Coupled Fluid-Structure Interaction Modeling of a Parafoil

Brandon Burnett

Follow this and additional works at: https://commons.erau.edu/edt

Part of the Aerospace Engineering Commons, and the Computational Engineering Commons

Scholarly Commons Citation
https://commons.erau.edu/edt/294

This Thesis - Open Access is brought to you for free and open access by Scholarly Commons. It has been accepted for inclusion in Dissertations and Theses by an authorized administrator of Scholarly Commons. For more information, please contact commons@erau.edu.
COUPLED FLUID-STRUCTURE INTERACTION

MODELING OF A PARAFOIL

A Thesis

Submitted to the Faculty

of

Embry-Riddle Aeronautical University

by

Brandon Burnett

In Partial Fulfillment of the

Requirements for the Degree

of

Master of Science in Aerospace Engineering

December 2016

Embry-Riddle Aeronautical University

Daytona Beach, Florida
COUPLED FLUID-STRUCTURE INTERACTION
MODELING OF A PARAFOIL

by

Brandon Burnett

A Thesis prepared under the direction of the candidate’s committee chairman, Dr. Richard Anderson, Department of Aerospace Engineering, and has been approved by the members of the thesis committee. It was submitted to the School of Graduate Studies and Research and was accepted in partial fulfillment of the requirements for the degree of Master of Science in Aerospace Engineering.

THESIS COMMITTEE

Chairman, Dr. Richard ‘Pat’ Anderson

Member, Dr. Scott Martin

Member, Dr. Mark Ricklick

Member, Dr. John Ekaterinatis

Graduate Program Coordinator, Dr. Magdy Attia

Dean of College of Engineering, Dr. Maj Mirmirani

Vice Chancellor, Academic Support, Dr. Christopher Grant

12-6-2016

12/8/16

12/8/16
ACKNOWLEDGMENTS

I would like to thank the entire faculty and staff of Embry-Riddle’s Eagle Flight Research Center for giving me the opportunity, experience and pleasure of working on very diverse and amazing programs throughout the years with such great people. I would like to give a special thanks to Dr. Richard ‘Pat’ Anderson for his exceptional teachings as professor, mentor and advisor during my time at EFRC as well as always having an open door for advice. Without his support, I likely would not be working as a flight controls engineer at Rockwell Collins today. I would also like to give a special thanks to Shirley Koelker for making every moment walking into the Eagle Flight Research Center an awesome and fun experience and treating me like family. I would like to thank Professor Glenn Greiner for sharing his wealth of engineering knowledge and expertise in aerospace over the years, and I would like to thank Dr. Scott Martin for always having an open door for advice as well as having the ultimate faith in his students for test flying an airplane that was dissembled and reassembled by graduate students. I would also like to thank Dr. Borja Martos for always providing positive support, friendly advice and providing astonishing knowledge in flight control testing. Special thanks to Dr. Mark Ricklick and Dr. John Ekaterinaris for joining my committee on such a short notice and helping me finish this thesis. I would like to thank Renjana Hareendran for all of her patience in dealing with the long hours of writing this thesis paper and most of all, my mother Robin and father John, for believing in me and providing unconditional support in all my endeavors to help me complete my work in aerospace engineering.
# TABLE OF CONTENTS

LIST OF TABLES ......................................................................................... v

LIST OF FIGURES ..................................................................................... vi

SYMBOLS ................................................................................................. xi

ABBREVIATIONS ....................................................................................... xii

ABSTRACT .................................................................................................. xiii

1. Introduction .......................................................................................... 1

1.1. Problem Statement ........................................................................... 2

1.2. Objective of Study ............................................................................ 2

1.3. Limitation of Study .......................................................................... 3

1.4. Organization of the Paper ................................................................. 3

2. Literature Review .................................................................................. 4

2.1. Past Methodologies .......................................................................... 9

2.2. Hypothesis ....................................................................................... 35

3. Methodology .......................................................................................... 36

3.1. Strategy ........................................................................................... 36

3.2. Engineering Tools ............................................................................ 39

3.3. Program Workflow .......................................................................... 40

3.4. Geometric Modeling ........................................................................ 42

3.5. Fabric / Material Modeling ............................................................... 59

3.5.1. Uniaxial Tensile Test ................................................................. 61

3.5.2. Biaxial Tensile Test ................................................................. 66

3.6. Fluid Structure Interaction Model .................................................... 75

3.6.1. Computational Fluid Dynamics Model ..................................... 77

3.6.2. Finite Element Analysis Model ................................................. 90

3.6.3. System Coupling ....................................................................... 95

3.7. Experimental Model ...................................................................... 102

4. Analysis .................................................................................................. 108

4.1. Experimental Results ..................................................................... 108

4.2. FSI Model ..................................................................................... 111

4.3. FSI and Experiment Comparison ................................................... 121

5. Conclusion and Recommendations ..................................................... 124

5.1. Conclusion ..................................................................................... 124

5.2. Recommendations and Further Study ............................................ 125

References .................................................................................................. 127

A. Experimental Specimen – Single–Cell Parafoil .................................... 130
LIST OF TABLES

Table 1: Ring-slot Material Properties (Xing-long, Qing-bin, Qian-gang, Tao, 2013) .... 32
Table 2: Modeling Tools ........................................................................................................ 39
Table 3: Re-meshing Limitations .......................................................................................... 88
LIST OF FIGURES

Figure 1: Parafoil Parachute (Performance Designs, Inc.) ............................................................... 2
Figure 2: Traditional Parachute Design ............................................................................................ 4
Figure 3: Experimental Drop Test (Gargana) .................................................................................. 4
Figure 4: CFD Parachute Design ...................................................................................................... 6
Figure 5: ANSYS Leaf Valve Artery (Scampoli) ............................................................................ 7
Figure 6: Aeroelastics on a Wing (Goodwin) .................................................................................. 8
Figure 7: Basic Parafoil Parachute Model (Lacroix, Bordenave, 1999) ........................................... 9
Figure 8: Fluid-Structural Mesh (Baskut, Akgul, 2011) ................................................................. 10
Figure 9: Iterative Fluid-Structural Interface .................................................................................. 10
Figure 10: Ram-Air Parachute Mesh (Kalro, Tezduyar, 2000) ......................................................... 12
Figure 11: Close-up of Voronoi Elements (Kalro, Tezduyar, 2000) ............................................... 12
Figure 12: Side Boundary of the Fluid Mesh (Kalro, Tezduyar, 2000) .......................................... 13
Figure 13: Fluid Modeling of Parachute Cells (Lacroix, Bordenave, 1999) ................................. 13
Figure 14: Simulation Pressure Distribution for Initial Assumed Shape (Left) and Final Equilibrium Condition (Right) (Kalro, Tezduyar, 2000) ............................................. 14
Figure 15: Simulated Velocity Vectors at Cross-Sections of Parafoil at Mid (Left) and End (Right) Sections (Kalro, Tezduyar, 2000) ................................................................. 14
Figure 16: Principal Stress Distribution Across Canopy Surface (Red-High Stress/White-Low Stress) (Kalro, Tezduyar, 2000) ................................................................. 14
Figure 17: Experimental Parafoil (Lacroix, Bordenave, 1999) ..................................................... 15
Figure 18: Airdrop Example (Kalro, Leonard, Accorse, 2000) ...................................................... 16
Figure 19: Structural and Fluid Meshes for Airdrop Parachute ...................................................... 17
Figure 20: Material Properties Used in Airdrop Parachute ............................................................. 17
Figure 21: Dynamic Analysis of Airdrop Parachute (where z is time in seconds) ....................... 18
Figure 22: Force vs Time of Airdrop Parachute (dashed line is the equilibrium condition) ....... 19
Figure 23: Position and Velocity Vs. Time for Airdrop Parachute ................................................. 20
Figure 24: Cross-Parachute Example ............................................................................................. 20
Figure 25: Material Properties Used in Cross-Parachute Example .............................................. 21
Figure 26: Fluid-Structural Vs. Experimental Model (Stein, Benney, Tezduyar, Potvin) 21
Figure 27: FSI vs. Experimental Model Results ............................................................................ 22
Figure 28: Parachute Porosity ........................................................................................................ 23
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>29</td>
<td>Parachute Inflation</td>
</tr>
<tr>
<td>30</td>
<td>Changes in Parachute Dynamics Due to Changes in Porosity</td>
</tr>
<tr>
<td>31</td>
<td>Ring-Sail Parachute</td>
</tr>
<tr>
<td>32</td>
<td>Fine-Gore Structural (Left) and Fluid Meshes (Right)</td>
</tr>
<tr>
<td>33</td>
<td>Cut-section of a Ring-Sail Parachute</td>
</tr>
<tr>
<td>34</td>
<td>Fluid-Structural Forces (Left-Fluid Mesh, Right-Structural Mesh)</td>
</tr>
<tr>
<td>35</td>
<td>Fluid Flow Field Simulation</td>
</tr>
<tr>
<td>36</td>
<td>Field Flow Simulation</td>
</tr>
<tr>
<td>37</td>
<td>Ring-Sail Slits and Gaps</td>
</tr>
<tr>
<td>38</td>
<td>Post-Processing Simulation</td>
</tr>
<tr>
<td>39</td>
<td>Ring-slot Parachute Example (Left-Packed Right-Inflated)</td>
</tr>
<tr>
<td>40</td>
<td>Ring-slot Parachute Canopy Inflation</td>
</tr>
<tr>
<td>41</td>
<td>Projected Area and Drag Force on Ring-slot Parachute</td>
</tr>
<tr>
<td>42</td>
<td>Flow Velocity Contour of Ring-slot Parachute</td>
</tr>
<tr>
<td>43</td>
<td>Close-up of Fluid Flow through Ring-slot Parachute Canopy</td>
</tr>
<tr>
<td>44</td>
<td>Fluid-Structural System Flow</td>
</tr>
<tr>
<td>45</td>
<td>Single-Cell Section of a Nine-Cell Parafoil</td>
</tr>
<tr>
<td>46</td>
<td>ANSYS Implicit Architecture</td>
</tr>
<tr>
<td>47</td>
<td>ANSYS Workbench Architecture</td>
</tr>
<tr>
<td>48</td>
<td>Performance Designs’ Drawing Views</td>
</tr>
<tr>
<td>49</td>
<td>Nine-Cell Parafoil</td>
</tr>
<tr>
<td>50</td>
<td>SolidWorks Geometric Model Views</td>
</tr>
<tr>
<td>51</td>
<td>Projected Airfoil on Extrusion</td>
</tr>
<tr>
<td>52</td>
<td>Stenciled Center Airfoil</td>
</tr>
<tr>
<td>53</td>
<td>Right Plane Creation</td>
</tr>
<tr>
<td>54</td>
<td>Right Airfoil</td>
</tr>
<tr>
<td>55</td>
<td>Three Airfoil Surfaces</td>
</tr>
<tr>
<td>56</td>
<td>Lofted Canopy</td>
</tr>
<tr>
<td>57</td>
<td>Right Seam Sketch</td>
</tr>
<tr>
<td>58</td>
<td>Right Seam Sketch Close-up</td>
</tr>
<tr>
<td>59</td>
<td>Right Seam Split Line</td>
</tr>
<tr>
<td>60</td>
<td>Right Seams</td>
</tr>
<tr>
<td>61</td>
<td>Center Seam</td>
</tr>
<tr>
<td>Figure</td>
<td>Image Description</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>62</td>
<td>Center Seam Cut</td>
</tr>
<tr>
<td>63</td>
<td>Top Seam Split Line</td>
</tr>
<tr>
<td>64</td>
<td>Bottom Seam Split Line</td>
</tr>
<tr>
<td>65</td>
<td>Top Seam Cut</td>
</tr>
<tr>
<td>66</td>
<td>Seam Cut</td>
</tr>
<tr>
<td>67</td>
<td>Line Attachment Points – Isometric</td>
</tr>
<tr>
<td>68</td>
<td>3D Sketch – Isometric</td>
</tr>
<tr>
<td>69</td>
<td>3D Sketch – Right</td>
</tr>
<tr>
<td>70</td>
<td>Nine-Cell Parafoil Geometry</td>
</tr>
<tr>
<td>71</td>
<td>Nine-Cell Parafoil Line Attachments</td>
</tr>
<tr>
<td>72</td>
<td>Single-Cell Close-up of Line Attachments</td>
</tr>
<tr>
<td>73</td>
<td>DM - Imported Model</td>
</tr>
<tr>
<td>74</td>
<td>Final Parafoil Model</td>
</tr>
<tr>
<td>75</td>
<td>3D Sketch – Right</td>
</tr>
<tr>
<td>76</td>
<td>Warp and Fill Unit Cell</td>
</tr>
<tr>
<td>77</td>
<td>FAAF Nylon Canopy Stiffeners</td>
</tr>
<tr>
<td>78</td>
<td>FAAF Nylon Canopy Specimen</td>
</tr>
<tr>
<td>79</td>
<td>Seam Construction</td>
</tr>
<tr>
<td>80</td>
<td>Uniaxial Tensile Test on FAAF Nylon Canopy</td>
</tr>
<tr>
<td>81</td>
<td>Uniaxial Tensile Test on FAAF Seam</td>
</tr>
<tr>
<td>82</td>
<td>Uniaxial Stress-Strain Test Data</td>
</tr>
<tr>
<td>83</td>
<td>Seam Samples</td>
</tr>
<tr>
<td>84</td>
<td>Top and Bottom Seam Stress Data</td>
</tr>
<tr>
<td>85</td>
<td>Biaxial Tensile Test (Galliot, Luchsinger)</td>
</tr>
<tr>
<td>86</td>
<td>Biaxial Test Setup</td>
</tr>
<tr>
<td>87</td>
<td>Biaxial Force Calibration</td>
</tr>
<tr>
<td>88</td>
<td>Biaxial Force Calibration Linear Data</td>
</tr>
<tr>
<td>89</td>
<td>Biaxial Test of FAAF Fabric</td>
</tr>
<tr>
<td>90</td>
<td>Biaxial Raw Test Data</td>
</tr>
<tr>
<td>91</td>
<td>Biaxial Stress-Strain Data</td>
</tr>
<tr>
<td>92</td>
<td>Mooney-Rivlin Curve-Fitting in ANSYS</td>
</tr>
<tr>
<td>93</td>
<td>FSI Structure</td>
</tr>
<tr>
<td>94</td>
<td>ANSYS Workbench Setup</td>
</tr>
</tbody>
</table>
Figure 95: Parafoil Fluid Domain Setup ................................................................. 77
Figure 96: ICEM - CFD Mesh ............................................................................. 78
Figure 97: ICEM - Parafoil Assembly Meshing .................................................. 79
Figure 98: ICEM – Isometric 3D Tetra-View..................................................... 79
Figure 99: Orthogonal Quality of a Tetrahedron Cell ....................................... 81
Figure 100: Skewness ....................................................................................... 81
Figure 101: Cusp Modeling .............................................................................. 82
Figure 102: ICEM – Orthogonal Statistics .......................................................... 83
Figure 103: ICEM – Skewness Statistics ............................................................ 83
Figure 104: Fluent .............................................................................................. 84
Figure 105: FEA Mesh ...................................................................................... 90
Figure 106: FEA Seams ................................................................................... 91
Figure 107: Parafoil Restraints ....................................................................... 92
Figure 108: Lift and Drag Forces ..................................................................... 93
Figure 109: Mapping Surfaces (Baskut, Akgul, 2011) ..................................... 95
Figure 110: System Coupling – Mapped Surfaces .......................................... 96
Figure 111: System Coupling – Mapping .......................................................... 97
Figure 112: System Coupling – Iterations ......................................................... 98
Figure 113: System Coupling – Layout ............................................................. 100
Figure 114: Single-cell Parafoil Experimental Setup ....................................... 102
Figure 115: Experimental Setup – Variable Span .......................................... 103
Figure 116: Experimental Setup – Front ........................................................... 103
Figure 117: Gas Powered Prop Front View ...................................................... 104
Figure 118: Gas Powered Prop Side View ....................................................... 105
Figure 119: Experimental Setup – Low-pressure Outlet .................................. 106
Figure 120: Experimental Setup – Honeycomb Inlet Exterior ...................... 107
Figure 121: Experimental Setup – Honeycomb Inlet Interior ....................... 107
Figure 122: Single-Cell Canopy Deformation ................................................ 108
Figure 123: Digitized Canopy Deformation ..................................................... 109
Figure 124: Trendline of Canopy Deformation ............................................... 109
Figure 125: Deflection of Canopy Deformation ............................................. 110
Figure 126: Simple Test Model ...................................................................... 111
Figure 127: FSI Test Model ............................................................................ 113
Figure 128: Parafoil FSI Model Convergence ................................................................. 114
Figure 129: Parafoil FSI Deformations ......................................................................... 115
Figure 130: Parafoil FSI Deformations – Side .............................................................. 115
Figure 131: Post CFD – 12 mph Max Displacements .................................................... 116
Figure 132: Velocity and Pressure Cut Planes ............................................................... 117
Figure 133: Velocity Streamlines – Center Rib ............................................................ 118
Figure 134: Pressure Contour – Center Rib ................................................................. 118
Figure 135: Velocity Streamlines – Quarter Cell ......................................................... 119
Figure 136: Pressure Contour – Quarter Cell .............................................................. 119
Figure 137: Velocity Streamlines – End Rib ............................................................... 120
Figure 138: Pressure Contour – End Rib .................................................................... 120
Figure 139: Side-View Comparison ............................................................................ 121
Figure 140: Canopy Deformations ............................................................................ 121
Figure 141: Change in Canopy Deformations ............................................................ 122
Figure 142: Extrapolated FSI Deformations ............................................................... 123
Figure 143: Explicit / Implicit Parachute Modeling ..................................................... 126
SYMBOLS

