A NUMERICAL ALGORITHM FOR HIGH REYNOLDS NUMBER
FLUID-STRUCTURE INTERACTION SIMULATIONS

A Thesis in
Aerospace Engineering
by
Erika Lieberknecht

© 2015 Erika Lieberknecht

Submitted in Partial Fulfillment
of the Requirements
for the Degree of

Master of Science

May 2015
The thesis of Erika Lieberknecht was reviewed and approved* by the following:

Robert Kunz
Senior Scientist and Head of Computational Mechanics Division, ARL
Professor of Aerospace Engineering
Thesis Advisor

Jonathan S. Pitt
Research Associate, ARL
Assistant Professor of Engineering Science and Mechanics

Scott T. Miller
Research Associate, ARL
Assistant Professor of Engineering Science and Mechanics

George A. Lesieutre
Professor of Aerospace Engineering
Head of the Department of Aerospace Engineering

*Signatures are on file in the Graduate School.
Abstract

A finite element based algorithm for high Reynolds number ($10^6 - 10^8$) flow is implemented to simulate fluid-structure interaction (FSI). Applicable to several engineering disciplines, FSI is the interaction between a deformable solid body and fluid flow, with particular interest in large deformations and fully coupled interactions. Numerical modeling of high Reynolds number flow requires care to preserve the FSI interface and fluid boundary layer.

High Reynolds number flows are naturally turbulent, so the governing equations for the fluid are time averaged, resulting in the Reynolds Averaged Navier-Stokes (RANS) equations. A turbulence model is added to the fluid formulation to close the system of equations. The resulting fluid equations are cast in an Arbitrary Lagrangian-Eulerian (ALE) frame of reference, and the solid and mesh governing equations remain in the Lagrangian frame of reference.

The solid and mesh formulations are discretized using the classical Galerkin FEM, while the fluid formulation is discretized using the Streamline Upwind Petrov-Galerkin (SUPG) method for stabilization of FEM based instabilities. A partitioned approach is presented for the mesh and fluid implementations followed by discussion for a monolithic approach. The method of manufactured solutions (MMS) is used to verify the mathematical accuracy of the implemented algorithm.

To show the effectiveness of the mesh formulation, a mixing problem is presented with appropriate results. To validate the fluid formulation, the fixed NACA0012 test case, from NASA, is simulated and compared to experimental results.
# Table of Contents

List of Figures ........................................................... vii  
List of Tables ............................................................ ix  
List of Symbols ................................................................ x  
Acknowledgments ............................................................ xiv  

Chapter 1  
Introduction .................................................................. 1  

Chapter 2  
Background and Literature Review .............................. 4  
2.1 Review of Computational FSI .................................. 4  
2.2 Problem Statement ............................................... 5  
2.3 Numerical Discretization ..................................... 6  
2.3.1 Stabilization Techniques ................................. 8  
2.3.2 Equal Order Fluid Approximations .................. 9  
2.4 Fluid-Structure Coupling .................................... 10  
2.5 Remarks .............................................................. 11  

Chapter 3  
Governing Equations ................................................. 13  
3.1 Averaging Methods ............................................. 13  
3.2 Domain .............................................................. 14  
3.3 Reynolds Averaged Navier-Stokes (RANS) ............ 15  
3.4 Turbulence Model ............................................... 17  
3.4.1 Turbulent Kinetic Energy ............................... 18
3.4.2 Specific Dissipation Rate $\omega$ ........................................ 18
3.5 ALE Reference Frame .......................................................... 19
3.6 Elasticity ............................................................................... 21
3.7 Mesh Motion .......................................................................... 22
3.8 Interface Conditions ............................................................... 23
3.9 Governing Equations Summary .................................................. 24

Chapter 4
Numerical Methods ................................................................. 27
4.1 Weak Formulation .................................................................. 27
4.2 Spatial and Temporal Discretization ......................................... 28
4.3 Fluid Formulation .................................................................. 29
4.4 FSI Formulation Summary ...................................................... 31
4.5 Implementation ....................................................................... 32
4.6 Method of Manufactured Solutions .......................................... 34

Chapter 5
Numerical Experimentation ......................................................... 40
5.1 Mesh Motion .......................................................................... 40
  5.1.1 Side-to-Side Motion ......................................................... 41
  5.1.2 Rotating Paddle ............................................................... 43
5.2 Flow Over a 2-D Airfoil .......................................................... 45

Chapter 6
Summary and Future Work .......................................................... 51
6.1 Summary ............................................................................... 51
6.2 Future Work .......................................................................... 53

Appendix A
Residual and Jacobian Terms ....................................................... 55

Appendix B
Variations in the SUPG Formulation and Further Exploration .... 60
B.1 Brooks and Hughes SUPG Formulation .................................. 61
  B.1.1 1D Advection-Diffusion Problem ..................................... 62
  B.1.2 2D Steady Advection-Diffusion Problem ........................ 68
B.2 Equal Order SUPG Formulation ............................................ 75
  B.2.1 Stokes Flow ..................................................................... 75
  B.2.2 Extension to the Navier-Stokes Equations ......................... 80
Appendix C

Monolithic Projection Scheme 87
  C.1 Verification via MMS . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 88
  C.2 Taylor Vortex Problem . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 98
  C.3 Comparison of Projection Parameter . . . . . . . . . . . . . . . . . . . . . . . 99

Bibliography 102
List of Figures

3.1 Fluid-Structure Interaction Domain ........................................ 15

4.1 Implementation of the Fluid Formulation using a Segregated Approach ................................................................. 33
4.2 Implementation of the Fluid Formulation using a Monolithic Approach ................................................................................. 34
4.3 Error Norm of the Fluid Velocity X Component .......................... 37
4.4 Error Norm of the Fluid Velocity Y Component .......................... 38
4.5 Error Norm of the Pressure Component ........................................ 39

5.1 Mesh Motion Domain for Test Cases ........................................ 41
5.2 Mesh Deformation of the Side-to-Side Moving Rectangle ............. 42
5.3 Fluid Velocity Magnitude at Maximum Deformation for Side-to-Side Motion ................................................................. 43
5.4 Pressure at Maximum Deformation for Side-to-Side Motion ........... 43
5.5 Mesh Deformation of a Rotating Rectangle ..................................... 44
5.6 Fluid Velocity Magnitude at Maximum Deformation of a Rotating Rectangle ................................................................. 45
5.7 Pressure at Maximum Deformation of a Rotating Rectangle ........... 45
5.8 Far View of the NACA0012 Airfoil Mesh ...................................... 46
5.9 Near View of the NACA0012 Airfoil Mesh ...................................... 46
5.10 $C_p$ vs. $x/c$ over a Fixed NACA0012 .......................................... 48
5.11 Pressure around a Fixed NACA0012 Airfoil ................................. 48
5.12 Fluid Velocity Magnitude around a Fixed NACA0012 Airfoil ........ 49
5.13 Turbulent Kinetic Energy around a Fixed NACA0012 Airfoil ......... 49
5.14 Specific Dissipation Rate around a Fixed NACA0012 Airfoil ......... 50

B.1 1D Advection/Diffusion SUPG Formulation vs. Exact Solution .......... 65
B.2 1D Advection/Diffusion Data from Plots from (Chaple, 2006) .......... 66
B.3 2D Advection/Diffusion with $Pe = 7.9057$ without SUPG ............. 70
B.4 2D Advection/Diffusion with $Pe = 7.9057$ without SUPG by Chaple (2006) ................................................................. 70
B.5 2D Advection/Diffusion with $Pe = 7.9057$ using SUPG ............. 71
B.6 2D Advection/Diffusion with $Pe = 7.9057$ using SUPG by Chaple (2006) ................................................................. 72
B.7 2D Advection/Diffusion with $Pe = 7.9057 \epsilon 5$ without SUPG .... 73
B.8 2D Advection/Diffusion with $Pe = 7.9057 \epsilon 5$ without SUPG by Chaple (2006) ................................................................. 74
B.9 2D Advection/Diffusion with $Pe = 7.9057 \epsilon 5$ using SUPG ....... 74
B.10 2D Advection/Diffusion with $Pe = 7.9057 \epsilon 5$ using SUPG by Chaple (2006) ............................................................... 75
B.11 Boundary Conditions for a Driven Cavity Flow Problem by Hughes et al. (1986) ............................................................. 76
B.12 Boundary Conditions for a Driven Cavity Flow Problem .......... 77
B.13 Pressure Elevation Comparison for a Driven Cavity Flow Problem 78
B.14 Pressure Contour Comparison for a Driven Cavity Flow Problem 79
B.15 Velocity Vector Comparison for a Driven Cavity Flow Problem ... 80
B.16 Boundary Conditions for a Driven Cavity Flow Problem .......... 81
B.17 Variation of $\alpha Re = 1,000$: $u_x$ vs. y distance at $x = 0.5$ .... 82
B.18 Variation of $\alpha Re = 1,000$: $u_y$ vs. x distance at $y = 0.5$ .... 82
B.19 Stabilization Methods $Re = 1,000$: $u_x$ vs. y distance at $x = 0.5m$ 83
B.20 Stabilization Methods $Re = 1,000$: $u_y$ vs. x distance at $y = 0.5$ 84
B.21 Stabilization Methods $Re = 5,000$: $u_x$ vs. y distance at $x = 0.5m$ 85
B.22 Stabilization Methods $Re = 5,000$: $u_y$ vs. x distance at $y = 0.5m$ 85
C.1 Error Norm of the Fluid Velocity X Component ..................... 91
C.2 X Velocity Computed Solution Using MMS ............................. 92
C.3 Error Norm of the Fluid Velocity Y Component ..................... 93
C.4 Y Velocity Computed Solution Using MMS ............................. 94
C.5 Error Norm of the Pressure Component ................................. 95
C.6 Pressure Computed Solution Using MMS ............................... 96
C.7 Error Norm of the Pressure Gradient X Component .................. 97
C.8 Pressure Gradient X Component Computed Solution Using MMS ... 98
C.9 Taylor Vortex Velocity Error Comparison ............................. 99
C.10 Comparison of the Projection Parameter $\alpha$ ......................... 101
List of Tables

4.1 Refinement Table of the Fluid Velocity X Component . . . . . . . 37
4.2 Refinement Table of the Fluid Velocity Y Component . . . . . . . 38
4.3 Refinement Table of the Pressure Component . . . . . . . . . . . . 39

B.1 Grid Refinement Table with SUPG for 1D Advection/Diffusion
   Problem . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 67
B.2 Grid Refinement Table without SUPG for 1D Advection/Diffusion
   Problem . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 68

C.1 Refinement Table of the Fluid Velocity X Component . . . . . . . 92
C.2 Refinement Table of the Fluid Velocity Y Component . . . . . . . 93
C.3 Refinement Table of the Pressure Component . . . . . . . . . . . . 95
C.4 Refinement Table for the X Component of the Pressure Gradient . . 97
List of Symbols

\( \alpha_k \)  Projection scheme coefficient
\( \beta_0 \)  Turbulence coefficient
\( \beta_1 \)  Turbulence coefficient
\( \beta^* \)  Turbulence coefficient
\( \Delta t \)  Time step
\( \gamma \)  Turbulence coefficient
\( \Gamma_D \)  Dirichlet boundary of a domain
\( \Gamma_N \)  Neumann boundary of a domain
\( \Gamma_{FS} \)  Fluid-structure interface on a boundary of a domain
\( \epsilon \)  Turbulent dissipation rate
\( \kappa \)  Karman constant
\( \lambda \)  Lamé's first parameter
\( \mu \)  Dynamic viscosity or Lamé's second parameter
\( \mu_t \)  Turbulent dynamic viscosity
\( \nu \)  Kinematic viscosity or Poisson's ratio
\( \nu_t \)  Turbulent kinematic viscosity
\( \rho \)  Mass density
\( \sigma_k \) Turbulence coefficient
\( \sigma_\omega \) Turbulence coefficient
\( \sigma_{\omega^2} \) Turbulence coefficient
\( \tau_R \) Reynolds stress tensor
\( \tau_w \) Wall stress
\( \omega \) Specific turbulent dissipation rate
\( \hat{\omega} \) Modified \( \omega \) for turbulent viscosity
\( \Omega \) Infinitesimal rotational tensor
\( \Omega \) Domain
\( \xi \) Projection of the pressure gradient
\( \chi_\omega \) Pope’s turbulence correction
\( b \) Body forces
\( C_{lim} \) Turbulent coefficient
\( d \) Infinitesimal strain tensor
\( d_m \) Mesh displacement
\( d_s \) Solid displacement
\( \hat{D} \) Variant of infinitesimal strain tensor
\( E \) Young’s modulus
\( F \) Deformation gradient
\( I \) Identity Tensor
\( k \) Turbulent kinetic energy
\( n \) Outward unit normal
\( p \) Pressure
\( P \) Turbulence production
\(\tilde{p}\) Pressure variation

\(S\) Strain invariant tensor or Second Piola-Kirchhoff stress tensor

\(u\) Instantaneous velocity

\(u'\) Fluctuating velocity

\(U\) Mean velocity

\(\bar{\phi}\) Time average of \(\phi\)

\(u_\tau\) Friction velocity

\(t\) Traction force

\(T\) Cauchy stress tensor

\(T_R\) First Piola-Kirchhoff stress tensor

\(T\) Select time for time averaging quantities

\(t\) time

\(x\) Position in the spatial frame

\(X\) Position in the reference frame

ALE Arbitrary Lagrangian-Eulerian

div Spatial frame divergence

Div Reference frame divergence

FSI Fluid-structure interaction

\(\text{grad}\) Spatial frame gradient

Grad Reference frame gradient

GWS Galerkin weak statement

RANS Reynolds Averaged Navier Stokes

PISO Pressure Implicit something else

SST Shear Strain something
SUPG Streamline Upwind Petrov-Galerkin

COMMON SUBSCRIPTS

f Fluid variables
κ ALE reference frame variables
m Mesh variables
s Solid variables
t Turbulence variables
Acknowledgments

I would like to thank the Applied Research Laboratory Walker Assistantship for financial support.

I would like to thank my advisor, Dr. Jonathan Pitt, for his support and guidance throughout my graduate school experience.

I would also like to thank Dr. Scott Miller for his input and support as well.

The Applied Research Laboratory has given me a wonderful opportunity. The employees and fellow students are continuously supportive and encouraging. Thank you.

The Aerospace faculty and staff have always been supportive of my ambitions, and it is greatly appreciated.

Lastly, I would like to thank those who have supported me through my graduate studies. Your encouragement and support have been instrumental in making this document a success. Words truly cannot express how thankful I am.
Chapter 1

Introduction

Aerospace, mechanical, and civil engineering are all industries with applications to fluid-structure interaction (FSI). Aeroelastic instabilities such as flutter on aircraft or windturbines, waves crashing into a ship, and the effect of air around a bridge or tall building are just a few physical examples. FSI is the interaction of a deformable body with fluid flow. The fluid interacts with the solid and deformations occur, then these deformations impact the surrounding fluid. This is known as two-way or fully coupled FSI. We have a particular interest in external flows with high Reynolds numbers, that are fully coupled with large solid deformation.

The governing equations are derived for the system using the balance laws and constitutive relations under the assumption of a continuum approach. This allows for the use of calculus. The governing equations are then differentiable and integratable. A weak form of the governing equations can then be presented.

The governing equations for the solid are naturally in a Lagrangian frame of reference. The fluid formulation is naturally in an Eulerian frame. For a fully coupled FSI simulation, a common frame work is needed to pass information between the fluid and solid along the interface. Therefore, the fluid formulation is transformed into the Arbitrary Lagrangian-Eulerian (ALE) frame, allowing information to be passed between the fluid and solid.

From the weak form of the governing equations, a numerical solution is approximated. The finite volume method is commonly used to approximate a fluid solution. However, to approximate a solution for the solid, the finite element method (FEM) is usually coupled with the finite volume method. This poses a
challenge when solving fully coupled FSI, where a common method to approximate both solutions is preferred. To obtain a numerical solution, for both the fluid and solid, a finite element method is introduced to discretize all of the FSI equations for high Reynolds number flow.

Using the standard classical Galerkin FEM to model high Reynolds number \((10^6 - 10^8)\) flow leads to well known instabilities in the solution. Stabilization is therefore needed for the fluid formulation to provide an accurate solution. Adding artificial diffusion to the fluid formulation would accommodate the instabilities that arise using FEM. We chose to implement the Streamline Upwind Petrov-Galerkin (SUPG) method, an upwind discretization scheme, for the fluid formulation. This will provide a stabilized approximation for the fluid solution.

High Reynolds number flow is inherently turbulent, tending to be irregular. A direct numerical simulation resolves the turbulent scales involved, but this is not computationally possible for the Reynolds numbers discussed. Neither is a large eddy simulation (LES) approach, which resolves the large turbulent scales. The Reynolds averaged Navier-Stokes (RANS), approach takes a time average of the fluid equations, so no turbulent scales are resolved. This is more appropriate due to computational costs; however, using an averaging approach leads to the classical closure problem of turbulence, having more equations than unknowns. A model representing the turbulence parameters is provided to sufficiently relate the turbulence to the RANS equations, closing the system. The \(k - \omega\) turbulence model is used in this work.

We are interested in a numerical solution for fully coupled FSI. The formulation can be implemented in a segregated fashion with a partitioned algorithm, or a monolithic algorithm can be used. A partitioned algorithm is a fully coupled approach, solving for the solid and fluid separately, and iterating with interface constraints until a convergence criteria is met. A monolithic algorithm keeps the fluid and solid strongly coupled when solving for the governing equations simultaneously. The strong coupling is a result of solving for all solution parameters at once for each timestep. In this work, a partitioned algorithm is presented for the implementation of the discretized governing equations. The framework for a monolithic algorithm is also presented as a future research topic.

Verification is provided via the method of manufactured solutions (MMS) show-
ing mathematical accuracy of the implemented fluid formulation. A mixing problem is presented to show the effectiveness of the mesh motion. Using a test case from a turbulence research database, the fluid formulation approximates flow around a fixed NACA0012 airfoil and compared to experimental results.

This work aims to enhance a current monolithic FSI solver. The combination of adding a turbulence model, discretizing the fluid formulation with the SUPG method and adding an equal order approximation method is a contribution to current literature by improving the fluid formulation to model high Reynolds number FSI simulations and allow for equal order approximations.

