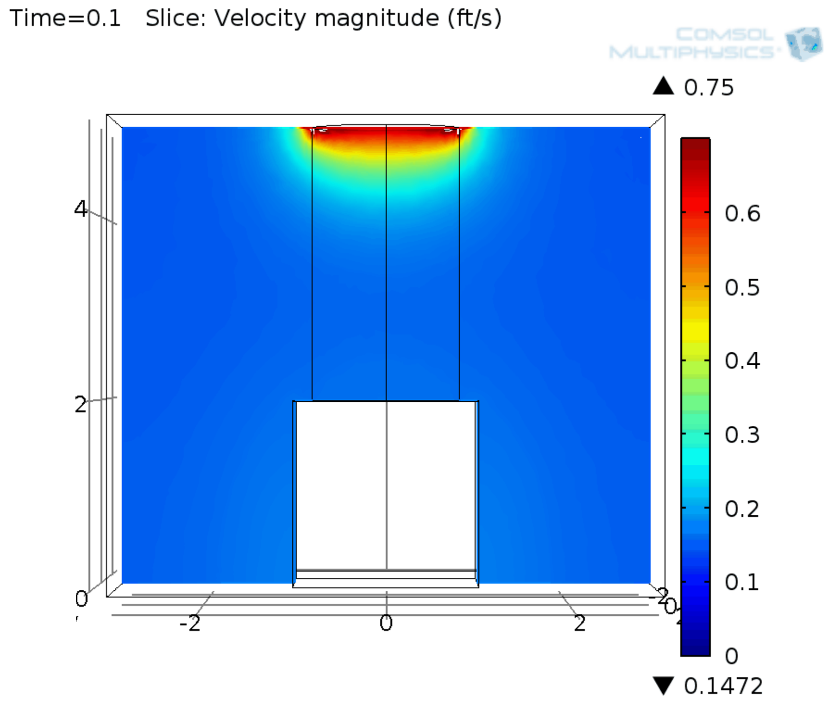
HOME

HOME

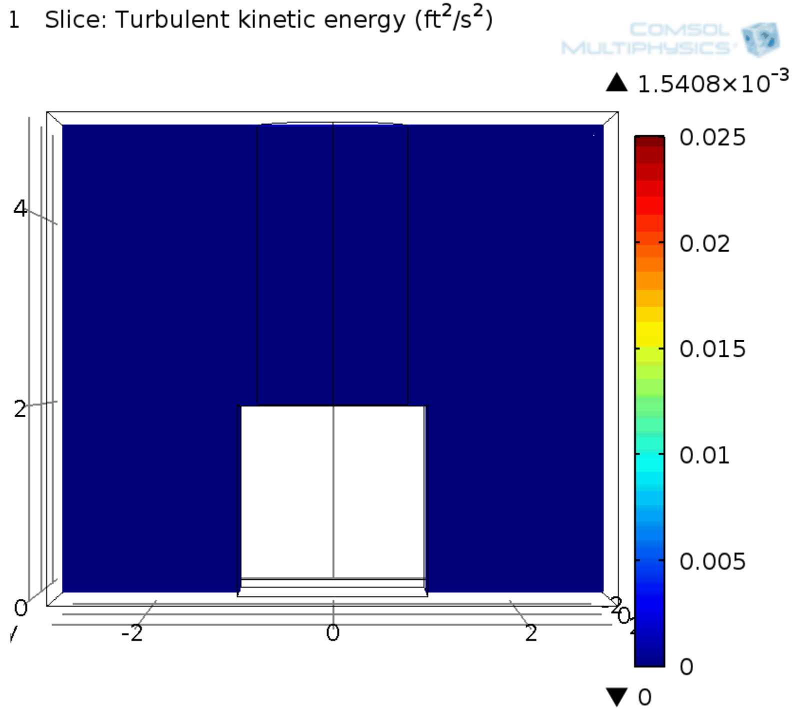
Time Dependent – YZ Plane



Velocity Contours



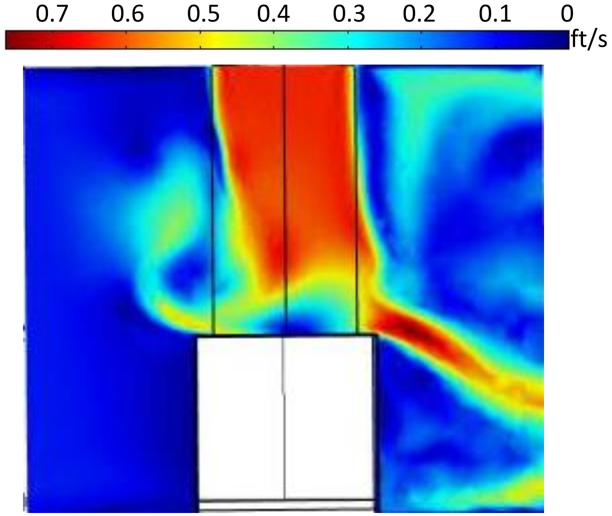
Velocity Magnitude Contours



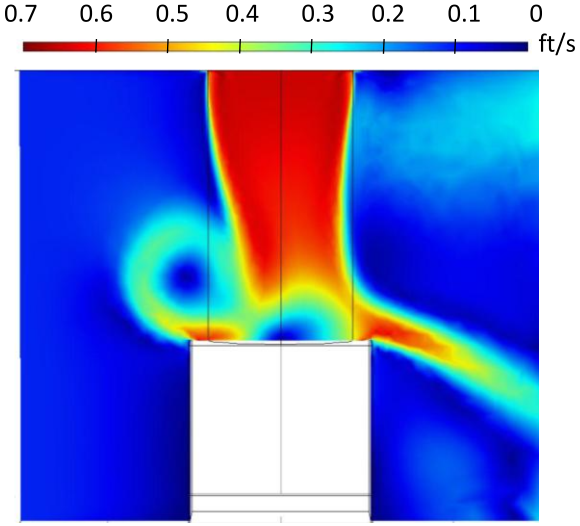
Turbulent Kinetic Energy Magnitude Contours