$\Delta t$  Change in Time  
$A$  Multidimensional Vector  
$C$  Courant Number  
$G_k$  Generation of Turbulence Kinetic Energy Due to Mean Velocity Gradients  
$m$  Mass  
$v$  Velocity  
$W$  Strain-Energy Density
## ABBREVIATIONS

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALE</td>
<td>Arbitrary Lagrangian Euler</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>FEA</td>
<td>Finite Element Analysis</td>
</tr>
<tr>
<td>FSI</td>
<td>Fluid-Structure Interaction</td>
</tr>
<tr>
<td>HMGP</td>
<td>Homogenized Modeling of Geometric Porosity</td>
</tr>
<tr>
<td>PISO</td>
<td>Pressure Implicit with Splitting of Operator</td>
</tr>
<tr>
<td>RAM</td>
<td>Random Access Memory</td>
</tr>
<tr>
<td>SSTFSI</td>
<td>Stabilized Space-time Fluid Structure Interaction</td>
</tr>
<tr>
<td>STT</td>
<td>Shear-Stress Transport</td>
</tr>
</tbody>
</table>
ABSTRACT

In the summer of 2014, Performance Designs, Inc. contacted Embry-Riddle Aeronautical University’s Eagle Flight Research Center to lead an investigation on square parachute design and optimization using modern computational methods to reduce costs in experimental testing. This thesis investigates the foundation for using an implicit fluid-structure interaction computational model to tackle the challenges of modeling a highly-flexible, porous fabric for design optimization of a parafoil parachute’s transient performance. Canopy deformations of a single-cell square parafoil using a fluid-structure interaction (FSI) model with nonlinear material modeling is compared to an experimental setup of matching geometry. The results of this thesis yielded a partial match of 25% between the experimental and FSI model deformations and thus asserts that fluid-structural modeling using ANSYS Multiphysics can be used to model square parachutes.
1. Introduction

Parachutes have a wide range of utility in today’s market from military, sport and even space applications such as landing on other planets or moons. Sport applications demand the highest performance and efficiency with high competition for designing the ultimate product. The goal of this paper is to introduce an implicit computational fluid-structural modeling approach that is applied to parachutes to help cut the costs of development. Experimental designs of parachutes are expensive and can be risky to test. Any method to reduce the number of experimental tests or to aid in the design process will lead to a more efficient and reliable product.

Modern computational modeling such as Computational Fluid Dynamics (CFD) utilizes a rigid (frozen) structure in space to analyze the pressures and velocities of the flow field surrounding the object ignoring the material properties. Finite Element Analysis (FEA) utilizes the material properties to determine the deformations of a structure under forces and pressures. Both CFD and FEA are difficult practices in their own and are very common among many industrial challenges today. However, by combining CFD and FEA in a coupled environment with non-linear, highly-flexible material whose very shape is defined by the flow fields surrounding it is an even bigger challenge that is ultimately the goal to accurately model a parachute.

Fluid-Structure Interaction (FSI) is the computational method of managing the exchange of data between CFD and FEA solvers for flexible structures under pressure flows. Fluid-structural modeling is a difficult computational process because computing power and precision of modeling are factors that must be analyzed for a quick and accurate model of a parachute with high fidelity. Therefore, FSI modeling is the answer
to a high fidelity model of a parachute.

The parafoil parachute shown in Figure 1 is an example of the complex geometry that a fluid-structure model will face in the design environment in the field of parachute engineering. This thesis is the first step to analyzing the performance of fluid-structural model used to analyze a single-cell subsection of a parafoil parachute.

Figure 1: Parafoil Parachute (Performance Designs, Inc.)

1.1. Problem Statement

Analyzing the performance and optimization of parachute design is currently limited to experimental testing and therefore expensive, difficult and can be dangerous to develop.

1.2. Objective of Study

The goal of this thesis is to research new alternatives in parachute engineering for design optimization of square parachutes.
1.3. Limitation of Study

- Computational limitations and time – FSI models are very computationally demanding and take an extensive amount of time to complete. This thesis is limited to the modeling of a single-cell subsection of a square parafoil using ANSYS Multiphysics FSI two-way coupling.

- The single-cell parafoil modeled in this thesis is constrained laterally to match the deformations of the experiment. An in-air stable flight of a single-cell subsection was unobtainable during experimental testing.

1.4. Organization of the Paper

*Chapter I Introduction*: This chapter deals with the problem, objective and limitation of the study.

*Chapter II Literature Review*: This chapter is crafted to bring the reader up to date on the prior experiments and computational modeling of parachutes in CFD, FEA and FSI over the past couple decades that lead up to the most current advances in parachute modeling.

*Chapter III Methodology*: The materials and methods used to construct the FSI modeling of a single-cell parafoil using ANSYS Multiphysics two-way coupling is described in this chapter.

*Chapter IV Analysis*: The results and analysis of the single-cell parafoil FSI model and experiment are presented in this chapter along with developmental modeling.

*Chapter V Conclusion and Recommendations*: The conclusion and comparison between the parafoil FSI model and experiment are discussed in this chapter along with further recommendations.
2. Literature Review

In the past, the design and optimization of parachutes can be summed up in the cyclic process shown in Figure 2, where the design engineer starts with modifying a generic shape that has already been experimentally proven to work in the field. This new design idea is tested in an experimental environment where the performance is evaluated such as the flight endurance, distance covered, and speed of the parachute using an onboard GPS or IMU device to monitor the position over time (Figure 3).

![Figure 2: Traditional Parachute Design](image)

![Figure 3: Experimental Drop Test (Gargana)](image)
This experimental process in Figure 2 can be very costly for optimizing a parachute’s geometry for optimal performance. As more modern computational methods emerge from the engineering world, these experimental costs can be reduced by taking analytical shortcuts to reduce the number of experimental tests by making more accurate design changes in parachute engineering. One shortcut is using computational fluid dynamics to determine the flight stability parameters of a canopy shape such as the coefficients of lift and drag that lead to the development of estimating the endurance, speed and accuracy of the parachute.

However, even with the power of computational fluid dynamics today, the material structure of the parachute is not taken into account and the parachute design engineer must make assumptions on whether the parachute will actually open or what the final shape of the inflated canopy will be. Therefore, this does not eliminate the need for experimental testing but rather reduces the number of parachute tests to reach the final goal. In the current parachute industry, adding CFD to the development process of a square parachute can reduce the number of experimental drop tests by a factor of 10 to 1. Figure 4 demonstrates the inner loop that the CFD analysis can offer in parachute design. Therefore, using CFD could reduce the costs of experimental testing by a significant factor. However, even with the implementation of CFD into the design process, it cannot resolve the transient problem of parachute inflation or the steady-state inflation deformation of the canopy.
Computational Fluid Dynamics takes geometric profiles in 3D space and freezes them into non-movable rigid bodies where the flow fields around the geometric profiles are solved for pressure, temperature and velocities. This situation is understandable in parachute design if the assumption is that the parachute canopy being analyzed is at its final shape and deformation (completed inflated) during the CFD analysis. However, this assumption requires several drop tests or wind tunnel tests to verify the shape. If there are any structural changes in the canopy design, such as a change in placement of a line attachment to the canopy, an experimental drop test must be done to ensure the new structure of the canopy profile will open and inflate.

In order to account for the structural dynamics of the parachute system, a finite element analysis must be administered on the canopy, but without the proper pressure distributions, how can one know if the structural dynamics can be trusted? And if so, given the nature of how flexible a parachute is, how will these pressure distributions change in response to the parachute inflation? The answer is a coupled fluid-structural model between the computational fluid dynamics solver and the finite element analysis. Fluid-structure Interaction (FSI) has become more popular in the last twenty years due to
the advances in implicit and explicit solvers and programs that make the exchange in data between these solvers seem more effortlessly. FSI is used primarily for materials that contain excessive deflections that are large enough to change the flow fields and change the corresponding pressure fields that act in and/or around the system. An example of such a situation is shown in Figure 5, where a FSI solver called ANSYS Coupling is used to simulate the displacement of an axisymmetric leaf valve artery composed of a biomembrane hyperelastic (very flexible) material.

Figure 5: ANSYS Leaf Valve Artery (Scampoli)

The pressures of the blood are passed through to the walls of the artery while the structure of the artery react to a change in displacement that results in changes to the flow of blood. This exchange of data between the pressures of the blood and the displacements of the artery create the cyclic process of the fluid-structure interaction until an approximate steady state condition of the artery structure is achieved.
Primarily, when fluid-structure interactions first became popular, it was used on solid isometric structures to calculate small displacements such as the change of lift distribution across the span of a wing after a wing loading was applied, as shown in Figure 6.