Outline of Thesis

The second chapter provides background information and a comprehensive review of current literature. The third chapter presents the governing equations for the problem structured in this thesis. Details of the turbulence model to close the system of equations and governing equations in the ALE reference frame, are presented. The governing equations for the solid and mesh are provided, along with necessary boundary conditions for FSI. The fourth chapter discusses the equal order interpolation scheme added to the fluid formulation. The finite element formulation is also discussed, where the SUPG formulation is used for the fluid formulation and the standard Galerkin method is used for the solid and mesh. Chapter five discusses the application of the proposed algorithm, and the last chapter summarizes and discusses future plans.
Background and Literature Review

As a concept FSI has been studied for ages in musical instruments and blood vessels as pulsating flow (Tijsseling, 1996). FSI applications have been limited by computational capabilities, but as computing power has increased so has the study of FSI. An analytical solution to an FSI problem is most often unobtainable presenting a challenging computational mechanics problem to find an approximate solution.

This chapter provides a review of the progress made in computational FSI. Challenges in computational FSI are presented and categorized. Current methods are discussed addressing some of the challenges faced in computational FSI with the expectation this work will make contributions to current progress.

2.1 Review of Computational FSI

It is standard to solve a structural mechanics problem using FEM (Cook et al., 2007), but it is standard to use either a finite difference scheme (Donea et al., 1982) or FVM for a fluid mechanics problem. If different numerical solutions are being approximated for the fluid and solid, then a problem arises as to how to pass information between the structural solver and the fluid solver. Setting a common framework, FEM provides a straightforward approach to approximate a numerical solution. FEM based hydro-structural problems were first published by Belytschko & Kennedy (1975) and Donea et al. (1976) in the 1970’s (Donea et al., 1982) using a pure Lagrangian description. FEM for the fluid was developed and coupled with
an existing solid solver to study the water hammer effect of pressure waves on the subassembly walls in a reactor core in Belytschko & Kennedy (1975). Early FSI advancements related to reactor technology assumed a rigid body to solve for the pressure in the first iteration. Proving to have unrealistic results, this lead to advancement in the coupling between the fluid and structure (Liu & Chang, 1985).

The immersed boundary method introduced by Peskin (1972) presents a different approach, circumventing complexities with mesh motion. A body force describes the fluid-solid interface Fadlun et al. (2000); Peskin (1972). Cristallo & Verzicco (2006) couples the immersed boundary method with large eddy simulations for high Reynolds number FSI.

Computational FSI is a complex computational mechanics problem. Bazilevs et al. (2012) categorize the challenges of computational FSI in the following manner: (1) problem statement, (2) numerical discretization method, and (3) coupling of the fluid and structure. The remainder of this chapter presents the challenges of FSI in each of these topics and discusses them below:

### 2.2 Problem Statement

Applications of FSI such as wind turbines, and general aircraft have a Reynolds number in the range of $10^6$ to $10^8$ (Lissaman, 1983), where airfoil flapping, oscillations, flutter predictions, and aerelastic instability occur. Understanding FSI phenomena can impact the efficiency of a design or avoid hazards, such as induced large vibrations (Zhu, 2007).

We are specifically interested in understanding high Reynolds number (ranging of $10^6$ to $10^8$), external FSI, which is inherently turbulent. Lund et al. (2003) discusses an optimization technique for FSI implementing the RANS equations with the algebraic Baldwin-Lomax turbulence model (Baldwin & Lomax, 1978) for the fluid formulation. The problem is discretized using both the classical Galerkin method and the SUPG method. Results are produced with Reynolds numbers on the order of $10^4$ (Lund et al., 2003). An algebraic model is a simple turbulence model. A two equation model provides a more developed framework. Such as the $k - \omega$ model (Wilcox, 1998) implemented in Sváček (2011) where the Galerkin Least Squares with the $k - \omega$ turbulence model provides numerical approximations
for FSI applications of Reynolds numbers on the range of $10^4 - 10^6$. Other methods like LES are implemented in the literature. For example, in de Sampaio et al. (2004) LES based techniques are used to model a bridge domain with Reynolds numbers on the order of $10^6 - 10^7$.

With the assumption of a continuum approach, the RANS equations are derived using balance laws and constitutive relations. A turbulence model is also implemented. A Lagrangian frame of reference is the natural description for the governing equations of the solid, while the fluid description is naturally in a deformed description or the Euler reference frame. For a fully coupled FSI simulation, a common frame work is needed to pass information between the fluid and solid along the interface. A fully Lagrangian see, Belytschko & Kennedy, 1975; Donea et al., 1976 or Eulerian frame of reference can be used, but Hughes et al. (1981) introduced the Arbitrary Lagrangian-Eulerian finite element description for incompressible, viscous flow providing a more efficient way to pass information between the fluid and solid domains. Therefore, the fluid formulation is transformed into the Arbitrary Lagrangian-Eulerian (ALE) frame, in this work, allowing information to be passed along the interface of the fluid and solid.

The interface is generally meshed one of two ways. An immersed boundary mesh can model the solid body. Fadlun et al. (2000) combines a finite difference method with an immersed boundary mesh using an LES turbulence model. The authors produce results with Reynolds numbers on the order of $10^9$. A few years later, Cristallo & Verzicco (2006) implement an immersed boundary method with an LES turbulence model with results having Reynolds numbers on the order of $10^4$. An immersed boundary method, being simpler to implement than a body fitted mesh, does not capture the details of the boundary layer imperative for this research. A body fitted mesh fits the mesh around the solid body, enabling the details of the boundary layer to be preserved.

### 2.3 Numerical Discretization

A common approach for discretizing a structural mechanics problem is to use the FEM. However, for fluid problems either a finite difference scheme or FVM are commonly used. Setting a common framework, the FEM provides a straight
forward approach to approximate a numerical solution. A system of partial differential equations is derived in which an analytical solution is not easily found. The FEM approximates a solution by taking piecewise interpolations of the dependent variables. Generally, the interpolations are polynomials, creating an order of accuracy corresponding to the degree of the polynomial (Cook et al., 2007).

The classical Galerkin method is a weighted residual method for the FEM, and when used for problems such as heat conduction produces a best approximation. This property is lost when applied to convective transport problems such as high Reynolds number FSI due to the inability to satisfy the inf-sup condition (Bathe et al., 2000). The condition ensures stability and determines the quality of a discretization scheme (Bathe et al., 2000; Brooks & Hughes, 1982).

The generic inf-sup condition is given as follows: Let $A(\eta, \psi)$ be the weak form of some function, where a solution for $\eta \forall U$ for all $\psi \in U$ is found and $U_h$ is the finite subspace of $U$,

$$\inf_{\eta \in U_h} \sup_{\psi \in U_h} \frac{A(\eta, \psi)}{\|\eta\|_s \|\psi\|_t} \geq \gamma.$$  \hspace{1cm} (2.1)

$\|\cdot\|_s$ is the norm for the solution and $\|\cdot\|_t$ is the norm for the weighting function. $\gamma > 0$, constant, and independent of crucial physical parameters and element size (Bathe, 2001).

Similar to equation (2.1), the inf-sup condition for the incompressible Navier-Stokes equations is commonly referred to as the Babuška-Brezzi condition (Bathe et al., 2000) and shown in equation (2.2):

$$\inf_{p_h \in P_h} \sup_{u_h \in U_h} \frac{\int_{\text{Vol}} p_h (\nabla \cdot u_h) d\text{Vol}}{\|u_h\|_{H^1} \|p_h\|_{L^2}} = \gamma_h \geq \gamma > 0.$$ \hspace{1cm} (2.2)

$U_h$ is the velocity interpolation space while $P_h$ is the pressure interpolation space. Again $\gamma$ must be constant and independent of element size, but the value of $\gamma$ is difficult to obtain analytically. Instead $\gamma_h$ is obtained by solving an eigenvalue problem for the smallest eigenvalue. If a spurious pressure mode is present in the problem $\gamma_h$ becomes zero, resulting in the inf-sup condition not being satisfied (Bathe, 2001). Finding a numerical solution to the Navier-Stokes equations poses challenges due to the coupling of the pressure and the velocity by the Babuška-Brezzi condition.
2.3.1 Stabilization Techniques

For problems such as heat exchange or beam bending the Galerkin method produces accurate results, but when considering advection problems, such as high Reynolds number flow, the Galerkin method may not achieve accurate results because the Babuška-Brezzi condition is not satisfied. Oscillations and instabilities occur in the solution. The mesh can be refined, but this can become cumbersome (Heil, 2004). A stabilization technique is needed to improve the solution. An upwind method would be used when accurate solutions are no longer achievable by the Galerkin method (Bathe, 2001; Brooks & Hughes, 1982).

The main objective of upwind methods is to add artificial diffusion to the formulation to eliminate oscillations that may occur in cases where the Galerkin method cannot produce the best approximate solution. Various upwinding methods were developed and implemented throughout the literature. Adding artificial diffusion to the fluid formulations is challenging because the accuracy of the solution decreases. Upwind methods have a reputation for decreasing the accuracy of the solution.

The Streamline Upwind Petrov-Galerkin (SUPG) method adds artificial diffusion to the formulation but does not have the loss of accuracy of other upwind methods. The SUPG method only adds diffusion in the streamline direction, increasing the accuracy of the solution, making the SUPG method more appealing than other types of upwinding methods (Brooks & Hughes, 1982).

Throughout the literature, the SUPG method is implemented in various ways. Heil (2004) implements the SUPG method. A preconditioning technique is used to solve a fully coupled FSI problem. The SUPG method is implemented to achieve results with a courser mesh. This allows a solution to be obtained more efficiently (Heil, 2004). Xu & Zhang (1991) make use of the SUPG method with a wave matrix method interpolating the pressure with four nodal points and velocity with eight nodal points. Chaple (2006) compares results of two test cases using the Galerkin method and the SUPG method. Details of Chaple (2006)’s work is discussed in more detail in Appendix B.

The SUPG method prevents oscillations and spurious instabilities that arise from convection dominated flow. The classical Galerkin weighting function is modified with a streamline upwind perturbation resulting in a consistent weighted residual but is not an equal order approximation (Brooks & Hughes, 1982). An
overview of the SUPG method is provided below:

Let \( B(u, \hat{u}) \) be the obtained weak form of the Navier-Stokes equations. \( F(u) \) are any external forces, and \( S(u, \hat{u}) \) is the SUPG addition. Adding the SUPG method to the weak form of the Navier-Stokes equations results in the following formulation:

\[
B(u, \hat{u}) + S(u, \hat{u}) = F(u) \quad \forall \hat{u} \in V_h,
\]

where

\[
V_h = \{ \hat{u} \in H_0^1(\Omega) | \hat{u} \in C^0(\Omega) \}
\]

(Chaple, 2006).

Equation (2.3) can also be written with a new weighting function shown in equation (2.5):

\[
B(u, \tilde{u}) = F(\tilde{u}).
\]

Where \( \tilde{u} \) is defined in equation (2.6):

\[
\tilde{u} = \hat{u} + f(\tau_{SUPG}),
\]

and \( f(\tau_{SUPG}) \) is a function of the SUPG parameter, \( \tau_{SUPG} \), and other terms (Brooks & Hughes, 1982).

In equation (2.6) the consistent SUPG formulation is applied to the weighting function of the Galerkin method affecting all terms in the Navier-Stokes equations (Brooks & Hughes, 1982; Idelsohn et al., 1998).

### 2.3.2 Equal Order Fluid Approximations

To make a finite element fluid formulation equal order, the velocity and pressure are interpolated separately and are of the same degree. This is usually done by decoupling the velocity and pressure in some fashion (Idelsohn et al., 1998; Rice & Schnipke, 1986). The solution method depends on the approach taken to obtain the pressure and keeping it explicit in the continuity equation.

The incompressibility constraint of the Navier-Stokes equations is typically modified related to the solution method used for solving the equations. It is common to derive a pressure equation from mass conservation. The pressure and
velocity are decoupled and the continuity constraint is applied.

The Poisson-type method and the fractional step method both derive a Poisson-type pressure equation (Idelsohn et al., 1998; Rice & Schnipke, 1986). The Poission-type equation results in the equal order interpolation (Rice & Schnipke, 1986). The fractional step method has advantages over the SIMPLE method because it does not require a mathematical approximation in the derivation of the pressure or the need to calculate pressure coefficients every iteration (Choi et al., 1997). The SIMPLER and SIMPLEST methods decouple the velocity and pressure by using pseudo-velocities. The pseudo-velocities are set equal to the nodal velocities and the nodal velocity-pressure coefficients are set to zero. The SIMPLEST method differs by making the non-symmetric velocity coefficient matrices symmetric. It was not clear whether one method was more adequate than the other. This was attributed to the low velocities used and the little to no upwind effect (Idelsohn et al., 1998).

The SIMPLE method makes the assumption the velocity connections adjacent to the pole are negligible. This leads to a simpler solution method than the PISO method and is adequate for solving steady state flows (Barton, 1998). A projection method specifically on the pressure gradient was used for the stabilization of the pressure term to accommodate equal order interpolation. It was implemented with a fractional step method as well as a monolithic approach (Codina & Blasco, 2000). An equal order interpolation method uses a local projection stabilization term comparable to the SUPG method. Two types of local projection methods are considered: i) a two-level approach and ii) pairs of spaces defined on the same mesh (enrichments) (Matthies et al., 2007).

2.4 Fluid-Structure Coupling

Heil (2004) has categorized most computational efforts as either partitioned methods or monolithic methods. The method used by Belytschko & Kennedy (1975) is considered a partitioned method, solving for the fluid and velocity separately, and iterating with an interface constraint until a convergence criteria is met. The information is iteratively passed between the fluid and solid interface. This method is used quite extensively due to ease of implementation. When applied to
high Reynolds number FSI, Bai et al. (2012) implement a partitioned approach to computationally solve the coupling of the fluid and solid. The authors approximate a solution using a finite element method for incompressible, high Reynolds number flow for flutter derivatives of a symmetric airfoil with a $k-\omega$ SST model to account for the turbulence. Glück et al. (2001) implements a finite element, partitioned algorithm with the $k-\epsilon$ turbulence model assuming a Reynolds number on the order of $10^6$.

In some cases a monolithic approach may be more cost effective and accurate. A monolithic approach solves for the fluid and solid simultaneously. Subiterations are not required. The monolithic approach keeps the fluid and structure coupled and simultaneously provides a solution that is inherently strongly coupled (Heil, 2004). Heil (2004) uses a preconditioning technique to solve a fully monolithic FSI problem with a maximum Reynolds number of 500. Exploring relaxation techniques, Küttler & Wall (2008) implements a monolithic approach with Reynolds numbers on the order of 100. Bazilevs et al. (2008) emphasizes an isogeometric analysis with a monolithic approach and applies it to a full scale wind turbine rotor in Bazilevs et al. (2011).

While a monolithic approach is appealing due to the lack of subiterations, Banks et al. (2014) proposes a partitioned approach with no subiterations. The method focuses on the interface constraints. Robin boundary conditions are used at the fluid-solid interface, providing a guess that does not require subiterations. Sheldon et al. (2014) conducted an adequate comparison of computational methodology. A framework was developed in the deal.ii finite element library (Bangerth et al., 2007). This allowed for a true comparison between a monolithic and partitioned approach. Sheldon et al. (2014) chose the benchmark case from Turek & Hron (2006) to compare computational methodologies. The results showed the monolithic approach to exceed other methods for this case.

### 2.5 Remarks

Previous high Reynolds number FSI models either implement a partitioned method (Bai et al., 2012), or a simpler turbulence model (Lund et al., 2003). It is an ambition of this work to provide a monolithic framework for FSI applications.
with high Reynolds numbers in the range of $10^6$ to $10^8$. Naturally, a common numerical approximation method must be implemented if the mindset is to have a monolithic approach. Providing this common framework, we used FEM to approximate a numerical solution. The SUPG method is implemented for stabilization of the fluid formation. The $k – \omega$ turbulence model is also implemented to account for turbulence. Codina & Blasco (2000)'s equal order projection method implementation is also discussed. This work provides the framework for high Reynolds number FSI problems with a monolithic approach by implementing FEM for both the fluid and solid formulations with SUPG stabilization for the fluid formulation and the addition of a turbulence model.
Governing Equations

With a particular interest in high Reynolds number FSI simulations, the derivation of a mathematical model in the form of PDEs is essential to understand the physics of the FSI. A continuum approach is assumed to derive the balance laws and constitutive relations. To account for the turbulent terms in the fluid formulation, an averaging method is appropriate, and the $k-\omega$ turbulence model is presented. The fluid formulation is then put into the Arbitrary Lagrangian-Eulerian (ALE) reference frame. The solid and mesh governing equations are presented as well. First, averaging methods are discussed.

3.1 Averaging Methods

Three types of averaging are used for turbulence modeling:

1. Ensemble averaging, is a general approach used for simple turbulence problems such as homogeneous turbulence. Turbulence is naturally inhomogeneous making a homogeneous assumption a special case (Wilcox, 1998).

2. Spatial averaging is also used for homogeneous turbulence due to the uniformity of the averaging in all spatial directions (Wilcox, 1998).

3. Time averaging does not change with time and is used for the Reynolds Averaged Navier-Stokes (RANS) equations (Wilcox, 1998). For this reason time averaging is used for this work and discussed in detail.
As time averaging is used for the Reynolds averaging it will be discussed in more detail. The instantaneous velocity is a sum of the mean velocity and the fluctuating velocity:

\[ u(X, t) = U(X) + u'(X, t), \]

where \( U \) is the mean velocity and \( u' \) is the fluctuating component. The mean velocity is defined in equation:

\[ U(X) = \lim_{T \to \infty} \frac{1}{T} \int_t^{t+T} u(X, t) \, dt, \]

where \( T \) is a select time considerably longer than maximum period of the velocity fluctuations. Taking the time average of the mean velocity is the time-averaged value:

\[ \overline{U}(X) = \lim_{T \to \infty} \frac{1}{T} \int_t^{t+T} U(X) \, dt = U(X). \]

The time average of the fluctuating part is zero:

\[ \overline{u'}(X, t) = \lim_{T \to \infty} \frac{1}{T} \int_t^{t+T} (u(X, t) - U(X)) \, dt = U(X) - \overline{U}(X) = 0. \]

### 3.2 Domain

Before the governing equations are discussed, the domain of a FSI problem is presented. From a kinematics point of view there is a reference domain and a spatial domain. The reference domain is considered fixed for all time, \( t \). A spatial domain is considered the current deformed domain. In an Eulerian description the spatial domain is fixed for all time and coincides with the reference domain. A fluid problem is generally solved using an Eulerian description. A Lagrangian description of a material describes the position in space of a material point. A material point describes a particles motion. This casts the governing equations in the reference domain when using a Lagrangian description.