Laminar CFD Model Results

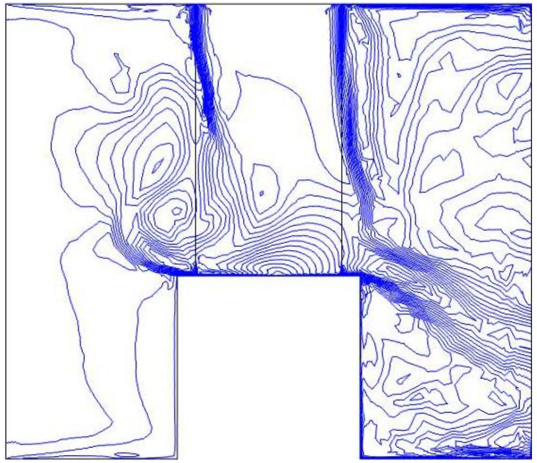
Velocity Magnitude Contours – Laminar Model



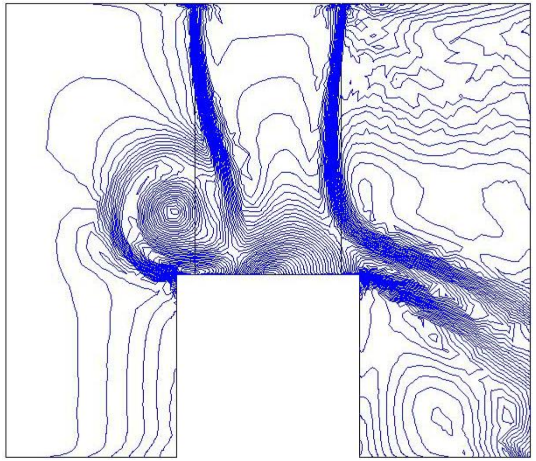
Velocity Magnitude Contours – Turbulent Model



Velocity Contours – Laminar Model



Velocity Contours – Turbulent Model

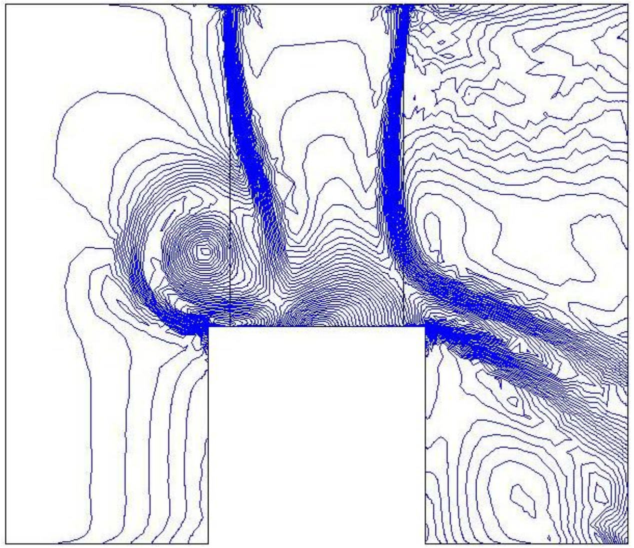


HOME

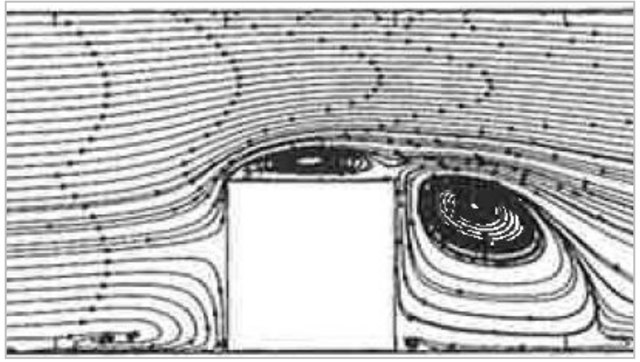
Comparison to Flow without Jet

- Flow with jet versus without
- Similarities
  - Low velocity point at top of cube:
    - Cross-flow induced
  - Up-wash in wake:
    - Cross-flow induced
- Differences
  - Horseshoe vortex:
    - Impinging jet in cross-flow only
  - High speed trailing edge:
    - Impinging jet in cross-flow only

Velocity Contours – Impinging Jet in Cross-flow



Velocity Contours – Mean Flow Around Cube



Source: Rodi, W., J. H. Ferziger, M. Breuer, and M. Pourquiée. 1997. "Status of Large Eddy Simulation: Results of a Workshop." Journal of Fluids Engineering 119.2: 256