![Aeroelastics on a Wing (Goodwin)](image)

Figure 6: Aeroelastics on a Wing (Goodwin)

The pressure fields create a force on the wing, the wing responds by displacing, which in turn, changes the pressures fields. As technology in FSI modeling advanced, this cyclic process can now be extended to highly elastic materials such as the flexible canopy of a parachute while taking into account the canopy’s material properties.
2.1. Past Methodologies

One of the first cases of FSI being used in parachute design was back in 1999, where the Airborne Research and Development Center of The French Ministry of Defense (Lacroix, Bordenave, 1999) developed a coupled fluid and structural modeling software they called ‘SINPA’ as a request by the French company SIMULOG. This coupled fluid-structure program was designed for a ram-air parachute (a parachute that takes its shape from a build-up of pressure i.e. a parafoil) using a Deforming-Spatial-Domain/Stabilized Space-Time (DSD/SST), or also known as Stabilized Space-Time Fluid-Structure Interaction (SSTFSI) technique, first introduced back in 1991. This technique was a series of constituent programs that make up the simulation process:

1. AUTOCAD – CAD Modeler
2. SIMAIL – Automatic Mesh Generator
3. SAMCEF/MECANO – Structural Mechanics Solver
4. N3S – Fluid Mechanics Solver
5. ENSIGHT – Postprocessor

The automatic mesh generator (dynamic mesh), structural and fluid mechanics solver were coupled together in a cyclic process to analyze the approximated steady-state stresses and deformations of the basic square parafoil geometry shown in Figure 7.

Figure 7: Basic Parafoil Parachute Model (Lacroix, Bordenave, 1999)
Solving the pressures and displacements of the parachute was accomplished using an iterative exchange of data between a structural and fluid mesh. These structural and fluid meshes were overlapped on the same boundary at the canopy surface of the parachute (where the canopy meets with the air, Figure 8). This fluid-structural overlapped mesh exchanges pressures and displacements of the computational fluid and structural domains throughout the cyclic process shown in Figure 9, until an approximate steady state equilibrium condition is reached on the structural solver.

![Fluid-Structural Mesh](image1)

*Figure 8: Fluid-Structural Mesh (Baskut, Akgul, 2011)*

![Iterative Fluid-Structural Interface](image2)

*Figure 9: Iterative Fluid-Structural Interface (Lacroix, Bordenave, 1999)*

This cyclic process between the automatic mesh generator, fluid and structural solvers is shown in the following logic:

1) The geometric model was created via AUTOCAD which defines the shape and material composition of the parachute layout including the following components:
   - Suspension Lines
   - Seams
   - Straps
   - Sail Surface
   - The Load on the Parachute (the Weight or Person)
2) The geometric model is loaded into the SIMAIL mesh generator from the CAD modeler (Figure 10 and Figure 11 display the mesh representation of the CAD geometry)

3) SIMAIL translated the mesh into comprehensible data for the preprocessors of the solid and fluid simulators (Figure 11 displays Voronoi elements for the fluid-structural interaction and Figure 12 shows the mesh generated for the flow field CFD around the parafoil)

4) Structural and fluid calculations take place and loop for each time step until an approximate steady-state response is met:
   - N3S performs the fluid simulation using the three dimensional Navier-Stokes equations (CFD)
   - SAMCEF/MECANO simulates the motion and dynamics of the parachute mechanical system (FEA)

5) ENSIGHT is used to create a transient analysis of the data for graphical animations (Figure 13 displays a flow field through the steady state equilibrium of the gliding parachute)

An orthogonal material model was used to model the parafoil canopy material. This implies that a Young’s Modulus was used for one planar direction of the canopy while a separate Young’s Modulus was used for the other planar dimension to create a membrane element. However, the orthogonal material model assumes a linear relationship between stress and strain, so the non-linearity within the parachute canopy will not be reflected in the results.

Introducing the material non-linearity would yield a true non-steady state solution for this problem because of the small deformations that would exist on the surface of the parafoil during flight. Imagine a ‘wrinkling’ effect where the parachute would overall appear to be not moving, but the canopy surface would be shifting due to the warping of the material. The non-linearity of a parachute canopy is modeled in this thesis.
Figure 10: Ram-Air Parachute Mesh (Kalro, Tezduyar, 2000)

Figure 11: Close-up of Voronoi Elements (Kalro, Tezduyar, 2000)
The final displacement from the fluid-structural model are shown in Figure 14 using the post-processing capabilities of the SINPA software. The velocity vectors showing the chord-wise flow of air across a cross-sectional cut of the parafoil is shown in Figure 15. The complete parafoil displacement with stress distribution is shown in Figure 16.
Figure 14: Simulation Pressure Distribution for Initial Assumed Shape (Left) and Final Equilibrium Condition (Right) (Kalro, Tezduyar, 2000)

Figure 15: Simulated Velocity Vectors at Cross-Sections of Parafoil at Mid (Left) and End (Right) Sections (Kalro, Tezduyar, 2000)

Figure 16: Principal Stress Distribution Across Canopy Surface (Red-High Stress/White-Low Stress) (Kalro, Tezduyar, 2000)
This computational FSI method developed (SINPA) was compared to an experimental model similar geometry (Figure 17) attained for the sole purpose of comparison to the computational model for validation. Comparing the results between the calculated and measured pressure fields for the FSI and experimental models yielded a difference of 6% in the lift-to-drag ratio, a difference of 0.2% in the vertical speed of the airflow and a difference of 5.8% in the horizontal speed of the airflow. Evidently, it was clear that the FSI model’s results were very similar to that of the experiment.

Figure 17: Experimental Parafoil (Lacroix, Bordenave, 1999)

Additionally in 1999, a NASA funded project for the US Army was conducting a 3D parachute simulation using the fluid-structure interaction method with the stabilized space-time formulation for an airdrop parachute. The procedure that was used was very similar to the ‘SINPA’ method developed by the Airborne Research and Development
Center of The French Ministry of Defense. The motion and displacement of the airdrop parachute shown in Figure 18 was analyzed using the SSTFSI model.

![Airdrop Example](image)

*Figure 18: Airdrop Example (Kalro, Leonard, Accorse, 2000)*

The code generation for the fluid and structural meshes along with the post processor and coupling program have been privately coded for this project, but the generated structural and fluid meshes for the airdrop parachute along with the material properties are shown in Figure 19 and Figure 20.
Figure 19: Structural and Fluid Meshes for Airdrop Parachute (Kalro, Leonard, Accorse, 2000)

<table>
<thead>
<tr>
<th>Material Group</th>
<th>Cables</th>
<th>Membranes</th>
<th>Concentrated Masses</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Suspension Lines</td>
<td>Radial Reinforcements</td>
<td>Risers</td>
</tr>
<tr>
<td>elements</td>
<td>780</td>
<td>360</td>
<td>40</td>
</tr>
<tr>
<td>thickness (area)</td>
<td>f_0 (ft^2)</td>
<td>0.0001</td>
<td>0.0001</td>
</tr>
<tr>
<td>density slugs/ft^3</td>
<td>6.0</td>
<td>6.0</td>
<td>6.0</td>
</tr>
<tr>
<td>Young's modulus</td>
<td>lb/ft^2</td>
<td>4.32x10^6</td>
<td>4.32x10^6</td>
</tr>
</tbody>
</table>

Figure 20: Material Properties Used in Airdrop Parachute (Kalro, Leonard, Accorse, 2000)
A time history of the coupled fluid and structural meshes working in association with the masses and material properties of the air drop parachute are shown in Figure 21 where the computational displacements of the parachute canopy work in tandem with the mass’ inertial forces as the parachute drops in air.

Figure 21: Dynamic Analysis of Airdrop Parachute (where z is time in seconds) (Kalro, Leonard, Accorse, 2000)
The recorded forces in the corresponding 'x' and 'y' dimensions (as identified in Figure 21) is shown below in Figure 22.

Figure 22: Force vs Time of Airdrop Parachute (dashed line is the equilibrium condition) (Kalro, Leonard, Accorse, 2000)
Using the stability coefficients from the simulated FSI model, the velocity and position of the parachute’s payload was graphed in Figure 23 to represent the motion of the parachute free-falling.

Figure 23: Position and Velocity Vs. Time for Airdrop Parachute (Kalro, Leonard, Accorse, 2000)

Two years later, another fluid-structure interaction model was implemented on a cross-parachute (as shown in Figure 24), also using the Deforming-Spatial-Domain/Stabilized Space–Time (DSD/SST) method.

Figure 24: Cross-Parachute Example (Stein, Benney, Tezduvar, Potvin, 2001)
The material properties used in this cross-parachute model are shown in Figure 25. Note the thickness of the cross-parachute modeled in this example is 0.0001 feet or 0.0012 inches thick.

<table>
<thead>
<tr>
<th>Material group:</th>
<th>Membranes</th>
<th>Cables</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Canopy</td>
<td>Suspension lines</td>
</tr>
<tr>
<td>Thickness (area)</td>
<td>0.0001 ft</td>
<td>0.0001 ft²</td>
</tr>
<tr>
<td>Density</td>
<td>3.75 slugs/ft³</td>
<td>0.85 slugs/ft³</td>
</tr>
<tr>
<td>Young’s modulus</td>
<td>$2 \times 10^9$ lb/ft²</td>
<td>$5.0 \times 10^6$ lb/ft²</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.3</td>
<td>—</td>
</tr>
</tbody>
</table>

Figure 25: Material Properties Used in Cross-Parachute Example
(Stein, Benney, Tezduyar, Potvin, 2001)

The drag, inflated shape and differential pressure behavior of the virtual cross-parachute model were compared to that of an actual experimental model in a wind tunnel as shown in Figure 26.

Figure 26: Fluid-Structural Vs. Experimental Model (Stein, Benney, Tezduyar, Potvin)
Figure 27 shows the calculated results of the FSI model versus a few measurements from the experimental model in the wind tunnel. Just as like the previously mentioned experiments, the FSI calculations are very close to the experimental results. The recorded measurements versus calculations of the FSI model differ anywhere from 3.5% to 12% overall in the results comparison. Something to note is that as the wind speed in the wind tunnel increases, so does the error between the calculated drag of the FSI model and the measured drag in the experiment.

<table>
<thead>
<tr>
<th>Simulation number</th>
<th>Cross-parachute model</th>
<th>Tunnel speed (miles/h)</th>
<th>Drag FSI (lb)</th>
<th>Experiment (lb)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>50-inch lines*</td>
<td>40</td>
<td>42.8</td>
<td>44.0</td>
</tr>
<tr>
<td>2</td>
<td>50-inch lines*</td>
<td>60</td>
<td>96.0</td>
<td>107.0</td>
</tr>
<tr>
<td>3</td>
<td>50-inch lines</td>
<td>80</td>
<td>170.5</td>
<td>–</td>
</tr>
<tr>
<td>4</td>
<td>45-inch lines</td>
<td>40</td>
<td>41.2</td>
<td>–</td>
</tr>
<tr>
<td>5</td>
<td>45-inch lines</td>
<td>60</td>
<td>92.6</td>
<td>–</td>
</tr>
<tr>
<td>6</td>
<td>45-inch lines</td>
<td>80</td>
<td>164.5</td>
<td>–</td>
</tr>
<tr>
<td>7</td>
<td>40-inch lines</td>
<td>40</td>
<td>39.6</td>
<td>41.0</td>
</tr>
<tr>
<td>8</td>
<td>40-inch lines</td>
<td>60</td>
<td>88.7</td>
<td>94.0</td>
</tr>
<tr>
<td>9</td>
<td>40-inch lines</td>
<td>80</td>
<td>157.5</td>
<td>–</td>
</tr>
<tr>
<td>10</td>
<td>28-inch lines</td>
<td>40</td>
<td>–</td>
<td>32.0</td>
</tr>
<tr>
<td>11</td>
<td>28-inch lines</td>
<td>40</td>
<td>–</td>
<td>62.0</td>
</tr>
</tbody>
</table>

*The experimental model had 51-inch lines.

One of the first cases of structural porosity was introduced by Yongsam Kim and Charles Peskin where the porosity of parachutes was conducted in the 2D domain (Kim, Peskin, 2006). Changes to the porosity of the canopy structure have led to changes in a parachute’s natural ‘rocking’ frequency as well as the overall inflation rate and deformation of the parachute. The proposed technique is called the ‘Immersed Boundary Method’ where the interaction of the following parameters can be tuned and the results can be post-processed and compared:

- Flexibility
- Elasticity
- Porosity (Figure 28)
- Mass of canopy, suspension lines, risers and payload
The inflation and performance characteristics can be post processed such as in Figure 29 where the time-lapse of the inflation sequence was plotted up to 2.88 seconds of simulated time.
Changing the porosity coefficients in the parachute canopy model resulted in a dynamic ‘rocking’ behavior of the parachute shown in Figure 30. The columns in Figure 30 correspond to different porosity settings while the rows correspond to different time samples. It seems that as the porosity of the parachute increases, the ‘rocking’ frequency of the dynamics decrease.

Figure 30: Changes in Parachute Dynamics Due to Changes in Porosity (Kim, Peskin, 2006)
Fast forwarding to 2012, after more modern developments in FSI accelerated the modeling capability, the Team for Advanced Flow and Simulation (TAFSM) successfully addressed some of the more prominent challenges of the Stabilized Space-Time FSI technique (SSTFSI). These challenges included modeling a quasi-direct and direct coupling technique that is more accurate to light canopy structures that are highly sensitive to fluid flows (Takizawa, Tezduyar, 2012).