Figure 3.1 illustrates the FSI domain with the FSI interface along with Dirichlet and Neumann boundary conditions for the solid and fluid. \( D \) denotes Dirichlet
boundary conditions. $N$ denotes Neumann boundary conditions. $f$ denotes the fluid domain, and $s$ denotes the solid domain.

Figure 3.1: The fluid-structure domain is illustrated with the FSI interface, shown in red, along with Dirichlet boundary conditions, denoted by $D$ and Neumann boundary conditions, denoted by $N$. $f$ denotes the fluid domain, and $s$ denotes the solid domain.

3.3 Reynolds Averaged Navier-Stokes (RANS)

The fluid motion is governed by the transient, incompressible Navier-Stokes equations. We start in the Eulerian reference frame with the balance of linear momentum:

$$
\rho \left( \frac{\partial \mathbf{u}}{\partial t} + \text{div} \, \mathbf{u} \otimes \mathbf{u} \right) = \text{div} \, \mathbf{T} + \rho \mathbf{b}_f \quad \forall \mathbf{X} \in \Omega_f. \quad (3.5)
$$

The Cauchy stress, $\mathbf{T}$ for a linearly viscous and incompressible fluid is given by:

$$
\mathbf{T} = -p \mathbf{I} + \mu \left( \text{grad} \mathbf{u} + (\text{grad} \mathbf{u})^T \right). \quad (3.6)
$$
The mass balance for an incompressible fluid prescribes the continuity condition:

\[ \text{div} \, \mathbf{u} = 0 \quad \forall \mathbf{X} \in \Omega_f. \]  

(3.7)

Equations (3.5) - (3.7) result in the commonly known Navier-Stokes equations:

\[
\frac{\partial \mathbf{u}}{\partial t} + \text{grad} \, \mathbf{u} \mathbf{[u]} - \nu \text{div} \left( \text{grad} \, \mathbf{u} \right) = -\text{grad} \, p + \mathbf{b}_f, \\
\text{div} \, \mathbf{u} = 0,
\]

(3.8)

when \( \nu \) and \( \rho \) are constant. As described in the previous sections high Reynolds number flows are inherently turbulent. To describe the turbulent nature, the pressure and velocity are written in terms of the mean and fluctuating components:

\[
\mathbf{u}(\mathbf{X},t) = \mathbf{U}(\mathbf{X}) + \mathbf{u}'(\mathbf{X},t), \\
p(\mathbf{X},t) = P(\mathbf{X}) + p'(\mathbf{X},t).
\]

(3.9)

Using time averaging, the instantaneous velocity and pressure are replaced with mean and fluctuating components, resulting in the Reynolds averaged Navier-Stokes (RANS) equations:

\[
\rho \left( \frac{\partial \mathbf{u}}{\partial t} + \text{grad} \, \mathbf{u} \mathbf{[u]} \right) = \text{div} \, \mathbf{T} + \rho \left( \mathbf{b}_f - \text{div} \left( \mathbf{u}' \mathbf{u}' \right) \right),
\]

(3.10)

where term, \( -\rho \left( \mathbf{u}' \mathbf{u}' \right) = \tau_R \), the Reynolds-stress tensor. The Reynolds-stress tensor leads to 9 more unknowns and not enough equations to solve for them. As discussed in previous sections, this is leads to a closure problem. Therefore, the Reynolds-stress tensor must be modeled to close the system of equations (Wilcox, 1998).

The Boussinesq approximation is considered valid and the Reynolds-stress tensor is given by:

\[
\tau_R = \mu_t \left( \text{grad} \, \mathbf{u} + \text{grad} \, \mathbf{u}^T \right) - \frac{2}{3} \rho k \mathbf{I},
\]

(3.11)
where the kinematic viscosity is shown in equation (3.12):

\[ \nu_t = \frac{\mu_t}{\rho} \] (3.12)

This results in:

\[ \rho \left( \frac{\partial \mathbf{u}}{\partial t} + \text{grad} \mathbf{u} \right) = \text{div} \mathbf{T} + \rho \mathbf{b}_f + \text{div} \tau_R, \] (3.13)

Substituting equations (3.6) and (3.11) results in the following:

\[ \rho \left( \frac{\partial \mathbf{u}}{\partial t} + \text{grad} \mathbf{u} \right) = \text{div} \left( -p \mathbf{I} + \mu \left( \text{grad} \mathbf{u} + (\text{grad} \mathbf{u})^T \right) \right) + \rho \mathbf{b}_f \]

\[ + \text{div} \left( \mu_t \left( \text{grad} \mathbf{u} + \text{grad} \mathbf{u}^T \right) - \frac{2}{3} \rho k \mathbf{I} \right), \] (3.14)

combining like terms gives:

\[ \frac{\partial \mathbf{u}}{\partial t} + \text{grad} \mathbf{u} = -\text{div} \left( \left( \frac{p}{\rho} + \frac{2}{3} k \right) \mathbf{I} \right) \]

\[ + \text{div} \left( \left( \nu + \nu_t \right) \left( \text{grad} \mathbf{u} + (\text{grad} \mathbf{u})^T \right) \right) + \mathbf{b}_f. \] (3.15)

Let \( \tilde{p} = \frac{p}{\rho} + \frac{2}{3} k \) this gives the commonly known RANS equations:

\[ \frac{\partial \mathbf{u}}{\partial t} + \text{grad} \mathbf{u} = -\text{div} \left( \tilde{p} \mathbf{I} \right) + \text{div} \left( \left( \nu + \nu_t \right) \left( \text{grad} \mathbf{u} + (\text{grad} \mathbf{u})^T \right) \right) + \mathbf{b}_f \] (3.16)

\[ \text{div} \mathbf{u} = 0. \] (3.17)

### 3.4 Turbulence Model

The RANS equations have been derived from the balance laws and the use of time averaging. To effectively close the system of equations, \( \nu_t \) must be appropriately defined via a turbulence model.

Turbulence models are often categorized by the modeling of the mixing length parameter. The simplest turbulence model is an algebraic model, but their applicability may be limited (Wilcox, 1998). A commonly used algebraic model is
the Baldwin-Lomax model (Baldwin & Lomax, 1978). One equation models such as the Spalart-Allmaras model (Spalart & Allmaras, 1994) are a relation of the turbulence length scale and some flow dimension. The \(k - \epsilon\) model introduced by Launder & Spalding (1974) gives a good representation of turbulent phenomena but at regions far from the wall. The \(k - \omega\) model proposed by Wilcox (1998) is numerically stable and relatively easy to implement. The model provides an accurate prediction of near wall turbulent flows (Wilcox, 1998).

3.4.1 Turbulent Kinetic Energy

The equation for turbulent kinetic energy as described by Wilcox (2008) is shown in equation (3.18):

\[
\frac{\partial (\rho k)}{\partial t} + \text{grad} (\rho k) \cdot \mathbf{u} = P - \beta^* \rho \omega k + \text{div} \left[ (\mu + \sigma_k \mu_t) \text{grad} k \right].
\]

For the standard model the production term \(P\) is denoted as follows:

\[
P = \tau_R : \text{grad} \mathbf{u}.
\]

3.4.2 Specific Dissipation Rate \(\omega\)

The rate of dissipation is the rate of transfer from the turbulent kinetic energy to the small scale eddies (Wilcox, 1998). The equation for specific dissipation rate as described by Wilcox (2008) is shown in equation (3.20):

\[
\frac{\partial (\rho \omega)}{\partial t} + \text{grad} (\rho \omega) \cdot \mathbf{u} = \frac{\gamma P}{\mu_t} - \beta_1 \rho \omega^2 + \text{div} \left[ (\mu + \sigma_\omega \mu_t) \text{grad} \omega \right] + \rho \frac{\sigma_\omega^2}{\omega} \text{grad} k \text{grad} \omega.
\]

Closure Coefficients

The turbulent viscosity is described as:

\[
\mu_t = \rho \nu_t = \frac{\rho k}{\omega},
\]
\[ \dot{\omega} = \max\left( \omega, C_{\text{lim}} \sqrt{\frac{2SS}{\beta^*}} \right), \quad (3.22) \]

\[ S = \frac{1}{2} \left( \text{grad} \, u + \text{grad} \, u^T \right). \quad (3.23) \]

Closure constants as described by Wilcox (2008) used are shown below:

\[ \beta^* = 0.09, \]
\[ \beta_0 = 0.0708, \]
\[ \gamma = 0.52, \]
\[ \sigma_k = 0.6, \]
\[ \sigma_\omega = 0.5, \]
\[ C_{\text{lim}} = 0.875, \]

\[ \sigma_{\omega2} = \begin{cases} 0, & \text{grad} \, \omega \, \text{grad} \, k \leq 0 \\ 0.125, & \text{grad} \, \omega \, \text{grad} \, k > 0 \end{cases}, \quad (3.25) \]

\[ \beta_1 = \beta_0 f_\beta, \]
\[ \beta_0 = 0.0708, \]
\[ f_\beta = \frac{1 + 85 \chi_\omega}{1 + 100 \chi_\omega} \quad (3.26) \]

For 2-D cases \( \chi_\omega = 0 \). For 3-D cases \( \chi_\omega \) is defined in equation (3.27):

\[ \chi_\omega = \frac{|\Omega \Omega^T \hat{D}^T|}{(\beta^* \omega)^3}, \]
\[ \Omega = \frac{1}{2} \left( \text{grad} \, u - \text{grad} \, u^T \right), \]
\[ \hat{D} = \frac{1}{2} \left( \text{grad} \, u + \text{grad} \, u^T \right) - \frac{1}{2} \text{div} \, u \mathbf{I} \quad (3.27) \]

### 3.5 ALE Reference Frame

The dilemma in solving an FSI problem is determining the frame of reference used to pass information between the fluid and solid. The Arbitrary Lagrangian-Eulerian description provides characteristics of both the Lagrangian and Eulerian
descriptions, allowing information to be passed between domains. The deformation gradient, $F$ is defined by:

$$F = \text{Grad } u_m$$  \hspace{1cm} (3.28)

Using the ALE derivative:

$$\frac{\partial \phi}{\partial t} = \frac{\partial \phi_\kappa}{\partial t} - \text{Grad } \phi_\kappa F^{-1} u_m,$$  \hspace{1cm} (3.29)

where $\phi$ can also be a vector, and the subscript $\kappa$ denotes the ALE reference frame, $F$ denotes the deformation gradient, and $u_m$ is the mesh velocity. The following relations are also needed:

$$\text{grad } \phi = \text{Grad } \phi_\kappa F^{-1},$$  \hspace{1cm} (3.30)

$$\text{div } T = \text{Grad } T_\kappa : F^{-T}.$$  \hspace{1cm} (3.31)

Transforming the RANS equations, shown in equations (3.16) and (3.17), from the deformed Eulerian configuration to the ALE reference frame results in:

$$\frac{\partial u_\kappa}{\partial t} + (\text{Grad } u_\kappa F^{-1}) [u_\kappa - u_m] =$$

$$-\text{Grad } (\bar{p} I) : F^{-T} + \text{Grad } ((\nu + \nu_t) (\text{Grad } u_\kappa F^{-1} + F^{-T} \text{Grad } u_\kappa ^T)) : F^{-T}$$

$$+ b_\kappa = 0 \hspace{1cm} \forall X \in \Omega_f(0),$$

$$F^{-1} : (\text{Grad } u_\kappa)^T = 0 \hspace{1cm} \forall X \in \Omega_f(0).$$  \hspace{1cm} (3.32)

The turbulent kinetic energy equation in the ALE reference frame is:

$$\frac{\partial k_\kappa}{\partial t} + (\text{Grad } k_\kappa F^{-1}) [u_\kappa - u_m] =$$

$$\frac{P_\kappa}{\rho} - \beta^* \omega_\kappa k_\kappa + \text{Grad } ((\nu + \sigma_k \nu_t) (\text{Grad } k_\kappa F^{-1})) : F^{-T} \hspace{1cm} \forall X \in \Omega_f(0).$$  \hspace{1cm} (3.33)

The specific dissipation rate equation in the ALE reference frame is:

$$\frac{\partial \omega_\kappa}{\partial t} + (\text{Grad } \omega_\kappa F^{-1}) [u_\kappa - u_m] =$$

$$\frac{\gamma P_\kappa}{\nu_t \rho} - \beta_1 \omega_\kappa^2 + \text{Grad } ((\nu + \sigma_\omega \nu_t) (\text{Grad } \omega_\kappa F^{-1})) : F^{-T}$$
\[ + \sigma_{\omega_2} \left( \text{Grad } k_\omega F^{-1} \right) \left( \text{Grad } \omega_\kappa F^{-1} \right) \forall X \in \Omega_f(0), \]

(3.34)

where, \( \frac{P_\kappa}{\rho} \) is:

\[
\frac{P_\kappa}{\rho} = \left( \nu_\kappa \left( \text{Grad } u_\kappa F^{-1} + F^{-T} \text{Grad } u_\kappa^T \right) F^{-T} - \frac{2}{3} k_\kappa I \right) : (\text{Grad } u_\kappa F^{-1}) .
\]

(3.35)

\( \hat{\omega} \) becomes:

\[
\hat{\omega}_\kappa = \max \left( \omega, C_{\text{lim}} \sqrt{\frac{2S_\kappa S_\kappa}{\beta^*}} \right),
\]

(3.36)

\[
S_\kappa = \frac{1}{2} \left( \text{Grad } u_\kappa F^{-1} + F^{-T} \text{Grad } u_\kappa^T \right).
\]

(3.37)

\( \sigma_{\omega_2} \) becomes:

\[
\sigma_{\omega_2} = \begin{cases} 
0, & \left( \text{Grad } k_\omega F^{-1} \right) \left( \text{Grad } \omega_\kappa F^{-1} \right) \leq 0 \\
0.125, & \left( \text{Grad } k_\omega F^{-1} \right) \left( \text{Grad } \omega_\kappa F^{-1} \right) > 0 
\end{cases}
\]

(3.38)

For 3-D cases \( \chi_{\omega_\kappa} \) is defined in equation (3.39):

\[
\chi_{\omega_\kappa} \equiv \left| \Omega_\kappa \Omega_\kappa^T \hat{D}_\kappa^T \right| \left( \beta^* \omega_\kappa \right)^3,
\]

\[
\Omega_\kappa = \frac{1}{2} \left( \text{Grad } u_\kappa F^{-1} - F^{-T} \text{Grad } u_\kappa^T \right),
\]

\[
\hat{D}_\kappa = \frac{1}{2} \left( \text{Grad } u_\kappa F^{-1} + F^{-T} \text{Grad } u_\kappa^T \right) - \frac{1}{2} \left( F^{-1} : (\text{Grad } u_\kappa^T) \right) I.
\]

(3.39)

3.6 Elasticity

The solid is governed by the equations of elasticity with the St. Venant Kirchhoff model in a Lagrangian reference frame. The velocity of the solid displacement is chosen to be a variable on the fluid-solid interface, requiring the two-field formulation of elastodynamics. To derive this formulation we start with the balance of linear momentum:

\[
\rho_s \frac{\partial u_s}{\partial t} - \text{Div } T_R = \rho_s b_s \quad \forall X \in \Omega_s.
\]

(3.40)
\( \mathbf{T}_R \) is given by:
\[ \mathbf{T}_R = \mathbf{F} \mathbf{S}. \]  
(3.41)

\( \mathbf{S} \), the second Piola-Kirchhoff stress tensor an isotropic material is given as:
\[ \mathbf{S} = \lambda \text{tr} (\mathbf{E}) \mathbf{I} + 2\mu \mathbf{E} \]  
(3.42)

where \( \lambda \) and \( \mu \) are Lamé’s first and second parameters. The strain tensor is expressed as a function of displacement, \( \mathbf{d}_s \):
\[ \mathbf{E} = \frac{1}{2} \left( \text{Grad} \mathbf{d}_s + (\text{Grad} \mathbf{d}_s)^T + (\text{Grad} \mathbf{d}_s)^T \text{Grad} \mathbf{d}_s \right). \]  
(3.43)

Due to separate displacement and velocity fields the kinematic compatibility condition is used:
\[ \frac{\partial \mathbf{d}_s}{\partial t} - \mathbf{u}_s = 0. \]  
(3.44)

This can also be shown in the one-field formulation:
\[ \rho_s \frac{\partial^2 \mathbf{d}_s}{\partial t^2} - \text{Div} \mathbf{T}_R = \rho_s \mathbf{b}_s \quad \forall \mathbf{X} \in \Omega_s \]  
(3.45)

A continuum mechanics text book, such as Gurtin (1982), can be used for details. The final problem statement becomes, find \( \mathbf{d}_s \) and \( \mathbf{u}_s \) such that:
\[ \rho_s \frac{\partial \mathbf{u}_s}{\partial t} - \text{Div} \mathbf{T}_R = \rho_s \mathbf{b}_s \quad \forall \mathbf{X} \in \Omega_s, \]  
(3.46)

\[ \frac{\partial \mathbf{d}_s}{\partial t} - \mathbf{u}_s = 0. \]  
(3.47)

### 3.7 Mesh Motion

The mesh was chosen as a linearly elastic material governed by an elastostatic equation:
\[ \text{Div} \mathbf{T}_R = 0 \quad \forall \mathbf{X} \in \Omega_f (0). \]  
(3.48)
Linearly elastic materials demand $\| \text{Grad } d_m \| \ll 1$. The infinitesimal strain tensor is a result from linearizing $E$ with respect to $\text{Grad } d_m$ and is given as:

$$e = \frac{1}{2} \left( \text{Grad } d_m + (\text{Grad } d_m)^T \right).$$

(3.49)

Thus the linearized first Piola-Kirchhoff stress with respect to $\text{Grad } d_m$ becomes:

$$T_R = \lambda_m \text{tr} (e) I + 2\mu_m e,$$

(3.50)

where $\lambda_m$ and $\mu_m$ are arbitrarily defined mesh properties.

