6-27-2013

Particle Image Velocimetry with Bluff Body and Jet in Cross-Flow Flame Holder Applications

Christopher Perron
chrisp.perron@gmail.com

Recommended Citation
https://opencommons.uconn.edu/gs_theses/456

This work is brought to you for free and open access by the University of Connecticut Graduate School at OpenCommons@UConn. It has been accepted for inclusion in Master's Theses by an authorized administrator of OpenCommons@UConn. For more information, please contact opencommons@uconn.edu.
Particle Image Velocimetry with Bluff Body and Jet in Cross-flow Flame Holder Applications

Christopher P. Perron
B.S. Rensselaer Polytechnic Institute

A Thesis
Submitted in Partial Fulfillment of the Requirements for a Degree of Master of Science at the University of Connecticut 2013
Master of Science Thesis

Particle Image Velocimetry with Bluff Body and Jet in Cross-flow Flame Holder Applications

Presented by:

Christopher P. Perron

Major Advisor: __________________________________________________

Dr. Michael W. Renfro

Associate Advisor: ________________________________________________

Dr. Baki M. Cetegen

Associate Advisor: ________________________________________________

Dr. Zhuyin Ren

University of Connecticut
2013
**Particle Image Velocimetry with Bluff Body and Jet in Cross-flow Flame Holder Applications**

In this study Particle Image Velocimetry (PIV) was used to acquire flow field and velocity information for a bluff body flame holder and a jet in cross-flow. The bluff body trials tested consisted of combustion of a premixed air and fuel flow with the purpose of improving boundary conditions. The jet in cross-flow was tested with a variety of jet to cross-flow momentum ratios. A tutorial is given on using MATLAB programming to use the PIV data to calculate fluid properties such as velocity, vorticity, and strain rates. Velocity vector fields are used to determine the velocity at different areas of the test section and to verify boundary conditions. The uncertainty of the PIV measurement is found using a correlation map, and the resulting error to the fluid calculations is found using a central differencing scheme. Conditioned Particle Image Velocimetry (CPIV) is introduced and used to find the flame edge. This thesis is meant to not only display some of the new results acquired, but also to serve as a reference for anyone doing research related to combustion or PIV.
Acknowledgements

I would like to thank my advisor, Dr. Michael Renfro for his guidance throughout the process. His patience and support has given me the confidence to become a better engineer.

Thanks to Dr. Baki Cetegen for all of his help and guidance throughout the research process.

Thanks to Dr. Zhuyin Ren for all of his help and for serving on my committee.

I would like to thank my coworkers on the rig, Jason Wagner and George Lapaan, for their hard work and for all of the help given to me in troubleshooting the rig, in homework, and in MATLAB coding.

I would like to thank my lab mates, Kathryn Gosselin, Marc Schneider, Martin Hawron, and Jarod Lake past and present for their technical support and comic relief.

I would like to thank my brother for his friendship.

I want to thank my father and mother for their love, and for supporting my decision to go to graduate school. You both are a huge inspiration to me.

I would like to thank Julie for her love and her support. I could never have accomplished this without you.
1.1 Flame Holders

In practical turbulent combustors, the reactants entering the combustion chamber are typically moving at higher speeds than the laminar flame speed, so flame holders are often necessary to stabilize combustion. Flame holders are used to create a flow separation which creates a slower speed recirculation zone that continuously ignites incoming mixtures in contact with the recirculation zone. There are different ways to achieve recirculation. Swirl combustors impart a tangential velocity. With sufficient swirl, a recirculation zone is created due to the pressure field. Step combustors typically use an expansion in area to create a recirculation zone that stabilizes a flame. Bluff body flame holders obstruct the flow to produce a recirculation zone behind the bluff body that ignites the incoming reactants. Flame holders can be a solid geometric object such as the case of the triangular bluff body and they can be also be fluidic such as the case of a jet in cross-flow. The geometric bluff body is typically used in secondary combustor applications. In these secondary combustors, hot products of lean combustion exit the turbine at high speed and relatively low pressure. More fuel is then added to the mixture which reacts with the excess oxygen to produce a flame. A geometric bluff body is used to slow the flow and create a recirculation zone for the premixture to combust and stabilize a flame. The following figure taken from Tuttle’s dissertation is represented here because it accurately shows the fluid flow and flame direction in the recirculation zone of a bluff body stabilized flame. (Tuttle, 2010)
Figure 1.1 Recirculation zone and streamlines for a bluff body-stabilized flame.

Because the secondary combustor in the engine is near atmospheric pressure, much more fuel is needed in order to produce equivalent amounts of thrust as the primary combustor. The inefficiency of fuel usage is tolerable since these secondary combustors are used for short periods of time. Other engines use jet in cross-flow combustion as a flame in products of a rich preburn. These jets typically consisting of mostly air are used to react with any unburned hydrocarbon fuel. The figure below by Samuelsen is a good representation of a typical engine utilizing the jet in cross-flow. (Samuelsen, n.d.)

Figure 1.2 Engine utilizing a jet flame in a reacting cross-flow.

In the diagram the fuel and air enter in an expanded area to allow for rich combustion. The hydrocarbon radicals produced by rich combustion help stabilize the flame. Richly burning flames also have relatively low temperatures which tend to produce lower nitrogen oxide emissions. As can be seen in the diagram an air jet is injected into the products of the rich preburn mixing with the unburned hydrocarbons from the rich preburn. The addition of oxygen is necessary to oxidize the carbon monoxide and to produce a secondary flame. The goal of this engine is to ideally have only major products of combustion and very trace amounts of pollutant emissions. (Samuelsen, n.d.) Different applications of the jet in cross-flow may require different amounts of mixing, so different jet arrangements and sizes have been researched.
1.2 Current Study

The current study is meant to explore the fluid properties of combustion applications with the use of Particle Image Velocimetry (PIV) as well as other fluid measurement techniques.

![Figure 1.3 Single image of a bluff body-stabilized flame acquired using high speed camera.](image)

Previous work in this lab with the bluff body includes concentrated fueling before the bluff body with and without preburned conditions. In this study the bluff body work is done with fueling much further upstream to provide for a fully premixed air and fuel for combustion on the bluff body.

Variations of the jet in cross-flow experiment have been studied in the past but the experiment has never been performed in this laboratory before. The initial goals of the experiment were to study the flow behavior of a single jet in a preburned cross-flow. The parameters tested were various jet to cross-flow momentum ratios for the jet and for the cross-flow.

PIV was performed on both experiments to extract the velocity measurements and fluid flow properties such as vorticity. This paper explores the use of this measurement technique to further understand the fluid physics occurring during the combustion reaction. The fundamentals of how MATLAB can be used to output relevant charts and calculations using PIV data is explained in a tutorial fashion. The
uncertainty associated with PIV measurements is also explored as well as the uncertainty propagation associated with the calculated parameters. Finally a technique referred to as Conditioned Particle Image Velocimetry (CPIV) uses the difference in seed density from burned and unburned regions to find the flame edge.
2. Literature Review

Extensive studies regarding jets in cross-flow as well as bluff body flames have been reported. In this Chapter, these works are reviewed to provide context for the experiment described in Chapter 3. The following sections describe different studies reviewed.

2.1 Fundamental Parameter Work

The jet in cross-flow physics problem has been studied with great interest for years. Different combustors were designed utilizing the jet in cross-flow concept to reduce nitric oxide emissions. The jet in cross-flow typically features non-premixed fuel and air burned rich which leaves unburned hydrocarbons as well as CO and NO\textsubscript{x}. The combustor features an air injection for a secondary burn to reduce these unwanted pollutants. In order to understand the physics of the problem there have been numerous studies performed on non-reacting jets in cross-flow. A.H. Lefebvre that relates the maximum jet penetration. (Lefebvre, 1999)

\[ Y_{\text{max}} = 1.15(D_j)(f^{0.5}) \sin(\theta) \quad (2.1) \]
Figure 2.1 Jet flame in vitiated cross-flow. Taken from Samuelsen (Samuelsen, n.d.)

In Eqn. 2.1 $Y_{\text{max}}$ is the maximum radial penetration of the jet, $D_j$ is the diameter of the jet, $J$ is the jet to cross-flow momentum ratio, and $\theta$ is the entry angle of the jet. The first figure in the image below represents an example of a single jet at a 90 degree angle to the bottom wall. The second figure represents $\theta = 75^\circ$.

Figure 2.2 Example of $\theta$ in Equation 2.1

The jet in cross flow momentum ratio, $J$, is defined in Eqn. 2.2 where $\rho J$ and $U_j$ are the density and velocity of the jet flow, and $\rho_M$ and $U_M$ are the density and velocity of the cross-flow.

$$J = \frac{\rho J \cdot U_j^2}{\rho_M \cdot U_M^2} \quad (2.2)$$

Holdeman’s work is useful in finding the ideal number of jets to use for optimum mixing. The following equation is valid for circular jets in a circular cross-flow.

$$\text{number of jets} = \frac{\pi \sqrt{2J}}{C} \quad (2.3)$$

Holdeman also developed a relationship for jet spacing in a rectangular cross-flow.
\[
\frac{S}{H} = \frac{C}{\sqrt{J}} \quad (2.4)
\]

In Eqn. 2.4, \(S\) is the orifice spacing, \(H\) is the channel height, \(J\) is the jet to cross-flow momentum ratio, and \(C\) is an empirical constant which is experimentally determined for various cases. He found that for a single jet penetration, \(C=2.5\) is optimal, and values less than and over the optimum result in under-penetration and over penetration respectively. He found that 1.25 and 5.0 are optimum values of \(C\) for in-line rows and staggered rows of opposed jets. (Holdeman, 1993) In the rectangular case the jets can be either inline or staggered. Samuelsen reports on more experiments where the jet air and the cross-flow are preheated separately. It was found that NO\(_x\) levels are higher when only the jet is preheated compared to when both the jet and cross-flow are preheated. Samuelsen concluded that while the jet in cross-flow set up does reduce the NO\(_x\) emissions, more research on the chemistry of combustion may be necessary for further understanding. (Samuelsen, n.d.)

Samuelsen has performed many studies with Holdeman. They performed an experimental study of reacting jets in a preburned cross-flow. In their studies together they had a fuel rich preburned cross-flow with oxygen jets injected around the perimeter. They determined that for their specific momentum flux ratio that when penetration is beyond optimal the jet air stays in the middle of the test section and the cross-flow products tend to stay towards the wall. They also found that in cases of under-penetration the jet air tends to stay by the wall and the cross-flow products tend to pass over it. Both of these cases are undesirable because they do not provide optimal mixing required for the desired jet combustion. (Leong & Samuelsen, 1999)

**2.1 Unsteady Flame-Wall Interactions in a Reacting Jet Injected into a Vitiated Cross-Flow**

Researchers at Georgia Institute of Technology performed an experiment to determine the interaction between the jet and the wall edge of the cross-flow. (Sullivan, et al., 2013) For low momentum flux
ratios the jet does not penetrate out of the cross-flow boundary layer. Therefore in these conditions the flame is often stabilized in the boundary layer and stays attached to the wall. The GIT researchers found that for jet flame J ratios of less than 20, the flame is unsteady and interacts with the wall. In these cases the wall is an important factor in the flame stabilization. They also reference an article that describes the flame structure at high jet momentum to cross-flow momentum ratios. The jet flame tends to propagate to the wall especially for very high temperature cross-flows.

2.2 Turbulent jet Flames in a Crossflow: Effects of Some jet, Crossflow, and Pilot-Flame Parameters on Emissions

Stephen Turns and Romarao Bandaru performed a study at Pennsylvania State University comparing jet behavior in a cross-flow compared to regular straight jets in no cross-flow. In their particular experimental setup, horizontal jet flames were introduced into a vertical cross-flow of air. They compared flame lengths and emissions data for different fuels with varying cross-flow velocities and varying jet velocities. The fuels used were methane, propane, ethylene, and a 95%CO/5%H₂, the cross-flow velocities were either 2.3 m/s or 4.3 m/s for each case, and the jet velocities varied for each fuel. They found that cross-flow jet flames are not as long as the straight jet flames of the same velocity. In general, for slower cross-flow velocities the jet flame is closer to the length of the straight jet flame, and for faster cross-flow velocities the jet flame is significantly shorter than the straight jet flame. This is mainly from the increased mixing rates from the higher cross-flow velocities. They also found that there were more CO, NO₂, and unburned hydrocarbons for the jet in cross-flow case than for the straight jet case. This is because there is some fuel which escapes near the nozzle and is unburned. It is also due to the vortices generated when cold air is introduced into the cross-flow. (Bandaru & Turns, 2000)

2.4 The Effect of Jet Mixing on the Combustion Efficiency of a Hot Fuel-Rich Cross-Flow
The jet in cross-flow problem has been investigated for the past years in an effort to fully understand the physics of this phenomenon. With a better understanding of the jet in cross-flow various goals such as higher efficiency combustion and pollutant reduction can be achieved. To fully understand the experiment performed it is imperative to investigate the experimental setups and results of previous experiments. Many researchers have different experiments which alter different variables including the contents of the cross-flow, the number of jets, and jet size. Boutazakhti, Thompson, and Lightstone had an experimental set up which contains a preburner in a cylindrical tube. Along the perimeter of the tube were lots of holes to act as the jet. The cross-flow contained combustion products of a fuel rich reaction between methane and air. Because the cross-flow had a rich equivalence ratio, $\Phi_M$ of 1.5, the combustion products contained about 7.2% of CO$_2$, 5.8% CO, and 7.2% H$_2$, as well as water vapor and some unreacted methane. The overall cross-flow temperature averaged at 1100°C which is higher than the autoignition temperature of H$_2$ (500°C), CO (609°C) and CH$_4$ (580°C). Further downstream air is injected into the jets which is then used to react with the unburned fuel in the cross-flow. The experimenters calculated optimum momentum flux ratio using an equation presented by Holdeman. Boutazakhti et al performed various tests with varied number of jets, various jet diameters, and varying cross-flow mass flow rates and jet mass flow rates. For certain cases the mass flow rates of the air jets were set to match the mass flow rates of the exhaust cross-flow. The following table has been taken directly from Boutazakhti et al because it is useful to portray the various experiments performed in this study. So many configurations are tested because it was important to see effects of varying momentum flux ratios at constant equivalence ratios.
Table 2.1 Jet in cross-flow conditions tested by Boutazakht et al. (Boutazakhti & Thomson, 2000)

Molar fractions of CO, CO$_2$, and H$_2$ were determined using gas analyzers. With these molar concentrations it was possible for the researchers to determine the unmixedness for each momentum flux ratio. The following equation is used to determine the unmixedness, where the $\Sigma$ is the unmixedness, $\sigma$ is the standard deviation of the carbon molar fraction, and $\mu$ is the mean value of the carbon fraction.

$$\Sigma = \frac{\sigma}{\mu}$$  \hspace{1cm} (2.5)

By taking measurements of the CO levels, a chart was created showing present [CO]/ [CO]$_D$ versus the overall equivalence ratio. In a way, the concentration ratio of CO is a measure of the combustion.
efficiency. It was found that combustion concentration of CO decreases for until an equivalence ratio between about 0.8, until it continues to increase. The configuration with 4 jets had the lowest concentration of CO at an equivalence ratio slightly less than 0.8. For 9 jets the low peak was about 0.8 and for 18 jets the low peak was about 0.85. This suggests that for richer cross-flows, having a greater number of jets increases the combustion efficiency, thereby giving a lower CO concentration. The interesting part of this finding is that for non-premixed systems the combustion efficiency is not related to equivalence ratio. The authors of the paper conclude that there must be some premixing of fuel and air before the chemical reaction takes place.