The Team for Advanced Flow and Simulation developed a method for calculating the slit and gap structure of FSI modeling along with introducing a porous canopy material model.

For application, the TAFSM applied their new SSTFSI technique to a ring-sail parachute design shown in Figure 31. Given its complexity with porous material and gaps, the ring-sail parachute is a difficult challenge for any fluid-structure model to simulate accurately.
Similar to the method first used in 1999 by the French DGA/CAP, the TAFSM used a coupled fluid-structural mesh interface that is composed of overlapped boundaries between the solid and fluid interfaces (Figure 32). The right-half of the ring-sail parachute in Figure 32 is a generated fluid mesh while the left-half is the structural mesh.

These fluid and structural meshes are overlapped at common nodes and exchange structural and fluid properties throughout the FSI cyclic iteration to determine the steady-state deformation of the parachute canopy when introduced to an airflow. Very similar to
the method first introduced in 1999, the structural and fluid meshes were generated from first creating a CAD model of the ring-sail parachute. Figure 33 shows an axis-symmetrical cut-section of the CAD model used for the mesh generation.

![Figure 33: Cut-section of a Ring-Sail Parachute (Takizawa, Tezduyar, 2012)](image)

Considering the model in Figure 33 implies that the canopy is a solid fabric, it is however, given porous characteristics that typical parachute canopy materials share. The geometric porosity is calculated using the Homogenized Modeling of Geometric Porosity (HMGP) technique [8] and [9]. After applying the external forces on the structural meshes, the vector force fields can be seen on both the fluid and structural meshes in Figure 34. These vector fields are combined with both the fluid and structural meshes to create the equilibrium state of the ring-sail parachute. Figure 35 and Figure 36 demonstrate an axis-symmetrical cut-section of the simulation of the structural and fluid meshes working together with the porosity effect added.
Figure 34: Fluid-Structural Forces (Left-Fluid Mesh, Right-Structural Mesh)  
(Takizawa, Tezduyar, 2012)

Figure 35: Fluid Flow Field Simulation  
(Takizawa, Tezduyar, 2012)
In the ring-sail example, the slits and gaps are created using a surface mesh with different porosity settings for canopy material and open slits (Figure 37).
A complete post-processed simulation at steady-state equilibrium condition is shown at the top of Figure 38 while the actual footage from a NASA drop test of a ring-sail parachute is shown at the bottom portion of Figure 38. As shown in Figure 38, the results of the experimental drop versus the computational model, the two show a very similar canopy displacement.

Figure 38: Post-Processing Simulation
(Takizawa, Tezduyar, 2012)
Finally in 2013, a numerical study on the inflation of a ring-slot parachute in a low speed airdrop environment was commenced using the fluid-structure interaction technique with the Arbitrary Lagrangian Euler (ALE) solver. The explicit FSI solver platform LS-DYNA was used to simulate the inflation of the ring-slot parachute. The initial geometric layout and inflated representation of the ring-slot parachute model before and after inflation are shown in Figure 39 (Xing-long, Qing-bin, Qian-gang, Tao, 2013).

![Figure 39: Ring-slot Parachute Example (Left-Packed Right-Inflated) (Xing-long, Qing-bin, Qian-gang, Tao, 2013).](image)

The fluid-structure coupling techniques used in ring-slot parachute FSI model include the following assumptions for the parachute model:

- Geometry of canopy before inflation is axially symmetric
- No pre-stress conditions exists
- The opening process uses infinite mass (zero canopy thickness) without considering gravity
- Air fluid is considered as incompressible flow at low velocity
- The fluid field is considered at quasi-state with a constant velocity at the inlet

With these assumptions in effect, the porosity of the fabric was taken into account using the Euclidian-Lagrangian penalty coupling method and the simple material properties for the individual components of the parachute as shown in Table 1.
Table 1: Ring-slot Material Properties (Xing-long, Qing-bin, Qian-gang, Tao, 2013)

<table>
<thead>
<tr>
<th>Name</th>
<th>Membranes</th>
<th>Cables</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thickness</td>
<td>Canopy</td>
<td>Suspension Lines</td>
</tr>
<tr>
<td>Density</td>
<td>0.0001 m</td>
<td>4 x 10^{-6} m²</td>
</tr>
<tr>
<td>Young’s Modulus</td>
<td>5880 kg/m³</td>
<td>5840 kg/m³</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>4.309 x 10⁸ Pa</td>
<td>1.2 x 10¹² Pa</td>
</tr>
<tr>
<td></td>
<td>0.001 m</td>
<td>0.001 m</td>
</tr>
<tr>
<td></td>
<td>6800 kg/m³</td>
<td>6800 kg/m³</td>
</tr>
<tr>
<td></td>
<td>4.309 x 10⁸ Pa</td>
<td>5.309 x 10⁸ Pa</td>
</tr>
</tbody>
</table>

A time history of the parachute canopy inflation process is shown at the following time steps in Figure 40.

Figure 40: Ring-slot Parachute Canopy Inflation (Xing-long, Qing-bin, Qian-gang, Tao, 2013).
Figure 41 shows the calculated projected area (area of parachute canopy as seen from above) and the drag force on the ring-slot parachute plotted as a function of time.

![Projected Area and Drag Force on Ring-slot Parachute](image)

Figure 41: Projected Area and Drag Force on Ring-slot Parachute (Xing-long, Qing-bin, Qian-gang, Tao, 2013).

Flow fluids through the computational fluid dynamics solver are shown in Figure 42 at several different time steps after full inflation occurs. A close-up on a planar flow field at full inflation is shown in Figure 43.

As shown over the past decade and a half, the advancements in FSI modeling since 1999 paved the way forward for advanced parachute design using more dependable and accurate models. The next step is to use these fluid-structural models to refine the design and development process for parachute engineering on a regular basis.
Figure 42: Flow Velocity Contour of Ring-slot Parachute (Xing-long, Qing-bin, Qian-gang, Tao, 2013).

Figure 43: Close-up of Fluid Flow through Ring-slot Parachute Canopy (Xing-long, Qing-bin, Qian-gang, Tao, 2013).
2.2. Hypothesis

The goal of this paper is to create the foundation of modeling a square parafoil using a single-cell highly-coupled implicit fluid-structure interaction model with non-linear material properties and validating the results to an experimental model of similar geometry to develop the first step in analyzing the transient performance of square parachutes that is easily modified and reusable for design optimization.
3. Methodology

3.1. Strategy

Similar to the methods used in the past, the first step to planning out the computational model of the aerodynamic and structural performance of a parafoil will be to design a program flow that the computational analysis will use as a roadmap. Like the previous work done on parachute modeling, the strategy will need the following programs to work synchronously: a geometric modeler, a meshing application, a CFD solver that works seamlessly with a FEA solver and vice versa, a method of incorporating the material properties of the canopy into the FEA solver, a coupling program that records the CFD and FEA solver’s input and output during each iteration of the fluid-structure interaction, and a post processor that will record the displacements and stresses of the parafoil canopy as well as the pressures and flow fields of the surrounding air. Figure 44 demonstrates the placement of these programs to complete the fluid-structure interaction roadmap that the computational analysis will need to follow to get an accurate model of a parafoil.

![Fluid-Structural System Flow Diagram]

Figure 44: Fluid-Structural System Flow
The computational model will also be completed using a standard PC of 16 GB RAM, 8 cores and a 3 GHz processor. A computer cluster would be the ultimate tool for tackling a FSI model of this difficulty. However, computer clusters are extremely expensive and are not of common practice among company infrastructure.

An implicit FSI model will be used as the primary engineering strategy for this thesis. The difference between an implicit and explicit FSI model is simply the number of cycles the FSI process completes per time step. If a FSI cycle only completes one transfer between the CFD and FEA solvers, then it is considered an explicit FSI model. If two or more iterations are used to reach a stabilized solution by reducing residuals, then the model is considered an implicit FSI model. The advantage of using an implicit model over an explicit model is that because an explicit model only uses one FSI cycle per time step which yields a typically high amount of residuals per time step, the time step must be very small. An implicit time step can be much larger. This makes explicit models exceptional for modelling a small duration time (i.e. in milliseconds) with high energy applications such as an explosion or a collision. Implicit FSI models are more efficient at models that require a longer amount of time to simulate such as a steady-state solution (i.e. greater than one second). Parachutes take around two seconds to unfold and inflate, and could take longer to reach an approximate steady-state performance. This puts parachutes right on borderline between implicit and explicit modeling.

Implicit modeling is very accurate in reducing residuals. High accuracy is required to model a parafoil because the slight miscalculation of a pressure component or displacement could drastically change the outcome of the parafoil shape and performance.
Parallel to the computational model, an experimental model is needed to validate the computational results for accuracy. Since the goal of this thesis will be to demonstrate the accurate modeling of a square parafoil using a fluid-structural model, it first must be proven to work on a smaller scale before modeling a full parafoil. Given the computational and experimental modeling limits of this thesis, the fluid-structure interaction model will be used to validate a single-cell subsection of a square parafoil to see if the computational model will imitate the deformations seen in the experiment. One cell of a parafoil is represented as the red section shown in Figure 45.

![Figure 45: Single-Cell Section of a Nine-Cell Parafoil](image)

The computational power needed to model a full parafoil will be drastically larger than that of a single-cell model and will slow down the development process and significantly increase the computational time. It will be proposed in the latter part of this thesis that a full scale parafoil model will be considered after validated results of a smaller model is accurate.

The material properties of the canopy material will need to be extracted in the form of a stress-strain relationship equation, linear or nonlinear (whichever the Nylon canopy experimental tests yield). So incorporation of this linear or nonlinear relationship between stress and strain will be added to the FEA model by using user-defined material stress-strain models.
3.2. Engineering Tools

After careful consideration among solvers including easiness to use, compatibility with other solvers and conforming to all the requirements set forth in the previous section, it was chosen to use the following software shown in Table 2.

Table 2: Modeling Tools

<table>
<thead>
<tr>
<th>Manufacturer</th>
<th>Program</th>
<th>Utility</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Dassault Systems</strong></td>
<td>SolidWorks</td>
<td>Geometric Modeler</td>
</tr>
<tr>
<td></td>
<td>Design Modeler</td>
<td>Geometric Modeler</td>
</tr>
<tr>
<td><strong>ANSYS</strong></td>
<td>ICEM Meshing</td>
<td>Manual / Automatic Meshing</td>
</tr>
<tr>
<td></td>
<td>Engineering Data</td>
<td>Material Properties Manager</td>
</tr>
<tr>
<td></td>
<td>FLUENT</td>
<td>CFD Modeler</td>
</tr>
<tr>
<td></td>
<td>Structural MAPDL</td>
<td>FEA Modeler</td>
</tr>
<tr>
<td></td>
<td>System Coupling</td>
<td>Data Transfer</td>
</tr>
<tr>
<td></td>
<td>Post CFD</td>
<td>Post Processing</td>
</tr>
</tbody>
</table>

Most of the programs listed in Table 2 belong to the ANSYS family of multi-physics software which adds to the easiness to pass information from one program to another. ANSYS has a program called Workbench that’s sole purpose is to connect the programs synchronously from one to another to allow swift data passage.

Dassault Systems SolidWorks will be used as the primary geometric modeler for the initial creation of the parafoil shape for ease of 3D surface modeling. The initial creation of the parafoil canopy and suspension lines will then be passed off to ANSYS Design Modeler where parametrics will be added to the geometry for design optimization. Parametrics allow a simple change to be made such as a length parameter or a line attachment location and instantly the geometric profile will be updated and the computation model can begin or restart. Once the geometric profile of the parafoil model is created and passed off to ANSYS Design Modeler, the FSI process can begin.
3.3. Program Workflow

Combining Figure 44 with the programs shown in Table 2 forms the final program workflow shown in Figure 46.

When organizing the structure of ANSYS products using ANSYS Workbench, the program flow is shown on a technical level in Figure 47 where the model is split into 4 parts: CFD Solver (Column A), FEA Solver (Column B), System Coupling (Column C) and Post Processing (Column D). The blue lines indicate the transfer of data between programs. A blue square (Geometry block at B3) indicates a one-way data transmission while a blue circle (System Coupling Setup block at C2) indicates a two-way data transmission.
Figure 47: ANSYS Workbench Architecture
3.4. Geometric Modeling

Performance Designs has supplied Embry-Riddle’s Eagle Flight Research Center with a single-cell subsection of a full-sized Nylon parafoil canopy by means of a non-dimensional schematic shown below in Figure 48.

Figure 48: Performance Designs’ Drawing Views
This schematic in Figure 48 contains the top, bottom, center, and side canopy layout design of the parafoil including the seams and attachment points. This single-cell section will be a representation of one-ninth of an axisymmetric parafoil model. A full parafoil is made up of nine single-cell subsections as illustrated below in Figure 49 (without line attachments).

![Figure 49: Nine-Cell Parafoil](image)

The first step to inserting the parafoil to the fluid-structure model is to integrate the drawing in Figure 48 into the geometry modeler of ANSYS (Design Modeler). This can be done using any computer-aided drafting and design program. For this particular step, Solidworks was used to draft a three-dimensional surface structure (not a solid structure) to create the non-dimensionalized geometry shown in Figure 50 (See Appendix A for the full set of dimensions for the single-cell square parafoil specimen). A surface implies a geometry that occupies no volume, only a surface comprised of a zero thickness wall exists at this stage in the model.
Figure 50: SolidWorks Geometric Model Views
To construct the surface geometry shown in Figure 50, the Performance Designs’ schematic from Figure 48 was projected onto zero-thickness plates using the projection method shown in Figure 51.