A continuum mechanics text book, such as Gurtin (1982), can be used for details. The final problem statement becomes, find $d_m$ such that:

$$\text{Div} (\lambda_m \text{tr} (e) I + 2\mu_m e) = 0 \quad \forall X \in \Omega_f(0).$$

(3.51)

### 3.8 Interface Conditions

The following variables are described as Dirichlet boundary conditions:

$$d_s|_{\Gamma_{DS}} = \tilde{d}_s(X, t),$$

(3.52)

$$u_s|_{\Gamma_{DS}} = \frac{\partial}{\partial t} \tilde{u}_s(X, t),$$

(3.53)

$$u_\kappa|_{\Gamma_{DF}} = \tilde{u}_\kappa(X, t).$$

(3.54)

The tractions are described as Neumann boundary conditions:

$$T_s[n_s]|_{\Gamma_{NS}} = \tilde{t}_s(x, t),$$

(3.55)

$$T_\kappa[n_\kappa]|_{\Gamma_{NP}} = \tilde{t}_\kappa(x, t).$$

(3.56)

To maintain conforming geometry for the mesh, a constraint is placed on the entire boundary for no normal mesh displacement:

$$(d_m \cdot n_f)|_{\Gamma_{DF} \cup \Gamma_{NF}} = 0.$$
Three additional conditions are imposed on the fluid-solid interface due to FSI coupling. The mesh displacement is governed by the solid displacement on the interface:

\[(d_m - d_s)|_{\Gamma_{DFS}} = 0, \quad (3.58)\]

the velocity field is considered continuous across the interface:

\[(u_\kappa - u_s)|_{\Gamma_{DFS}} = 0, \quad (3.59)\]

the tractions are also continuous across the interface:

\[ (T_s [n_s] + T_\kappa [n_\kappa])|_{\Gamma_{NFS}} = (T_s - T_\kappa) [n_s]|_{\Gamma_{NFS}}, \quad (3.60) \]

where \( n_s = -n_\kappa. \)

### 3.9 Governing Equations Summary

The derived governing equations are summarized below:

1. For the fluid formulation find \( u_f, p, k, \) and \( \omega \) such that:

\[
\frac{\partial u_\kappa}{\partial t} + (\text{Grad} u_\kappa F^{-1}) [u_\kappa - u_m] = \\
- \text{Div} \left( \bar{p} I F^{-T} \right) + \text{Div} \left( (\nu + \nu_t)(\text{Grad} u_\kappa F^{-1} + F^{-T} \text{Grad} u_\kappa^T) F^{-T} \right) \\
+ b_\kappa \quad \forall X \in \Omega_f(0),
\]

\[
F^{-1} : (\text{Grad} u_\kappa)^T = 0 \quad \forall X \in \Omega_f(0). \quad (3.61)
\]

\[
\frac{\partial k_\kappa}{\partial t} + (\text{Grad} k_\kappa F^{-1}) [u_\kappa - u_m] = \\
\frac{P_\kappa}{\rho} - \beta^* \omega_\kappa k_\kappa + \text{Div} \left( (\nu + \sigma_k \nu_t)(\text{Grad} k_\kappa F^{-1}) F^{-T} \right) \quad \forall X \in \Omega_f(0).
\]

\[
(3.62)
\]
\[
\frac{\partial \omega_k}{\partial t} + (\text{Grad } \omega_k \mathbf{F}^{-1}) [\mathbf{u}_k - \mathbf{u}_m] = \\
\gamma \frac{P_k}{\nu_t \rho} - \beta_1 \omega_k^2 + \text{Div } \left( (\nu + \sigma_\omega \nu_t)(\text{Grad } \omega_k \mathbf{F}^{-1}) \mathbf{F}^{-T} \right) \\
+ \frac{\sigma_\omega^2}{\omega_k} (\text{Grad } k_n \mathbf{F}^{-1}) (\text{Grad } \omega_k \mathbf{F}^{-1}) \quad \forall \mathbf{X} \in \Omega_f(0).
\]

(3.63)

2. For the solid formulation find \( \mathbf{d}_s \) and \( \mathbf{u}_s \) such that:

\[
\rho_s \frac{\partial \mathbf{u}_s}{\partial t} - \text{Div } (\mathbf{F} (\lambda \text{tr } (\mathbf{E}) \mathbf{I} + 2\mu \mathbf{E})) = \rho_s \mathbf{b}_s \quad \forall \mathbf{X} \in \Omega_s.
\]

(3.64)

The strain tensor is expressed as a function of displacement, \( \mathbf{d}_s \):

\[
\mathbf{E} = \frac{1}{2} \left( \text{Grad } \mathbf{d}_s + (\text{Grad } \mathbf{d}_s)^T + (\text{Grad } \mathbf{d}_s)^T \text{Grad } \mathbf{d}_s \right).
\]

(3.65)

The kinematic compatibility condition is:

\[
\frac{\partial \mathbf{d}_s}{\partial t} - \mathbf{u}_s = 0.
\]

(3.66)

3. For the mesh formulation find \( \mathbf{d}_m \) such that:

\[
\text{Div } (\lambda_m \text{tr } (\mathbf{e}) \mathbf{I} + 2\mu_m \mathbf{e}) = 0 \quad \forall \mathbf{X} \in \Omega_f(0).
\]

(3.67)

Where, \( \mathbf{e} \) is given as:

\[
\mathbf{e} = \frac{1}{2} \left( \text{Grad } \mathbf{d}_m + (\text{Grad } \mathbf{d}_m)^T \right).
\]

(3.68)

4. The following interface conditions are used: Mesh constraint:

\[
(\mathbf{d}_m \cdot \mathbf{n}_f)|_{\Gamma_{D_F} \cup \Gamma_{N_F}} = 0.
\]

(3.69)
Mesh and solid displacement:

\[(d_m - d_s)\big|_{\Gamma_{DFS}} = 0, \quad (3.70)\]

Continuous velocity and tractions on the interface:

\[(u_\kappa - u_s)\big|_{\Gamma_{DFS}} = 0, \quad (3.71)\]

\[(T_s [n_s] + T_\kappa [n_\kappa])\big|_{\Gamma_{NFS}} = (T_s - T_\kappa) [n_s]\big|_{\Gamma_{NFS}}. \quad (3.72)\]

In summary, time averaging is utilized to derive the Reynolds Averaged Navier-Stokes equations. The RANS equations govern the fluid domain. The \(k - \omega\) turbulence model is implemented to close the fluid system of equations. The fluid formulation is put in the ALE reference frame. The solid governing equations are derived, in a Lagrangian reference frame, using the theory of elasticity with a St. Venant Kirchoff model. Mesh motion was chosen to be described by a linear elastic material. This provides a mathematical model in the form of PDEs for the entire FSI domain. An analytical solution to the governing equations is not known. Therefore, an approximation can be made using a numerical method discussed in the next chapter.
Numerical Methods

In Chapter 3, a system of PDEs are derived with an unobtainable analytic solution. The FEM provides a straightforward approach to approximating the fluid and solid allowing for a common numerical discretization. The SUPG method is presented for stabilization of the fluid solution.

This chapter derives the weak form of the governing equations from the strong form, developing the spatial and temporal discretization. The fluid formulation is discussed in thorough detail in this chapter. Details of the solid and mesh formulations are in Appendix A. The formulations are implemented with a partitioned and monolithic approach using an open source finite element library known as deal.ii (Bangerth et al., 2007).

4.1 Weak Formulation

Chapter 3 derives the governing equations in the form of a system of PDEs, known as the strong form. An integral form of the governing equations, known as the weak form, will enable the finite element method to be used (Fish & Belytschko, 2007).

To obtain the weak formulation for the fluid, solid, and mesh, each domain is weighted by a weighting function and integrated over the domain. This is notated by \((a,b)_{\Omega} \equiv \int_{\Omega} a \cdot b \, d\Omega\), where \(\Omega\) is the given domain. The weighting function is determined by the method used. The Galerkin method is used for the solid and mesh formulations and discussed in more detail by Sheldon et al. (2014). The
SUPG method is used for the fluid formulation and details are discussed in the following sections.

4.2 Spatial and Temporal Discretization

The fluid formulation is nonlinear, making the system of equations more complex. The Newton-Raphson method is employed to linearize the system of equations. The weak form is discretized into piecewise interpolations to obtain the residual and corresponding Jacobian terms needed for the Newton-Raphson method. First the spatial and temporal discretization is discussed.

To create a partially discretized problem, the dependent variables are approximated with the following form:

$$\phi \approx \phi_h = \sum \Phi_j \hat{\phi}_j, \quad (4.1)$$

where $\hat{\phi}$ denotes the weighting function. The Galerkin method sets the weighting function equivalent to the basis function:

$$\hat{\phi} = \sum \Phi_i \hat{\phi}_i. \quad (4.2)$$

The Galerkin method is sufficient for the solid and mesh formulations, due to high Reynolds number flow instabilities may occur in the fluid solution. To eliminate instabilities that may occur the SUPG method adds artificial diffusion to the governing fluid equations but does not have the loss of accuracy of other upwind methods when adding artificial diffusion making it more appealing. This is due to the artificial diffusion being added only the streamline direction. The streamline perturbation prevents oscillations that result from high Reynolds numbers or convection dominated flow (Brooks & Hughes, 1982).

The SUPG method is implemented as follows. Let $B(u, \hat{u})$ be the weak form of the Navier Stokes equation. Using the SUPG method the fluid formulation becomes:

$$B(u, \hat{u}) + S(u, \hat{w}) = 0, \quad (4.3)$$
where \( \mathbf{\dot{w}} \) is:

\[
\mathbf{\dot{w}} = \tau_{\text{SUPG}} \mathbf{u} \cdot \nabla \mathbf{\hat{u}}.
\] (4.4)

The SUPG method can also be written as:

\[
B (\mathbf{u}, \mathbf{\tilde{u}}) = 0,
\] (4.5)

where,

\[
\mathbf{\tilde{u}} = \mathbf{\hat{u}} + \mathbf{\dot{w}}.
\] (4.6)

Appendix B provides test problems using the SUPG method.

To obtain a fully discretized problem a numerical time scheme is introduced. Taking a general ODE, \( \dot{\eta} = f(\eta, t) \), to show how a finite difference scheme is used to approximate time derivatives. The ODE when integrated with respect to time can be approximated as an explicit scheme is shown in equation (4.7):

\[
\frac{\eta^{n+1} - \eta^n}{\Delta t} = f(\eta^n, t).
\] (4.7)

Or it can be approximated as an implicit scheme shown in equation (4.8):

\[
\frac{\eta^{n+1} - \eta^n}{\Delta t} = f(\eta^{n+1}, t).
\] (4.8)

A combined implicit/explicit scheme known as the \( \theta \)-method where a weighting parameter, \( \theta \) is used. The \( \theta \)-method is shown in equation (4.9):

\[
\frac{\eta^{n+1} - \eta^n}{\Delta t} = \theta f(\eta^{n+1}, t) + (1 - \theta) f(\eta^n, t).
\] (4.9)

If \( \theta = 0 \) the scheme becomes fully explicit and if \( \theta = 1 \) the scheme is fully implicit. An implicit time scheme is used for the developed algorithm, but can be changed if necessary.

### 4.3 Fluid Formulation

This results in a discretized non-linear fluid formulation, linearized by the Newton-Rhapson method. The Newton-Rhapson method takes an initial guess
and approximates a solution to the equation using the functions’ derivatives:

$$\phi^{n+1} = \phi^n - \frac{R(\phi^n)}{J(\phi^n)}. \quad (4.10)$$

The Jacobian, $J$, is defined as:

$$J(\psi) = \frac{\partial R_i}{\partial \psi_j}. \quad (4.11)$$

The fluid formulation is of the form:

Find \( \{ u_h^k, p_h^k, \xi_h^k \} \) such that

$$J \left( \frac{\partial u_k}{\partial t} + (\text{Grad} \ u_k F^{-1}) \left[ u_k - u_m \right] - b_k, \tilde{u}^i_f \right)_{\Omega_f} - J \left( \tilde{p}_k F^{-1}, \text{Grad} \ \tilde{u}^i_f \right)_{\Omega_f}$$

$$+ J \left( (\nu + \nu_t) \left( \text{Grad} \ u_k F^{-1} + F^{-1} \text{Grad} \ u_k^T \right) F^{-T}, \text{Grad} \ \tilde{u}^i_f \right)_{\Omega_f} = 0$$

$$\forall X \in \Omega_f(0), \quad (4.12)$$

$$J \left( F^{-1} : \text{Grad} \ u_k^T, \tilde{p}_j^i \right)_{\Omega_f} = 0 \quad \forall X \in \Omega_f(0), \quad (4.13)$$

$J$ is the determinant of the deformation gradient. It is obtained from transferring from the Eulerian reference frame to the ALE reference frame:

$$\int_{\Omega_f(t)} f(x) dx = \int_{\Omega_f(0)} f_k(X) JdX \quad (4.14)$$

This results in a system of equations expressed in matrix form below:

$$\begin{bmatrix}
J_{uu} & J_{up} & J_{uk} & J_{uw} \\
J_{pu} & J_{pp} & J_{pk} & J_{pw} \\
J_{ku} & J_{kp} & J_{kk} & J_{kw} \\
J_{wu} & J_{wp} & J_{wk} & J_{ww}
\end{bmatrix}
\begin{bmatrix}
\Delta u \\
\Delta p \\
\Delta k \\
\Delta \omega
\end{bmatrix} = -
\begin{bmatrix}
R_u \\
R_p \\
R_k \\
R_\omega
\end{bmatrix}, \quad (4.15)$$

where all residual and Jacobian terms are shown in Appendix A.
4.4 FSI Formulation Summary

The full FSI system of equations can also be shown in matrix form:

\[
\begin{bmatrix}
J & \begin{pmatrix}
\Delta d_s \\
\Delta u_s \\
\Delta u_f \\
\Delta p_f \\
\Delta k_f \\
\Delta \omega_f \\
\Delta d_m \\
\end{pmatrix}
\end{bmatrix} = - \begin{bmatrix}
R_{d_s} \\
R_{u_s} \\
R_{u_f} \\
R_{p_f} \\
R_{k_f} \\
R_{\omega_f} \\
R_{d_m} \\
\end{bmatrix}
\]

(4.16)

where \( J \) is:

\[
J = \begin{bmatrix}
J_{d_s,d_s} & J_{d_s,u_s} & J_{d_s,u_f} & J_{d_s,p_f} & J_{d_s,k_f} & J_{d_s,\omega_f} & J_{d_s,d_m} \\
J_{u_s,d_s} & J_{u_s,u_s} & J_{u_s,u_f} & J_{u_s,p_f} & J_{u_s,k_f} & J_{u_s,\omega_f} & J_{u_s,d_m} \\
J_{u_f,d_s} & J_{u_f,u_s} & J_{u_f,u_f} & J_{u_f,p_f} & J_{u_f,k_f} & J_{u_f,\omega_f} & J_{u_f,d_m} \\
J_{p_f,d_s} & J_{p_f,u_s} & J_{p_f,u_f} & J_{p_f,p_f} & J_{p_f,k_f} & J_{p_f,\omega_f} & J_{p_f,d_m} \\
J_{k_f,d_s} & J_{k_f,u_s} & J_{k_f,u_f} & J_{k_f,p_f} & J_{k_f,k_f} & J_{k_f,\omega_f} & J_{k_f,d_m} \\
J_{\omega_f,d_s} & J_{\omega_f,u_s} & J_{\omega_f,u_f} & J_{\omega_f,p_f} & J_{\omega_f,k_f} & J_{\omega_f,\omega_f} & J_{\omega_f,d_m} \\
J_{d_m,d_s} & J_{d_m,u_s} & J_{d_m,u_f} & J_{d_m,p_f} & J_{d_m,k_f} & J_{d_m,\omega_f} & J_{d_m,d_m} \\
\end{bmatrix}
\]

(4.17)

and the residual and Jacobian terms are shown in Appendix A.

Several Jacobian terms can be reduced to zero:

\[
J = \begin{bmatrix}
J_{d_s,d_s} & J_{d_s,u_s} & 0 & 0 & 0 & 0 & 0 \\
J_{u_s,d_s} & J_{u_s,u_s} & 0 & 0 & 0 & 0 & 0 \\
0 & J_{u_f,u_s} & J_{u_f,u_f} & J_{u_f,p_f} & 0 & 0 & J_{u_f,d_m} \\
0 & J_{p_f,u_s} & J_{p_f,u_f} & 0 & 0 & 0 & J_{p_f,d_m} \\
0 & 0 & 0 & J_{k_f,k_f} & J_{k_f,\omega_f} & 0 \\
0 & 0 & 0 & J_{\omega_f,k_f} & J_{\omega_f,\omega_f} & 0 \\
0 & 0 & 0 & 0 & 0 & J_{d_m,d_m} \\
\end{bmatrix}
\]

(4.18)

With the system of equations in matrix form an approximate solution is now
obtainable. The formulation is implemented in C++ using deal.ii, an open source finite element library and details are discussed in the following section.

4.5 Implementation

The discussed numerical formulation is computed in C++ using an open source finite element library known as deal.ii. Deal.ii allows for the details of mesh usage and refinement, degrees of freedom usage and more to be handled by the library (Bangerth et al., 2007).

A common approach to implement a turbulence model takes a segregated approach as in Kuzmin et al. (2007), where the utilization of the $k - \epsilon$ model is discussed in detail. The author’s work expands previous research with specific interest in the near-wall region, where modeling techniques are generally implemented due to the known near-wall tendencies of the $k - \epsilon$ model (Kuzmin et al., 2007).

To numerically approximate a solution for the system of equations with a segregated approach the coupling of the velocity, pressure, or mesh with the turbulent terms are neglected. The mesh is solved first followed by the fluid velocity and pressure. The fluid velocity solution obtained is used as a constant to solve for the turbulent kinetic energy, $k$, and the specific dissipation rate, $\omega$. The updated turbulent values obtain a new value for $\nu_t$. Using this new value the solution of the fluid velocity and pressure are updated followed by updating the turbulent terms again. The coupling loop of the fluid and turbulent terms continues until a convergence criteria is met as represented in Figure 4.1.
Figure 4.1: A diagram of the implementation of the fluid formulation using a segregated approach. The solution for the fluid velocity is obtained and set constant to solve for the turbulence terms. A new value for $\nu_t$ is obtained and used to update the fluid solution. The iteration between the fluid and turbulent terms continues until a convergence criteria is met. Several subiterations are required.