Another test was done with all of the 9 jet modules. Each module has the same $J_{\text{OPT}}$ the authors presented earlier from Holdeman et al, but have jet diameters ranging from 2.05mm to 3.97mm. The CO concentration was measured for each of these cases for varying momentum flux ratios.

![Combustion efficiency vs. momentum ratio for various jet diameters.](image)

**Figure 2.3 Combustion efficiency vs. momentum ratio for various jet diameters.**

In general, as the momentum flux ratio increases, the combustion efficiency increases until a maximum peak, which then the efficiency decreases for increasing ratios. The peaks range from $J=20$ to $J = 180,$
and in general the larger jets have the maximum combustion efficiency at lower momentum flux ratios. However of all of the cases only one of them has the greatest combustion efficiency at optimum momentum flux ratio predicted from the equation used previously.

**Figure 2.4 Combustion efficiency vs. equivalence ratio for various jet diameters.**

However when the combustion efficiency for the same jet diameters are tested against overall equivalence ratio instead of momentum flux ratio, they all seem to have a peak between ratios of 0.75 to 0.85. This suggests that the equivalence ratio is more of a dominating factor for combustion efficiency than the momentum flux ratio. To prove the point further, the researchers did another test where the equivalence ratio is fixed at 0.80, the ratio where most of the jets had the highest efficiency. The J ratio for each of the jet diameters is plotted. It is shown that the higher J ratios have higher combustion efficiency and a decrease in emissions. It is also shown that the combustion efficiency increases with increasing numbers of jets. (Boutazakhti & Thomson, 2000)

**2.5 Stratified jet Flames in a Heated (1390K) Air Cross-Flow with Autoignition**
James Driscoll and Daniel Micka performed a similar jet in cross flow study, but with different objectives. They performed a study with a preheated lean mixture cross-flow and autoignition of jet flames. The researchers wanted to discover the answer to three questions in particular. They wanted to know if the jet flame is affected if the cross-flow temperature is hotter than the autoignition temperature of the jet flame’s fuel. They were specifically concerned with the effect of the temperature on the autoignition delay. The second issue they were trying to resolve is the how the flame structure is affected with faster cross-flow velocities. Lastly they measured flame lengths and determined that flame length increases with increasing fuel mass flow rate. From this finding they determined air velocity, not fuel velocity, is the dominate factor in mixing. (Micka & Driscoll, 2012)

2.6 Simultaneous Measurements of Velocity and CH Distribution. Part II: Deflected Jet Flames

Many times the jet is perpendicular to the cross-flow, but this is not always the case. Han and Mungal performed a study at Stanford University which observed jet in cross-flow behavior with angled jets. Their experimental set up consisted of a vertical test section with a cross sectional area of 50cmx50cm. They used two concentric tubes for the jet, one tube to provide the fuel and one tube to provide pilot fuel to stabilize the flame if needed. The governing equation for a transverse jet penetrating into a cross-flow is as follows.

\[
\frac{x}{rd} = A \left(\frac{y}{rd}\right)^B \quad (2.6)
\]

In this equation \( r \) is the square root of the momentum ratio, which is the jet density times jet velocity squared over cross-flow density times cross-flow velocity squared. The constants \( A \) and \( B \) are experimental fits, and are generally accepted to be about between 1.5 and 2.0 and 0.25 to 0.38 respectively. In addition to a perpendicular jet case, they performed an experiment with the jet tilted +45° and -45° from the perpendicular case. In all cases the jet had a mole fraction 0.25 ethylene and
0.75 nitrogen, the square root of the momentum ratio was 10, and the cross-flow velocity was between 1.7 and 1.8 m/s. Interestingly enough the jet at -45° (aiming towards the cross-flow) does not need a pilot to keep a stable flame. The researchers used simultaneous CH Planar Laser Induced Fluorescence (PLIF) and Particle Image Velocity (PIV). These studies were used to compare images and calculate strain rates. The strain rates are calculated here using the traditional fluid dynamics equations.

\[ s_{xy} = \frac{1}{2} * \left( \frac{du}{dy} + \frac{dv}{dx} \right) \tag{2.7} \]

There were some conclusions that could be made regarding the jet angles. It was found that the +45° jet had a much longer flame than the -45°. It can be concluded that negatively angled jets have better mixing than positively angled jets and neutral jets. Another conclusion to make is the principle strains rates in the CH layer decrease for increasing distances away from the jet. (Han & Mungal, 2003)

### 2.7 Mixing Characteristics and Emissions of Strongly-Forced Non-Premixed and Partially-Premixed Jet Flames in Crossflow

K.C. Marr, N.T. Clemens, and O.A. Ezekoye performed a study where pulsed jet flames in an unheated cross-flow to determine the effect pulsing has on emissions levels and mixing between fuel and air. Their experiment consisted of a square cross-sectional area of 0.16m² and a length of 1m. The air blower in the cross-flow was used to produce a cross-flow velocity ranging from 1.5m/s to 2.0m/s, 1.7m/s in most cases. The test section consisted of a single circular jet 6.35mm in diameter which was capable of injecting non-premixed fuel as well as a premixture of fuel and air. The jet was also equipped with a 270Hz speaker which was used to send pulses into the jet flow. The researchers first conducted a control experiment using an unforced non-premixed jet flame, which produced a bright sooty flame, as expected. The experiment was next performed for pulsed non-premixed flames. It was found that the introduction of the pulsing speaker created flames which were less bright and also had less orange
emission from soot radiation than the unforced non-premixed flame. The reduction in soot was quantified by measuring the luminosity and observing the color of the flame. The researchers discovered that with stronger pulses there is an increased reduction in soot. For larger pulses, air from the cross-flow can often be sucked into the jet, partially premixing the fuel and air. When the partial premixture is released it results in a non-sooty flame which accelerates turbulent mixing near the jet exit. They also found for higher amplitude ratios in the jet speaker, the jet flames tend to lift asymmetrically due to the momentum of the cross-flow. In addition the researchers performed emissions testing on the unforced non-premixed jet flames, the forced non-premixed jet flames, and the unforced partially-premixed jet flames. The emissions tests agreed with the above results. Emissions results were reported in the form of an Emissions Index term which essentially is the ratio of the mass of emissions produced compared to the mass of the fuel burned. The NOx emissions decreased as the amplitude of the pulses was increased. They also found that the pulse amplitudes need to be high enough so that the combustion residence time is small enough to take care of the thermal NOx increase. It was also determined that fuels that produce more soot need higher pulse amplitudes. The emissions results for the CO and unburned hydrocarbons (UHC) were very different. For a specific amplitude ratio of 5, they found that the CO emissions were 3 times greater than the unforced case and the UHC were 7 times greater. The researchers concluded that the CO emissions increased due to the vortex structures produced with the pulsing. With higher pulsing, the vortex structures have a more rapid mixing of cold air and fuel, which tends to rapidly cool the flow. The unburned hydrocarbons level increases with higher pulse amplitudes. This is because with higher pulse amplitudes the jet flame tends to lift and unburned fuel can escape through the bottom of the flame. The takeaway fundamentals from the study are that with a pulsed jet flame it is often possible to reduce NOx emissions and increase premixing, but there is a sacrifice with increased CO emissions as well as unburned hydrocarbon emissions. It is much more effective to premix
the fuel and air before combustion, however due to dangers of flashback and blowout it is not always practical. (Marr & Clemens, 2012)

2.8 Autoignition of Hydrogen/Nitrogen Jets in Vitiated Air Crossflows at Different Pressures

J.M. Fleck et al performed a study where they observed the autoignition properties of hydrogen/nitrogen jets. They determined that autoignition of the jet was prohibited by the lower temperatures tested. Ignition delay time is influenced by temperature. They also found that at higher pressures (about 15 bar) the jet would not autoignite. At lower cross-flow velocities the pressure seemed to have no impact on the autoignition. They found that turbulence and pressure have affects related to the ignition chart. Turbulent cross-flow helps autoignition in the second ignition limit however does not above the second limit. They also determined that for higher pressures ignition jump from the first limit to the second occurs at higher temperatures. (Fleck, et al., 2012)

2.9 Combustor Flowfield Measurements of a Transverse Jet Holder

Kareem Ahmed and David Forliti performed a study examining the reacting and non-reacting flow of the jet in cross-flow. They concluded that the reacting case has higher velocities and a shear region created by the heat release from the combustion products. They found that higher momentum ratios result in more heat released. (Ahmed & Forliti, 2009)

Recently there has been some interest in comparing the bluff body style flame holder to the jet in cross-flow. The bluff body is referred to as a submerged flame holder while the jet is referred to as a fluidic flame holder. Kareem Ahmed and David Forliti performed an experimental study at the State University of New York in Buffalo where measurements were taken for the jet in cross-flow case and compared to results for a bluff body flame holder. A single slot jet was used with a dimension of 0.279 mm to inject methane fuel at a flow rate to make an equivalence ratio between the jet and cross-flow $\phi=1$. The
results were compared to that of a wall bounded bluff body. The researchers compared side by side instantaneous PIV images of the bluff body flame holder and the jet in cross-flow flame holder. The vortices resulting from the bluff body push the reactants downwards resulting in upwards propagating flame. The result is that the reaction stays in the lower part of the combustion chamber. The flame eventually propagates upwards and into the main flow. The vortices produced behind the bluff body seem to be periodic rotating structures. In the jet different observations were made. The jet interacting with the cross-flow initially produces small regions of positive vorticity in the flow above the jet and negative vorticity in the recirculation zone of the jet. The positive vorticity helps to create faster flame speed and push the reaction up towards the top wall. The vortices in the jet seem to be much more random in comparison with the periodic vortices produced by the bluff body. The researchers noticed that the flame holder efficiency and area covered is much greater for the jet flame holder than the bluff body. The researchers also use vorticity to explore flame wrinkling and its effects. The conclusion was that flame wrinkling increases the size of the flame and the flame reaction rate, which therefore increases the flame efficiency. The material derivative of the vorticity vector can be expanded into separate components which are used to describe flame behavior.

\[
\frac{D\vec{\omega}}{Dt} = \frac{1}{\rho^2} (\nabla \rho \times \nabla p) - \vec{\omega} \nabla \cdot \vec{v} + \nabla \cdot \nabla \vec{v} + \nu \nabla^2 \vec{\omega} \quad (2.8)
\]

In this equation the first component on the right hand side is the baroclinic torque, followed by the gas expansion term, the vorticity stretching term, and the vorticity diffusion term. An examination of probability density function plots for vorticity for each flame holder showed whether each flame holder had a positive or negative vorticity bias. The main conclusions to draw from the investigation are that the jet in cross-flow created positive vorticity in the flame which allowed the flame to propagate to the top wall. This differs from the bluff body which created negative vorticity. Through the vorticity studies,
they concluded that the jet in cross-flow flame holder allowed for more efficient combustion. (Ahmed & Forliti, 2010)

2.10 Basic Description of Bluff Body Stabilized Flames

Turbulent flames are often stabilized with the use of a geometric object called a bluff body. The principle behind the bluff body the creation of a recirculation consisting of combustion products near adiabatic flame temperature. The hot products circulating behind the bluff body act as an ignition source continuously burning the reactants. Flame blowoff occurs when the flame is unable to stabilize itself on the bluff body and extinguishes. (Turns, 2000)

![Chemiluminescence Image of a bluff body-stabilized flame. Flow is left to right.](image)

Figure 2.6 Example image of bluff body-stabilized flame used in the present experiment. Flow is left to right. (Kopp-Vaughan, 2011)
2.11 Early Bluff Body Work

In 1953 Spalding published an article which describes flame stabilization by a recirculating wake. He describes that the blowoff velocity is related to the gas density, the flame holder diameter, and the flame propagation velocity. (Spalding, 1953)

\[
\frac{V}{d \rho S_u^2} = constant \quad (2.9)
\]

In 1955, Zukoski and Marble performed work on bluff body flames and proved the influence of wake transition to bluff body flame stabilization. They tested gasoline-air systems as well as methane-air systems on a cylindrical rod bluff body. From the blowoff curves they found that maximum blowoff velocities occur when the fuel to air mixture is close to stoichiometric. Temperature is a great influence on reaction time, and because stoichiometric equivalence ratios yield the highest flame temperature, the flames tend to stabilize at higher velocities at the stoichiometric condition. They determined that for Reynolds numbers below the transition region the bluff body flame is predominantly stabilized by molecular transport of the fuel and air. They also found that for very high Reynolds numbers the maximum blowoff velocity corresponds to the square-root of the bluff body diameter, but they did not come up with a clear explanation as to why that is true. (Zukoski & Marble, 1955)

2.12 Syngas Fuel Composition Sensitivities of Combustor Flashback and Blowout

Lieuwen et al. performed an experiment where they tested different fuel combinations of H₂, CO, and CH₄ at different inlet pressures and temperatures to try to confirm the equivalence ratio blowout limit as calculated using a Damköhler number. The work is not specifically about bluff body flame stabilization but it is important because the concepts studied regarding blowout can be applied to bluff body studies. The chemical reaction time scale was estimated using equation 2.10, where \( \alpha \) is the thermal diffusivity and \( S_L \) is the laminar flame speed.
\[ \tau_{\text{chemical}} = \frac{\alpha}{S^2_L} \quad (2.10) \]

The Damköhler number was used to estimate the blowout equivalence ratio of the system using Eqn. 2.11, where the residence time is the d/U_{ref} (characteristic length divided by a reference velocity).

\[ Da = \frac{\tau_{\text{residence}}}{\tau_{\text{chemical}}} = \frac{S^2_L d}{\alpha U_{ref}} \quad (2.11) \]

Using an adjusted equivalence ratio defined as the local equivalence ratio plus a constant related to the ratio of mass diffusivities of fuel to oxygen, they calculated Damköhler numbers for their data sets of different syngas fuels which had a value of 2.1, as seen in Fig. 2.7. Fig. 2.8 shows the predicted equivalence ratio versus the actual equivalence ratio for the data sets tested. The researchers concluded that the Damköhler number can be used to accurately predict the blowout equivalence ratio within 10% accuracy.