![Projected Airfoil on Extrusion](image1)

Figure 51: Projected Airfoil on Extrusion

From this projected schematic, an airfoil for both the center and side airfoils were drawn and used as the parafoil profile. The non-dimensionalized center stenciled airfoil is shown in Figure 52.

![Stenciled Center Airfoil](image2)

Figure 52: Stenciled Center Airfoil

The placement of the right and left planes of the single-cell parafoil were defined by the top and bottom schematics from Figure 48. Figure 53 demonstrates how the sizing of the width of the single-cell parafoil was placed.
From the creation of the right plane, the right side airfoil was created in Figure 54 in the similar fashion as the center airfoil. With the right airfoil defined, the left airfoil was mirrored from the right shown in Figure 55.
A zero-thickness surface was lofted from the newly defined airfoils shown in Figure 56 that would define the complete structure of the single-cell parafoil.

To add detail to the parafoil canopy, seam lines were projected onto the canopy to define the boundary between the different material properties between the Nylon fabric and the Nylon seams. To do this, the seam sketches were created from the seam profiles.
on the schematics from Figure 48. Figure 57 and Figure 58 show the seam profile built from the side airfoil schematic. The two-dimensional seam sketches were projected onto the parafoil canopy as shown in Figure 59. The ‘split line’ technique was used to make the seams which cuts through the three-dimensional surface using the seam profile as the ‘knife’. The resulting cut section is shown in Figure 60.

These newly created cut sections can now be segregated from the rest of the parafoil model and be later identified as a seam to ANSYS.

![Figure 57: Right Seam Sketch](image-url)
An example of the split line creation process is shown in Figure 61 where one of the center airfoil seams is being projected onto the center airfoil. The pink profile in Figure 61 is the seam sketch and the blue section is the specific section of the parafoil that receives the cut. The result is a parafoil with multiple defined surfaces shown in Figure 62.
The top and bottom seams were created using the same split line technique shown in Figure 63 and Figure 64.
The resulting geometry after the seam lines were created is shown in Figure 65 and Figure 66. No dimensions have been indicated because this model has been created from a dimensionless schematic. In the model definition, scaling has been added to the model to provide matching geometric dimensions to the parafoil canopy supplied by Performance Designs for the experiment. See Appendix A for the dimensionalized single-cell parafoil drawing used in the experimental and FSI model.
The last step in creating the parafoil computational model is to model the line attachments and suspension lines to imitate holding the parafoil in place. The first line attachment was created at the front vertices as shown by the orange lines in Figure 67. From these points, parameters were used to define and control the spacing between the line attachments on the parafoil model. These parameters can be changed for modeling optimization if needed in the future.
Between each line created in Figure 67 was a connection point that was used to draw the three dimensional line segments shown in Figure 68 and Figure 69. These line segments were rotated about the axis shown in Figure 68.
The analysis in this thesis will only be used on a single-cell model of the parachute. Figure 48 draws out the entire parafoil design if extrapolated to the full-size nine-cell layout to get a better picture of what the entire parafoil would look like.

Figure 70: Nine-Cell Parafoil Geometry
Attaching lines from the parafoil canopy to a fixed point in space would yield the results shown in Figure 71 while Figure 72 displays a close-up of the line attachments on the one parafoil cell. Yet only a single-cell is modeled using side wall-mounted restraints in this thesis, the next step would be to model the entire parafoil in FSI using Figure 71 as the initial condition geometry.

Figure 71: Nine-Cell Parafoil Line Attachments

Figure 72: Single-Cell Close-up of Line Attachments
One more step before the geometric profile is complete is to import the SolidWorks parafoil into ANSYS Multiphysics using ANSYS Design Modeler. Figure 73 demonstrates the importation process including the canopy profile and suspension lines. At this point, parametrics could be added to the parafoil model for later design optimization of the fluid-structure model.

![DM - Imported Model](image)

Figure 73: DM - Imported Model

One drawback using three-dimensional surfaces in CFD modeling is that only one force reaction can take place on one side of a three-dimensional surface. This means that only one side of a surface can receive a force. In other words, it would be the same as modeling the lift or drag of an airfoil using only the pressures on either the top or the bottom surface. This is a completely unacceptable assumption for this model, and therefore, adjustments need to be made to the surface body for further analysis. To overcome this drawback, each surface in the single-cell model was duplicated. The
duplicate surfaces were bonded together as one but remain separate surfaces which occupy the same space. Each surface will receive one side of the force reactions from the fluid-structure interaction and since the duplicated surfaces are bonded together, they will both move in unison with a resulting deflection. As of now, these surfaces occupy no volume. A thickness will be applied to these surfaces in the structural solver.

The last step before the model is transferred over to the FEA and CFD solvers for meshing is to create the CFD fluid domain. Figure 74 shows the final model setup for the single-cell parafoil model with a conical CFD fluid domain with standard CFD practice dimensions of 10c x 15c x 5c where c is the chord length of the parafoil.

![Figure 74: Final Parafoil Model](image)

The complete structure of the single-cell parafoil, including line attachments, can be rotated about a fixed point in space to be analyzed for different angles of attack. The referenced used on the parafoil with respect to angle of attack is the chord line. Or the
line between the very front nose of the parafoil and the cusp in the back. This is different from the angle between the bottom of the parafoil and the velocity, which is not used when referring to angle of attack.

Figure 75: 3D Sketch – Right
3.5. Fabric / Material Modeling

The experimental single-cell parafoil provided by Performance Designs is composed of a Nylon thin membrane with added strengthening stiffeners for tear resistance. Since the canopy membrane has these bi-directional strengthening stiffeners, shown up close by Figure 77, the relationship between force and displacement will change depending on the direction a force is applied to these stiffeners. Figure 76 displays a graphical representation of the warp and fill dimensions that make up the parafoil canopy material. Applying a force in line with these stiffeners say, for example, in the direction of the ‘warp’ in Figure 76, a smaller deformation will occur in the canopy than a force applied at an angle of 45 degrees from the warp dimension (φ in Figure 76). Therefore, at the very minimal, an orthogonal material model will need to be used to accurately represent the stress-strain relationship for the Nylon canopy.

Another difficulty in modeling a material with bi-directional stiffeners is that the relationship between stress and strain may change in one dimension of the material while another force is applied at another orthogonal direction. For example, if a force is applied on the warp dimension, the relationship between stress and strain may change when a force is then applied on the fill dimension. Notice in Figure 77, there are more strengthening stiffeners along the fill direction than there are in the warp direction. This should yield a larger deformation in the warp dimension when the same force is applied in line with the fill dimension.
Gathering the material properties from the Nylon canopy was accomplished using a uniaxial stress-strain test to extract the varying deformation versus force behavior from different angular rotations of the warp and fill directions. A biaxial stress-strain test was used to experimentally determine the changing deformations that would occur from applying one force along the warp direction, while another force was applied along the fill direction. Gathering the stress-strain data from these experiments yielded the nonlinear material model for the Nylon canopy used in the FSI model.
3.5.1. Uniaxial Tensile Test

Square specimens (Figure 78) and seam samples (Figure 79) were taken from the raw FAAF Nylon canopy material used in the construction of the parafoil. Figure 80 demonstrates the uniaxial test completed on one of the sample square segments of the FAAF Nylon fabric canopy and Figure 81 demonstrates the same test completed on one of the seam samples.

Figure 78: FAAF Nylon Canopy Specimen
Figure 79: Seam Construction

Figure 80: Uniaxial Tensile Test on FAAF Nylon Canopy
Figure 81: Uniaxial Tensile Test on FAAF Seam

The resulting stress-strain relationships for the Nylon square specimens at four different angular rotations from the fill dimension are shown in Figure 82. Notice the stress-strain curve for the ‘fill’ dimension has the largest slope, which implies the dimension with the greatest resistance to deformation. This is expected because the fill direction of the canopy defined in Figure 77 has the most number of strengthening stiffeners per cross-sectional length. The warp dimension also has a large number of strengthening stiffeners, so the slope of the stress-strain curve is larger than that of the 22.5 and 45 degree offsets from the fill dimension. The 45 degree offset contains the largest deformation for small pressures, because at an angle of 45 degrees from the fill direction, there is the least amount of tension strength in the Nylon canopy.
Two seam specimens were used in uniaxial testing to determine the difference (if any) in the stress-strain relationship between the top and bottom seams of the single-cell parafoil defined by the blue highlighted lines in Figure 83.
The resulting stress and strain relationship for the top and bottom seams showed a very similar relationship as shown in Figure 84.

Figure 84: Top and Bottom Seam Stress Data
3.5.2. Biaxial Tensile Test

To accurately model the change in the stress-strain relationship for different forces acting at different orthogonal directions of the Nylon canopy, a biaxial tensile test was conducted on square specimens of the Nylon canopy. Figure 85 demonstrates the theoretical experimental test setup of the biaxial tensile test.

![Biaxial Tensile Test Diagram](image)

Figure 85: Biaxial Tensile Test (Galliot, Luchsinger)

As shown in the center of Figure 85, if there exists a force \( F_x \) in the x dimension, a change in deformation in the x direction \( \Delta L_x \) will exist. However, if there was another force already present in the y direction \( F_y \), a \( \Delta L_{45^\circ} \) will occur and alter the linear or nonlinear relation between stress and strain in the x dimension. The goal of the biaxial test is to extract the \( \Delta L_{45^\circ} \) when applying two forces orthogonal to each other on the parafoil canopy specimen at the same time.

Figure 86 shows the actual biaxial test setup used on the canopy specimens
courtesy of JJ Menen of Performance Designs. During testing, the voltage count (voltage required to move the motors shown in Figure 86) and number of motor steps (rotation of the motor) are recorded at 100 Hz. The voltage count is related to the force acting on the material and motor steps is a measure of the deformation of the material being tested.

Figure 86: Biaxial Test Setup

To relate voltage count to force, the biaxial test was calibrated using an in-line force measuring device shown in Figure 87 where the correlation between voltage count and force was connected by a constant of proportionality. Shown in Figure 88, the force versus voltage count was plotted to distinguish the relationship between voltage count and force applied in the biaxial tests.
Figure 87: Biaxial Force Calibration

Figure 88: Biaxial Force Calibration Linear Data
The slope for the fill dimension was determined to be approximately 0.0466 pounds per voltage count while the slope for the warp dimension was determined to be about 0.0459 pounds per voltage count. This yielded the following relationships between motor voltage count and stress applied to the fill and warp dimensions of the canopy:

\[\text{Fill Stress} = \frac{\text{Fill Slope} \times \text{Fill Motor Voltage Count}}{L_C \times \text{Nylon Thickness}}\]  
(1)

\[\text{Fill Stress} = \frac{0.0466 \times \text{Fill Motor Voltage Count}}{3 \text{ in} \times 0.003 \text{ in}}\]  
(2)

\[\text{Fill Stress} = 5.178 \times \text{Fill Motor Voltage Count}\]  
(3)

\[\text{Warp Stress} = \frac{\text{Warp Slope} \times \text{Warp Motor Voltage Count}}{L_C \times \text{Nylon Thickness}}\]  
(4)

\[\text{Warp Stress} = \frac{0.0459 \times \text{Warp Motor Voltage Count}}{3 \text{ in} \times 0.003 \text{ in}}\]  
(5)

\[\text{Warp Stress} = 5.100 \times \text{Fill Motor Voltage Count}\]  
(6)

where:

\[L_C = \text{Length of Clamp (d' in Figure 85)}\]

The motor steps were measured using a micrometer and it was determined that for every 200 motor steps recorded, 0.0372441 inches were displaced by the motor’s pull on the material. Figure 89 demonstrates the setup of the biaxial test completed a square 9in x 9in sample of the canopy material.

\[\text{Strain} = \frac{0.0372441 \text{ inches}}{200 \text{ motor steps}} \times \text{Motor Steps}\]  
(7)

\[\text{Strain} = \frac{0.0372441 \text{ inches}}{200 \text{ motor steps}} \times \text{Motor Steps}\]  
(8)

\[\text{Strain} = 2.069 \times 10^{-5} \times \text{Motor Steps}\]  
(9)

where:

\[L_o = \text{Original square length of the square specimen}\]
Smaller forces were applied to the biaxial test than the uniaxial test to avoid deformation due to overstraining of the Nylon. Overstraining the Nylon would result in false data points or a ‘shift’ of the stress-strain curve and would not accurately extract the dimensional deflection change. To associate the relationship between different deformations caused by orthogonal forces acting at one time, five force ratio scenarios were acted on the canopy specimens along the warp and fill dimensions: 5:1, 2:1, 1:1, 1:2 and 1:5 (Warp:Fill). For example, in the first case of 5:1, a force was applied in the fill direction while a force of five times greater was applied in the warp direction. Shown in Figure 90 is the raw test data from the five different force cases for the warp and fill directions. Figure 91 is the same test data, but after the voltage count and motor steps were converted to stress and strain using Equations 3, 6 and 9 to determine the stress-strain plots of the canopy material.
Figure 90: Biaxial Raw Test Data
Figure 91: Biaxial Stress-Strain Data
To integrate the stress-strain relationship of the Nylon canopy and seams into the FSI model, data points were taken from the stress-strain plots for both the uniaxial and biaxial tests and entered via tabular data points into ANSYS Engineering Data tool. The data points were then curve-fitted into a non-linear material model using the 9 parameter Mooney–Rivlin hyperelastic curve-fitting technique to determine the material constants for the strain-energy density function as shown in Equation 10 below. Strain-energy density is related to stress and strain by Equation 11.

\[
W = \sum_{p,q=0}^{N} C_{pq} (\bar{I} - 3)^p (\bar{I}_2 - 3)^q + \sum_{m=1}^{M} D_m (J - 1)^{2m} \quad (10)
\]

\[
W = \frac{1}{2} \sigma_{xx} \varepsilon_{xx} \quad (11)
\]

where:
- \( W \) – Strain energy potential
- \( \sigma_{xx} \) – Stress
- \( \varepsilon_{xx} \) – Strain
- \( C_{pq} \) are the material constants
- \( D_m \) are volumetric material constants

Since the Nylon canopy material is not compressible, the volumetric parameters (all D parameters) in Equation 10 were set to zero. Figure 92 demonstrates the process for incorporating the seam test data into the ANSYS Engineering Data tool using the Nylon density of 1.2 gram per cubic centimeters.
Figure 92: Mooney-Rivlin Curve-Fitting in ANSYS
3.6. Fluid Structure Interaction Model

The fluid-structure parafoil model will encompass the data exchange between the CFD and FEA solvers governed by the ANSYS System Coupling program as part of the program flow shown in Figure 93. The coupling process starts with the CFD solver to determine the initial pressures acting on the parafoil structure. Then a displacement occurs in the FEA solver which in turn changes the flow fields around the parafoil. Each of the solvers will reach a solution in one cycle of the FSI model until an equilibrium in the change in displacement of the parafoil model is met.