A monolithic approach may be more cost effective and accurate by solving for all FSI terms simultaneously. The monolithic approach eliminates the iterative loop between the fluid terms and the turbulent terms. All variables are approximated in the same iteration, eliminating subiterations, as shown in Figure 4.2.
Figure 4.2: A diagram of the implementation of the fluid formulation using a monolithic approach. This approach eliminates the iterative loop between the fluid terms and the turbulent terms. All variables are approximated in the same iteration.

The coupled fluid-turbulence solution is relaxed in the following manner (Küttler & Wall, 2008; Sheldon, 2012):

$$\phi^{n+1} = \phi^n + a^{n+1} \left( \phi^{n+1} - \phi^n \right),$$

(4.19)

where $\phi$ is representative of the fluid and turbulent solution terms and $a^{n+1}$ is a relaxation factor. Here $a^{n+1} = a = 0.5$.

### 4.6 Method of Manufactured Solutions

MMS uses an arbitrary solution for verification of mathematical accuracy and ignores any physics related to the problem (Roache, 2002). The Navier-Stokes equations, in the Eulerian reference frame, are verified with MMS.
An exact solution is manufactured when one cannot be obtained analytically. The method is described specific for this work, but can be generalized if necessary:

1. A non-trivial, continuous solution is produced by solving a PDE, specifically the continuity equation:

   \[ u^* = \begin{pmatrix} \sin(4\pi x) \sin(4\pi y) \sin(4\pi t) \\ \cos(4\pi x) \cos(4\pi y) \sin(4\pi t) \end{pmatrix} \]. \hfill (4.20)

   The pressure could be any solution, and was chosen to be:

   \[ p^* = \sin(4\pi x) \sin(4\pi y) \sin(4\pi t) \], \hfill (4.21)

   and let:

   \[ \phi^* = \begin{pmatrix} u^* \\ p^* \end{pmatrix} \]. \hfill (4.22)

2. A source term is computed from the manufactured solutions of \( u^* \) and \( p^* \):

   \[ R_1 = 4\pi \cos(4\pi t) \sin(4\pi x) \sin(4\pi y) + 4\pi \cos(4\pi x) \sin(4\pi t) \sin(4\pi y) + 16\pi^2 \sin(4\pi x) \sin(4\pi y) + 4\pi \cos(4\pi x) \cos(4\pi y)^2 \sin(4\pi t)^2 \sin(4\pi x) + 4\pi \cos(4\pi x) \sin(4\pi t)^2 \sin(4\pi x) \sin(4\pi y)^2 \] \hfill (4.23)

   \[ R_2 = 4\pi \cos(4\pi t) \cos(4\pi x) \cos(4\pi y) + 4\pi \cos(4\pi y) \sin(4\pi t) \sin(4\pi x) + 16\pi^2 \cos(4\pi x) \cos(4\pi y) \sin(4\pi t) - 4\pi \cos(4\pi x)^2 \cos(4\pi y) \sin(4\pi t)^2 \sin(4\pi y) - 4\pi \cos(4\pi y) \sin(4\pi t)^2 \sin(4\pi x) \sin(4\pi y)^2 \] \hfill (4.24)

3. Evaluate by computing a solution, \( \phi_h \), using the source term. The domain is a 1x1 unit with \( \nu = 1.0 \), and dirichlet boundary conditions are used, meaning the exact solution governs the boundaries of the domain.

4. Compute the error using grid refinement. The following norms are utilized for convergence rate studies:
The error of $\phi_h$ in the $L_2$ norm:

$$\text{error}_{L_2} = \left( \int_{\Omega} |\phi^* - \phi_h|^2 \, dx \right)^{\frac{1}{2}} \quad (4.25)$$

The error of $\phi_h$ in the $H^1$ semi-norm:

$$\text{error}_{H_1} = \left( \int_{\Omega} \left| \frac{d\phi^*}{dx} - \frac{d\phi_h}{dx} \right|^2 \, dx \right)^{\frac{1}{2}} \quad (4.26)$$

The error of $\phi_h$ in the $L_\infty$ norm:

$$\text{error}_{L_\infty} = \max_{x \in \Omega} |\phi^* - \phi_h| \quad (4.27)$$

5. Using the described norms the computed solution is compared to the manufactured solution for each component of the fluid velocity and the pressure using the following equation:

$$m = \frac{\text{error}}{\ln \frac{1}{h}} \quad (4.28)$$

where $m$ is the slope of the line and $h$ is the cell width of the current grid refinement.

Figure 4.3 shows the calculated norms using equation 4.28 for the X component of the fluid velocity and Table 4.1 shows the convergence rates.
Figure 4.3: Plot of the calculated norms using equation 4.28 for the X component of the fluid velocity. The X component of the fluid velocity converges at the expected convergence rate for 2nd order.

Table 4.1. Refinement table of the fluid velocity X component. The X component of the fluid velocity converges at the expected convergence rate for 2nd order.

<table>
<thead>
<tr>
<th>cycle</th>
<th># cells</th>
<th># dofs</th>
<th>L2-error</th>
<th>H1-error</th>
<th>Linfty-error</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>16</td>
<td>187</td>
<td>8.135e-05</td>
<td>-</td>
<td>1.052e-03</td>
</tr>
<tr>
<td>1</td>
<td>64</td>
<td>659</td>
<td>2.027e-05</td>
<td>2.00</td>
<td>1.584e-03</td>
</tr>
<tr>
<td>2</td>
<td>256</td>
<td>2467</td>
<td>2.055e-06</td>
<td>3.30</td>
<td>3.160e-04</td>
</tr>
<tr>
<td>3</td>
<td>1024</td>
<td>9539</td>
<td>2.521e-07</td>
<td>3.03</td>
<td>6.490e-05</td>
</tr>
<tr>
<td>4</td>
<td>4096</td>
<td>37507</td>
<td>3.219e-08</td>
<td>2.97</td>
<td>1.605e-05</td>
</tr>
</tbody>
</table>

Figure 4.4 shows the calculated norms for the Y component of the fluid velocity and Table 4.2 shows the convergence rates.
Figure 4.4: Plot of the calculated norms using equation 4.28 for the Y component of the fluid velocity. The Y component of the fluid velocity converges at the expected convergence rate for 2$^{nd}$ order.

Table 4.2. Refinement table of the fluid velocity Y component. The Y component of the fluid velocity converges at the expected convergence rate for 2$^{nd}$ order.

<table>
<thead>
<tr>
<th>cycle</th>
<th># cells</th>
<th># dofs</th>
<th>L2-error</th>
<th>H1-error</th>
<th>Linfty-error</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>16</td>
<td>187</td>
<td>3.74e-04</td>
<td>-</td>
<td>1.430e-02</td>
</tr>
<tr>
<td>1</td>
<td>64</td>
<td>659</td>
<td>2.402e-05</td>
<td>3.96</td>
<td>1.822e-03</td>
</tr>
<tr>
<td>2</td>
<td>256</td>
<td>2467</td>
<td>2.074e-06</td>
<td>3.53</td>
<td>3.072e-04</td>
</tr>
<tr>
<td>3</td>
<td>1024</td>
<td>9539</td>
<td>2.530e-07</td>
<td>3.04</td>
<td>6.488e-05</td>
</tr>
<tr>
<td>4</td>
<td>4096</td>
<td>37507</td>
<td>3.220e-08</td>
<td>2.97</td>
<td>1.605e-05</td>
</tr>
</tbody>
</table>

Figure 4.5 shows the calculated norms for the pressure and Table 4.3 shows the convergence rates.
Figure 4.5: Plot of the calculated norms using equation 4.28 for the pressure component of the fluid formulation. The pressure component of the fluid formulation converges at the expected convergence rate for 2\textsuperscript{nd} order.

Table 4.3. Refinement table of the pressure component, converging at the expected rate for 2\textsuperscript{nd} order.

<table>
<thead>
<tr>
<th>cycle</th>
<th># cells</th>
<th># dofs</th>
<th>L2-error</th>
<th>H1-error</th>
<th>Linfty-error</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>16</td>
<td>187</td>
<td>7.097e-04</td>
<td>-</td>
<td>1.180e-02</td>
</tr>
<tr>
<td>1</td>
<td>64</td>
<td>659</td>
<td>3.642e-04</td>
<td>0.96</td>
<td>6.979e-03</td>
</tr>
<tr>
<td>2</td>
<td>256</td>
<td>2467</td>
<td>6.322e-05</td>
<td>2.53</td>
<td>2.846e-03</td>
</tr>
<tr>
<td>3</td>
<td>1024</td>
<td>9539</td>
<td>6.709e-06</td>
<td>3.24</td>
<td>1.279e-03</td>
</tr>
<tr>
<td>4</td>
<td>4096</td>
<td>37507</td>
<td>1.327e-06</td>
<td>2.34</td>
<td>6.339e-04</td>
</tr>
</tbody>
</table>
Chapter 5

Numerical Experimentation

We have interest in developing tools to better understand high Reynolds FSI. The RANS equations are presented in Chapter 3 along with the $k - \omega$ turbulence model. The implementation of the RANS equations and the turbulence model are developed in Chapter 4.

Similarly in Chapter 3 the governing equations for the mesh are derived and implemented in Chapter 4. In this chapter the mesh motion is numerically observed using two test cases. The first case is a side-to-side deformation case. The second case is a rotating paddle case.

The NASA NACA0012 validation study (Rumsey, 2014) case is an adequate high Reynolds number FSI problem in aerospace applications. Providing results to compare to the validation study will further validate the implementation of the formulation, but more importantly it will be a progressive step towards a flexible 2-D airfoil, numerical test case.

5.1 Mesh Motion

The objective of the following numerical experiments are to ensure adequate mesh motion implementation. The same domain is used for both cases, a 5x5 square outer domain is given with an inner 1x0.5 rigid, rectangle domain shown in Figure 5.1.
Figure 5.1: A figure of the domain used for two mesh motion simulations. A 5x5 square outer domain is given with an inner 1x0.5 rigid, rectangle domain.

The boundary conditions imposed on the outer square domain are the same for both simulations and prescribed below:

\[ d_m = 0, \]  
\[ (5.1) \]

\[ u_f = 0, \]  
\[ (5.2) \]

\[ p = 0. \]  
\[ (5.3) \]

### 5.1.1 Side-to-Side Motion

For the first case, the rectangle, initially at rest, is forced to move side-to-side. The boundary conditions imposed on the inner rectangle are:

\[ \theta = \frac{\pi}{4} t \]  
\[ (5.4) \]

\[ d_{m,x} = \frac{1}{8} (1 - \cos \theta) \]  
\[ (5.5) \]
\[ d_{m,y} = 0 \quad (5.6) \]
\[ u_f = \dot{d}_m \quad (5.7) \]
\[ u_{f,x} = \frac{\pi}{32} \sin \theta \quad (5.8) \]
\[ \nabla p \cdot \hat{n} = 0 \quad (5.9) \]

Results and Conclusions

A maximum deformation occurs on the right side of the domain shown in Figure 5.2.

Figure 5.2: A figure depicting the maximum deformation occurring on the right side of the domain for side-to-side motion of the rigid, inner rectangle.

The rigid body’s motion causes the flow to be relocated. Figure 5.3 shows the magnitude of the fluid velocity with the deformed mesh at maximum deformation.
Figure 5.3: A plot of the fluid velocity magnitude at maximum deformation for side-to-side motion of the rigid, inner rectangle.

The pressure is also shown with the deformed mesh at maximum deformation in Figure 5.4.

Figure 5.4: A plot of the pressure at maximum deformation for side-to-side motion of the rigid, inner rectangle.

5.1.2 Rotating Paddle

A second mesh motion case explored is a rotating paddle. The domain from Figure 5.1 is used. The inner rectangle, initially at rest, is rotated about the origin.
by $\pm 5^\circ$. Therefore, the boundary conditions imposed on the inner rectangle are:

$$\theta (t) = 5 \frac{\pi}{180} \sin (\pi t) \quad (5.10)$$

$$d_{m,x} = x \cos \theta - y \sin \theta - x \quad (5.11)$$

$$d_{m,y} = x \sin \theta + y \cos \theta - y \quad (5.12)$$

$$u_{f,x} = 5 \frac{\pi}{180} \cos (\pi t) (-x \sin \theta - y \cos \theta + y) \quad (5.13)$$

$$u_{f,y} = 5 \frac{\pi}{180} \cos (\pi t) (x \cos \theta - y \sin \theta - x) \quad (5.14)$$

**Results and Conclusions**

The rotated rectangle at maximum rotation is shown in Figure 5.5

Figure 5.5: A figure depicting the maximum deformation occurring when the rigid, inner rectangle is rotated counter-clockwise $\pm 5^\circ$.

Figure 5.6 shows the magnitude of the fluid velocity caused by the rigid body rotation at maximum deformation while Figure 5.7 shows the pressure.
5.2 Flow Over a 2-D Airfoil

High Reynolds number FSI is applicable to several engineering disciplines and several types of test cases would be acceptable. For this work, a comparison is made to the NASA NACA0012 validation study (Rumsey, 2014). This will suffice
as further validation of the implementation of the turbulence model and appropriate transformation into the ALE formulation.

The 225x65 grid is used from Rumsey (2014). Figure 5.8 is the entire grid domain and while Figure 5.9 is a near view picture of the NACA0012 airfoil in the domain, being it is too small to see in Figure 5.8.

![Figure 5.8](image1.png)

Figure 5.8: A far view of the NACA0012 airfoil mesh taken from (Rumsey, 2014) to be used as validation study.

![Figure 5.9](image2.png)

Figure 5.9: A near view of the NACA0012 airfoil mesh taken from (Rumsey, 2014) to be used as validation study.
The velocity is uniformly parallel to the airfoil:

\[ \mathbf{u}_{in} = \begin{cases} u_x = U_\infty \\ u_y = 0 \end{cases}, \quad (5.15) \]

where the freestream velocity is given as:

\[ U_\infty = 10.0 \left[ \frac{m}{s} \right]. \quad (5.16) \]

The freestream conditions were defined to be the same as the inlet conditions, freestream conditions = inlet conditions. The the inlet and airfoil parameters for \( k \) and \( \omega \) are described below:

\[ k_{in} = 10^{-9} \left[ \frac{m^2}{s^2} \right], \]
\[ \omega_{in} = 0.5 \left[ \frac{1}{s} \right], \quad (5.17) \]
\[ k_{wall} = 0, \]
\[ \omega_{wall} = 20^{-4} \left[ \frac{1}{s} \right]. \]

To find the pressure coefficient, \( C_p \), the pressure across the airfoil is non-dimensionalized in equation (5.18):

\[ C_p = \frac{p}{(1/2) \rho u_\infty^2} = \frac{\tilde{p} - \frac{2}{\gamma} k}{(1/2) u_\infty^2}, \quad (5.18) \]

where \( \tilde{p} \) is the modified pressure solved for numerically.

**Results and Conclusions**

\( C_p \) is plotted over the airfoil non-dimensionally by \( x/c \) and compared to data from Gregory & O’Reilly (Jan 1970) shown in Figure 5.10.
Figure 5.10: A plot of $C_p$ versus x/c over a fixed NACA0012 at ±0° angle of attack, $\alpha$. This compares the turbulence model implemented in deal.ii to experimental data provided by (Rumsey, 2014). (Rumsey, 2014) is a compressible test case, but can be used to compare against incompressible data.

Rumsey (2014) notes the validation cases provided are for compressible flow. Incompressible equations can be used, but the results may vary slightly, contributing to the disagreement in the results of Figure 5.10.

The pressure is plotted in Figure 5.11 once the solution reached steady state. The results show a high pressure region at the leading edge of the airfoil. The pressure is much lower for the rest of the airfoil, comparable to 5.10.

Figure 5.11: A plot of the pressure over a fixed NACA0012 at ±0° angle of attack, $\alpha$. The results show a high pressure region at the leading edge of the airfoil, at the stagnation point, and a lower pressure for the rest of the airfoil.

The steady state conditions for the magnitude of the fluid velocity is also plotted in Figure 5.12. The wake region is clearly visible, but could be resolved further with a finer grid. A course grid was used due to computational time. A finer grid should be used for future calculations and is discussed in the next chapter.
Figure 5.12: A plot of the fluid velocity magnitude over a fixed NACA0012 at $\pm 0^\circ$ angle of attack, $\alpha$. The wake region is clearly visible, but could be resolved further with a finer grid. A course grid was used due to computational time.

The turbulence terms are plotted over the NACA0012 airfoil. Figure 5.13 shows the turbulent kinetic energy, $k$. Using the RANS equation results in the turbulent scales being averaged instead of being computed, leaving out the deatails of the wake of the turbulent kinetic energy and having an average of zero.

Figure 5.13: A plot of the turbulent kinetic energy, $k$, over a fixed NACA0012 at $\pm 0^\circ$ angle of attack, $\alpha$. The RANS equations result in the turbulent scales being averaged instead of being computed, leaving out the deatails of the wake of the turbulent kinetic energy and having an average of zero.
Figure 5.14 is a plot of the specific dissipation rate, \( \omega \). A high dissipation rate is seen at the leading edge.

Figure 5.14: A plot of the specific dissipation rate, \( \omega \), over a fixed NACA0012 at \( \pm 0^\circ \) angle of attack, \( \alpha \). The RANS equations result in the turbulent scales being averaged instead of being computed, leaving out the details of the wake. A high dissipation rate is seen at the leading edge.
The finite element method is utilized to further develop an algorithm for high Reynolds number FSI. The governing equations are derived, implemented, and verified via numerical experimentation in this work. This chapter provides a detailed summary of the work completed providing a thorough review of each chapter. A proposal for future work is also provided.