![Figure 2.7 Damköhler number versus % H₂. Experiments had U₀=6 m/s (approach velocity), inlet temperature and pressure of 300 K and 1.7 atm. (Noble, et al., 2006)](image)
2.13 Blowoff Dynamics of Bluff Body Stabilized Turbulent Premixed Flames

Flame stability from a bluff body is a topic that has been researched for many years. Particularly much of the previous research has been based around flame dynamics just before and during blowoff. The goal of studying the flame blowoff behavior is to prevent engine failure. Michael Renfro and Baki Cetegen have extensively studied this phenomenon. They used a round conical shaped burner which supported a vertical flame. The flame was stabilized on a disk shaped bluff body 10mm in diameter. The air flow rate tested in this case was 10 m/s and was verified by a hot wire anemometer test. Along with Chaudhuri and Kostka, they performed particle image velocimetry to see the velocity vector fields along the flame and flame edge. From the velocity fields vorticity and strain rate were calculated. Vorticity ($\omega$) is calculated as the curl of the velocity vector.

$$\vec{\omega} = \nabla \times \vec{U} \quad (2.12)$$
Considering flow in only the x and y directions, vorticity is present around only the z direction.

\[
\vec{\omega} = \frac{dv}{dx} - \frac{du}{dy}
\] (2.13)

They also used OH planar laser induced fluorescence to find the flame edge. They found that the local strain rates exceed extinction stretch rates which lead to extinction of the flame in the shear layer. After extinction the new reactants mix in the shear layers and react in the recirculation zone. With the reignition of the reactants, the shear layers can often reignite, and sometimes this ignites the whole flame again, if only momentarily. Using the high speed chemiluminescence the researchers established that before blowoff there is overlapping of Kelvin Helmholtz vortices. (Chaudhuri, et al., 2010)

These researchers along with Kristin Kopp-Vaughan and Trevor Jensen performed more blowoff measurements, this time in vitiated flow. They found that for lower equivalence ratios at the bluff body flame there is an increasing amount of small sections of extinction in the flame. They performed PIV and used the data to find the aerodynamic stretch rates on the flame surface. They found that flame blowoff occurs at much lower equivalence ratios when in the presence of a vitiated cross-flow than in a non-vitiated cross-flow. In hotter cross-flows the combustion reaction rate at the bluff body is increased which helps keep the flame lit for lower equivalence ratios. It was found that equivalence ratios close to blowoff had an increased flame stretch rate and also increased flame instability. They found that when under a vitiated cross-flow the flame’s Benard-von Karman caused the instability related with blowoff. In non-vitiated cross-flow the flame had Kelvin-Helmholtz instabilities which caused blowoff.

2.14 Dynamics of a Longitudinally Forced, Bluff Body Stabilized Flame

Tim Lieuwen’s research group at Georgia Institute of Technology performed extensive research of bluff body stabilized flames. In this particular experiment a flame was produced under cross-flow temperatures ranging from 297 to 870 K with velocities from 38 to 170 m/s. High speed videos of the
flame were taken and analyzed to come up with a mathematical model to describe the flame. They defined an $x$ and $y$ ordinate system where the positive $x$-axis is in the center of the bluff body and pointing in the direction of the flow. The $y$-axis spreads in either direction perpendicular to the $x$-axis.

$$G(x, y, t) = L(x, t) - y = 0 \quad (2.14)$$

In this equation, $L(x, t)$ is a function that describes how far the flame extends on the $y$-axis as a function of time and the distance away from the bluff body. In general, distances further from the bluff body have larger values for the $L$ function. The $G(x, y, t)$ function is defined for convenience to express the size and shape of the flame depending on the three variables. The researchers derived a differential equation to relate the flame shape to the velocities in the $x$ and $y$ directions.

$$\frac{dL}{dt} + u \frac{dL}{dx} - v = S_L \sqrt{\left(\frac{dL}{dx}\right)^2 + 1} \quad (2.15)$$

In this equation $u$ is the flame velocity in the $x$ direction, $v$ is the flame velocity in the $y$ direction, and $S_L$ is the laminar flame speed. Lastly, to model the fluctuations of the flame, they defined $L$, $u$, and $v$ in terms of temporal mean and fluctuation parts. They concluded that the flame becomes larger with increasing $x$ value, then peaks and decays with increasing $x$ value. The increase in flame size is due to the bluff body anchoring as well as flame wrinkles created by velocity oscillations. The peak and decay are attributed to the flame propagation outwards at local flame speeds. (Shin, et al., 2001)
3. Experimental Setup and Fundamental Background Information

Background information for the bluff body experiment, the jet in cross-flow (JICF) experiment, Particle Imaging Velocimetry (PIV), and measurement uncertainty calculations are discussed.

3.1 Bluff Body Experimental Setup

Figure 3.1 3-D model of the bluff body experiment.

Figure 3.2 Cutaway drawing of the inside geometry of the bluff body experiment.

In the experiment the air from a compressor enters through the back and enters the preburner section, as seen in Figs. 3.7, 3.8, and 3.9. The fuel is injected in the bottom of the old preburner and mixes with the air before entering the settling section. The preburner in this experiment is not used for burning. The settling section measures 6 in. tall by 12 in. wide and 20 in. long was used to allow the fuel and air
to premix on the way to the combustion chamber. The settling section as well as the convergent section contained KAST-O-LITE 97L purchased from ANH Refractories/A.P. Green Industries to reduce heat loss to the atmosphere. After the settling section a convergent section (Figs. 3.13-3.17) was used to reduce the flow area to the area of the test section and to increase the reactants speed. Measuring 12 in. long, the convergent section reduces the area from 6 in. tall by 12 in. wide to 1.5 in. tall to 3 in. wide. A 1in. outer diameter (OD) tube welded to the side of the convergent section was used to inject the alumina particles when performing PIV. After the convergent section a smaller settling section was used, as can be seen in Figs. 3.18-3.21. In the past the small convergent section was used mainly for fuel injection near the bluff body, but in the current experiment it serves mainly as extra distance for the mixing of the reactants before combustion. From the small settling section the reactants enter the test section which happens to be the only part where the flow is visible, as seen in Figs. 3.22-3.24. The test section has a cross-sectional area of 3in. by 1.5 in. and 8 in. long. The front and back walls are removable steel plates with the dimensions shown in Figs. 3.5 and 3.6. The top and front walls are removable quartz windows made from S1-UV fused silica ordered from ESCO Products with the same dimensions seen in Figs. 3.5 and 3.6. The quartz windows allow for camera and laser access when performing PIV.

The test section contains the bluff body (Fig. 3.4) which is shaped like an equilateral triangle with a side length of 9.6 mm (about 0.38 in.). The front of the bluff body is located 0.5 in. from the front edge of the quartz window. A pilot jet flame located within an inch behind the bluff body (not shown in diagrams) was used to light the flame. After the test section is the cooling section of the rig. The round section on the right side of Fig. 3.22 is the cooling section which allows multiple water tubes and a drain tube to be connected to douse the flame before entering the exhaust section. It is important to not open the water valve all the way open when using low air flow rates, as water may spill over into the test section. When performing an experiment it is recommended to open the water valve very until water is seen in the test
section (if it is seen at all). The exhaust section (Figs. 3.25 and 3.26) allows for the combustion products to flow upwards and out a vent located on the roof of the building.

In previous work, the bluff body flame had a non-uniform velocity profile which made comparison make a more uniform velocity profile the experiment had to be changed. It was decided to allow for a much greater distance for the air and fuel to mix before combustion, effectively increasing the time allowed for mixing. The experiment was adapted so that instead of fueling through airfoils at the bluff body the fuel was introduced through the old preburner section as seen in Figs. 3.7, 3.8, and 3.9. Because the airfoils were not being used they were removed to prevent unnecessary obstruction of the bluff body. The velocity profile (Fig. 3.3) has velocity deficits which correspond to the locations of the airfoils.

![Figure 3.3 Old bluff body velocity profiles.](image)

In the experiment described in this thesis, the upstream air pressure from the compressor was 50 psi, which resulted in about 15 m/s average velocity in the test section. The experimental conditions are shown in Table 3.1.

<table>
<thead>
<tr>
<th>Experimental Conditions</th>
<th>Equivalence Ratio φ</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.76</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>------------------------------------</td>
<td>-----</td>
</tr>
<tr>
<td>Air Density [kg/m$^3$]</td>
<td>1.1614</td>
</tr>
<tr>
<td>Fuel Density [kg/m$^3$]</td>
<td>1.7858</td>
</tr>
<tr>
<td>Test Section Velocity [m/s]</td>
<td>15</td>
</tr>
<tr>
<td>Test Section Mass Flow Rate [kg/s]</td>
<td>0.0518336</td>
</tr>
<tr>
<td>Air Mass Flow Rate [kg/s]</td>
<td>0.0494234</td>
</tr>
<tr>
<td>Fuel Mass Flow Rate [kg/s]</td>
<td>0.00241026</td>
</tr>
<tr>
<td>Air Volumetric Flow Rate [SLPM]</td>
<td>2553.30</td>
</tr>
<tr>
<td>Fuel Volumetric Flow Rate [SLPM]</td>
<td>80.9810</td>
</tr>
</tbody>
</table>

Table 3.1 Chart displaying conditions for bluff body experiment.

Figure 3.4 Bluff body diagram.

Figure 3.5 Optical window located on the front and back of the test section. Dimensions are in inches.
Figure 3.6 Laser window located on the top and bottom of the test section. The hole shown is only for the JICF experiment and does not exist on the windows used for the bluff body experiment. Dimensions are in inches.
Figure 3.7 Front view of preburner section. Preburner used for fuel injection only. Dimensions are in inches.

Figure 3.8 Cutaway view of preburner. Dimensions are in inches.
Figure 3.9 Bottom view of settling section. Fuel is added through hole on the bottom. Dimensions are in inches.
Figure 3.10 Settling section. Dimensions are in inches.
Figure 3.11 Cutaway view of settling section. Dimensions are in inches.
Figure 3.12 Side view of settling section. Dimensions are in inches.
Figure 3.13 Front view of convergent section. Dimensions are in inches.
Figure 3.14 Cutaway view of convergent section. Flow is from left to right. Dimensions are in inches.
Figure 3.15 Side View of convergent section. Flow is into the page. Dimensions are in inches.
Figure 3.16 Side View of convergent section. Flow is out of the page. Dimensions are in inches.

Figure 3.17 Detail view of exit shown in Fig. 3.16. The horizontal seed tube as well as the seed tube exit can be seen. The air and fuel mixture flows around the tube. Flow is out of the page. Dimensions are in inches.
Figure 3.18 Front view of small settling section. Dimensions are in inches.

Figure 3.19 Cutaway view of small settling section. Dimensions are in inches.
Figure 3.20 Top view of small settling section. Dimensions are in inches.

Figure 3.21 Side view of small settling section. Dimensions are in inches.
Figure 3.22 Dimensions of the bluff body test section. Dimensions are in inches.
Figure 3.23 Cutaway view of the bluff body test section. Dimensions are in inches.

Figure 3.24 Side views of the bluff body test section. Dimensions are in inches.
Figure 3.25 Exhaust system diagram. Exhaust stack is 129.5 in. The left plate connects to the exit of the test section. Flow is left right. Dimensions are in inches.
Figure 3.26 Cutaway view of exhaust system. Flow is left right and up.
3.2 Experimental Setup JICF

A slightly altered experimental set up was used for the jet in cross-flow experiment. Some of the components remained the same, but others had to be removed or replaced.

Figure 3.27 3-D model of JICF experiment. Flow is left right.

A new preburner had to be designed in order to stabilize a flame to produce a vitiated cross-flow velocity of 10 m/s or lower. The old preburner was not capable of achieving such low flow rates. The new burner is designed with a cross-sectional area equal to that of the test section, 3 in. x 1.5 in. The design can be seen in Figs. 3.29-3.31. Air from the flow bench comes in a 1.25 in. tube in the back of the burner at a mass flow rate specified for the desired cross-flow velocity. This experiment aims to average a 15 m/s cross-flow velocity. Fuel was injected through both sides of a 0.25 in. tube with small holes facing the incoming air flow. A standard K type thermocouple was used to monitor the cross-flow temperature. After the preburner, the convergent section from the bluff body experiment was used
(Figs. 3.13-3.17). The main advantage of reusing this section is to utilize its seeding capabilities. The small settling section was also reused to add extra settling distance before combustion. The small settling section is displayed in Figs. 3.18-3.21. The test section (Fig. 3.32) is the same test section used in the bluff body experiment except the bluff body is removed and the bottom plate is replaced with a plate that can hold the jet parts. As seen in Fig. 3.6 the jet is located 1.7 in. in from the upstream edge of the bottom window. The jet design can be seen in more detail in Figs. 3.33 and 3.34.

A fuel tube extends through the width of the burner, and fuel is injected from both sides. In order to maximize fuel and air mixing in the preburner, the holes on the fuel tube inject fuel towards the airflow. A spark plug is then used to light the premixed fuel and air and stabilize a flame on the fuel tube. A standard K type thermocouple in the preburner is used to monitor the flame temperature. The equivalence ratio in the preburner is calculated using the air flow rate and fuel flow rate. The air flow rate is controlled using the orifice bench and the fuel is controlled using a single mass flow controller. Steel wool sandwiched in between two pieces of mesh was used to straighten the flow and keep the flame profile uniform. Using K type thermocouples the current temperature achieved from the preburner is 1040°C (1313K). ChemKin has calculated the adiabatic flame temperature of propane for a 0.5 equivalence ratio to be about 1510K. The difference in temperature can be attributed to incomplete mixing as well as heat loss through the preburner. After the heat loss, the cross-flow temperature is measured to be 820°C (1093K).

KAST-O-LITE 97L sections are used in between the preburner and the test section. The settling section has a few goals. The first goal of the section is provide a distance between the preburner flame and the test section while keeping the temperature of the combustion products uniform through the cross sectional area. KAST-O-LITE is used to minimize heat loss in the section to keep the combustion products
temperature as close to the adiabatic flame temperature as possible. Another purpose of the settling
section is to provide a tube for seed to be injected when performing PIV.

The test section was fitted with a jet set up on the bottom plate. The jet consists of a ½ in. air tube. Air in
the jet is controlled using a mass flow controller controlled by a Labview program.

Figure 3.28 Cutaway of JICF experiment. Flow is left right.
Figure 3.29 3-D model of JICF preburner.

Figure 3.30 JICF preburner drawing and cutaway. Flow is from left to right. Dimensions are in inches.
Figure 3.31 Side view of JICF preburner. Flow is into the page. Dimensions are in inches.

Figure 3.32 JICF test section.
Figure 3.33 Front view of the air tube. Flow is from bottom to top. Dimensions are in inches.
Figure 3.34 Detail view of jet tubes right before test section. The top of the outside tube shown is flush with bottom side of the test section. Dimensions are in inches.

3.4 JICF Operating Conditions

The operating conditions tested for the JICF setup are a preburner equivalence ratio of 0.5 and momentum ratios of 25, 50, and 75. The momentum ratio depends on the cross-flow velocity. By selecting a desired momentum ratio one can calculate the required mass flow rates jet air to use. The cross-flow velocity was estimated using a mass flow rate balance and a temperature of the combustion products entering the test section.

\[ \dot{m}_{in} = \dot{m}_{out} \quad (3.9) \]

The mass flow rate into the experiment is calculated simply by adding the mass flow rate of the air and the mass flow rate of the fuel entering the preburner.

\[ \dot{m}_{in} = \dot{m}_{air} + \dot{m}_{fuel} \quad (3.10) \]
The average velocity of the products into the test section is calculated a rearranged form of Eqn. 3.3. The area of the test section is known to be 0.00290 m² (3 in. x 1.5 in), the density of the combustion products is assumed to be the density of air as a function of the temperature (Fig. 3.36), and the mass flow rate is calculated in Eqn. 3.10.

![Density vs Temperature](image)

Figure 3.36 Cross-Flow density as a function of cross-flow temperature.

<table>
<thead>
<tr>
<th>Experimental Conditions</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>φ Preburner</td>
<td>0.5</td>
</tr>
<tr>
<td>Preburner Temperature [K]</td>
<td>1400</td>
</tr>
<tr>
<td>Mean Velocity Test Section [m/s]</td>
<td>10.6</td>
</tr>
<tr>
<td>Mean Temperature Test Section [K]</td>
<td>820</td>
</tr>
</tbody>
</table>

Table 3.2 Experimental Conditions for the cross-flow.