![Figure 93: FSI Structure](image)

The program flow of the FSI coupling process shown above in Figure 93 is the same as Figure 94 when viewed in the ANSYS Workbench environment by the user. To better understand the ANSYS environment, a blue line with a square in Figure 94 indicates a one-way exchange of data such as importing the geometry of the parafoil model into Design Modeler for both cells B2 and C3 while a blue line with a circle
indicates a two-way exchange of data such as the forces and displacements leaving the CFD and FEA solvers, while crossing through the system coupling manager. The complete flow of data in the ANSYS Workbench architecture shown in Figure 94 can be summed up in the following order with the cells marked in parenthesis:

- The parafoil geometry is imported in ANSYS Design Modeler (B2 and shared with C2).
- ANSYS Engineering Data exports the material properties from the test data to ANSYS APDL (A2 to C1).
- The parafoil geometry in ANSYS Design Modeler is passed off to ANSYS ICEM where parallel meshing occurs for both ANSYS Fluent and APDL (B3 – fluid mesh and C4 – structural mesh).
- The fluid domain is created and initialized in ANSYS Fluent (B5).
- The ANSYS APDL FEA structural setup is initialized (C5).
- ANSYS System Coupling is initialized (D2).
- The first iteration of the FSI cycle starts:
  a. ANSYS Fluent in B5 completes a converged cycle.
  b. The forces acting on the nodes of the parafoil surface are passed through System Coupling to be mapped to APDL and recorded in ANSYS Post CFD for a data step.
  c. ANSYS APDL solves for the displacements of the parafoil canopy.
  d. The displacements are recorded in ANSYS Post CFD and passed back through System Coupling to Fluent where a new converged cycle is solved.
  e. This cycle continues until the final time setup in System Coupling is reached.
- Post Processing of Data in ANSYS Post CFD

Figure 94: ANSYS Workbench Setup
3.6.1. Computational Fluid Dynamics Model

The initial mesh structure for the CFD solver (ANSYS Fluent) was created using ANSYS ICEM using local and global automatic tetrahedron meshing techniques for different bodies and faces of the parafoil surface model. Figure 95 and Figure 96 demonstrate the CFD fluid domain environment both in theory and in ICEM mesher.

![Figure 95: Parafoil Fluid Domain Setup](image-url)
Tetrahedrons cells were used in the construction of the fluid domain mesh because of the limitations of the ANSYS software. A structural mesh would have been ideal for the parafoil shape, however, tetrahedrons are more flexible for morphing geometry in a dynamic environment.

A growth rate of 1.2 was used to scale the sizing of the tetrahedrons in the CFD domain from the parafoil mesh surface. This implies that the tetrahedrons will grow in size by a factor of 1.2 for every layer away from the parafoil model in the CFD domain. This keeps the precision of cells high around the parafoil surface while keeping the overall number of tetrahedron cells for the entire model low. This growth rate of fluid cells is more easily visualized from the isometric view of the CFD domain in Figure 96.
A close-up cut section of the parafoil canopy (cut at the center airfoil) in the CFD domain mesh near the parafoil surface is shown in Figure 97 through Figure 98. The size of the tetrahedron cells is a function of the curvature of the parafoil. For example, on sharp corners, the size/volume of the tetrahedron cells are smaller.

Figure 97: ICEM - Parafoil Assembly Meshing

Figure 98: ICEM – Isometric 3D Tetra-View
The process of refining the mesh parameters by helping the CFD solver to converge with less error and fewer iterations was driven by keeping the orthogonal quality ratio high and skewness low for the overall CFD domain. The orthogonal quality of a mesh cell is most simply explained by taking the smallest dimension of one of the sides of a cell and dividing it by the largest dimension whereas skewness is the measure of angular twist the overall shape of a mesh cell. For example, a perfect cube has an orthogonal quality of 1 and a skewness of 0, because all the sides are the same length and there is no twist. A rectangle would have an orthogonal quality of less than 1 because it has a larger and smaller side but would still have a skewness of zero because there is no twist or warp to its shape. An example of a cell with a low orthogonal quality and a high skewness would be one in the shape of a very thin twisted plate. This makes meshing a thin surface very difficult in a fluid domain and is why the parafoil is treated as a zero-thickness plate in the CFD mesh so that the cells of the parafoil canopy do not occupy any cells in the CFD domain and simply just connect to the pressure contours around the parafoil. In ANSYS Multiphysics, orthogonal quality and skewness are defined by Equation 12 and Equation 13 respectfully.

\[
Orthogonal\ Quality = \min \left( \frac{A_i^*d_i}{|A_i||d_i|}, \frac{A_i^*c_i}{|A_i||c_i|} \right) \tag{12}
\]

where:

\(A\) – Normal direction of a face  
\(c\) – Three dimensional distance vector between center and \(A\)  
\(d\) – Three dimensional distance vector between center and midpoint of side
Figure 99: Orthogonal Quality of a Tetrahedron Cell

\[
\text{Skewness} = \max \left( \frac{\phi_{\text{max}} - \phi_0}{180 - \phi_0}, \frac{\phi_0 - \phi_{\text{min}}}{\phi_0} \right)
\]  

(13)

Figure 100: Skewness

An approximation was applied to the cusp of the parafoil to keep the orthogonal quality high. Shown in Figure 101, the parafoil cusp was rounded in the geometric model and a local mesh size control was applied to the keep the mesh cells’ orthogonal quality high and skewness low. The mesh cells in and around the cusp were controlled to be 0.1 inches in length (meaning each linear edge of every tetrahedron was scaled to be at least 0.1 inches long).
Analyzing the mesh metrics of the CFD is absolute key in setting up a good FSI model that will not end up with large residuals for each time step. The mesh statistics of the parafoil model after CFD meshing yielded the distribution of orthogonally cells and skewness shown in Figure 102 and Figure 103 for the overall CFD model. No mesh cells were under an orthogonal quality of 0.15 or over a skewness of 0.95 to allow for optimal dynamic performance in the FSI model. The averages for both the overall orthogonal quality and skewness were 0.9 and 0.175 respectfully.

In addition, a mesh sensitivity analysis was completed using these mesh settings discussed to optimize the CFD mesh using the smallest number of mesh elements and nodes without reducing the deformation resolution of the end result. It was found that with the single-cell parafoil model used in this thesis, no less than 270,000 elements and no less than 38,000 nodes on the parafoil surface is sufficient to model the single-cell parafoil deformations. Any further adjustment to the mesh refinement was found to either result in deformed mesh geometry or high pixelated deformations in the FSI process.

This mesh model for the CFD domain was then loaded into ANSYS Fluent CFD solver for the first iteration of the FSI cycle.
Figure 102: ICEM – Orthogonal Statistics

Figure 103: ICEM – Skewness Statistics
Once the mesh was imported in ANSYS Fluent, the Navier-Stokes equations for conservation of mass and momentum (Equations 14 and 15) combined with the transient shear-stress transport (SST) $k-\omega$ model was used for the computation of the fluid flow around the parafoil using the turbulent kinetic energy and specific dissipation rate equations (Equations 16 and 17). Equations 18 through 28 are used to help simplify the form of Equations 16 and 17. The benefit of using the $k-\omega$ SST model is its effectiveness for low Reynolds number flows and flexibility for adverse pressure gradients and separating flow. This helps with modeling the unpredictable turbulent flow around a parachute. The $k-\omega$ SST model also switches to a $k-\varepsilon$ model in the free-stream sections of airflow, thereby avoiding the common $k-\omega$ turbulence model problem where the solution is too sensitive to the inlet free-stream turbulence sensitivity. A velocity inlet with turbulence intensity of 10% and a hydraulic diameter of 0.5 feet was used to model the inflow of the parafoil CFD domain, while a pressure outlet was used to model the outflow as shown back in Figure 95 for defining the inlet and outlet of the CFD domain.
\[
\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} = 0
\]  
(14)

\[
\frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \nabla \vec{u} = -\frac{1}{\rho} \nabla p + v \nabla^2 \vec{u} + \frac{1}{3} v \nabla (\nabla \cdot \vec{u}) + \vec{g}
\]  
(15)

where:
\[
\begin{align*}
\rho &= \text{Density} \\
\vec{u} &= \text{Flow field} \\
v &= \text{Component of flow field} \\
p &= \text{Pressure} \\
\vec{g} &= \text{Body accelerations}
\end{align*}
\]

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + \bar{G}_k - Y_k + S_k
\]  
(16)

\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_i} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_i} \right) + G_\omega - Y_\omega + D_\omega + S_\omega
\]  
(17)

\[
\Gamma_k = \mu + \frac{\mu_t}{\sigma_k}
\]  
(18)

\[
\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega}
\]  
(19)

\[
\mu_t = \left( \frac{\rho k}{\omega} \right) \max \left( \frac{1}{\alpha^*}, \frac{SF_2}{a_1 \omega} \right)
\]  
(20)

\[
\alpha^* = \alpha^*_o \left( \frac{\alpha^*_o + Re_t / \Omega_k}{1 + Re_t / \Omega_k} \right)
\]  
(21)

\[
\sigma_k = \left( \frac{F_1}{\sigma_{k,1}} \right) + \left( \frac{1 - F_1}{\sigma_{k,2}} \right)
\]  
(22)

\[
\sigma_\omega = \left( \frac{F_1}{\sigma_{\omega,1}} \right) + \left( \frac{1 - F_1}{\sigma_{\omega,2}} \right)
\]  
(23)

\[
F_1 = \tanh \phi_1^4
\]  
(24)
\[
\phi_1 = \min \left[ \max \left( \frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho y^2 \omega} \right), \frac{4\rho k}{\sigma_{\omega,2} D_\omega^+ y^2} \right]
\] (25)

\[
D_\omega^+ = \max \left[ 2\rho \left( \frac{1}{\sigma_{\omega,2}} \right) \left( \frac{1}{\omega} \right) \left( \frac{d k}{d x_j} \right) \left( \frac{d \omega}{d x_j} \right), 10^{-10} \right]
\] (26)

\[
F_2 = \tanh \phi_2^2
\] (27)

\[
\phi_2 = \max \left[ 2\rho \frac{\sqrt{k}}{0.09\omega y}, \frac{500\mu}{\rho y^2 \omega} \right]
\] (28)

where:

\(\vec{G}_k\) = *Generation of turbulence kinetic energy due to mean velocity gradients*

\(G_\omega\) = *Generation of \(\omega\)*

\[
Re_t = \frac{\rho k}{\mu \omega}
\]

\[
R_k = 6
\]

\[
\alpha_\infty = 0.52
\]

\[
\sigma_{k,2} = 1.0
\]

\[
\alpha_o = \frac{1}{9}
\]

\[
\beta_i = 0.072
\]

\[
\sigma_{\omega,1} = 2.0
\]

\[
\sigma_{\omega,2} = 1.168
\]

Properly defining the dynamic mesh properties are crucial in a successful FSI model, especially a model with a high amount of free-roaming displacement such as a parachute. Smoothing, layering and re-meshing techniques were used to dynamically model the motion of the CFD domain and smoothing constraints were used to keep the mesh cells as clean and as possible for motion. The region of the CFD mesh that represents the air volume was dynamically smoothed using a diffusion-based dynamic mesh, shown in Equations 29 through 31 allowing for a uniform displacement of CFD cells. Where a diffusion parameter (\(\alpha\)) of 1.5 was used to relate the normalized boundary distance and volume of the mesh cells.