6.1 Summary

This research discusses numerical computations of fully coupled, external, high Reynolds number flow interacting with solids having large deformations. The finite element method provides a common framework to solve for the fluid and solid. This allows for either a monolithic or a partitioned approach to be implemented. High Reynolds number, FEM based fluid approximations are subject to instabilities occurring in the solution, producing inaccurate solutions. The fluid formulation requires turbulence modeling due to the natural nature of high Reynolds number flow. The motivation for this work is influenced by the following:

1. instabilities occurring in the fluid solution,
2. adequate modeling of the turbulence due to the high Reynolds numbers, and
3. multi-order accuracy for the velocity and pressure terms in the fluid formulation.
Chapter 2 gives a comprehensive review of current literature, aiding in background knowledge of this work. In Chapter 3 the balance laws are used with constitutive relations to arrive at the governing equations. The governing equations for the fluid are time averaged to account for the mean and fluctuating components of turbulent flow. This results in the RANS equations. Using the RANS equations presents the classical turbulence closure problem having more unknowns than equations. The $k-\omega$ turbulence model is used to close the system of equations accounting for the Reynolds-stress tensor. The resulting fluid governing equations are cast in an ALE frame of reference, and the solid and mesh governing equations remain in a Lagrangian frame of reference. This provides the framework to solve for the fluid and solid in common reference frame.

In Chapter 4 the solid and mesh formulations are discretized using the classical Galerkin method while the fluid formulation is discretized using the SUPG method for stabilization of high Reynolds number, FEM based instabilities. An equal order projection scheme is added to the fluid formulation to allow for equal order accuracy of the fluid velocity and pressure. The formulation is implemented using an open source finite element library, deal.ii. A partitioned approach is presented for the mesh and fluid implementation followed by discussion for a monolithic approach.

Chapter 5 presents numerical experiments with the implemented algorithm. Horizontal, uniform flow is modeled over a flat plate. The solution is compared to the commonly known law of the wall. The results show the $k-\omega$ turbulence model having a good approximation of the viscous layer, but not deviating from the expected results in the log layer. Mesh motion is shown using two test cases. The first case oscillates a rigid body to the left and right of the domain. The second cases is a mixing problem, rotating the rigid body about the center to a maximum of five degrees. Both test cases show the implementation of the mesh motion. A fixed NACA0012 airfoil case from a turbulence research database is reproduced and compared to experimental results. An extension of this work is then to use a flexible NACA0012 airfoil to show application of FSI.

In conclusion, this work has accomplished the following:

1. the use of the SUPG method for the fluid formulation to circumvent insta-
bilities caused by high Reynolds number, FEM based approximations,

2. modeling of turbulent flow by implementing the $k - \omega$ turbulence model in the fluid formulation, and

3. discussed a projection method to provide equal order approximations of the fluid formulation.

The combination of the listed items provides enhancement to current literature, but further work of the implemented algorithms is imperative to contribute to high Reynolds number FSI simulations.

6.2 Future Work

Further understanding of the implemented algorithms is necessary for this research to provide valuable results. This section provides insights for future work to enhance the research already completed.

- A partitioned approach was implemented in Chapter 4 and applied in Chapter 5 due to the implementation of the turbulence model. The goal of this work is to provide a framework for a monolithic approach to ultimately use a monolithic approach. Numerical challenges specifically with the turbulence model need to be addressed in order to have a monolithic approach.

- Terms that are considered constant in the segregated approach would no longer be constant in a monolithic approach. This eliminates lagging in the turbulence, but causes instability in the Jacobian and makes the problem less likely to converge with a reasonable timestep. Lund et al. (2003) implements a monolithic approach with an algebraic turbulence model, but approximates the Jacobian for computational costs. More effort into approximation of the Jacobian and the accuracy of the solution may provide an answer to a monolithic turbulence formulation.

- The SUPG and equal order formulations both have parameters dependent on the mesh size, thus dependent on the grid. If a poor grid is used leading to poor mesh size values this effects the SUPG and equal order parameters in
turn effecting the formulations and solutions. Each formulation is effective with a uniform grid which is discussed in Appendices B and C However, having a better understanding to avoid mesh dependency would help future FSI investigations.

- A more resolved grid for the NACA0012 numerical experiment would improve the results and may also improve converge rates, but time was a limiting factor. Extending this case to a flexible airfoil is of interest as well. Self excitations occur naturally, but challenging to computationally resolve (Bai et al., 2012). Inital forced oscillations have been explored for flexible airfoils as numerical experiments in other literature such as Bai et al. (2012) and Bazilevs et al. (2012).

A finite element based algorithm is proposed, providing the layout for a monolithic approach to simulating high Reynolds number FSI. A turbulence model is introduced with verification and validation case studies provided. A significant amount of progress was made, but further investigation should be done.
Residual and Jacobian Terms

Following the work in Sheldon (2012), the identities used are from Gurtin (1982). The deformation gradient, $\mathbf{F}$, and the derivative of the deformation gradient, $\mathbf{F}'$ are:

$$ \mathbf{F} = \text{Grad} \mathbf{d}, \quad (A.1) $$

$$ \mathbf{F}' = \text{Grad} \mathbf{d}_m. \quad (A.2) $$

Other identities include:

$$ \mathbf{F}^{-1'} = -\mathbf{F}^{-1} \mathbf{F}' \mathbf{F}^{-1}, \quad (A.3) $$

$$ \mathbf{F}^{-T'} = \left( \left( \mathbf{F}^{-1} \right)' \right)^T, \quad (A.4) $$

$$ J = \det (\mathbf{F}) \quad (A.5) $$

$$ J' = \text{Jtr} (\mathbf{F}' \mathbf{F}^{-1}). \quad (A.6) $$

Using the above the identities the residuals and Jacobians are found.

The solid displacement, $\mathbf{d}_s$, residual is:

$$ R_{d_s} = \left( \frac{\partial \mathbf{d}_s}{\partial t}, \mathbf{d}_s' \right)_{\Omega_s} + \left( \mathbf{u}_s, \mathbf{d}_s' \right)_{\Omega_s}. \quad (A.7) $$
The Jacobians resulting from the solid displacement, $d_s$, residual equation:

$$J_{d_s,d_s} = \begin{pmatrix} \frac{\partial \hat{d}_j^i}{\partial t} & \hat{d}_i^j \end{pmatrix}_{\Omega_s}, \quad \text{(A.8)}$$

$$J_{d_s,u_s} = \begin{pmatrix} \hat{u}_j^i & \hat{d}_i^j \end{pmatrix}_{\Omega_s}. \quad \text{(A.9)}$$

The solid velocity, $u_s$, residual is:

$$R_{d_s} = \rho_s \begin{pmatrix} \frac{\partial u_s^j}{\partial t} & \hat{u}_j^i \end{pmatrix}_{\Omega_s} + \left( T_R, \text{Grad} \hat{u}_s^i \right)_{\Omega_s}, \quad \text{(A.10)}$$

where $T_R$ is defined in equation (3.41).

The Jacobians resulting from the fluid velocity, $u_f$, residual equation:

$$J_{u_f,d_s} = \left( T_R^j, \hat{u}_s^i \right)_{\Omega_s}, \quad \text{(A.11)}$$

where,

$$T_R^j = \text{Grad} \hat{d}_j^i (\lambda \text{tr} (E) I + 2\mu E) + \left( \text{Grad} \hat{d}_s + I \right) \left( \lambda \text{tr} (E^f) I + 2\mu E^f \right), \quad \text{(A.12)}$$

and,

$$E = \frac{1}{2} \left( \text{Grad} d_s^j + (\text{Grad} d_s^i)^T + (\text{Grad} d_s^j)^T \text{Grad} d_s + (\text{Grad} d_s)^T \text{Grad} d_s \right). \quad \text{(A.13)}$$

$$J_{u_f,u_s} = \rho_s \begin{pmatrix} \frac{\partial \tilde{u}_s^j}{\partial t} & \tilde{u}_s^i \end{pmatrix}_{\Omega_s}. \quad \text{(A.14)}$$

The fluid velocity, $u_f$, residual is:

$$R_u = J \begin{pmatrix} \frac{\partial u_\kappa}{\partial t} + (\text{Grad} u_\kappa F^{-1}) [u_\kappa - u_m] - b_\kappa, \tilde{u}_f^i \end{pmatrix}_{\Omega_f} - J \begin{pmatrix} \rho_\kappa F^{-T}, \text{Grad} \tilde{u}_f^j \end{pmatrix}_{\Omega_f}$$

$$+ J \begin{pmatrix} (\nu + \nu_t) (\text{Grad} u_\kappa F^{-1} + F^{-T} \text{Grad} u_\kappa^T) F^{-T}, \text{Grad} \tilde{u}_f^i \end{pmatrix}_{\Omega_f}. \quad \text{(A.15)}$$

The Jacobians resulting from the fluid velocity, $u_f$, residual equation:
\[ J_{u_f d_f} = J \left( \frac{\partial \tilde{u}_f^i}{\partial t} \right) + J \left( \left[ \text{Grad} \, \tilde{u}_f^j \text{F}^{-1} \right] [u_\kappa - u_m], \tilde{u}_f^i \right)_{\Omega_f} \]
\[ + J \left( \left[ \text{Grad} \, u_\kappa \text{F}^{-1} \right] [\tilde{u}_f^j], \tilde{u}_f^i \right)_{\Omega_f} \]
\[ + J \left( (\nu + \nu_t) \left( \text{Grad} \, \tilde{u}_f^j \text{F}^{-1} + \text{F}^{-T} \text{Grad} \, \tilde{u}_f^j \text{T} \right) \text{F}^{-T}, \text{Grad} \, \tilde{u}_f^i \right)_{\Omega_f}, \quad (A.16) \]

\[ J_{u_f p_f} = -J \left( \hat{p}_f^j \text{F}^{-T}, \text{Grad} \, \tilde{u}_f^i \right)_{\Omega_f} \quad (A.17) \]

\[ J_{u_f d_m} = J' \left( \frac{\partial u_\kappa}{\partial t}, \tilde{u}_f^i \right)_{\Omega_f} - J' \left( \left( \text{Grad} \, u_\kappa \text{F}^{-1} \right) [u_m], \tilde{u}_f^i \right)_{\Omega_f} \]
\[ - J \left( \left( \text{Grad} \, u_\kappa \text{F}^{-1} \right)' [u_m], \tilde{u}_f^i \right)_{\Omega_f} - J \left( \left( \text{Grad} \, u_\kappa \text{F}^{-1} \right)' [u_m], \tilde{u}_f^i \right)_{\Omega_f} \]
\[ + J' \left( \left( \text{Grad} \, u_\kappa \text{F}^{-1} \right)' [u_\kappa], \tilde{u}_f^i \right)_{\Omega_f} + J \left( \left( \text{Grad} \, u_\kappa \text{F}^{-1} \right)' [u_\kappa], \tilde{u}_f^i \right)_{\Omega_f} \]
\[ - J' \left( \hat{p}_\kappa \text{F}^{-T}, \text{Grad} \, \tilde{u}_f^i \right)_{\Omega_f} - J \left( \hat{p}_\kappa \left( \text{F}^{-T} \right)', \text{Grad} \, \tilde{u}_f^i \right)_{\Omega_f} \]
\[ + J' \left( (\nu + \nu_t) \left( \text{Grad} \, u_\kappa \text{F}^{-1} \right) \text{F}^{-T}, \text{Grad} \, \tilde{u}_f^i \right)_{\Omega_f} \]
\[ + J \left( (\nu + \nu_t) \left( \text{Grad} \, u_\kappa \text{F}^{-1} \right) \text{F}^{-T}, \text{Grad} \, \tilde{u}_f^i \right)_{\Omega_f} \]
\[ + J \left( (\nu + \nu_t) \left( \text{F}^{-T} \text{Grad} \, u_\kappa \text{T} \right) \text{F}^{-T}, \text{Grad} \, \tilde{u}_f^i \right)_{\Omega_f} \]
\[ + J \left( (\nu + \nu_t) \left( \text{F}^{-T} \text{Grad} \, u_\kappa \text{T} \right) \text{F}^{-T}, \text{Grad} \, \tilde{u}_f^i \right)_{\Omega_f} \]
\[ + J \left( (\nu + \nu_t) \left( \text{F}^{-T} \text{Grad} \, u_\kappa \text{T} \right) \text{F}^{-T}, \text{Grad} \, \tilde{u}_f^i \right)_{\Omega_f} \right). \quad (A.18) \]

The fluid pressure, \( p_f \), residual is:

\[ R_p = -J \left( \text{F}^{-1} : \text{Grad} \, u_\kappa \text{T}, \hat{p}_f^j \right)_{\Omega_f}. \quad (A.19) \]

The Jacobians resulting from the fluid pressure, \( p_f \), residual equation:

\[ J_{p_f u_f} = -J \left( \text{F}^{-1} : \text{Grad} \, \tilde{u}_f^j \text{T}, \hat{p}_f^j \right)_{\Omega_f}, \quad (A.20) \]
\[
J_{p/d_m} = -J' \left( \mathbf{F}^{-1} : \nabla \mathbf{u}_\kappa^T, \hat{p}_f \right)_\Omega_f - J \left( \left( \mathbf{F}^{-1} \right)' : \nabla \mathbf{u}_\kappa^T, \hat{p}_f \right)_\Omega_f . \tag{A.21}
\]

The fluid turbulent kinetic energy, \( k_f \), residual is:

\[
R_k = J \left( \frac{\partial k_\kappa}{\partial t} - \frac{P_\kappa}{\rho} + \beta^* \omega_\kappa \hat{k}_f, \hat{k}_f \right)_\Omega_f \\
+ J \left( \left( \nabla k_\kappa \mathbf{F}^{-1} \right) \left[ \mathbf{u}_\kappa - \mathbf{u}_m \right] + \left( \mathbf{F}^{-1} : \nabla \mathbf{u}_\kappa \right) \hat{k}_f, \hat{k}_f \right)_\Omega_f \\
+ J \left( (\nu + \sigma_k \nu_t) \left( \nabla k_\kappa \mathbf{F}^{-1} \right) \mathbf{F}^{-T}, \nabla \hat{k}_f \right)_\Omega_f . \tag{A.22}
\]

The Jacobians resulting from the fluid turbulent kinetic energy, \( k_f \), residual equation:

\[
J_{k_f k_f} = J \left( \frac{\partial \hat{k}_f^j}{\partial t} + \beta^* \omega_\kappa \hat{k}_f^j, \hat{k}_f \right)_\Omega_f \\
+ J \left( \left( \nabla \hat{k}_f^j \mathbf{F}^{-1} \right) \left[ \mathbf{u}_\kappa - \mathbf{u}_m \right] + \left( \mathbf{F}^{-1} : \nabla \mathbf{u}_\kappa \right) \hat{k}_f^j, \hat{k}_f \right)_\Omega_f \\
+ J \left( (\nu + \sigma_k \nu_t) \left( \nabla \hat{k}_f^j \mathbf{F}^{-1} \right) \mathbf{F}^{-T}, \nabla \hat{k}_f \right)_\Omega_f , \tag{A.23}
\]

\[
J_{k_f \omega_f} = J \left( \beta^* \omega_f \hat{k}_f, \hat{\omega}_f \right)_\Omega_f . \tag{A.24}
\]

The fluid turbulent specific dissipation rate, \( \omega_f \), residual is:

\[
R_\omega = J \left( \frac{\partial \omega_\kappa}{\partial t} - \frac{\gamma P_\kappa}{\nu_t \rho} + \beta_1 \omega_\kappa \omega_f, \hat{\omega}_f \right)_\Omega_f \\
+ J \left( \left( \nabla \omega_\kappa \mathbf{F}^{-1} \right) \left[ \mathbf{u}_\kappa - \mathbf{u}_m \right] + \left( \mathbf{F}^{-1} : \nabla \mathbf{u}_\kappa \right) \omega_\kappa, \hat{\omega}_f \right)_\Omega_f \\
+ J \left( (\nu + \sigma_\omega \nu_t) \left( \nabla \omega_\kappa \mathbf{F}^{-1} \right) \mathbf{F}^{-T}, \nabla \hat{\omega}_f \right)_\Omega_f \\
- J \left( \frac{\gamma \omega_\kappa}{\omega_\kappa} \left( \nabla k_\kappa \mathbf{F}^{-1} \right) \left( \nabla \omega_\kappa \mathbf{F}^{-1} \right), \hat{\omega}_f \right)_\Omega_f . \tag{A.25}
\]

The Jacobians resulting from the fluid turbulent specific dissipation rate, \( \omega_f \), resid-
ual equation:

\[
J_{\omega f k f} = -J \left( \frac{\sigma \omega^2}{\omega \kappa} \left( \text{Grad} k^j \mathbf{F}^{-1} \right) \left( \text{Grad} \omega_f \mathbf{F}^{-1} \right), \hat{\omega}_j \right)_{\Omega_f}, \tag{A.26}
\]

\[
J_{\omega_f \omega_f} = J \left( \frac{\partial \hat{\omega}_j}{\partial t} + 2\beta_1 \omega \kappa \hat{\omega}_j, \hat{\omega}_j \right)_{\Omega_f}
+ J \left( (\text{Grad} \hat{\omega}_j \mathbf{F}^{-1}) \left[ \mathbf{u}_\kappa - \mathbf{u}_m \right] + (\mathbf{F}^{-1} : \text{Grad} \mathbf{u}_\kappa) \hat{\omega}_j, \hat{\omega}_j \right)_{\Omega_f}
+ J \left( (\nu + \sigma \omega) \left( \text{Grad} \hat{\omega}_j \mathbf{F}^{-1} \right) \mathbf{F}^{-T}, \text{Grad} \hat{\omega}_j \right)_{\Omega_f}
+ J \left( \frac{\sigma \omega^2}{\omega \kappa} \left( \text{Grad} k^j \mathbf{F}^{-1} \right) \left( \text{Grad} \omega_f \mathbf{F}^{-1} \right) \hat{\omega}_j, \hat{\omega}_j \right)_{\Omega_f}
- J \left( \frac{\sigma \omega^2}{\omega \kappa} \left( \text{Grad} k^j \mathbf{F}^{-1} \right) \left( \text{Grad} \hat{\omega}_j \mathbf{F}^{-1} \right), \hat{\omega}_j \right)_{\Omega_f}. \tag{A.27}
\]

The mesh displacement, \( \mathbf{d}_m \), residual is:

\[
R_{\mathbf{d}_m} = \left( \lambda_m \text{Div} \mathbf{d}_m, \text{Div} \hat{\mathbf{d}}_m^i \right)_{\Omega_f}
+ \left( \mu_m \left( \text{Grad} \mathbf{d}_m + \text{Grad} \mathbf{d}_m^T \right), \text{Grad} \hat{\mathbf{d}}_m^i \right)_{\Omega_f}. \tag{A.28}
\]

The Jacobians resulting from the mesh displacement, \( \mathbf{d}_m \), residual equation:

\[
J_{\mathbf{d}_m \mathbf{d}_m} = \left( \lambda_m \text{Div} \hat{\mathbf{d}}_m^i \text{Div} \hat{\mathbf{d}}_m^i \right)_{\Omega_f}
+ \left( \mu_m \left( \text{Grad} \hat{\mathbf{d}}_m^i + \text{Grad} \hat{\mathbf{d}}_m^i \right), \text{Grad} \hat{\mathbf{d}}_m^i \right)_{\Omega_f}. \tag{A.29}
\]
Fluid solutions using finite element formulations often obtain oscillations and instabilities. This may occur because the Babuška-Brezzi condition is not met. This may also occur from advection dominant flow (Olshanskiĭ et al., 2009). Various methods have been developed and discussed throughout the literature addressing one or both of the challenges mentioned and offering countermeasures to overcome them. This work has focused on the SUPG method. However, there are variations of the SUPG method as well. In particular, two SUPG variations are discussed below. Both formulations have been implemented for various test cases and the results are provided.