<table>
<thead>
<tr>
<th>J ratio</th>
<th>25</th>
<th>50</th>
<th>75</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid</td>
<td>Air</td>
<td>Air</td>
<td>Air</td>
</tr>
<tr>
<td>Diameter [in]</td>
<td>0.375</td>
<td>0.375</td>
<td>0.375</td>
</tr>
<tr>
<td>Density [kg/m³]</td>
<td>1.19E+00</td>
<td>1.19E+00</td>
<td>1.19E+00</td>
</tr>
<tr>
<td>Velocity [m/s]</td>
<td>3.93E+01</td>
<td>5.56E+01</td>
<td>6.81E+01</td>
</tr>
<tr>
<td>Mass Flow Rate [kg/s]</td>
<td>1.74E-03</td>
<td>2.46E-03</td>
<td>3.01E-03</td>
</tr>
<tr>
<td>Volumetric Flow Rate [m³/s]</td>
<td>1.49E-03</td>
<td>2.11E-03</td>
<td>2.58E-03</td>
</tr>
<tr>
<td>-----------------------------</td>
<td>----------</td>
<td>----------</td>
<td>----------</td>
</tr>
</tbody>
</table>

Table 3.3 Experimental conditions for the jet.

3.5 Particle Imaging Velocimetry

Particle Imaging Velocimetry (PIV) is commonly used to take velocity measurements in a plane defined by a laser sheet and to determine various flow field characteristics in that two-dimensional plane. Usually a double cavity Nd-YAG laser is coupled with a charge-coupled device (CCD) camera to track seed particles injected in the flow upstream of the test section. The experiments described in this thesis utilize a FlowMaster 3S pass filter, a New Wave (Solo PIV III) dual-cavity Nd:YAG, and DaVis 7.0 post-processing software, as shown in Fig. 3.37.

![PIV experimental setup](image)

**Figure 3.37 PIV experimental setup.** (Tuttle, et al., 2013)

When a laser pulse hits a seed particle the light spreads in all directions in a phenomenon called Mie scattering. Two laser pulses are used together separated by a known time difference. The PIV camera containing a frame-straddling 1024x1280 CCD chip records images at the two pulses and uses a cross correlation algorithm to determine the particle movement between the two pulses. From the particle
displacement information and the known time difference between the pulses, velocity fields can be computed (Adrian & Westerweel, 2011).

![Raw Mie scattering image from DaVis](image)

**Figure 3.38 Raw Mie scattering image from DaVis**

The DaVis software divides the image area into interrogation windows consisting of 32x32 pixels. The cross-correlation algorithm looks at all of the particles in the interrogation window, and tries to determine the particle displacement within the window. The most common displacement of all of the particles in a window is used to determine a single velocity vector for that window. When this process is done for every interrogation window the total test section area is represented with velocity vectors. (Adrian & Westerweel, 2011)

There are a few fundamental pieces of equipment necessary for properly executing PIV. The camera which usually has frame straddling capabilities is needed. Frame straddling is a fast method of saving information to the CCD chip in two back to back images without having to transfer the information to a
computer. A dual cavity Nd-YAG laser is required to create the pulses separated by single to tens of microseconds depending on the average velocity. Optics and mirrors are needed to direct the laser beam to the test section. A cylindrical lens is used to spread the laser beam into a thin laser sheet which is used to illuminate the entire length of the test section. Quartz windows are needed to allow the laser to transmit through the test section. Seed particles are also needed for injection in the flow. The experiments described in this thesis utilize Micro-alumina #1 (Al₂O₃) particles which measure 1 micron in diameter and have a melting point high enough to withstand the flame. The seed used in the cross-flow was contained in a TSI 9310 fluidized bed aerosol generator and was injected in a seed tube in the convergent section contained in both experimental setups, as seen in Fig. 3.39.

![Convergent Section Diagram](image)

**Figure 3.39 Convergent Section where seed was injected in both experiments.**

Once the velocity vectors have been computed using DaVis, as described in more detail subsequently, it is convenient to further analyze the vector field data using user written programs in MATLAB. It is possible to output the velocity vector information to MATLAB for further analysis using the readimx('filename.VC7') and showimx functions, as discussed later. MATLAB is useful for eliminating stray velocity vectors outside the size of the test section, for displaying colored velocity vector plots,
creating streamlines for the velocity plots, calculating strain rate, calculating and plotting vorticity, as well as any calculation that can be done with known velocity fields.

Figure 3.40 Interrogation windows with initial and final particle locations. The big arrows in the windows represent the vectors resulting from the cross-correlation. (Adrian & Westerweel, 2011) PIV software creates peaks to determine possible ending locations for a particle. Using the cross correlation the correct particle ending location is determined with the largest peak, as can be seen in Fig. 3.40.
Figure 3.41 First image shows possible particle movement locations. Second image shows resulting correlation map. Third image shows a peak of the most probable particle location. Taken directly from (Westerweel & Poelma)

A peak ratio is defined to compare the height of the highest peak to the height of the smaller peaks and noise, as shown in Eqn. 3.11.

\[
Q = \frac{P_1 - min}{P_2 - min} > 1 \quad (3.11)
\]

Q is the peak ratio, \(P_1\) is the highest peak, \(P_2\) is the second highest peak, and \(min\) is the common noise background. If the peak ratio is close to 1, then the highest peak and second highest peak are the same height, so there is a great deal of uncertainty associated with the vector produced. Typical peak ratios are on the order of 1.2 to 1.5. In the DaVis algorithm, an interrogation window with only one peak is given a Q ratio of 100. This shows that there the program is relatively certain that particles are correctly matched between the two images. (LaVision, 2002)

There are different rules of thumb to be considered in performing of PIV. Generally it is desirable to have about 8 particles in a single interrogation window. (Gharib & Dabiri, 2012) Too few particles and the dominant peak in the cross-correlation will not be averaged over similar particle velocities but will be of the same magnitude as the peak from individual particles, leading to low Q ratios. Too many
particles and noise floor in the cross-correlation can become of similar magnitude to the particle peaks, again leading to low Q ratios. It is also desirable for particles to move approximately one quarter of the interrogation window size between laser pulses. This is done to ensure the majority of particles in the interrogation window for the first frame are also in the same interrogation window for the second frame. If a particle leaves the interrogation window between the two images then it cannot be used for displacement calculation. (Gharib & Dabiri, 2012) The distance a particle moves between frames can be altered by changing the interrogation window size or by altering the time difference between the laser pulses. By increasing the interrogation window size there will be more particles window which results in decreased uncertainty, however the amount of vectors produced will decrease as well. (Westerweel & Poelma) It is also necessary to determine proper flow conditions for the seeding system. It is desirable to have the seed as evenly distributed through the flow as possible. After performing PIV, seed can accumulate on the quartz window which blocks the camera’s view of the flow. It is important to keep the windows clean in between experiments.

PIV images can be used in an analysis called conditioned particle image velocimetry (CPIV), as discussed in detail in Chapter 6. This process works on the fact that in the flame, the density is much lower than out of the flame. The difference in density reduces the amount of seed in the volume which results in a lowered intensity. The intensity data needs to be corrected and normalized by the mean. When plotting the probability density function of the intensity there should be two peaks formed. The first peak corresponds to the reactants and the second peak corresponds to the products. There is a minimum in between that corresponds to the flame level value. (Tuttle, et al., 2013). The use of CPIV to condition velocity statistics along the flame front is discussed in Chapter 6.

3.6 Measurement Uncertainty
If a variable is dependent on other variables that all have an associated uncertainty, it is important to find how the error propagates to the dependent variable. Figliola and Beasley describe the proper method for calculating error propagation.

\[
u_R = \pm \left[ \sum_{i=1}^{L} \left( \theta_i u_{x_i} \right)^2 \right]^{1/2} \quad (3.12)
\]

\[
\theta_i = \frac{dR}{dx_i} \quad i = 1,2 \ldots L \quad (3.13)
\]

To find the uncertainty in the variable R, it is necessary to find the partial derivatives of R with respect to all of variables. The overall uncertainty is equal to the square root of the sum of the squares of the partial derivatives multiplied by the uncertainty of that variable. In the example of the PIV experiment it can be assumed that the only variables of uncertainty are the horizontal and vertical velocities. It is assumed that the X and Y directional vectors describing the position of each pixel in the image are known with absolute certainty. Many properties of fluids can be derived using the velocity information such as vorticity and strain rate can be derived from the velocity information. If the uncertainty of velocity is known, then it is possible to determine the uncertainty of the vorticity and strain rates. (Figliola & Beasley, 2006) The derived equation for vorticity uncertainty is show in Eqn. 3.14. Likewise the derivations for strain rate uncertainties are shown in Eqn. 3.15, 3.16, and 3.17.

\[
\omega' = \sqrt{\left( \frac{d\omega}{dU} \right)^2 \cdot U' + \left( \frac{d\omega}{dV} \right)^2 \cdot V'} \quad (3.14)
\]

\[
e'_{xy} = e'_{yx} = \sqrt{\left( \frac{d\varepsilon_{xy}}{dU} \right)^2 \cdot U' + \left( \frac{d\varepsilon_{xy}}{dV} \right)^2 \cdot V'} \quad (3.15)
\]

\[
e'_{xx} = \sqrt{\left( \frac{d\varepsilon_{xx}}{dU} \right)^2 \cdot U'} \quad (3.16)
\]
\[ e'_{yy} = \sqrt{\left(\frac{d y_{yy}}{dV}\right)^2} \cdot V' \] (3.17)

4. Introduction to Particle Image Velocimetry Tools in MATLAB

The DaVis commercial software was used to analyze the raw Mie scattering image pairs and calculate the corresponding vector fields. It is possible to do some further analyzing in DaVis, but it is more convenient to perform calculations in MATLAB. This chapter serves as a tutorial to anyone looking to use MATLAB to further process data from Particle Image Velocimetry (PIV) explains in much greater detail. The readimx and showimx functions can be used to extract the vectors showing the horizontal and vertical components of velocity for each horizontal and vertical location.

\[
[\text{Variable}] = \text{readimx('file path');} \quad (4.1)
\]

\[
[X,Y,U,V] = \text{showimx(variable);} \quad (4.2)
\]

4.1 Velocity Vector Plots

Colorized vector plots using MATLAB were created using the vfield function.

\[ \text{vfield}(X,Y,U,V,\text{scale}) \] (4.3)

The inputs X, Y, U, and V represent the horizontal direction, the vertical direction, the horizontal velocity, and the vertical velocity. The input “scale” represents the length of the vectors output. For example inputting “U/10” for the scale would output vectors with values one tenth the size of the actual value. The plots shown in this thesis use the horizontal velocity variable (U) as the scale. The vectors in the plot show the direction of the flow at each specific location and the color of the vectors correspond to the vector velocity magnitude, as shown in Fig. 4.A
The two vertical lines near the 0 and 10 mm marks are user entered in MATLAB, and are used to show the location of the jet hole as found by the raw images. Due to the nature of turbulent flow, it is often convenient to analyze both the average and statistical variations in the velocity images. The average image can be produced by MATLAB or DaVis, and shows the average velocity magnitudes and directions for each interrogation window. Comparing Fig. 4.2 to Fig. 4.1 shows the difference between an instantaneous image and an average image. The average images for the J=50 and J=75 cases are displayed in Figs. 4.3 and 4.4. A similar analysis can be done with the bluff body experiment as shown in Fig. 4.5 and Fig. 4.6.
Figure 4.2 Average vector plot with $J=25$ and $\phi_{Pre}=0.5$. Jet location is marked with vertical lines.

Figure 4.3 Average vector plot with $J=50$ and $\phi_{Pre}=0.5$. Jet location is marked with vertical lines.
Figure 4.4 Average vector plot with $J=75$ and $\phi_{Pre}=0.5$. Jet location is marked with vertical lines.

Figure 4.5 Instantaneous vector plot with $\phi=0.76$. Bluff body location is marked with lines.
Figure 4.6 Instantaneous vector plot with $\phi=0.76$. Bluff body location is marked with lines.

4.1 Applications of Velocity Vector Plots

Using the MATLAB software it is possible to extract line plots from the vector plots produced. As an example, if an estimate of the angle of jet flow was desired, one could measure the angle of the jet velocities.

$$\theta = \cos^{-1} \frac{U}{|\vec{U}|} \quad (4.4)$$

In the previous equation $\theta$ is the angle with respect to the vertical axis and $|\vec{U}|$ is the magnitude of the velocity for that interrogation window. Once $\theta$ is calculated for every interrogation window, it is possible to make a plot of the angle versus X direction for a constant Y value. In this case the angles are examined for a Y value of 5.48mm for the J=25 case.
Figure 4.7 Velocity angle plot of $J=25$ and $\phi_{\text{Pre}}=0.5$. $Y=5.48$ mm.

Looking at the vector plot, at $Y=5.48$mm the center of the jet appears to be around $X=10$mm. At this value the angle the velocity has with the vertical axis is about $38^\circ$. Performing a similar analysis for the $J=50$ and $J=75$ case the angles with the vertical $\theta=26^\circ$ and $\theta=13^\circ$ respectively. When plotting the results (Fig. 4.8) it can be seen that increased momentum ratios have more resistance to the cross-flow.

Figure 4.8 Velocity angle for various momentum ratios.
As another example a similar analysis can be done for the velocity fields as well. If it were desired to know the horizontal velocity profile 25 mm after the bluff body, a chart of Y direction versus horizontal velocity for X=30 mm can be produced, as shown in Fig. 4.9. As expected, the horizontal velocity is much faster around the bluff body flame, and is much slower in the bluff body flame.

![Graph showing vertical direction versus horizontal velocity profile for X=30mm.](image)

Figure 4.9 Vertical direction versus horizontal velocity profile for X=30mm. Image used is the average image.

### 4.2 Streamlines of Flow

It is possible to use the velocity information to calculate many fluid properties. Streamlines are curves that show the direction of the flow and are defined to be tangent to the flow direction.

\[
\frac{dx}{U} = \frac{dy}{V} \quad (4.5)
\]
Figure 4.10 Streamlines for average image of $J=25$ and $\phi_{pre}=0.5$.

Figure 4.11 Streamlines for average image of $J=50$ and $\phi_{pre}=0.5$. 
Figure 4.12 Streamlines for average image of \( J=75 \) and \( \phi_{Pr}=0.5 \).

MATLAB has a built-in function which allows the user to create streamlines given the input of velocity and directional data.

\[
[Variable] = streamline(X,Y,U,V,startX,startY) \quad (4.6)
\]

In eqn. 4.6, \( X \) is the horizontal direction, \( Y \) is the vertical direction, \( U \) is the horizontal velocity, \( V \) is the vertical velocity, \( startX \) is the starting horizontal location of the streamlines and \( startY \) is the starting vertical location of the stream lines. If \( startX \) and \( startY \) are vectors, multiple streamlines can be produced. Using streamlines one can note that the velocity upstream of the jet tend to curve upwards and align parallel with the jet. One can also note that in areas with low seed density such as recirculation zones, the streamlines are generally inconclusive. A similar plot can be produced for the bluff body flame, as shown in Fig. 4.13.
4.3 Vorticity Plots

Vorticity plots are interesting in determining the amount of circulation in the fluid test section.

\[
\bar{\omega} = \nabla \times \vec{U} \quad (4.7)
\]

If velocity is only considered in the X and Y direction, then vorticity is defined as rotating about the Z axis.

\[
\bar{\omega} = \frac{dV}{dX} - \frac{dU}{dY} \quad (4.8)
\]

One way to calculate vorticity in MATLAB is by using the curl function. In eqn. 4.9, \( \omega_z \) is the curl in the Z direction, CAV is the calculated angular velocity, X is the horizontal distance, Y is the vertical distance, U is the horizontal velocity, and V is the vertical velocity.

\[
[\omega_z, CAV] = \text{curl}(X, Y, U, V) \quad (4.9)
\]
In this thesis however a central differencing method is used to calculate the derivatives of velocity with respect to distance, which are then used to calculate vorticity using eqn. 4.10. The central differencing method is discussed in greater detail in Chapter 6.

\[
\omega_{i,j} = \frac{V_{i,j+1} - V_{i,j-1}}{2\Delta X} - \frac{U_{i+1,j} - U_{i-1,j}}{2\Delta Y} \quad (4.10)
\]

Once the vorticity is calculated it is most easily plotted using the filled contour plot function in MATLAB.

\[
\text{contourf}(X, Y, W) \quad (4.11)
\]

In eqn. 4.11 X is the horizontal distance, Y is the vertical distance, and W is the parameter to be plotted in X and Y, which in this case is the vorticity.