\[
\vec{x}_{\text{new}} = \vec{x}_{\text{old}} + \vec{u} \Delta t
\] (29)

\[
\nabla (\gamma \nabla \vec{u}) = 0
\] (30)

\[
\gamma = \frac{1}{d\alpha} = \frac{1}{V\alpha}
\] (31)
where:
\[ \bar{x} = \text{Mesh node displacement (from center of cell)} \]
\[ \bar{u} = \text{Mesh displacement velocity} \]
\[ \gamma = \text{Diffusion coefficient} \]
\[ \alpha = \text{Diffusion parameter (adjustable by user)} \]
\[ d = \text{Normalized boundary distance} \]
\[ V = \text{Normalized cell volume} \]

Dynamic layering was another mesh feature that was applied to the CFD domain which encompasses how the mesh domain reacts to being ‘compressed’ or ‘stretched’. Therefore, compression and tension restraints were set to the corresponding mesh cells in the CFD domain called the **split factor** and **collapse factor** as defined in Equations 32 and 33. A split factor in dynamic layering is a value that determines the limit to the height-to-base ratio of a stretched mesh cell before the mesh cell is split in half to create two cells. A collapse factor sets the height requirement for a layer of mesh cells until they are merged with other mesh cells in the event that the mesh layer is compressed. A split factor of 0.4 and a collapse factor of 0.2 (the defaults of ANSYS dynamic layering) resulted in a good dynamic mesh layering response between the CFD domain and the parafoil canopy mesh cells.

\[
\begin{align*}
    h_{\text{min}} &> (1 + \alpha_s) h_{\text{ideal}} \\
    h_{\text{min}} &< \alpha_c h_{\text{ideal}}
\end{align*}
\]

where:
\[ h_{\text{ideal}} = \text{Average height of a cell layer} \]
\[ h_{\text{min}} = \text{Height of a mesh layer after tension or compression} \]
\[ \alpha_s = \text{Dynamic layer split factor} \]
\[ \alpha_c = \text{Dynamic layer collapse factor} \]

Finally, re-meshing techniques were applied to the CFD domain under the following conditions shown in Table 3 below. This implies that for every instance a CFD cell breaches one of the values specified in Table 3, the ICEM mesher will re-mesh the
specific section of the CFD domain that violated the conditions in Table 3.

Table 3: Re-meshing Limitations

<table>
<thead>
<tr>
<th>Re-Meshing Dimension</th>
<th>Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum Length Scale (ft)</td>
<td>0.003</td>
</tr>
<tr>
<td>Maximum Length Scale (ft)</td>
<td>8</td>
</tr>
<tr>
<td>Maximum Cell Skewness</td>
<td>0.95</td>
</tr>
<tr>
<td>Maximum Face Skewness</td>
<td>0.9</td>
</tr>
<tr>
<td>Size Remeshing Interval</td>
<td>1</td>
</tr>
</tbody>
</table>

The PISO pressure-velocity scheme was used to help aid the solution of the solver by using a Courant controlling technique called ‘solution steering’. Solution steering is a capability new to ANSYS that allows the Courant number to change based on the change in size and volume of the CFD cells (after dynamic re-meshing, layering and smoothing occurs). Solution steering works by starting the solver with an initial Courant number and after the first iteration, the PISO solution steering will change the Courant number based on the change of cell size. Equations 34 and 35 demonstrate the calculation of the Courant number both for n dimensions and three dimensions respectively.

\[ C = \Delta t \sum_{i=1}^{n} \frac{u_{x_i}}{\Delta x_i} \]  \hspace{1cm} (34)

\[ C = \frac{u_x \Delta t}{\Delta x} + \frac{u_y \Delta t}{\Delta y} + \frac{u_z \Delta t}{\Delta z} \]  \hspace{1cm} (35)

where:
- \( C \) = Courant number
- \( \Delta t \) = Time step
- \( \Delta x, \Delta y, \Delta z \) = Average mesh cell size in one dimension
- \( u \) = Velocity in one dimension
An initial Courant number of 1 was used for the dynamic modeling of the parafoil and if the velocity of the airflow (12 mph from experiment) and initial mesh cell size (0.1 ft) is known, Equation 34 can be solved for the time step required for the implicit FSI model in the system coupling as shown in Equation 36.

$$\Delta t = \frac{C \Delta x \Delta y \Delta z}{u_x \Delta y \Delta z + u_y \Delta x \Delta z + u_z \Delta x \Delta y}$$ (36)

When using a wind speed of 17.2 ft per second (12 mph) as $u_x$, a Courant number of 1 and cell size of 0.1 ft as $\Delta x$, a $\Delta t$ of 0.0005 sec was calculated to be the maximum time step that is needed to properly model the FSI parafoil. To be conservative, a time step of 0.0004 sec was used as the time step in the FSI coupling process.
3.6.2. Finite Element Analysis Model

The parafoil was originally created using surface elements that occupy no volume in the geometric modeler, however during the FEA meshing process in ANSYS ICEM, these surface elements are thickened outwards to occupy a thin shell volume. The resulting mesh of the parafoil canopy is 0.003 inches thick using ANSYS APDL shell elements from ANSYS ICEM mesher as shown in Figure 105. 0.003 inches is the same thickness as the Nylon parafoil canopy provided by Performance Designs Inc. The Nylon lines attachments also have been extracted outwards to occupy a cylinder volume sharing common nodes from end to end.

Figure 105: FEA Mesh
For the FEA analysis, the seam sections of the canopy geometry have been identified as a different type of Nylon (seam sections shown in Figure 106) and have been assigned the corresponding mechanical properties as identified by the stress-strain tests in Section 3.5. As a result of this, the mesh shell sections in the FEA model share common nodes and edges at the connection lines between the canopy and seam segments allowing for a higher fidelity model of the canopy. The mesh resolution was kept to match the same density and number of nodes as in the CFD mesh to keep the mesh mapping quality as high as possible. The mesh sensitivity study conducted on the CFD mesh keeps the mesh resolution as high as possible without causing problems with deformations.

![Figure 106: FEA Seams](image)

Three different restraints were tested in the FEA model to observe the behavior of the parafoil canopy displacements. Shown on the top left in Figure 107, the first type includes restraining the parafoil on either side of the canopy, which ultimately fixes the sides of the parafoil in place (keeping the sides rigid and motionless) allowing the top, bottom and center airfoil sections to move freely. Another restraint (shown on the top right of Figure 107) is to fix the parafoil at the 10 line attachment points to allow motion
throughout the entire canopy except where the Nylon suspension lines would be attached. Lastly, and most useful to the designer, is to include the Nylon suspension lines of the parafoil and restrain the model only at one point (where the load of the parafoil would naturally be) as shown at the bottom of Figure 107. These three restraint configurations were created in the event that the experimental test would be unable to meet a stable deformation of the parafoil canopy using one point in fixed space (the bottom configuration of Figure 107).
Restraining the parafoil at the location of the load (at one point) would allow for the entire parafoil to move freely in all dimensions when exposed to the free stream velocity. Using ANSYS APDL to solve for the normal and parallel reaction forces to hold the parafoil in place at this single point in space with respect to the free stream velocity of the inflow would ultimately yield the lift and drag forces of the parafoil as shown in Figure 108.

Figure 108: Lift and Drag Forces
The finite element model for the parafoil utilizes the Mooney-Rivlin constants derived from the method in Section 3.5 to establish the relationship between stress-strain and strain-energy density to solve for the displacements of the shell elements that make up the FEA mesh using the Lagrangian conservation of momentum from Equation 37. Additionally, solution stabilization was added to the structural solver in the form of constant force dampening. It was found through computational testing that in the event of high-pressure fields from the CFD solver that excess forces would cause excessive deformations in the canopy profile. To fix this, a constant damping factor of 0.1 was applied to the structural mesh elements that failed to converge after the first few sub steps of the structural solver.

\[
\rho \frac{\partial^2 \ddot{x}}{\partial t^2} = \nabla \cdot \dot{\tau} + \rho \ddot{\mathbf{g}}
\]

(37)

where:
- \( \ddot{x} \) = Structural displacements
- \( \rho \) = Structure density
- \( \dot{\tau} \) = Cauchy stress tensor
- \( \ddot{\mathbf{g}} \) = Body forces
3.6.3. System Coupling

For the FEA and CFD solvers to exchange force and displacement data from one solver to the other, they need a common zone or boundary to be identified and created to share data. This zone in FSI applications is called the *mapping zone*, which is a collection of nodes within the surfaces that exchange data from one solver to the other. These mapped surfaces are identified in each solver and are later used in the system coupling application of ANSYS to be overlaid, one on top of the other, where an exchange of data commences as shown in Figure 109.

![Mapping Surfaces](image)

**Figure 109: Mapping Surfaces (Baskut, Akgul, 2011)**

Similar to Figure 109, the parafoil model has a structural mapped zone and a fluid mapped zone that share the same mesh properties in each of the solvers. Both the fluid and structural mapped zones occupy the same surface on the parafoil canopy and share nodes where an exchange of data commences during the FSI coupled process. Shown in Figure 110, the properties of the CFD mapped zone mesh are similar to the tetrahedrons used to create the CFD domain. The FEA mapped surface is comprised of a structural mesh. Both of the mesh surfaces in Figure 110 exist at the same location and share nodes for the data transmission between the two solvers.
The resolution of these mapped surfaces is the same as the surface properties of the meshes created in the ICEM for the FEA and CFD solvers and it’s very important that these meshes have similar resolution. In other words, the mapped mesh surfaces need to have the approximate same number of nodes and cells as the other. The higher the resolution of cells in the mesh, the more accurate the motion of the canopy.

At the start of the system coupling process, the mapped cells for each coupled mesh are analyzed and displayed to the user as a percentage of what how many cells from one solver are in contact with the other. Figure 111 demonstrates the mapping analysis at the startup of an example FSI application. The mapping is only as good as the lowest number on the system coupling summary. In Figure 111, the lowest percentage is 91. This implies that only 91 percent of the entire nodes for one mapped surface is in contact with the other, i.e. only 91 percent of the forces or displacement data (depending on which mapped zone) is transferred to the other mapped surface. This number for any analysis should be as high as possible. Anything below 100 percent is a risk leading to bad accuracy within the modeling.
The parafoil model used in this analysis had a minimum mapped accuracy of 96 percent. Some shapes are hard to get 100 percent mapping because of the nature of the mesh properties. The parafoil model has a combination of structural and tetrahedron mesh cells and it’s unlikely that all the mapped mesh surfaces share a common node at the more complicated mesh surfaces such as the cusp and seam connections. The cusp and seam connections have far more nodes and cellular mesh density than a typical face of the canopy, therefore, these are the tricky locations to get mapping surfaces to meet properly. Local mesh controls at the cusp, seams and boundaries were used to increase the percentage of mapped nodes for the parafoil.

The coupled interaction between the CFD and FEA solvers in ANSYS are recorded in ANSYS Post CFD. A record of whether or not each solver has reached convergence successfully and properly sent a data exchange between each solver is recorded as shown in the solution information window of ANSYS in Figure 112.
Solution controls are used in the FSI process to control the number of times a FSI cycle is completed to reach convergence. For one complete iteration of a FSI cycle to converge successfully, the FEA and CFD solvers may run multiple times until convergence is reached (implicit FSI modeling). Explicit FSI coupling is when the number of FEA and CFD cycles is kept to one and regardless of the residuals, the time step moves on. Because explicit coupling only uses one FSI cycle per time step, the time step must be set very low. Since this thesis is using residual convergence to run multiple FSI cycles per time step (implicit), the time step can be set much higher than that of an
explicit solver.

System Coupling allows for the user to set the number of FSI iteration attempts. This means that after one full FSI iteration (the FEA and CFD solvers reached a solution for one time step and exchanged data) if the residual values are too low, the entire cycle may be re-run with new initial conditions from the previously unacceptable FSI cycle. Allowing ANSYS to re-run cycles over and over can lead to an extremely long time to reach a FSI solution.

For example, the FEA solver may need to run 5 times to reach convergence and the CFD solver may need to run 20 times to reach convergence. This would imply that one FSI cycle (one time step) will take a total of 100 FEA and CFD cycles combined. If the residuals are not below a desired threshold at the end of this FSI cycle, the entire time step will be run again using updated initial conditions from the prior FEA and CFD runs. If a particular FSI cycle takes 3 runs to finish, then that means the FEA and CFD solvers will have a combined 300 times to reach convergence on one time step.

These numbers are not consistent with each step due to varying forces and deformations. Adjusting the convergence criteria (controlling at what limit the residual is ‘acceptable’ for each solver) and setting the maximum number of iterations allowed for each solver to reach converge by ANSYS System Coupling will control the total time a FSI takes to run. Ideally, setting a large number of allowable iterations and a high residual limit will yield good results but may take a long time and may even be unnecessary for reaching a good conclusion.

Shown in Figure 113, the residuals from the data transfers are shown and plotted to the user. Each FSI iterative cycle starts with a high residual, re-runs the solver with the
updated initial conditions then updates the plot. As the solvers reach a convergence with a lower residual, the FSI cycle starts a new cycle with the next time step. Cycles that have failed to meet the convergence criteria for one FSI cycle are shown as repeated residual values (i.e. coupling iterations 100 through 120 of Figure 113). These FSI cycles are repeated until the either the residuals from one of the solvers is low enough or the maximum number of allowable FSI iteration attempts is reached. In the case of Figure 113, the residuals from the FEA solver became too high and were rejected from that cycle and the FSI cycle was repeated until the maximum number of allowable FSI iterations was reached.