Recall, $\tilde{u}$ is defined in equation (2.6):

$$\tilde{u} = \hat{u} + f(\tau_{\text{SUPG}}),$$

(B.1)

where $f(\tau_{\text{SUPG}})$ is a function of the SUPG parameter, $\tau_{\text{SUPG}}$, and other terms (Brooks & Hughes, 1982).

The first formulation discussed is used in works such as Brooks & Hughes (1982) and Chaple (2006). The formulation is as follows:

$$f(\tau_{\text{SUPG}}) = \tau_{\text{SUPG}} \nabla \hat{u}[u],$$

(B.2)
resulting in:
\[ \tilde{u} = \hat{u} + \tau_{\text{SUPG}} \nabla \hat{u}[u]. \] (B.3)

The definition for \( \tau_{\text{SUPG}} \) is:
\[ \tau_{\text{SUPG}} = \frac{h_e}{2|u|} \left( \coth Pe_e - \frac{1}{Pe_e} \right) \] (B.4)

where \( h_e \) is the element length and the Peclet number, \( Pe \), is:
\[ Pe_e = \frac{|u|h_e}{2\nu}. \] (B.5)

In general for Peclet numbers greater than one, \( Pe > 1 \), \( \tau_{\text{SUPG}} \) becomes:
\[ \tau_{\text{SUPG}} = \frac{h_e}{2|u|}. \] (B.6)

Note \( \tau \) is a function of space (Brooks & Hughes, 1982; Chaple, 2006).

A second SUPG formulation discussed is proposed by Hughes et al. (1986). This SUPG formulation allows for equal order interpolation of the fluid velocity and pressure. The proposed equal order SUPG formulation is as follows:
\[ f(\tau_{\text{SUPG}}) = \frac{\alpha h^2}{2\mu} \nabla \hat{p}, \] (B.7)

where \( \alpha \) is some constant and \( h = \frac{\Omega_e}{\sqrt{sd}} \). \( \Omega_e \) is the diameter of the element, and \( sd \) is the space dimension. Resulting in:
\[ \tilde{u} = \hat{u} + \frac{\alpha h^2}{2\mu} \nabla \hat{p}. \] (B.8)

### B.1 Brooks and Hughes SUPG Formulation

We want to show the accuracy of the SUPG method, originally proposed by Brooks & Hughes (1982). The problems chosen show the SUPG method is independent of problem parameters as well as the effectiveness of the SUPG method compared to the classical Galerkin method when used with a convection dominated problem. Therefore, appendix B compares 1D and 2D advection-diffusion
problems presented in Chaple (2006). MMS is also used in 1D to provide code verification.

B.1.1 1D Advection-Diffusion Problem

A one-dimensional problem shown in equation (B.9) is used to show the effectiveness of the SUPG method.

\[ U \frac{d\phi}{dx} - k \frac{d^2\phi}{dx^2} = x^2, \]
\[ \Omega = (0, 1), \phi(0) = 0, \phi(1) = 0. \]

Here \( U = 200 \) and constant. The one-dimensional problem is multiplied by a weighting function and the SUPG equation is also added giving the form shown in equation (B.10):

\[ B(\phi, \dot{\phi}) + S(\phi, \dot{\phi}) = F(\dot{\phi}), \]

where

\[ B(\phi, \dot{\phi}) = (U\phi_x, \dot{\phi}) - (k\phi_{xx}, \dot{\phi}), \]
\[ F(\dot{\phi}) = (x^2, \dot{\phi}). \]

The SUPG term is defined by taking the advection operator with respect to the weighting function and multiplying it by the advection-diffusion-reaction operator with respect to, \( \phi \), and integrating over the domain. The term is then multiplied by the SUPG parameter, \( \tau k \), shown in equation (B.12):

\[ S(\phi, \dot{\phi}) = \tau k (U\phi_x - k\phi_{xx} - x^2, U\dot{\phi}_x). \]

Taking equation (B.10) and putting the respective terms from equations (B.11) and (B.12) results in the residual shown in equation:

\[ R^i = (U\phi_x, \dot{\phi}^i) - (k\phi_{xx}, \dot{\phi}^i) + \tau k (U\dot{\phi}^i_x, U\phi_x) - \tau k (U\dot{\phi}^i_x, k\phi_{xx}) \\
- \tau k (U\dot{\phi}^i_x, x^2) - (x^2, \dot{\phi}^i) = 0 \]
\( \phi \) is then discretized as well as its derivatives to find the Jacobian and shown in equation (B.14):

\[
\begin{align*}
\phi & \approx \sum \Phi_j \hat{\phi}_j, \\
\phi_x & \approx \sum \Phi_j \hat{\phi}_{xj}, \\
\phi_{xx} & \approx \sum \Phi_j \hat{\phi}_{xxj}.
\end{align*}
\] (B.14)

From equation (B.13) and (B.14) the Jacobian is determined and shown in equation (B.15):

\[
J^{ij} = \frac{\delta R^i}{\delta \Phi_j} = (U \hat{\phi}_x^i, \hat{\phi}_x^i) - (k \hat{\phi}_{xx}^i, \hat{\phi}_x^i) + \tau^k (U \hat{\phi}_x^i, U \hat{\phi}_x^i) - \tau^k (U \hat{\phi}_x^i, k \hat{\phi}_{xx}^i) \quad (B.15)
\]

It can be shown that the SUPG terms and the weighting function can be combined to create a new weighting function, \( \tilde{w} \), for the residual and the Jacobian shown in equations (B.16) and (B.17) respectively:

\[
R^i = (U \phi_x^i, \tilde{\phi}_x^i) - (k \phi_{xx}^i, \tilde{\phi}_x^i) - (x^2, \tilde{\phi}_x^i) = 0 \quad (B.16)
\]

\[
J^{ij} = (U \hat{\phi}_x^i, \tilde{\phi}_x^i) - (k \hat{\phi}_{xx}^i, \tilde{\phi}_x^i) \quad (B.17)
\]

Where \( \tilde{\phi} \) is shown in equation (B.18):

\[
\tilde{\phi}_x^i = \hat{\phi}_x^i + \tau^k U \hat{\phi}_x^i. \quad (B.18)
\]

**Results and Conclusions**

An exact solution was obtained by taking equation (B.9) and solving for the homogeneous solution and the particular solution. The homogeneous solution is shown in equation (B.19):

\[
\phi_h = c_1 e^{\frac{Ux}{k}} + c_2. \quad (B.19)
\]

While the particular solution was solved by the method of undetermined coefficients and shown in equation (B.20):

\[
\phi_p = \frac{2k^2x}{U^3} + \frac{kx^2}{U^2} + \frac{x^3}{3U}. \quad (B.20)
\]
This results in equation (B.21):

\[ \phi = \phi_p + \phi_h, \]

\[ \phi_p = \frac{2k^2 x}{U^3} + \frac{kx^2}{U^2} + \frac{x^3}{3U} + c_1 e^\frac{Ux}{k} + c_2. \]

Using the boundary conditions from (B.9) constants \( c_1 \) and \( c_2 \) are found and shown in equation (B.22):

\[ c_1 = -\frac{2k^2}{U^3} + \frac{k}{U^2} + \frac{1}{3U}, \]

\[ c_2 = -c_1. \]

The exact solution determined above matches the numerical solution produced in deal.ii. Figure B.1 shows the exact solution plotted with the numerical solution for \( Pe = 10^3 \) for the top plot, \( Pe = 10^5 \) for the middle plot, and \( Pe = 10^{10} \) for the bottom plot.
Figure B.1: A plot of the 1D advection/diffusion SUPG formulation versus the exact solution for $Pe = 10^3$ for the top plot, $Pe = 10^5$ for the middle plot, and $Pe = 10^{10}$ for the bottom plot. The numerical solution agrees with the exact solution and is independent of the Peclet number used.
The results are independent of the Peclet number agreeing with the results given by Chaple (2006) shown in Figure B.2. Again \( Pe = 10^3 \) for the top plot, \( Pe = 10^5 \) for the middle plot, and \( Pe = 10^{10} \) for the bottom plot.

Figure B.2: A plot from (Chaple, 2006) of the 1D advection/diffusion SUPG formulation versus the exact solution for \( Pe = 10^3 \) for the top plot, \( Pe = 10^5 \) for the middle plot, and \( Pe = 10^{10} \) for the bottom plot. The results from Figure B.1 agree with the findings from (Chaple, 2006).
It should be noted $e^{\frac{U}{k}}$ was so large it was producing an infinite result when coded. This also lead to the constants $c_1$ and $c_2$ to be approximately zero and only the particular solution to be coded.

To provide further verification of the added SUPG terms the method of manufactured solutions (MMS) was also used. MMS uses an arbitrary solution for verification of mathematical accuracy ignoring any physics of the problem (Roache, 2002). Here $\phi$ is defined in equation (B.23):

$$\phi^* = \sin(4\pi x)^2. \quad (B.23)$$

This produces a right hand side (RHS) shown in equation (B.24):

$$RHS = 8\pi U \cos(4\pi x) \sin(4\pi x) - 32k\pi^2(\cos(4\pi x)^2 - \sin(4\pi x)^2). \quad (B.24)$$

Taking the right hand side, equation (B.24), and the boundary conditions used in equation (B.9), a solution is computed and compared to the exact solution by finding the error, calculated using the $L_2$ norm, $H^1$ semi-norm, and $L_\infty$ norm. The norms and convergence rates were calculated using the added SUPG terms as well as without them. The results are shown in Table B.1 and Table B.2 respectively:

**Table B.1.** Refinement table of a 1D advection/diffusion problem using SUPG stabilization. The $L_2$ norm and $L_\infty$ norm have lower convergence rates with SUPG stabilization than without SUPG stabilization.

<table>
<thead>
<tr>
<th>cycle</th>
<th># cells</th>
<th># dofs</th>
<th>$L_2$ - error</th>
<th>$H^1$ - error</th>
<th>$L_\infty$ - error</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>128</td>
<td>257</td>
<td>3.507e-02</td>
<td>-</td>
<td>8.710e-01</td>
</tr>
<tr>
<td>1</td>
<td>256</td>
<td>513</td>
<td>1.745e-02</td>
<td>1.01</td>
<td>4.371e-01</td>
</tr>
<tr>
<td>2</td>
<td>512</td>
<td>1025</td>
<td>8.713e-03</td>
<td>1.00</td>
<td>2.188e-01</td>
</tr>
<tr>
<td>3</td>
<td>1024</td>
<td>2049</td>
<td>4.354e-03</td>
<td>1.00</td>
<td>1.095e-01</td>
</tr>
<tr>
<td>4</td>
<td>2048</td>
<td>4097</td>
<td>2.177e-03</td>
<td>1.00</td>
<td>5.480e-02</td>
</tr>
</tbody>
</table>
Table B.2. Refinement table of a 1D advection/diffusion problem without the use of SUPG stabilization. The $L_2$ norm and $L_\infty$ norm have higher convergence rates without SUPG stabilization than with SUPG stabilization.

<table>
<thead>
<tr>
<th>cycle</th>
<th># cells</th>
<th># dofs</th>
<th>$L^2$ − error</th>
<th>$H^1$ − error</th>
<th>$L_\infty$ − error</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>128</td>
<td>257</td>
<td>1.476e-03</td>
<td>-</td>
<td>2.140e-03</td>
</tr>
<tr>
<td>1</td>
<td>256</td>
<td>513</td>
<td>3.692e-04</td>
<td>2.00</td>
<td>5.387e-04</td>
</tr>
<tr>
<td>2</td>
<td>512</td>
<td>1025</td>
<td>9.250e-05</td>
<td>2.00</td>
<td>1.378e-04</td>
</tr>
<tr>
<td>3</td>
<td>1024</td>
<td>2049</td>
<td>2.332e-05</td>
<td>1.99</td>
<td>3.652e-05</td>
</tr>
<tr>
<td>4</td>
<td>2048</td>
<td>4097</td>
<td>6.020e-06</td>
<td>1.95</td>
<td>1.030e-05</td>
</tr>
</tbody>
</table>

Table B.1 shows the convergence rates for each norm to be of the same order, $O(1)$. Table B.2 shows the $L^2$ and $L_\infty$ error to be approximately degree $+1$, $O(2)$, while the $H^1$ error is still $O(1)$. This is expected when the SUPG terms are not added to the weighting function.

B.1.2 2D Steady Advection-Diffusion Problem

A problem showing the effectiveness the SUPG method in two dimensions is also considered. Equation (B.25) shows the problem considered.

\[-\nabla \cdot (k \nabla \phi) + \mathbf{u} \cdot \nabla \phi = 0,\]
\[\Omega = (0,1) \times (0,1).\]  

(B.25)

Here $U = (1,3)$ and constant. While the boundary conditions are shown in equation

\[\phi(0, y) = 1,\]
\[\phi(1, y) = 0,\]
\[\phi(x, 1) = 0,\]

(B.26)
\[\phi(0, y) = 1,\]
\[\Omega = (0,1) \times (0,1).\]
Multiplying by the weighting function and integrating over the domain yields the equation (B.27):

\[ B(\phi, w^i) = (u \cdot \phi_x, w^i) - (k\phi_{xx}, w^i). \]  

(B.27)

Following the same equation form as for the 1D case shown in equation (B.10) and the SUPG term is the same for the 2D case resulting in equation (B.28):

\[ S(\phi, w^i) = \tau^k(u \cdot w^i_x, u \cdot \phi_x - k\phi_{xx}). \]  

(B.28)

The residual is shown in equation (B.29) by using equations (B.27) and (B.28):

\[ R^i = (u \cdot \phi_x, w^i) - (k\phi_{xx}, w^i) + \tau^k(u \cdot w^i_x, u \cdot \phi_x) - \tau^k(u \cdot w^i_x, k\phi_{xx}) = 0. \]  

(B.29)

Again, as in the 1D case a new weighting function can be used, \( \tilde{w} \), shown in equation (B.18). The residual is then written in terms of \( \tilde{w} \) and shown in equation (B.30):

\[ R^i = (u \cdot \phi_x, \tilde{w}^i) - (k\phi_{xx}, \tilde{w}^i) = 0. \]  

(B.30)

The Jacobian is determined and shown in equation (B.31):

\[ J^{ij} = (u \cdot w^j_x, \tilde{w}^i) - (k w^j_{xx}, \tilde{w}^i). \]  

(B.31)

**Results and Conclusions**

The numerical solutions obtained without the SUPG method are compared to the solutions obtained by Chaple (2006) and to the solutions using the SUPG method. Figure B.3 shows the numerical solution produced for \( Pe = 7.90575 \) without the use of SUPG, with a grid size of 16384 elements. The numerical solution does not match the results given by Chaple (2006) shown in Figure B.4 because of instabilities occurring in the solution.
Figure B.3: A plot of the 2D advection diffusion solution with $Pe = 7.9057$ without the SUPG stabilization. The numerical solution does not match the results given by Chaple (2006) because of instabilities occurring in the solution.

Figure B.4: A plot of the 2D advection diffusion solution with $Pe = 7.9057$ without the SUPG stabilization provided by Chaple (2006) as a visual comparison.

This does not mean the results are incorrect but that the Galerkin method is not
very accurate in convection dominant cases. It should be noted to produce better results higher order elements and quadrature formula were used in the numerical solution for both numerical solutions. Figure B.5 shows the solution with \( Pe = 7.9057 \) when using the SUPG method.

![Figure B.5: A plot of the 2D advection/diffusion case with \( Pe = 7.9057 \) using the SUPG method. The numerical solution is smoother and a desirable result is obtained.](image)

The results shown in Figure B.5 agree with the results provided by Chaple (2006) shown in Figure B.6, where 1600 elements were used.
A higher Peclet number, $Pe = 7.9057e^5$, was simulated for the same problem. Results with and without SUPG stabilization are compared with each other and to Chaple (2006). Figure B.7 shows the numerical solution produced for $Pe = 7.90575e^5$. Again the numerical solution does not match the results given by Chaple (2006) shown in Figure B.8. This shows the instabilities of the Galerkin method especially at large Peclet numbers. It did not make sense to give a 3D representation of Figure B.7 due to the variance in $\phi$. It was better depicted with a 2D plot.
Figure B.7: A plot of the 2D advection diffusion solution with $Pe = 7.9057e^5$ without the SUPG stabilization. The numerical solution does not match the results given by Chaple (2006) because of several instabilities occurring in the solution. The plot shows instabilities occurring in an advection/diffusion solution when using the Galerking method especially at large Peclet numbers. The plot was better depicted in 2D due to the variance in $\phi$. 
Figure B.8: A plot of the 2D advection diffusion solution with $Pe = 7.9057e^5$ without the SUPG stabilization provided by Chaple (2006). The numerical solution is very oscillatory and does not resemble a desired solution.

Figure B.9: A plot of the 2D advection diffusion solution with $Pe = 7.9057e^5$ with the SUPG stabilization. The numerical solution does provide a desirable solution, but instabilities still exist in the crosswind diffusion of the SUPG method because the SUPG method only stabilizes in the streamline direction (Chaple, 2006).
Figure B.9 shows instabilities in the crosswind diffusion of the SUPG method. This is because the SUPG method only stabilizes in the streamline direction (Chaple, 2006). It is more noticeable at larger Peclet numbers. The results match the results given by Chaple (2006) shown in Figure B.10 again 1600 elements were used.