Average vorticity plots are useful for making general observations of the overall vorticity trends in some of the test section locations.

![Average vorticity plot](image)

**Figure 4.14 Average vorticity plot of J=25 and \( \phi_{pc}=0.5 \).**
Figure 4.15 Average vorticity plot of $J=50$ and $\phi_{\text{pre}}=0.5$.

Figure 4.16 Average vorticity plot of $J=75$ and $\phi_{\text{pre}}=0.5$. 
Figure 4.17 Average vorticity plot of bluff body flame at $\phi=0.76$.

Average vorticity plots however are not useful for observing random vortex structures due to turbulence. If the sign of the vorticity varies in a certain location for multiple images then the overall vorticity in that area will not be accurately represented in the average image. It is for this purpose that it is worthwhile looking at the vorticity plots for sample instantaneous images. A sample image is shown below in Figs. 4.18 and 4.19.
Figure 4.18 Instantaneous vorticity plot of $J=25$ and $\phi_{pre}=0.5$.

Figure 4.19 Instantaneous vorticity plot of bluff body flame at $\phi=0.76$.

4.3 Extension of Vorticity
An advantage of using vorticity plots it that they can be used to estimate the rotational direction of the velocity and to help evaluate mixing. To further analyze mixing it can be convenient to define an average vorticity in the X direction, the Y direction, and also the total field averaged vorticity. The average vorticity at each X location can be calculated the in the following manner.

$$\omega_{\text{ave}}(x) = \int \omega \, dy = \left( \frac{1}{n} \sum_{j}^{n} \omega_{i,j} \right) n \Delta y = \Delta y \sum_{j}^{n} \omega_{i,j} \quad (4.12)$$

![Figure 4.20 Average Vorticity along X-Direction for average image of J=25 and $\phi_{pr}=0.5$.](image)

It can be seen from Fig. 4.20 that the average vorticity is positive upstream of the jet, becomes negative downstream of the jet and then evens out to be about 0.

Similarly, the average vorticity at each Y location can be calculated.

$$\omega_{\text{ave}}(y) = \int \omega \, dx = \left( \frac{1}{m} \sum_{i}^{m} \omega_{i,j} \right) m \Delta x = \Delta x \sum_{i}^{m} \omega_{i,j} \quad (4.13)$$

Finally, it is convenient to define a field averaged vorticity which can be useful in estimating mixing.
\[ \bar{\omega} = \iiint |\omega| dx dy = \int \omega_{\text{ave}}(y) dy = \Delta x \Delta y \sum_i^m \sum_j^n |\omega_{i,j}| \quad (4.14) \]

**Figure 4.21 Field Averaged Vorticity Plot**

The plot shows that increasing momentum ratios lead to a higher field averaged vorticity which promotes mixing. The trend appears to be linear, but more data points at different momentum ratios are required to determine the true trend. This is a topic of interest which should be explored in the future.

### 4.4 Strain Rate

Another important fluid property in combustion is strain rate. Strain rate shows the deformation of a fluid. In tensor notation strain rate can be defined using eqn. 4.15.

\[
e_{ij} = \frac{1}{2} \left( \frac{du_i}{dx_j} + \frac{du_j}{dx_i} \right) \quad (4.15)
\]

In two dimensions the four primary strain terms simplify to eqns. 4.16, 4.17, and 4.18.

\[
e_{xx} = \frac{dU}{dX} \quad (4.16)
\]
As explained by Kundu and Cohen, strain rate can be separated into symmetric and antisymmetric stress tensors. (Kundu & Cohen, 2008)

\[
\frac{d\bar{U}}{d\bar{X}} = \begin{bmatrix}
    e_{xx} & \frac{1}{2} e_{xy} \\
    \frac{1}{2} e_{yx} & e_{yy}
\end{bmatrix} + \begin{bmatrix}
    0 & \frac{1}{2} \omega_z \\
    -\frac{1}{2} \omega_z & 0
\end{bmatrix} \tag{4.19}
\]

Using the central differencing scheme it is possible to estimate the strain rate values fields. Again more details on the central differencing scheme will be discussed in Chapter 6.

\[
e_{xx} \rightarrow e_{i,j} = \frac{U_{i,j+1} - U_{i,j-1}}{\Delta X} \tag{4.20}
\]

\[
e_{yy} \rightarrow e_{i,j} = \frac{V_{i,j+1} - V_{i,j-1}}{\Delta Y} \tag{4.21}
\]

\[
e_{xy} = e_{yx} \rightarrow e_{i,j} = \frac{1}{2} \left( \frac{V_{i,j+1} - V_{i,j-1}}{2\Delta X} + \frac{U_{i+1,j} - U_{i-1,j}}{2\Delta Y} \right) \tag{4.22}
\]

In MATLAB, the strain rate plots can be plotted in a similar fashion to vorticity.
Figure 4.22 Contour plot of normal strain in the x direction.

Figure 4.23 Contour plot of normal strain in the y direction.
Figure 4.24 Contour plot of shear strain.
5. Current Bluff Body Study

The previous studies on the bluff body were slightly different than the studies shown in this thesis. In the past, the focus of the Particle Image Velocimetry (PIV) was on the wake of the bluff body and in the recirculation zone.

![Average vector plot showing former PIV field of view.](image)

The preburner shown in Figure 3.AA and 3.AB burned fuel at a lean equivalence ratio for a hot vitiated cross-flow to simulate the products from a primary combustor. Fuel was added through airfoils which were inserted in the small settling section shown in Figure 3.AM. The extra fuel was added 5 in. before the bluff body and reacted with the unused air from the first combustor. One of the primary purposes of the experiment was to use the velocity profiles to match with computational fluid dynamics simulations (CFD) to verify the accuracy of the simulations performed.

It was found that the inlet conditions for the previous bluff body work were not sufficiently uniform to enable comparisons to some bluff body flame models. To better understand the velocity profiles entering the test section, Kopp-Vaughan performed a PIV study which included an area 0.5 bluff body
widths upstream of the bluff body location. She performed PIV for four different upstream pressure cases graphed horizontal velocity profile versus the vertical distance at the location 0.5 bluff body widths (<\(\frac{1}{4}\) in.) upstream of the bluff body. (Kopp-Vaughan, 2011)

![Velocity profile graph](image)

**Figure 5.2 Velocity along the vertical axis upstream of the bluff body.** Negative and positive distances denote area below and above the bluff body. (Kopp-Vaughan, 2011)

As can be seen in Figure 5.2 the upstream conditions are not very uniform. The lines appear to almost have an “M” shape having a slower velocity in the vertical location of the bluff body and faster velocities just above and below it. Specifically the 50 PSI case reaches a maximum velocity of about 25 m/s seen about 7 mm below the bluff body and a minimum at the bluff body location of about 21 m/s. From her study, Kopp-Vaughan noticed that these dips in the velocity profile line up with the vertical locations of the fuel injecting airfoils. (Kopp-Vaughan, 2011) At the time this non-uniformity in the velocity profile was acceptable because the main focus of the experiment was the recirculation zone of the bluff body.

In the current bluff body study, changes were made to the experiment to try to achieve more uniform boundary conditions. Fuel and air were added in the preburner section to allow the whole length of the experiment for settling and mixing. Since they were no longer necessary, the fuel injecting air foils were removed as well so they would not obstruct the flow before the bluff body.
As can be seen from Figure 5.3, the flow still resembles an “M” shape, but it looks better than the previous study.

Figure 5.3 Average horizontal velocity profiles just after the bluff body, in the middle of the test section, and at the end of the test section. Upstream pressure is 50 psi.

To get a better view of the inlet conditions, PIV was done with the bluff body removed. Figure 5.4 shows that above and below the bluff body the velocity is relatively constant at about 27 m/s. Figure 5.5 shows a vector plot which appears to have very smooth and constant inlet conditions as well as a symmetric velocity profile. The improved inlet conditions shown in this thesis show provide for better data that can more accurately be compared with CFD models.
Figure 5.4 Average horizontal velocity profiles with the bluff body removed. Upstream pressure is 70 psi.

Figure 5.5 Average image with improved boundary conditions.
6. Particle Image Velocimetry Uncertainty Analysis

For all of the PIV experiments done in the laboratory, it is desirable to know the error associated with the PIV setup. Any error associated with the experimental results will propagate further as an analysis of the results is done. For the results to have any credibility it is important to know the degree of the associated uncertainty to ultimately see how much confidence can be placed in the final analysis.

6.1 Determining Particle Image Velocimetry Uncertainty

After DaVis performs a cross-correlation on the images, a peak correlation peak is formed for every interrogation window. DaVis can export a correlation map so that the peaks for every interrogation window are shown. The correlation map is displayed in terms of different intensities within the interrogation window. The following figure is a 2-D image of a correlation map for a single image in the PIV experiment.

![Correlation Map](image)

**Figure 6.1 Correlation map of a jet in cross-flow image.**
As can be seen, the correlation map is divided into boxes representing every single interrogation window. The concept can be easier understood by zooming in to a single interrogation window.

![Correlation Map](image)

**Figure 6.2 Single interrogation window from the correlation map in Figure 6.1.**

This figure represents a contour plot for a single interrogation window. Each interrogation window is normally 32 pixels wide by 32 pixels high, but DaVis adds a blank column at the end to make 33 columns and a blank row at the beginning to make 33 rows. Therefore the analysis was done for interrogation windows of 33x33 pixels. The contour plot shows that there are many different mini peaks, but only one very large peak. In the image the large peak is in the middle square and is represented by the highest intensity values as shown by the color bar on the right. For interrogation windows where there the particle destination is uncertain, the cross correlation will produce a shorter, wider peak. On the contrary, cross correlations with very low uncertainty will have a single tall and thin peak. (Adrian & Westerweel, 2011) In order to determine the uncertainty it is necessary to determine the peak thickness and height. To do this a program was written that finds the row number containing the peak, and makes a plot of the intensity in that row versus the pixel number. The following chart shows the result.
Figure 6.3 Row of pixels containing the maximum peak.

As can be seen by the chart there is one large peak surrounded by noise on both sides. In order to find the uncertainty, it is necessary to find the half peak half width. This is done by finding the height of the peak minus the height of the surrounding noise, and then finding width of the peak at half of that value. The program determines the highest level of intensity in the row, and defines the peak as that pixel plus and minus 2 pixels. For example, in this image the summit of the peak is located at pixel 17, so the peak is defined to exist between pixels 15 to 19. The intensity of all of the pixels not in the peak are averaged to find the average noise level. The half peak height is defined as half of the difference of the peak height and the average noise height, plus the average noise height.

\[
Half \ Peak \ Height = \frac{Peak \ Height - Average \ Noise \ Height}{2} + Average \ Noise \ Height \quad (6.1)
\]
Figure 6.4 Diagram showing the value for $\frac{1}{2}$ peak height.

Because each half of the peak consists of only three points, the width of the half peak height is estimated using a linear regression. The uncertainty of the PIV measurement is defined at plus or minus the half peak half width. This represents the uncertainty of the distance the PIV particle travels with distance measured in number of pixels. It was found that uncertainty varies throughout the image by ±0.5 pixels to ±1.0 pixels. For convenience, the average uncertainty of ±0.75 pixels is used to characterize the velocity uncertainty throughout the image. By knowing the pixel length is 6.7µm from calibration of the system, the uncertainty can be converted to meters. The nominal distance the particle travels is calculated by using the velocity magnitude for the interrogation window.

$$Velocity\ Magnitude = \sqrt{U^2 + V^2} \quad (6.2)$$

The nominal distance the particle travels is equal to the velocity magnitude multiplied by the time in between the two images. In this experiment the time difference is 4µs.

$$Distance = Velocity\ Magnitude \cdot Time\ Delay \quad (6.3)$$
For the purposes of velocity computation, it is convenient to determine the percentage uncertainty for each interrogation window. The percentage uncertainty is determined by dividing the uncertainty distance by the nominal distance.

\[
\% \text{Uncertainty} = \frac{\text{Uncertainty Distance}}{\text{Nominal Distance}} \times 100 \quad (6.4)
\]

Using the percentage uncertainty it is easy to find the uncertainty of the velocity in the vertical and horizontal directions.

\[
U' = \% \text{Uncertainty} \times U \quad (6.5)
\]

\[
V' = \% \text{Uncertainty} \times V \quad (6.6)
\]

### 6.2 Error Propagation and Analysis of Uncertainty

Using the program to find the uncertainty for each interrogation window it is possible to compare the uncertainty for different parts of the test section. Interrogation windows in each part of the section were sampled and averaged. The average uncertainty of the data at the bottom of the jet is ±0.70 pixels, the average uncertainty for the jet recirculation zone is ±0.76 pixels and the average uncertainty in the cross-flow is ±0.75 pixels. It is interesting to note that the uncertainty appears to be independent of J ratio. A statistical analysis was done for all of the interrogation windows for a single J=25 image pair. In the analysis, the sections where uncertainty values are 0 or not a number have been eliminated. Also all values that occur at Y values of greater than 19 mm or less than -19 mm are considered to be outside of the test section and have been eliminated as well. The average uncertainty is ±0.7528 pixels with a standard deviation of 0.1359.
Figure 6.5 Contour plot of uncertainty in pixels for jet in cross-flow.

For the purposes of this analysis, 0.75 has been taken to be the uncertainty for all interrogation windows. The uncertainty of a distance that a particle travels in the interrogation window is calculated knowing that a pixel is 6.7 μm.

\[
\pm 0.75 \text{Pixel} \times \frac{6.7 \mu m}{\text{Pixel}} = \pm 5.025 \mu m \quad (6.7)
\]

Knowing there is 4.0 μs in between the images it is possible to find the uncertainty of the velocity in the interrogation window.

\[
\pm 5.025 \mu m \times \frac{1}{4.0 \mu s} = \pm \frac{1.256 m}{s} \quad (6.8)
\]

This calculated velocity corresponds to the magnitude of the velocity in the interrogation window. It is desirable to know the error in distance propagates to the corresponding error in velocity and other parameters that are a function of velocity. The velocity percent uncertainty per interrogation window is a comparison that essentially calculates percent error of the velocity magnitude as compared to the
velocity magnitude for that interrogation window. By this definition, if the uncertainty of the velocity magnitude is assumed to be the same for every interrogation window then the interrogation windows corresponding to the lowest velocities will have the highest uncertainty percentage. In the following contour plot this is seen to be true where the highest uncertainty falls in the recirculation zone of the jet and the lowest uncertainty is in the cross-flow and the jet itself. A more accurate way to measure this would be to use the actual uncertainty of each interrogation window instead of assuming 0.75 pixels, but due to missing data and blank spots on the uncertainty in pixels contour plot this isn’t possible.