![Figure 113: System Coupling – Layout](image)

The FSI System Coupling parameters setup for the single-cell parafoil model were run at time step increments of 0.4 milliseconds (derived from Courant Number analysis) for a total of 0.192 seconds of simulated time. This implies that 480 FSI cycles took place between the FEA and CFD solvers. At each time step, displacements and force
reactions for three different restraint configurations (Figure 107) were run for three different parallel models to use for comparison to the experimental test. As mentioned earlier, the three different restraint configurations were modelled in the event that the single-cell parafoil specimen was unable to reach an approximate steady-state equilibrium position in free space.

The computation power required to model a FSI parafoil is extensive depending on the total number of mesh cells and cell density of the FEA and CFD meshes. A mesh sensitivity study was completed on the single-cell parafoil model for both the FEA and CFD meshes to reduce the total number of mesh elements and nodes as much as possible without suffering any resolution loss in deformation. The model in this thesis was optimized as much as possible to run on a typical PC of 16 GB RAM, 8 core, 3 GHz processor. However, using a standard PC, the parafoil model used in this thesis still takes about four days to model a complete FSI cycle to an approximate steady-state condition.

Ultimately, a steady-state equilibrium between the parafoil’s displacements and force reactions is the goal of the FSI model, but due to the complexity of the hyperelastic material at a very thin shell thickness of 0.003 inches, the steady-state FSI modelling capability of ANSYS Multiphysics has a very hard time to each any convergence. Even using a time varying transient approach to the FSI modelling is difficult for ANSYS Multiphysics to reach a convergence. After several attempts to adjust modelling and meshing techniques, dynamic mesh parameters and motion study, the parafoil model was still unable to reach full convergence even after 480 FSI iterations.
3.7. Experimental Model

Restraining the single cell parafoil subsection at one point using suspension lines described in Section 3.6.2 was unobtainable because the parafoil specimen was unstable in flow with only one cell. Therefore, collecting the force reactions for the entire single-cell parafoil proved too difficult to empirically determine. Instead, the Nylon canopy was housed inside the experimental setup shown in Figure 114 to collect the longitudinal and vertical canopy deformations to use in comparison with the FSI model. This experimental setup consists of a wooden housing with dimensions of 4ft x 8ft and a variable span of up to 28.25in for the single-cell parafoil (the full width of the parafoil).

Figure 114: Single-cell Parafoil Experimental Setup
Figure 115: Experimental Setup – Variable Span

Figure 116: Experimental Setup – Front
The airflow is controlled by a gas-powered prop (Figure 117) at the diffuser end of the parafoil wind tunnel simulating a low-pressure out-flow (Figure 119), similarly used in the ANSYS FLUENT CFD model. The benefit of a low-pressure outflow is that the air is drawn through the experiment in a more uniform profile without the pressure differential created from the rotational flow by the propeller.

Figure 117: Gas Powered Prop Front View
(Eagle Flight Research Center, Embry-Riddle Aeronautical University 2014)
Figure 118: Gas Powered Prop Side View
(Eagle Flight Research Center, Embry-Riddle Aeronautical University 2014)
For additional aid in straightening the inflow, a four in thick honeycomb panel was installed at the inlet of the experimental setup shown in Figure 120 and a close-up of the honeycomb cells in Figure 121.
Figure 120: Experimental Setup – Honeycomb Inlet Exterior

Figure 121: Experimental Setup – Honeycomb Inlet Interior
4. Analysis

4.1. Experimental Results

One side of the experimental setup consisted of a Plexiglass window (Figure 122) that allowed for a view of the span of the parafoil for measurements while being inflated by the air without interrupting the airflow. On the wooden panel behind the parafoil in Figure 122 was a grid consisting of one-inch squares for scaling of the parafoil deformation. Once the airflow reached a slow and steady 12 mph, pictures were taken of the parafoil deflections and digitized for analysis.

![Figure 122: Single-Cell Canopy Deformation](image)

Using the grid as the scalar unit for an inch in the digitized pictures, the outline of the side airfoil shape of the parafoil was overlapped with the outline of the inflated center cell of the parafoil as shown in Figure 123.
The top section of the parafoil outline was rotated to be horizontal where trendlines were added to the deflated and inflated profiles (Figure 124). The difference of these profiles was taken and divided by the normalized unit of an inch from the captured picture. The resulting deflection as a function of chord length (left is the trailing edge) was plotted in Figure 125.
Figure 125 depicts the approximate inflation as a function of chord length of the parafoil at 12 mph in inches and can be used to compare with the computational model.

Figure 125: Deflection of Canopy Deformation
4.2. FSI Model

The single-cell FSI model takes anywhere from four to five days to over a week to run on a typical PC with at least eight cores, 16 GB of RAM and a 3 GHz processor, depending on the resolution of the CFD and FEA mesh. Tuning of the FSI parameters such as the dynamic mesh settings, maximum number of iterations run per solver, FLUENT, APDL and FSI coupling settings have been tested and run on a far more simplistic model for time saving. Only on a successful completion of a simple model would yield a test for the single-cell parafoil model to produce accurate results. This process was repeated to finely tune the single-cell parafoil FSI model.

Such an example of a simplified model is shown below in Figure 126. This model is comprised of two 12 inch x 12 inch plates that would act as restraints (shown in blue on the left side of Figure 126) holding one curved 12 x 24 inch sheet. This curved sheet would simulate a piece of the parafoil canopy that was held at a 90-degree angle to airflow at the same speed as experienced in the experimental model. This simplified model is comprised of the same bonded-surface geometry defined in Chapter 3 including the same material properties of the FAAF Nylon canopy at a total of 0.003 inches thick.

![Figure 126: Simple Test Model](image)
A 12 mph free stream velocity is introduced perpendicular to the curved plate to induce a motion in the canopy. After the FSI model runs for approximately 20 hours real time simulating 0.3 sec of model time, the mesh motion is captured in Figure 127 where a maximum deflection of 9.22 inches occurs at the center of the curved canopy.

The deflections witnessed in this simplified plate model are how the dynamic mesh and structural solver settings were determined on a small scale before being implemented in the single-cell parafoil model. The only difference between this simplified plate model and the parafoil is the meshing size and complexity of the parafoil’s geometry. Simply put, if the settings for the FSI work in the simplified plate, then the FSI model of the parafoil should behavior very similar.
Figure 127: FSI Test Model
The single-cell parafoil FSI model was run for 371 time steps, each at 0.5 milliseconds for a total of 0.1855 seconds of simulated time. During this solution, the implicit FSI model required approximately 1400 coupling iterations to complete (almost each time step required about 5 FSI cycles to reach convergence). The model was stopped when an equilibrium value of the data transfers between the structural and CFD solvers was detected in the ANSYS System Coupling monitor shown in Figure 128 (the repeated pattern of data transfers after coupling iteration 900).

![Figure 128: Parafoil FSI Model Convergence](image)

The resulting deformations of the single-cell parafoil canopy after the FSI model has reached equilibrium is shown in Figure 129 and Figure 130.
The maximum displacement of the parafoil canopy is plotted as a function of normalized chord length in Figure 131 (similar to the analysis for the experimental setup in Section 4.1).
Figure 131: Post CFD – 12 mph Max Displacements

The figures on the next few pages plot the velocity streamlines and pressure contours across three cross-sectional planes defined using SolidWorks in Figure 132. These three planes are normal to the span of the parafoil and run parallel from the inlet to the output of the fluid domain. These three planes are defined as the ‘center rib’, ‘quarter cell’ and ‘end rib’ which correspond to the center of the parafoil, the halfway point between the center rib and an end rib, and the end rib respectfully.

The velocity flow fields for the center rib and quarter cell cut sections contain a smooth, laminar flow. However, the velocity flow fields located at the end rib, where the parafoil is held at the boundary, contains some flow separation on the top of the parafoil canopy. This flow separation could possibly be the result of the boundary interference between the wall restraint and the parafoil canopy.
Figure 132: Velocity and Pressure Cut Planes
Figure 133: Velocity Streamlines – Center Rib

Figure 134: Pressure Contour – Center Rib
Figure 135: Velocity Streamlines – Quarter Cell

Figure 136: Pressure Contour – Quarter Cell
Figure 137: Velocity Streamlines – End Rib

Figure 138: Pressure Contour – End Rib
4.3. FSI and Experiment Comparison

Over-plotting the maximum deformations of the experiment versus the FSI model yield the results shown in Figure 140 (front of parafoil is on the right). A similar trend can be observed between the two models as they both follow a similar curve in the maximum deformation. However, the deformation curve is more pronounced in the experiment than in the FSI model.

![Side-View Comparison](image)

Figure 139: Side-View Comparison

![Canopy Deformations](image)

Figure 140: Canopy Deformations

The differences between the experiment and the FSI model become more prevalent when the changes between the deflated and inflated canopies are compared between the experimental and FSI models. More canopy displacement is shown to occur
in the front chord section (the mouth of the parafoil), while most of the canopy deformation in the experimental model occurred at approximately halfway through the chord length.

![Change in Canopy Deformations](image)

**Figure 141: Change in Canopy Deformations**

Overall, there is observed to be more canopy displacement in the experimental model versus the FSI model given the same environmental conditions. After review of the change in canopy displacements before and after inflation, there is approximately four times the amount of canopy displacement in the experimental model than seen in the FSI model. Flow separation is apparent towards the cusp end of the FSI parafoil model shown by the velocity flow fields in Figure 133 through Figure 138. This flow separation could be the reason the FSI model did not inflate towards the cusp end of the parafoil as much as observed in the experiment. These differences in deformation could also be due to slight differences in boundary conditions at the wall where the parafoil is restrained. Lastly, measurement error from deformations in the experimental model could also contribute to the differences in canopy deformation. Despite this, both the deformations
in the FSI model and the experiment show similar trends. For example, extrapolating the FSI parafoil model deformations by four times yields a close match to the experiment as shown in Figure 142.

Figure 142: Extrapolated FSI Deformations
5. Conclusion and Recommendations

5.1. Conclusion

The single-cell parafoil modeled in this report using ANSYS Multiphysics FSI two-way coupling on a standard PC of 16 GB RAM, 8 cores, 3 GHz processor showed to have similar trends in vertical canopy displacement, but still showed slightly different maximum canopy deformations. The FSI model underpredicts the deformation by approximately 25% of the true deflection observed in the experiment. Since it was impossible to create a controlled environment with the single-cell parafoil in a stable and sustained flight through a wind tunnel, the force reactions were not able to be recorded for model comparison. Instead the canopy displacements were recorded from the single-cell parafoil using non-destructive means that do not obstruct the wind flow. The canopy deformations of the single-cell square parafoil FSI model showed to have similar trends in vertical canopy displacement but still differed from the results observed in the experimental model. This difference in canopy displacement could be due to measurement error from the experiment or slight differences in boundary conditions between the two models. Comparing the results of the computational model to the experimental measurements yielded a partial match and thus asserts that fluid-structural modeling using ANSYS Multiphysics can be used to model parachutes.
5.2. Recommendations and Further Study

Bridging the gap between a single-cell parafoil and a full parafoil is the next step. The complexity of a parafoil parachute is an obstacle in developing an accurate mesh with precise dynamic mesh settings. Using the FSI settings defined in this thesis will require more computing power to reach convergence on a FSI cycle for a full nine-cell parafoil. This will lead to another study of how to reduce the mesh settings and optimize the CFD settings to retrieve an accurate result in a minimal amount of time for such a task.

Another recommendation will be to analyze the same model but using an explicit FSI solver. Explicit modelers are more effective at modeling high-energy simulations such as explosions or high-speed impacts. Modeling the impact shock of a parafoil opening could be such a task for an explicit solver. The ALE method uses fixed CFD mesh cells that occupy a dimension in space. As the structural model passes through these cells, the air properties are passed to the surfaces of the FEA model. These removes the unnecessary need to re-mesh or change the CFD mesh during the FSI process.

Another avenue to investigate would be the explicit modeling of a parachute inflation phase coupled with the implicit modeling of the same parachutes approximate steady-state performance phase. Since an explicit model is more accurate at modeling high energy transients at smaller time steps, it could be used to model the initial inflation phase of a parachute, then the resulting deformations could be used as the initial conditions of an implicit model to solve for the approximate steady-state conditions as shown in the ANSYS architecture in Figure 143. ANSYS Multiphysics makes this process more seamless with the transfer of explicit and implicit mesh modeling.
Figure 143: Explicit / Implicit Parachute Modeling
References


C. Galliot, R.H. Luchsinger. Uniaxial and biaxial mechanical properties of ETFE foils. EMPA - Swiss Federal Laboratories for Materials Science and Technology, Center for Synergetic Structures Ueberlandstrasse 129, CH-8600 Duebendorf, Switzerland


Gargana, Bill, Guided Parafoil High Altitude Research, Airborne Systems North America of CA, Inc.


Mosseev, Yuri. SOFTWARE TOOLS FOR THE PARAGLIDER COMPUTER-AIDED DESIGN GUIDE. Moscow, Russia: American Institute of Aeronautics and Astronautics, 2001. DOC.


Scampoli, Steve, ANSYS Inc., Lead Product Manager


A. Experimental Specimen – Single-Cell Parafoil

[Diagram showing dimensions and measurements related to a single-cell parafoil.]