![Contour plot of percentage uncertainty.](image)

**Figure 6.6 Contour plot of percentage uncertainty.**

Once the percentage velocity uncertainty is known, it is possible to determine the U and V velocity uncertainties for each interrogation window. If the angle of the velocity magnitude is known, this can also be done using the velocity magnitude and trigonometric relationships. As mentioned in the uncertainty analysis fundamentals section, the uncertainty of the vorticity can be calculated using an error propagation technique.
Finding the derivative of vorticity with respect to the U and V velocities in MATLAB required the use of writing a form of a central differencing scheme. In the differencing scheme, the derivative of a velocity in an interrogation window is approximated to equal to the difference of the velocity of the windows next to it divided by the difference in length between the neighboring windows. Vorticity can be calculated in the following way.

\[
\omega_{i,j} = \frac{V_{i,j+1} - V_{i,j-1}}{2\Delta X} - \frac{U_{i+1,j} - U_{i-1,j}}{2\Delta Y} \tag{6.10}
\]

The next step is to find the derivative of vorticity with respect to the U and V velocities. Again this is done in a similar fashion.

\[
\frac{d\omega_{i,j}}{dV_{i,j+1}} = \frac{1}{2\Delta X} \tag{6.11}
\]

\[
\frac{d\omega_{i,j}}{dV_{i,j-1}} = \frac{-1}{2\Delta X} \tag{6.12}
\]

\[
\frac{d\omega_{i,j}}{dU_{i+1,j}} = \frac{1}{2\Delta Y} \tag{6.13}
\]

\[
\frac{d\omega_{i,j}}{dU_{i-1,j}} = \frac{-1}{2\Delta Y} \tag{6.14}
\]

Using these estimated values for the derivative of vorticity with respect to a velocity field, the uncertainty in vorticity can be calculated using the propagation of error technique.

\[
\omega'_{i,j} = \sqrt{\left(\frac{d\omega_{i,j}}{dV_{i,j+1}} \cdot V'\right)^2 + \left(\frac{d\omega_{i,j}}{dV_{i,j-1}} \cdot V'\right)^2 + \left(\frac{d\omega_{i,j}}{dU_{i+1,j}} \cdot U'\right)^2 + \left(\frac{d\omega_{i,j}}{dU_{i-1,j}} \cdot U'\right)^2} \tag{6.15}
\]
Substituting in values for the derivatives allows for a simplified equation.

\[
\omega'_{i,j} = \sqrt{\left( \frac{V'}{2\Delta X} \right)^2 + \left( \frac{-V'}{2\Delta X} \right)^2 + \left( \frac{U'}{2\Delta Y} \right)^2 + \left( \frac{-U'}{2\Delta Y} \right)^2}
\]  

(6.16)

For convenience the vorticity percent uncertainty is defined as the vorticity in the interrogation window divided by the maximum vorticity in the test section. Figure 6.7 shows a contour plot of the vorticity percent uncertainty. Again as can be expected it is shown that the regions with low velocity have the most uncertainty, and the high velocity regions have the lowest uncertainty levels.

![Contour plot of percentage uncertainty of vorticity.](image)

Figure 6.7 Contour plot of percentage uncertainty of vorticity.

In a similar fashion to vorticity, the uncertainties associated with strain rate calculations can be derived.

\[
\frac{de_{xx,i,j}}{dU_{i+1,j}} = \frac{1}{\Delta X}
\]  

(6.17)

\[
\frac{de_{xx,i,j}}{dU_{i-1,j}} = \frac{-1}{\Delta X}
\]  

(6.18)
\[ e'_{xx\ ij} = \sqrt{\left(\frac{d_{xx\ ij}}{dU_{i+1\ j}} \cdot U'\right)^2 + \left(\frac{d_{xx\ ij}}{dU_{i-1\ j}} \cdot U'\right)^2} = \sqrt{\left(\frac{U'}{\Delta X}\right)^2 + \left(-\frac{U'}{\Delta X}\right)^2} \quad (6.19) \]

\[ \frac{d_{yy\ ij}}{dV_{i+1\ j}} = \frac{1}{\Delta Y} \quad (6.20) \]

\[ \frac{d_{yy\ ij}}{dV_{i-1\ j}} = \frac{-1}{\Delta Y} \quad (6.21) \]

\[ e'_{yy\ ij} = \sqrt{\left(\frac{d_{yy\ ij}}{dV_{i+1\ j}} \cdot V'\right)^2 + \left(\frac{d_{xx\ ij}}{dV_{i-1\ j}} \cdot V'\right)^2} = \sqrt{\left(\frac{V'}{\Delta Y}\right)^2 + \left(-\frac{V'}{\Delta Y}\right)^2} \quad (6.22) \]

\[ \frac{d_{xy\ ij}}{dV_{i+1\ j}} = \frac{1}{2\Delta X} \quad (6.23) \]

\[ \frac{d_{xy\ ij}}{dV_{i-1\ j}} = \frac{-1}{2\Delta X} \quad (6.24) \]

\[ \frac{d_{xy\ ij}}{dU_{i+1\ j}} = \frac{1}{2\Delta Y} \quad (6.25) \]

\[ \frac{d_{xy\ ij}}{dU_{i-1\ j}} = \frac{-1}{2\Delta Y} \quad (6.26) \]

\[ e'_{xy\ ij} = \sqrt{\left(\frac{d_{xy\ ij}}{dV_{i+1\ j}} \cdot V'\right)^2 + \left(\frac{d_{xx\ ij}}{dV_{i-1\ j}} \cdot V'\right)^2 + \left(\frac{d_{xy\ ij}}{dU_{i+1\ j}} \cdot U'\right)^2 + \left(\frac{d_{xy\ ij}}{dU_{i-1\ j}} \cdot U'\right)^2} = \sqrt{\left(\frac{V'}{2\Delta X}\right)^2 + \left(-\frac{V'}{2\Delta X}\right)^2 + \left(\frac{U'}{2\Delta Y}\right)^2 + \left(-\frac{U'}{2\Delta Y}\right)^2} \quad (6.27) \]
Figure 6.8 Contour plot of percentage uncertainty of strain rate in the x-x plane.

Figure 6.9 Contour plot of percentage uncertainty of strain rate in the y-y plane.
Figure 6.10 Contour plot of percentage uncertainty of strain rate in the x-y plane.

Figure 6.11 Contour plot of uncertainty in pixels for the bluff body.
Figure 6.12 Contour plot of percentage uncertainty of velocity.

Figure 6.13 Contour plot of percentage uncertainty of vorticity.
Figure 6.14 Contour plot of percentage uncertainty of strain rate in the x-x plane.

Figure 6.15 Contour plot of percentage uncertainty of strain rate in the y-y plane.
Figure 6.16 Contour plot of percentage uncertainty of strain rate in the x-y plane.
7. Conditioned Particle Image Velocimetry

It many combustion applications it can be desirable to find the edge of the flame. Typically this is done using OH planar laser induced fluorescence (OH PLIF). When OH PLIF is not readily available then a technique called Conditioned Particle Image Velocimetry (CPIV) can be used. CPIV takes advantage of the fact that in flame regions or in hot/cold boundaries the density is much less and the seed density is lowered as well. The lowered seed density is captured in the raw Mie scattering images and can be used to extract the flame edge or the boundary between a hot and cold flow.

7.1 Conditioned Particle Image Velocimetry Method and Results

To begin a CPIV analysis, it is necessary to examine the raw image pair to be used. Of the pair the first image is usually used because it has the least exposure time to the laser pulse and therefore has less interference from chemiluminescence. The two images can be separated using the data field produced from the readimx function (4.1). The “variable.Data” field contains a matrix of intensities which has the same number of rows as a single image but twice the number of columns. Basic programming logic can be used to separate the intensities from the two images, as seen in the appendix. The matrix read from MATLAB is inverted, inverting the intensity matrix will bring it right side up. As can be seen in Figure 7.A, the image is easier viewed by trimming the top and bottom of the image so that only areas in the test sections can be seen. There are a number of ways to do this, but in this analysis a new matrix was defined that only contained the test section. For details refer to the MATLAB program in the appendix.
Figure 7.A Raw Mie Scattering image.

Figure 7.B Raw image produced in MATLAB.
Due to scattering, the laser power decreases with increasing path length. It is necessary to correct the image to compensate for the laser power differences. This can be done by linearly equalizing the mean intensity of the bottom of the image so that it is equal to the mean intensity of the top of the image. The image then needs to be normalized by the mean intensity value of the image to give consistent values for each image pair. To do this, a region on the top of the image and a region on the bottom of the image need to be defined. For example, the Y-span vectors can be defined to be from 1 to 100 on the bottom, and from 500 to 600 for the top (if 600 is close to the top of the image). The X-span vector can be defined to be from 1 to the size of the column. After both regions are defined as two matrices of intensities, it is necessary to find the mean value of each one.

\[
\begin{align*}
\text{Int}_T &= \text{Mean} \left( \text{Mean} \left( I(\text{yspan}_T, \text{xspan}_T) \right) \right) \\
\text{Int}_B &= \text{Mean} \left( \text{Mean} \left( I(\text{yspan}_B, \text{xspan}_T) \right) \right)
\end{align*}
\]

Figure 7.C Sample selection of top and bottom regions for linear normalization.

In equation 6.1, IntT is the mean intensity for the top, IntB is the mean intensity for the bottom, I is the intensity matrix, yspanT and yspanB are the Y-spans for the top and bottom regions defined earlier, and xspan is the X-span defined earlier.

\[
\begin{align*}
\text{Int}_T &= \text{Mean} \left( \text{Mean} \left( I(\text{yspan}_T, \text{xspan}_T) \right) \right) \\
\text{Int}_B &= \text{Mean} \left( \text{Mean} \left( I(\text{yspan}_B, \text{xspan}_T) \right) \right)
\end{align*}
\]
Once the mean intensities for the top and bottom regions are found, it is time to linearly normalize them. Two more variables are defined to represent the corrected top intensity (IcT) and the corrected bottom intensity (IcB), as shown in Equation 7.2.

\[
IcT = \frac{IntT}{IntT} \quad IcB = \frac{IntT}{IntB} \quad (7.2)
\]

This way the mean of the top region is normalized to equal 1 and the mean of the bottom region is normalized to equal a fraction between 0 and 1. The goal from here is to find an equation for a line that connects these two mean values. Equation 7.3 calculates the slope and vertical intercept of this line.

\[
Icm = \frac{IcT - IcB}{Mean(yspanT) - Mean(yspanB)} \quad IcB = IcT - Icm * Mean(yspanT) \quad (7.3)
\]

Now that a line representing the normalized means throughout the image has been created, the line needs to be applied to the original intensity matrix, effectively normalizing all of the intensity values. From there the result is an image which is linearly normalized by the top and bottom intensities. From there the image needs to be normalized by the mean value. The best way to do this is to find the average values from the top and bottom regions defined earlier, and then to average them. Every value in the intensity matrix should be normalized by this value. The resulting image is shown in Figure 7.D.
Next it is important to define a matrix of intensities to be the region of interest. The purpose of the region of interest is to create a small area where it is clear to see the difference in intensities between the hot and the cold regions of the flow. It is necessary in MATLAB in order to speed the program processing time. The following image shows an example of a region of interest chosen. Figure 7.E shows a sample region of interest chosen which contains both reacting and non-reacting regions.
Figure 7.E Sample region of interest chosen containing flame and non-reacting regions.

Because the region of interest contains both reacting and non-reacting regions, the intensities it contains should vary as such. They are most easily computed using a histogram plot in MATLAB.

\[ \text{hist(Values, # of bins)} \]

For convenience in this calculation, the matrix of intensities in the region of interest was converted into a vector containing all of the rows. The histogram was done using this vector of intensities and 100 bins.
Figure 7. F Histogram produced from the region of interest.

From the histogram plot it is clear to see that there are two prominent peaks. The first peak, at high intensities, corresponds to areas with no flame and high density, and the second peak at low intensity corresponds to the areas where the flame exists and the density is low. It is necessary from here to determine an intensity value which shall be determined to be the edge of the hot/cold boundary. The value is referred to as the global reactedness threshold (GRT) and can be determined by either finding the average intensity between the two peaks, or more appropriately by the value of the minimum between the two peaks. Finding the minimum value between the two peaks is most easily performed by writing a code to begin at the maximum value of the first peak and to examine the values of the following peaks until they stop decreasing and begin to increase. Again, for more details refer to the MATLAB code attached in the appendix. Once the GRT is defined it can be used to further separate the image into reacting and non-reacting regions. A code is used to evaluate every pixel value and redefine every value above the GRT to be equal to 1, and every value below the GRT equal to 0. From here the
image needs to be filtered with a Gaussian filter. The easiest way to do this is using a built-in function in MATLAB as shown in Equation 7.4.

\[
\text{Variable} = \text{imfilter}(I, fspecial('gaussian', filter size, standard deviation)} \quad (7.4)
\]

The equation for a Gaussian filter is displayed in Equation 7.5 (Tuttle, et al., 2013).

\[
h_{i,j} = \frac{\exp\left(-\frac{i^2 + j^2}{2\sigma^2}\right)}{\sum_{i}^{N} \sum_{j}^{N} \exp\left(-\frac{i^2 + j^2}{2\sigma^2}\right)} \quad (7.5)
\]

![Figure 7.G Resulting filtered image.](image)

A similar analysis was performed for the bluff body. The shortened raw image used in this analysis is shown below.
Figure 7.1 Raw Mie Scattering image.

For simplicity the bluff body was removed from the image as to not interfere with the intensity of the burned and unburned regions. From PDF, the GRT was found and the image intensities were edited about that point. After the Gaussian filter was performed the burned and unburned regions can be seen. The flame edge is more visibly seen using a contour plot.
Figure 7.J Resulting filtered image.
Figure 7. K Contour plot of flame edge.

7.2 Conditioned Particle Image Velocimetry Alternative Filter Method

Again as can be seen with the previous images there is some uncertainty as to the location of the flame edge due to the Gaussian filter which changes the binarized image into a continuous function of intensity values. In an alternative method it can be valuable to perform the Gaussian filter before continuing with the rest of the analysis. After the image is shortened to the desired size, the Gaussian filter is performed and a PDF of the intensity is then found. The following histogram is for a jet in cross-flow image.
Figure 7.12 Histogram produced from region of interest.

The histogram produced is not as clean as the one produced before, but it can be seen that there is a minimum value at about 1.2, which is the minimum value selected in the previous analysis. When using this as the GRT and redefining the intensities below and above that threshold, the plot looks much cleaner and has a more defined flame edge.

When the same plot is applied to the bluff body image the results are similar. The histogram again shows a minimum between the two peaks of about 1.2.
Figure 7.14 Histogram produced from region of interest.

The produced contour plot has a defined flame edge, but is not very clean. The red values in the unburned region can be attributed to possible uneven seeding conditions or seed build up on the wall.

Figure 7.15 Resulting contour plot showing the flame edge.
It can be concluded that CPIV is a useful technique for estimating the flame edge, but as Tuttle describes, it is not as accurate as planar laser induced fluorescence. (Tuttle, et al., 2013) None the less, its convenience makes it an important technique to consider for studies where the location of the flame edge is desired.
8. Conclusions and Future Work

8.1 Conclusions

Particle Image Velocimetry (PIV) was performed on a bluff body flame holder and a jet in cross-flow flame holder. A tutorial on how to use MATLAB software to process the PIV data is given for the benefit of future students. The bluff body experiment was modified to improve the boundary conditions which were verified using PIV.

An uncertainty analysis was performed on the PIV images and it was found that the typical uncertainty in the cross-correlation peak was about 0.75 pixels. This results in a 10% velocity error in the cross-flow of about 10%, an error less than 10% in the jet, and an error of 80% or above in the recirculation zone. The high error corresponds to not enough seed entering the flow in the area. The propagation of error analysis showed the vorticity and strain rate uncertainties as well. The uncertainties were divided by the highest vorticity or strain rate value, which explains why the percent error for vorticity and strain rate is lower than that for velocity. Vorticity error was found to be about 7% in the cross-flow, jet, and some areas of the recirculation zone. The uncertainty of strain rate in the x-x direction tends to be about 14% in the cross-flow and about 4% in the jet and parts of the recirculation zone. The uncertainty of strain rate in the y-y direction tends to be about 2% in the cross-flow and up to 20% in the jet and parts of the recirculation zone. The uncertainty in the x-y plane tends to be about 12% for the cross-flow, the jet and the recirculation zone.

A MATLAB code for Conditioned Particle Image Velocimetry (CPIV) was used to find the edge of a flame in the case of bluff body studies or a hot/cold boundary in the case of the air jet-in-
crossflow. The code was written in a user-friendly fashion such that future students should be able perform CPIV and use it to find fluid properties at the flame edge.

8.2 Future Work

There are many areas of future work that are recommended to further the knowledge of these particular flame holders. It would be interesting to perform PIV measurements with differently shaped bluff body flame holders. In practice the geometric flame holder is often shaped like an elliptical with a long trailing edge.

Much of the desired data from the bluff body flame holder is located in the recirculation zone. Unfortunately due to the low density of the fluids in the flame it is very difficult to properly seed the recirculation zone. A recommendation would be to develop a geometric flame holder that is capable of seeding from the trailing edge.

Also in this paper only one single round jet configuration was tested. There are so many different jet in cross-flow combinations that have not been tested in this laboratory. This could also be done with cross-flows of different temperatures to see how temperature and excess air affect the jet.

It would also be interesting to perform PIV simultaneously with OH-PLIF. This would allow the researchers to see the velocity profiles of the flame and to also to see the flame edge. CPIV can
be performed in the mean time to roughly estimate the flame edge, PLIF is a far better measure.
Works Cited


%This code is used to find the vector plots, streamlines and vorticity plots for a .VC7 image from DaVis. It also finds average vorticity stats

%Clear Command Window***************************************************
clear
clc
close all
%**********************************************************************

%Read PIV Image*************************************************************

    JJ=0; % 1 for sample
    J=25;
    liney1=-17;
    liney2=-20;
    if J==25
        linex=5.56;
    elseif J==50
        linex=10.4496;
    elseif J==75
        linex=10.857;
    end

    if J==25
        if JJ==1
            [A]=readimx('C:\Users\cpp05001\Documents\MATLAB\ThesisPIV\JICF\reacting\Seeded_jet_J25_Phi1.65\Trial_5\Diff\PIV_MP(32x32_50%ov_ImgCorr)_PostProc\B00010.VC7');
        else
            [A]=readimx('C:\Users\cpp05001\Documents\MATLAB\ThesisPIV\JICF\reacting\Seeded_jet_J25_Phi1.65\Trial_5\Diff\PIV_MP(32x32_50%ov_ImgCorr)_PostProc\TimeMeanQF_Vector\B00001_Avg V.VC7');
        end
    elseif J==50
        if JJ==1
            [A]=readimx('C:\Users\cpp05001\Documents\MATLAB\ThesisPIV\JICF\reacting\Seeded_jet_J50_Phi1.65\Trial_3\Diff\PIV_MP(32x32_50%ov_ImgCorr)_PostProc\B00005.VC7');
        else
            [A]=readimx('C:\Users\cpp05001\Documents\MATLAB\ThesisPIV\JICF\reacting\Seeded_jet_J50_Phi1.65\Trial_3\Diff\PIV_MP(32x32_50%ov_ImgCorr)_PostProc\TimeMeanQF_Vector\B00001_Avg V.VC7');
        end
    elseif J==75
        if JJ==1
            [A]=readimx('C:\Users\cpp05001\Documents\MATLAB\ThesisPIV\JICF\reacting\Seeded_jet_J75_Phi1.65\Trial_3\Diff\PIV_MP(32x32_50%ov_ImgCorr)_PostProc\B00050.VC7');
        else
            [A]=readimx('C:\Users\cpp05001\Documents\MATLAB\ThesisPIV\JICF\reacting\Seeded_jet_J75_Phi1.65\Trial_3\Diff\PIV_MP(32x32_50%ov_ImgCorr)_PostProc\TimeMeanQF_Vector\B00001_Avg V.VC7');
        end
    else
    end
end
end
[X,Y,U,V] = showimx(A);

%Resize Image to size of test section and Invert**********************
[rowsize, colsize] = size(X);
K = 1;
for i = 1: colsize
    if Y(1,i) > 19 || Y(1,i) < -19
        remove(K) = i;
        K = K + 1;
    end
end
X(:, remove) = []; Y(:, remove) = []; U(:, remove) = []; V(:, remove) = [];
X = X'; Y = Y'; U = U'; V = V';

% PLOT VECTOR FIELD**************************************************
figure
vfield(X, Y, U/10, V/10, U);
xlabel('mm')
ylabel('mm')
c = colorbar;
ylabel(c, 'm/s')
title('J=25 Average')
line([linex; linex], [liney1; liney2])
line([linex-9.525; linex-9.525], [liney1; liney2])

% Plot Streamlines**************************************************
figure
vfield(X, Y, U/10, V/10, U);
xlabel('mm')
ylabel('mm')
title('J=25 Average')
line([linex; linex], [liney1; liney2])
line([linex-9.525; linex-9.525], [liney1; liney2])

startx = -10.*ones(1, 7);
starty = -15:5:15;
h = streamline(X, Y, U, V, startx, starty);

startx = 10*ones(1, 7);
starty = -15:5:15;
i = streamline(X, Y, U, V, startx, starty);

startx = 20*ones(1, 7);
starty = -15:5:15;
j = streamline(X, Y, U, V, startx, starty);
startx=25*ones(1,7);
starty=-15:5:15;
k=streamline(X,Y,U,V,startx,starty);

startx=30*ones(1,7);
starty=-15:5:15;
l=streamline(X,Y,U,V,startx,starty);

startx=35*ones(1,7);
starty=-15:5:15;
m=streamline(X,Y,U,V,startx,starty);

startx=40*ones(1,7);
starty=-15:5:15;
n=streamline(X,Y,U,V,startx,starty);

startx=0:2:4;
starty=-15*ones(1,3);
o=streamline(X,Y,U,V,startx,starty);

%*******************************************************************
%Calculate Derivatives**********************************************
X=X/1000;
Y=Y/1000;
[rowsize,colsiz]=size(U);
delx=X(1,2)-X(1,1);
dely=Y(1,1)-Y(2,1);

%Calculate the derivative of U and V WRT Y*************************
for i=1:rowsize
    for j=1:colsiz
        if i==1
            dUdY(i,j)=(U(i+1,j)-U(i,j))/delx;
            dVdY(i,j)=(V(i+1,j)-V(i,j))/delx;
        elseif i==rowsize
            dUdY(i,j)=(U(i,j)-U(i-1,j))/delx;
            dVdY(i,j)=(V(i,j)-V(i-1,j))/delx;
        else
            dUdY(i,j)=(U(i+1,j)-U(i-1,j))/(2*delx);
            dVdY(i,j)=(V(i+1,j)-V(i-1,j))/(2*delx);
        end
    end
end
%This calculates the derivative of U and V WRT X
for i=1:rowsize
    for j=1:colsiz
        if j==1
            dUdX(i,j)=(U(i,j+1)-U(i,j))/dely;
            dVdX(i,j)=(V(i,j+1)-V(i,j))/dely;
        elseif j==colsiz
            dUdX(i,j)=(U(i,j)-U(i,j-1))/dely;
            dVdX(i,j)=(V(i,j)-V(i,j-1))/dely;
        else
            dUdX(i,j)=(U(i+1,j)-U(i-1,j))/(2*dely);
            dVdX(i,j)=(V(i+1,j)-V(i-1,j))/(2*dely);
        end
    end
end
dUdX(i,j) = (U(i,j+1) - U(i,j-1)) / (2*delx);
dVdX(i,j) = (V(i,j+1) - V(i,j-1)) / (2*delx);

end
end
end

%*********************************************************************
%Calculate and Plot Vorticity****************************************

W = dVdX - dUdY;  %Calculate Vorticity (1/s)
X = X*1000;
Y = Y*1000;
figure
contourf(X,Y,W)
line([linex;linex],[liney1;liney2])
line([linex-9.525;linex-9.525],[liney1;liney2])
c=colorbar;
ylabel(c,'1/s')
title('J=75 Sample')
xlabel('mm')
ylabel('mm')
%**********************************************************************

%Average Vorticity and Yield Average Vorticity*************************
%Average Vorticity Along X-Direction
sum=0;
delY=Y(1,1)-Y(2,1);
[rowsize,colsize]=size(W);
for k = 1:colsize
    for L = 1:rowsize
        sum=sum+W(L,k);
    end
    wx(1,k) = sum*delY;
    sum=0;
end
figure, plot(X,wx)
title('Average Vorticity Along X-Direction')
xlabel('X mm')
ylabel('Vorticity (1/s)')

%Average Vorticity Along Y-Direction
sum=0;
delX=X(1,2)-X(1,1);
[rowsize,colsize]=size(W);
for k=1:rowsize
    for L=1:colsize
        sum=sum+W(k,L);
    end
    wy(1,k)=sum*delX;
    sum=0;
end
figure, plot(Y,wy)
title('Average Vorticity Along Y-Direction')
xlabel('Y mm')
ylabel('Vorticity (1/s)')

%Field Averaged Vorticity
sum=0;
for j=1:colsizex
    for i=1:rowsize
        sum = sum+abs(W(i,j));
    end
end
FAV=delX*delY*sum;
Xvec = X(1,:);
Yvec = Y(:,1);
Wvec = trapz(Yvec,W);
Wbar = trapz(Xvec,Wvec);

% Strain Rate Equations and Plots
exx=dUdX;
exy=0.5*(dUdY+dVdX);
eyx=exy;
eyy=dVdY;

figure
contourf(X,Y,exx)
c=colorbar;
ylabel(c,'1/s')
title('J=25 Average e_x_x')
xlabel('mm')
ylabel('mm')
figure
contourf(X,Y,eyy)
c=colorbar;
ylabel(c,'1/s')
title('J=25 Average e_y_y')
xlabel('mm')
ylabel('mm')
figure
contourf(X,Y,exy)
c=colorbar;
ylabel(c,'1/s')
title('J=25 Average e_x_y')
xlabel('mm')
ylabel('mm')

%**********************************************************************
Appendix A2 – Uncertainty

%This code takes an .IM7 image from DaVis and finds the uncertainty in the measurement for each interrogation window.

%Clear Command Window*****************************************************************************
clear

count=0;
close all

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%Import Images%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
CorrelationMap=readimx('C:\Users\cpp05001\Documents\MATLAB\ThesisPIV\Correlation\J75Trial3Image1\CorrelationMap\B00001.im7');
count=count+1; figure(count), [xp,yp,I]=showimx(CorrelationMap);
[rowsize, colsize]=size(I);
count=0;
[B]=readimx('C:\Users\cpp05001\Documents\MATLAB\ThesisPIV\JICF\reacting\Seeded_jet_J75_Phi1.65\Trial_3\Diff\PIV_MP(32x32_50%ov_ImgCorr)_PostProc\B00001.VC7');
[X,Y,U,V]=showimx(B);
X=X'; Y=Y'; U=U'; V=V'; X=X/1000; Y=Y/1000;
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%Define Correlation Map into Individual Interrogation Windows%%%%%%%%%%%
%Row 1-8 are blank
%Row 61-64 are blank
unpix=zeros(60,80);
for b=1:80%5:10
column=b;
for a=1:64%14:1:54
count=count+1
row=a;

istart=1+33*(row-1);

iend=33*row;

jstart=1+33*(column-1);
jend=33*column;
a=0; b=1;
IW=zeros(33,33);
for i=istart:iend

a=a+1;
b=1;

for j=jstart:jend

IW(a,b)=I(i,j);
b=b+1;
end
end

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%Make A Contour Plot for******************************************
x=zeros(1,1); y=zeros(1,1);

for i=1:33 %defines x variable [X1 X2 X3; X1 X2 X3; X1 X2 X3]

for j=1:33

x(i,j)=j;

end
for j=1:33
    % defines y variable [Y3 Y3 Y3; Y2 Y2 Y2; Y1 Y1 Y1]
    for i=1:33
        y(i,j)=34-i;
    end
end

% figure(4), contour(x,y,IW) % makes contour plot
% colorbar % adds colorbar
% grid on % grids
% grid minor % minor axis grids
% xlabel('Pixels')
% ylabel('Pixels')
%**********************************************************************
% Find column and row of maximum value************************************
IW(33,1)=0; % This needs to be set to zero, Davis Makes it 1
MX=max(max(IW));
[num idx] = max(IW(:));
[rowmax colmax] = ind2sub(size(IW),idx);

% figure(5), plot([1:33],IW(rowmax,:))
% xlabel('Pixels')
% ylabel('Intensity')
%**********************************************************************
% Find Half Peak Width***************************************************
% find Half Peak
% Define Baseline
% Define Baseline as every pixel not within 2 pixels of peak.
var=5; % Used to create row with less terms
varpeak=2; % Used to define +- terms of peak
xx=x(rowmax,:);
y=IW(rowmax,xx);

% Finds the Peak baseline
yy=y;
[size1,size2]=size(x);
    if colmax < varpeak+1
        unpix(row,column)=0;
        unpix2(row,column)=0;
    else
        for i=colmax-varpeak:colmax+varpeak
            yy(1,i)=0;
        end
    end
peakbase=sum(yy)/(size2-2*varpeak-1);

% Peak Height
HP=0.5*(max(y)-peakbase);
PeakHeight=max(y);
% Location of HP
HPYLoc=HP+peakbase;
% Linear Regression may help us estimate x value for HPY
% First Line Linear Reg using interp1 function
xbeg(1,1)=x(1,colmax-1);
xbeg(1,2)=x(1,colmax);
ybeg(1,1)=y(1,colmax-1);
ybeg(1,2)=y(1,colmax);
HPXLoc(1,1)=interp1(ybeg,xbeg,HPYLoc,'linear');
% Second Line Linear Regression using interp1 function
xbeg(1,1)=x(1,colmax+1);
xbeg(1,2)=x(1,colmax);
ybeg(1,1)=y(1,colmax+1); % ybeg(1,2)=y(1,colmax);
HPXLoc(1,2)=interp1(ybeg,xbeg,HPYLoc,'linear');

% Calculate the Uncertainty in Pixels
Unc(1,1)=colmax-HPXLoc(1,1);
Unc(1,2)=HPXLoc(1,2)-colmax;
unpix(row,column)=Unc(1,2); % + or - uncertainty in pixels
unpix2(row,column)=Unc(1,1);

figure
contourf(X,Y,unpix)
c=colorbar;
title('Uncertainty in Pixels')
xlabel('meters')
ylabel('meters')
ylabel(c,'Pixels')
% This code takes a .VC7 image from DaVis and calculates the corresponding 
% error in velocity and vorticity.

% Clear Command Window
clc
clear
close all

% Read in Vector Image
B=readimx('C:\Users\cpp05001\Documents\MATLAB\ThesisPIV\JICF\reacting\Seeded_jet_J25_Phil1.65\Trial_5\Diff\PIV_MP(32x32_50%ov_ImgCorr)_PostProc\B00004.VC7');
figure, [X,Y,U,V]=showimx(B);
X=X'; Y=Y'; U=U'; V=V'; % Invert to align vectors with interrogation windows
X=X/1000; Y=Y/1000; % Convert X&Y from mm to meters

% Determine Percent Error of Velocity
unpix=.75; % Uncertainty in Pixels
dtime=4*10^-6; % Time between images (seconds)
velmag=sqrt(U.^2+V.^2); % m/s % Velocity Magnitude
distance=velmag*dtime; % meters % Distance Particle Travels in IntWindow
pixdistance=6.7*10^-6; % meters/pixel % FROM DAVIS
undistance=unpix*pixdistance; % Uncertainty distance in meters
uncvelmag=undistance/dtime;
[rowsize,colsiz]=size(U);
uncpercent=100*undistance./distance;

% Eliminate Percentages over 100
for i=1:rowsize
    for j=1:colsiz
        if uncpercent(i,j)>100
            uncpercent(i,j)=100;
        end
    end
end
uncU=uncpercent.*U/100;
uncV=uncpercent.*V/100;

% Use Central Differencing Scheme to find Derivatives of velocity
[rowsize,colsiz]=size(U);
delx=X(1,2)-X(1,1);
dely=Y(1,1)-Y(2,1);
% Calculates the derivative of U and V WRT Y
for i=1:rowsize
    for j=1:colsiz
        if i==1
            dUdY(i,j)=(U(i+1,j)-U(i,j))/delx;
            dVdY(i,j)=(V(i+1,j)-V(i,j))/delx;
        elseif i==rowsize
            dUdY(i,j)=(U(i,j)-U(i-1,j))/delx;
            dVdY(i,j)=(V(i,j)-V(i-1,j))/delx;
        else

Appendix A3 – Propagation of Error
%This calculates the derivative of U and V WRT X
for i=1:rowsize
    for j=1:colsize
        if j==1
            dUdX(i,j)=(U(i,j+1)-U(i,j))/dely;
            dVdX(i,j)=(V(i,j+1)-V(i,j))/dely;
        elseif j==colsize
            dUdX(i,j)=(U(i,j)-U(i,j-1))/dely;
            dVdX(i,j)=(V(i,j)-V(i,j-1))/dely;
        else
            dUdX(i,j)=(U(i,j+1)-U(i,j-1))/(2*delx);
            dVdX(i,j)=(V(i,j+1)-V(i,j-1))/(2*delx);
        end
    end
end

%**********************************************************************

%Calculate Vorticity (1/s)**********************************************
W=dVdX-dUdY;
%**********************************************************************

%Error Propagation******************************************************
for i=1:rowsize
    for j=1:colsize
        if i==1 || j==1 || i==rowsize || j==colsize
            uncW(i,j)=0;
        else
            uncW(i,j)=sqrt((uncV(i,j+1)/(2*delx)).^2 + (uncV(i,j-1)/(2*delx)).^2 + (uncU(i+1,j)/(2*dely)).^2 + (uncU(i-1,j)/(-2*dely)).^2);
        end
    end
end
%************************************************************************

%Strain Rate Calculations**********************************************
%Calculate e'xx
exx=dUdX;
unexx=zeros(1,1);
for i=1:rowsize
    for j=1:colsize
        if i==1 || j==1 || i==rowsize || j==colsize
            unexx(i,j)=0;
        else
            unexx(i,j)=sqrt((uncU(i+1,j)/(delx)).^2 + (uncU(i-1,j)/(delx)).^2);
        end
    end
end
%Calculate e'yy
eyy=dVdY;
unceyy=zeros(1,1);
for i=1:rowsize
    for j=1:colsize
        if i==1 || j==1 || i==rowsize || j==colsize
            unceyy(i,j)=0;
        else
            unceyy(i,j)=sqrt((uncV(i,j+1)/(dely)).^2 ... 
            + (-1*uncV(i,j-1)/(dely)).^2);
        end
    end
end
%Calculate e'xy
exy=.5*(dUdY+dVdX);
uncexy=zeros(1,1);
for i=1:rowsize
    for j=1:colsize
        if i==1 || j==1 || i==rowsize || j==colsize
            uncexy(i,j)=0;
        else
            uncexy(i,j)=sqrt((uncV(i,j+1)/(2*delx)).^2 + ... 
            (-1*uncV(i,j-1)/(2*delx)).^2 + (uncU(i+1,j)/(2*dely)).^2 ... 
            + (uncU(i-1,j)/(-2*dely)).^2);
        end
    end
end
%**********************************************************************
%Propagation of Error
%Eliminates Outliers
%PercentW=100*abs(uncW./W);
PercentW=100*abs(uncW./(max(max(abs(W)))));
%PercentW=100*abs(uncW./abs(W));
% for i=1:rowsize
%     for j=1:colsize
%         if PercentW(i,j)>100
%             PercentW(i,j)=0;
%         end
%     end
% end

%Calculate and Plot Percent Uncertainty of Vorticity****************
PercentW=100*abs(uncW./(max(max(abs(W)))));
figure
contourf(X,Y,PercentW)
c=colorbar;
title('Vorticity Percent Uncertainty')
xlabel('meters')
ylabel('meters')
ylabel(c, 'Percent')

%Strain Rate Equations**************************************************************************
PercentExx = 100 * abs(uncexx ./ (max(max(abs(exx)))));
figure
contourf(X,Y,PercentExx)
c=colorbar;
title('e_x_x Percent Uncertainty')
xlabel('meters')
ylabel('meters')
ylabel(c,'Percent')

PercentEyy = 100 * abs(unceyy ./ (max(max(abs(eyy)))));
figure
contourf(X,Y,PercentEyy)
c=colorbar;
title('e_y_y Percent Uncertainty')
xlabel('meters')
ylabel('meters')
ylabel(c,'Percent')

PercentExy = 100 * abs(uncexy ./ (max(max(abs(exy)))));
figure
contourf(X,Y,PercentExy)
c=colorbar;
title('e_x_y Percent Uncertainty')
xlabel('meters')
ylabel('meters')
ylabel(c,'Percent')

%***********************************************************************
AA = uncpercent(:,:,1:Inf);
uncpercent = uncpercent.*AA;
% for i=1:rowsize
% for j=1:colsiz
% if uncpercent(i,j)>100
% uncpercent(i,j)=100;
% end
% end
% figure
% contourf(X,Y,uncpercent)
% xlabel('meters')
% ylabel('meters')
% c=colorbar;
% ylabel(c,'Percent')

%Plot Velocity Percent Error****************************************************************
figure
contourf(X,Y,uncpercent)
title('Velocity Percent Error Per Interrogation Window')
xlabel('meters')
ylabel('meters')
c=colorbar;
ylabel(c,'Percent')

%***********************************************************************

Appendix A4 – CPIV

132 | Page
This code takes a .IM7 image from DaVis and uses the CPIV technique to determine the Flame Edge

Clear Command Window
clc
clear
close all

Read in the Image to be used
A=readimx('C:\Users/cpp05001\Documents\MATLAB\ThesisPIV\JICF\reacting\Seeded_jet_U25_Phi1.65\Trial_5\Diff\B00020.IM7');

This code separates the first image from the second image
[rowsize, colsize]=size(A.Data);
for i=1:rowsize
    for j=1:0.5*colsize
        I1(i,j)=A.Data(i,j);
    end
end
for i=1:rowsize
    a=i; b=0;
    for j=0.5*colsize+1:colsize
        b=b+1;
        I2(a,b)=A.Data(i,j);
    end
end

Define the Intensity Matrix as first image and invert
I=I1;
I=I';

shorten the image
[rowsize, colsize]=size(I);
a=1; b=0;
for i=300:900
    b=0;
    for j=1:colsize
        b=b+1;
        W(a,b)=I(i,j);
    end
    a=a+1;
end
[rowsize, colsize]=size(W);
for i=rowsize:-1:601
    W(i,:)=[];
end
I=ones(1,1);
I=W;
[rowsize, colsize]=size(I);
% Gaussian Filter
\[
% \text{Gauss} = \text{imfilter}(I, \text{fspecial('gaussian',19,3))};
% \text{figure}, \text{imshow(Gauss)}
% \text{Gauss2} = \text{imfilter}(\text{Gauss}, \text{fspecial('gaussian',19,3))};
% I = \text{Gauss2};
\]

% This code makes power correction from top to bottom
\[
[\text{rowsize}, \text{colsize}] = \text{size(I)};
\text{yspanT} = 1:100;
\text{yspanB} = 500:600;
\text{xspan} = 1:\text{rowsize};
\text{IntT} = \text{Mean(Mean(I(\text{yspanT},xspan))});
\text{IntB} = \text{Mean(Mean(I(\text{yspanB},xspan))));}
\text{IcT} = \text{IntT/IntT}; \text{\% determines top mean to be 'correct', adjust bottom}
\text{IcB} = \text{IcT/IntB};
\text{Icm} = (\text{IcT-IcB})/(\text{Mean(yspanT)}-\text{Mean(yspanB)}); \text{\% calculates slope}
\text{Icb} = \text{IcT-Icm*Mean(yspanT)}; \text{\% calculates vertical intercept}
\text{Ic} = \text{Icm*(1:rowsize)+Icb};
\text{ImgIc} = \text{Ic'*ones(1, colsize).*double(I)};
\]

% Normalize by mean value
\[
\text{ImgIc} = \text{ImgIc./Mean(Mean([\text{ImgIc(yspanT,xspan); \text{ImgIc(yspanB,xspan)]]}})};
\text{I} = \text{ones(1,1)};
\text{I} = \text{ImgIc};
\text{x1} = 1; \text{x2} = \text{colsize};
\text{y1} = 1; \text{y2} = 100;
\text{y3} = 500; \text{y4} = 600;
\text{figure}, \text{imshow(I)}
\text{line([x1,x2],[y1,y1])}
\text{line([x1,x2],[y2,y2])}
\text{line([x1,x1],[y1,y2])}
\text{line([x2,x2],[y1,y2])}
\text{line([x1,x2],[y3,y3])}
\text{line([x1,x2],[y4,y4])}
\text{line([x1,x1],[y3,y4])}
\text{line([x2,x2],[y3,y4])}
\]

% Define a Region of Interest (ROI)
\[
\text{yspan} = 1:200;
\text{xspan} = 101:200;
\text{ROI} = I(\text{yspan}, \text{xspan});
\]

% Make a PDF to find Min and Max Peaks
\[
[\text{rowsize}, \text{colsize}] = \text{size(ROI)};
\text{a} = 0;
\text{for} \text{i} = 1:\text{rowsize}
    \text{for j} = 1:\text{colsize}
        \text{a} = \text{a} + 1;
        \text{PDLine(1,a)} = \text{ROI(i,j)};
    \end{for}
\end{for}
end
end

bins=100;
figure, hist(PDFLine,bins)
ylabel('Frequency')
xlabel('Intensity')
title('Histogram Plot of Pixel Intensity')
[N,X]=hist(PDFLine,bins);

%*****************************************************************
%Find GRT**********************************************************
[rowsize,colsiz]=size(N);
j=1;
w=0;
while w==0;
a=N(1,j);
b=N(1,j+1);
if a<b
    j=j+1;
else
    firstmax=j;
    w=1;
end
end
w=0;
a=0; b=0;
j=firstmax;
while w==0
    a=N(1,j);
b=N(1,j+1);
    if a>b
        j=j+1;
    else
        minimum=j;
        w=1;
    end
end
GRT=X(1,j)

%****************************************************************
%GRT, set values to 1 and 0**************************************
[rowsize,colsiz]=size(I);
for i=1:rowsize
    for j=1:colsiz
        if ge(I(i,j),GRT) %ge => greater or equal to
            I(i,j)=0;
        else
            I(i,j)=1;
        end
    end
end
%*********************************************************************
%Gaussian Filter*****************************************************
Gauss=imfilter(I,fspecial('gaussian',19,3));
figure, imshow(Gauss)
Gauss2=imfilter(Gauss,fspecial('gaussian',19,3));
I=Gauss2;

%********************************************************************
%Flip Image Upside Down**********************************************
[rowsize, colsize]=size(I);
Iflip=zeros(rowsize,colsize);
a=0;
for i=rowsize:-1:1
  a=a+1;
  b=0;
  for j=1:colsize
    b=b+1;
    Iflip(a,b)=I(i,j);
  end
end
I=Iflip;
figure

%********************************************************************
contourf(I)
Appendix A5 – High Speed Images

%This code takes in an AVI file from a high speed camera and
%seperates the images for further processing

%Clear Command Window************************************************
clc
clear
close all
%********************************************************************

%Bring in movie******************************************
mov = aviread('C:\Users\cpp05001\Documents\MATLAB\HighSpeed\03_19_13\J30_Ph0.8_300C
HJ_920CCF_Trial 1.avi');
%*****************************************************************

%Find number of frames in the movie************************
numberOfframes=size(mov,2)
%****************************************************************

%Define Variables Used********************************************
% % [Variable,map] = frame2im(move(frame#))
%Find the size of an individual frame
[image1,map]=frame2im(mov(1));
[sizeY,sizeX]=size(image1);
%Initialize imagesum variable and set it to zero
imagesum=zeros(sizeY,sizeX);
%****************************************************************

%Begins a loop to define image variables, and to add the image******
for counter=56:57
string='image';
 imvar=[string,int2str(counter)];
 [imvar,map]=frame2im(mov(counter));
for x=1:sizeX
 for y=1:sizeY
   imagesum(y,x)=imagesum(y,x)+imvar(y,x);
 end
end
aveimage=imagesum/counter;
im2double(aveimage);
imshow(aveimage)
colorbar
colormap(map)
%****************************************************************

%This code takes in an AVI file from a high speed camera and
%seperates the images for further processing

%Clear Command Window************************************************
clc
clear
close all
%********************************************************************

%Bring in movie******************************************
mov = aviread('C:\Users\cpp05001\Documents\MATLAB\HighSpeed\03_19_13\J30_Ph0.8_300C
HJ_920CCF_Trial 1.avi');
%*****************************************************************

%Find number of frames in the movie************************
numberOfframes=size(mov,2)
%****************************************************************

%Define Variables Used********************************************
% % [Variable,map] = frame2im(move(frame#))
%Find the size of an individual frame
[image1,map]=frame2im(mov(1));
[sizeY,sizeX]=size(image1);
%Initialize imagesum variable and set it to zero
imagesum=zeros(sizeY,sizeX);
%****************************************************************

%Begins a loop to define image variables, and to add the image******
for counter=56:57
string='image';
 imvar=[string,int2str(counter)];
 [imvar,map]=frame2im(mov(counter));
for x=1:sizeX
 for y=1:sizeY
   imagesum(y,x)=imagesum(y,x)+imvar(y,x);
 end
end
aveimage=imagesum/counter;
im2double(aveimage);
imshow(aveimage)
colorbar
colormap(map)
%****************************************************************