Figure B.10: A plot of the 2D advection diffusion solution with $Pe = 7.9057e^5$ with the SUPG stabilization provided by Chaple (2006) are a visual comparison to Figure B.9 with agreeable results.

## B.2 Equal Order SUPG Formulation

### B.2.1 Stokes Flow

A driven cavity flow test case was produced using steady state stokes flow with a weak formulation as follows:

$$
R = \left( -\text{div} \ (\bar{\mu}I) + \text{div} \ \left( (\nu + \nu_t) \left( \text{grad} \ u + (\text{grad} \ u)^T \right) \right) + \mathbf{b}_f, \tilde{u} \right) - \left( \text{div} \ u, \hat{p} \right) = 0,
$$

yielding:

$$
R_u = - (p, \text{div} \ \hat{u}) + 2\mu \left( \left( \text{grad} \ u + (\text{grad} \ u)^T \right), \text{grad} \ \hat{u} \right),
$$

(B.33)
and
\[ R_p = \left( \text{grad} \rho, \frac{\alpha h^2}{2\mu} \text{grad} \tilde{p} \right) - \left( \text{div} u, \tilde{p} \right). \] (B.34)

**Results and Conclusions**

Results for a 24x24 mesh with Q1/Q1 elements and \( \alpha = 0.5 \) were provided by Hughes et al. (1986). The boundary conditions provided by Hughes et al. (1986) are shown in Figure B.11. The problem was reproduced with the boundary conditions shown in Figure B.12, but certain fluid parameters were not defined in Hughes et al. (1986). Therefore, they were assumed, \( U_x = 1.0, \ Re = 1,000, \) and \( \rho = 1.2 \frac{kg}{m^3}. \) The results are provided for a visual comparison in Figures ?? - ??.

![Figure B.11](image_url)

Figure B.11: A figure provided by Hughes et al. (1986) of the boundary conditions used for a driven cavity flow problem to test an equal order SUPG fluid formulation.
Figure B.12: An illustration of the boundary conditions for a driven cavity flow problem used to duplicate the results provided by Hughes et al. (1986).

The pressure elevation was provided by Hughes et al. (1986) and plotted with the reproduced results shown in Figure B.13.
Figure B.13: Plots of the pressure elevation are shown for a driven cavity flow problem. The reproduced solution (b) is compared to the solution provided by Hughes et al. (1986) (a) as a visual comparison with agreeable results.

The pressure contour was also provided by Hughes et al. (1986) and plotted with the reproduced results shown in Figure B.14
Figure B.14: Plots of the pressure contour are shown for a driven cavity flow problem. The reproduced solution (b) is compared to the solution provided by Hughes et al. (1986) (a) as a visual comparison with agreeable results.

A velocity vector plot was also provided by Hughes et al. (1986) and shown with the reproduced results in Figure B.15.
Figure B.15: Plots of velocity vectors are shown for a driven cavity flow problem. The reproduced solution (b) is compared to the solution provided by Hughes et al. (1986) (a) as a visual comparison with agreeable results.

**B.2.2 Extension to the Navier-Stokes Equations**

Applying the equal order SUPG formulation to the Navier-Stokes equations results in the following weak formulation:

\[
R_u = \left( \frac{\partial \mathbf{u}}{\partial t} + \text{grad} \mathbf{u} \mathbf{[u]}, \mathbf{\hat{u}} \right) - (p, \text{div} \mathbf{\hat{u}}) + 2\mu \left( \left( \text{grad} \mathbf{u} + (\text{grad} \mathbf{u})^T \right) , \text{grad} \mathbf{\hat{u}} \right),
\]

(B.35)

and

\[
R_p = \left( \frac{\partial \mathbf{u}}{\partial t} + \text{grad} \mathbf{u} \mathbf{[u]} , \frac{\alpha h^2}{2\mu} \text{grad} \hat{p} \right) - \left( \text{grad} p , \frac{\alpha h^2}{2\mu} \text{grad} \hat{p} \right) - (\text{div} \mathbf{u}, \hat{p}).
\]

(B.36)

The formulation was compared to another stabilization technique produced by Hachem et al. (2010). The test problem is a cavity driven flow problem with the boundary conditions shown in Figure B.16.
Figure B.16: An illustration of the boundary conditions for a driven cavity flow problem used to compare results to Hachem et al. (2010).

An inlet velocity of $U_x = 1$ is used, $Re = 1,000$, and $\rho$ is assumed to be $1.2 \frac{kg}{m^3}$.

**Results and Conclusions**

The parameter $\alpha$ was investigated using different values mentioned by Hughes et al. (1986). The x velocity, $u_x$ was plotted versus the y distance at $x = 0.5m$. Similarly the y velocity, $u_y$ was plotted versus the x distance at $y = 0.5m$. This was done to compare results to Hachem et al. (2010) shown in Figures B.17 and B.18.
Figure B.17: The x velocity is plotted versus the y distance at $x = 0.5m$. The results are compared to Hachem et al. (2010) and are comparable when $\alpha = 0.001$.

Figure B.18: The y velocity is plotted versus the x distance at $y = 0.5m$. The results are compared to Hachem et al. (2010) and are comparable when $\alpha = 0.001$.

Figures B.17 and B.18 show $\alpha = 0.001$ produce the most comparable results to Hachem et al. (2010). According to Hughes et al. (1986), all values of $\alpha$ chosen produced reasonable results, but oscillations in the pressure were reported for $\alpha = 0.001$ (Hughes et al., 1986).
Figures B.19 and B.20 are a comparison of no stabilization, the SUPG method proposed by Brooks & Hughes (1982), the equal order SUPG method by Hughes et al. (1986), and the reference data from Hachem et al. (2010) from the same cavity driven flow problem.

Figure B.19: A comparison of no stabilization, the SUPG method proposed by Brooks & Hughes (1982), the equal order SUPG method by Hughes et al. (1986) when $\alpha = 0.001$, and the reference data from Hachem et al. (2010) for cavity driven flow at $Re = 1,000$. Plotting $u_x$ vs. $y$ distance at $x = 0.5m$. 
Figure B.20: A comparison of no stabilization, the SUPG method proposed by Brooks & Hughes (1982), the equal order SUPG method by Hughes et al. (1986) when $\alpha = 0.001$, and the reference data from Hachem et al. (2010) for cavity driven flow at $Re = 1,000$. Plotting $u_y$ vs. $x$ distance at $y = 0.5m$.

Figures B.19 and B.20 show the SUPG method proposed by Brooks & Hughes (1982) over diffuses the solution. The equal order SUPG method with $\alpha = 0.001$ proposed by Hughes et al. (1986) is comparable if not better than the no SUPG results. Based on the results from Figures B.17 and B.18 it is possible the results could improve with further investigation of the $\alpha$ parameter.

Since reasonable results were produced at $Re = 1,000$ with $\alpha = 0.001$, the same problem was simulated with $Re = 5,000$ varying $\alpha$. The solution diverged when no stabilization was used and when $\alpha = 0.001$, and $\alpha = 0.1$. This could be due to the oscillations occurring in the pressure solution. A solution was obtainable for $\alpha = 0.5$, and $\alpha = 1.0$ and shown in Figures B.21 and B.22.
Figure B.21: A comparison of the Hughes et al. (1986) stabilization varying the parameter $\alpha$ at $Re = 5,000$ for cavity driven flow problem where $u_x$ is plotted versus $y$ distance at $x = 0.5m$.

Figure B.22: A comparison of the Hughes et al. (1986) stabilization varying the parameter $\alpha$ at $Re = 5,000$ for cavity driven flow problem where $u_y$ is plotted versus $x$ distance at $y = 0.5m$.

When there is a large amount of SUPG added to the formulation the solution becomes stable but also very diffused as shown in Figures B.21 and B.22. More
investigation into the parameter $\alpha$ and the equal order SUPG formulation should be done.
Monolithic Projection Scheme

Further investigation is needed on the projection method. The current progress on the projection method is discussed in this appendix.

If the current fully discretized scheme would be solved using the Newton-Raphson method, Q2/Q1 elements would be needed for the velocity and pressure respectively to satisfy the Babuška-Brezzi condition. To ensure the velocity and pressure are solved with the same order of accuracy, an equal order scheme is implemented. The introduction of the pressure gradient onto the velocity by Codina & Blasco (2000) creates an equal order solution, but it also creates a third unknown. The method stabilizes the pressure to accommodate equal order interpolation and can be applied to a monolithic approach as well as a fractional step time integration method (Codina & Blasco, 2000).

The projection method adds the following terms to continuity and a third equation. In the Eulerian reference frame they are:

$$\alpha_k^2 (\nabla \tilde{p}, \nabla \tilde{p}) - \alpha_k \left( \xi^{n+\beta} \nabla \hat{p} \right) + (\nabla \mathbf{u}, \hat{p}) = 0, \quad (C.1)$$

$$-\alpha_k \left( \nabla \tilde{p}, \xi \right) + \left( \xi, \xi \right) = 0, \quad (C.2)$$

where $\xi$ is the third unknown, the gradient of the pressure, $\beta$ is either 0 or 1 to uncouple or couple equations (C.1) and (C.2) respectively, and $\alpha_k$ is a stabilization parameter given by:

$$\alpha_k = \left( \frac{4\nu}{\alpha_0 h_k^2} + \frac{2\nu_k}{h_k} \right)^{-\frac{1}{2}}, \quad (C.3)$$
where \( \nu \) is the kinematic viscosity, \( h_k \) is the element size, \( \alpha_0 = 1.0 \) is a given constant, and \( v_k \) is a representative velocity of that element at that current time step.

Transforming the above equations into the ALE reference frame gives:

\[
\alpha_k^2 \left( \text{Grad} \tilde{p}_k \mathbf{F}^{-1}, \text{Grad} \tilde{p} \mathbf{F}^{-1} \right) - \alpha_k \left( \xi_{\kappa}^{n+\beta}, \text{Grad} \tilde{p} \mathbf{F}^{-1} \right) + (\mathbf{F}^{-1} : \text{Grad} \mathbf{u}_\kappa^T, \tilde{p}) = 0, \quad \text{(C.4)}
\]

\[
-\alpha_k \left( \text{Grad} \tilde{p}_k \mathbf{F}^{-1}, \dot{\xi} \right) + \left( \xi_{\kappa}, \dot{\xi} \right) = 0. \quad \text{(C.5)}
\]

This provides an equal order projection method to add to the discretized Navier-Stokes equations. Now the fluid formulation can be discussed in detail and the residual and Jacobian terms are found:

\[
\begin{bmatrix}
J_{uu} & J_{up} & J_{u\xi} & J_{uk} & J_{uw} \\
J_{pu} & J_{pp} & J_{p\xi} & J_{pk} & J_{pw} \\
J_{\xi u} & J_{\xi p} & J_{\xi \xi} & J_{\xi k} & J_{\xi w} \\
J_{ku} & J_{kp} & J_{k\xi} & J_{kk} & J_{kw} \\
J_{\omega u} & J_{\omega p} & J_{\omega \xi} & J_{\omega k} & J_{\omega w}
\end{bmatrix}
\begin{bmatrix}
\Delta \mathbf{u} \\
\Delta p \\
\Delta \xi \\
\Delta k \\
\Delta \omega
\end{bmatrix}
= -
\begin{bmatrix}
\mathbf{R}_u \\
\mathbf{R}_p \\
\mathbf{R}_\xi \\
\mathbf{R}_k \\
\mathbf{R}_\omega
\end{bmatrix}, \quad \text{(C.6)}
\]

where all residual and Jacobian terms are shown in Appendix A.

### C.1 Verification via MMS

MMS uses an arbitrary solution for verification of mathematical accuracy and ignores any physics related to the problem (Roache, 2002). The Navier-Stokes equations with the equal order projection terms, in the Eulerian reference frame, are verified with MMS. Similar to Chapter 4:

1. A non-trivial, continuous solution is produced by solving a PDE:

\[
u^* = \begin{pmatrix} \sin(4\pi x) \sin(4\pi y) \sin(4\pi t) \\ \cos(4\pi x) \cos(4\pi y) \sin(4\pi t) \end{pmatrix}. \quad \text{(C.7)}
\]
Here continuity is not solved for because of the added projection terms to the continuity equation.

The pressure was chosen to be:

\[ p^* = \sin(4\pi x) \sin(4\pi y) \sin(4\pi t). \] \hspace{1cm} (C.8)

We have the third unknown, as well, set as an independent solution:

\[ \xi^* = \begin{pmatrix} 4\pi \sin(4\pi x) \sin(4\pi y) \sin(4\pi t) \\ 4\pi \sin(4\pi x) \sin(4\pi y) \sin(4\pi t) \end{pmatrix}, \] \hspace{1cm} (C.9)

and let:

\[ \phi^* = \begin{pmatrix} u^* \\ p^* \\ \xi^* \end{pmatrix}. \] \hspace{1cm} (C.10)

2. A source term is computed from the manufactured solutions of \( u^* \) and \( p^* \):

\[ R_1 = 4\pi \cos(4\pi x) \sin(4\pi t) \sin(4\pi y) + 4\pi \cos(4\pi t) \sin(4\pi x) \sin(4\pi y) + 32\pi^2 \sin(4\pi t) \sin(4\pi x) \sin(4\pi y) + 4\pi \cos(4\pi x) \sin(4\pi x) \cos(4\pi y) \sin(4\pi y)^2 \sin(4\pi t)^2 + 4\pi \cos(4\pi x) \sin(4\pi x) \sin(4\pi y)^2 \sin(4\pi t)^2 \] \hspace{1cm} (C.11)

\[ R_2 = 4\pi \cos(4\pi t) \cos(4\pi x) \cos(4\pi y) + 4\pi \cos(4\pi y) \sin(4\pi t) \sin(4\pi x) + 32\pi^2 \cos(4\pi x) \cos(4\pi y) \sin(4\pi t) - 4\pi \cos(4\pi x)^2 \cos(4\pi y) \sin(4\pi y) \sin(4\pi t)^2 - 4\pi \sin(4\pi x)^2 \cos(4\pi y) \sin(4\pi y) \sin(4\pi t)^2 \] \hspace{1cm} (C.12)

\[ R_3 = 4\pi \cos(4\pi y) \sin(4\pi x) \sin(4\pi t) + 4\pi \cos(4\pi x) \sin(4\pi y) \sin(4\pi t) \]
\[ + 32\pi^2 \sin(4\pi t) \sin(4\pi x) \sin(4\pi y) \] (C.13)

\[ \mathbf{R}_4 = 4\pi \cos(4\pi x) \sin(4\pi y) \sin(4\pi t) - \sin(4\pi x) \sin(4\pi y) \sin(4\pi t) \] (C.14)

\[ \mathbf{R}_5 = 4\pi \sin(4\pi x) \cos(4\pi y) \sin(4\pi t) - \sin(4\pi x) \sin(4\pi y) \sin(4\pi t) \] (C.15)

3. Evaluate by computing a solution, \( \phi_h \), using the source term. The domain is a 1x1 unit with \( \nu = 1.0 \) and \( \alpha = 1.0 \). Dirichlet boundary conditions are used, meaning the exact solution governs the boundaries of the domain.

4. Compute the error using grid refinement. The following norms are utilized for convergence rate studies:

   The error of \( \phi_h \) in the \( L_2 \) norm:
   \[
   \text{error}_{L_2} = \left( \int_{\Omega} |\phi^* - \phi_h|^2 \, dx \right)^{\frac{1}{2}}
   \] (C.16)

   The error of \( \phi_h \) in the \( H^1 \) semi-norm:
   \[
   \text{error}_{H_1} = \left( \int_{\Omega} \left| \frac{d\phi^*}{dx} - \frac{d\phi_h}{dx} \right|^2 \, dx \right)^{\frac{1}{2}}
   \] (C.17)

   The error of \( \phi_h \) in the \( L_{\infty} \) norm:
   \[
   \text{error}_{L_{\infty}} = \max_{x \in \Omega} |\phi^* - \phi_h|
   \] (C.18)

5. Using the described norms the computed solution is compared to the manufactured solution for each component of the fluid velocity and the pressure.
using the following equation:

\[ m = \frac{\text{error}}{\ln \frac{1}{h}}, \quad (C.19) \]

where \( m \) is the slope of the line and \( h \) is the cell width of the current grid refinement.

Figure C.1 shows the calculated norms using equation C.19 for the X component of the fluid velocity and Table C.1 shows the convergence rates.

Figure C.1: A plot of the calculated norms for the X component of the fluid velocity. The X component of the fluid velocity does not converge as well as expected.
Table C.1. Refinement table of the fluid velocity X component. The X component of the fluid velocity does not converge as well as expected.

<table>
<thead>
<tr>
<th>cycle</th>
<th># cells</th>
<th># dofs</th>
<th>L2-error</th>
<th>H1-error</th>
<th>Linfty-error</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>64</td>
<td>405</td>
<td>4.349e-02</td>
<td>-</td>
<td>1.703e+00</td>
</tr>
<tr>
<td>1</td>
<td>256</td>
<td>1445</td>
<td>8.128e-03</td>
<td>2.42</td>
<td>4.551e-01</td>
</tr>
<tr>
<td>2</td>
<td>1024</td>
<td>5445</td>
<td>2.944e-03</td>
<td>1.47</td>
<td>1.233e-01</td>
</tr>
<tr>
<td>3</td>
<td>4096</td>
<td>21125</td>
<td>1.157e-03</td>
<td>1.35</td>
<td>3.521e-02</td>
</tr>
</tbody>
</table>

Figure C.2 shows the computed solution \( \phi_h \) for the X velocity component.

Figure C.2: A plot of the computed solution \( \phi_h \) for the X velocity component using MMS.

Figure C.3 shows the calculated norms for the Y component of the fluid velocity and Table C.2 shows the convergence rates.

Figure C.3 shows the calculated norms for the Y component of the fluid velocity and Table C.2 shows the convergence rates.
Figure C.3: A plot of the calculated norms for the Y component of the fluid velocity. The Y component of the fluid velocity does not converge as well as expected.

Table C.2. Refinement table of the fluid velocity Y component. The Y component of the fluid velocity does not converge as well as expected.

<table>
<thead>
<tr>
<th>cycle</th>
<th># cells</th>
<th># dofs</th>
<th>L2-error</th>
<th>H1-error</th>
<th>Linfinity-error</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>64</td>
<td>405</td>
<td>5.942e-02</td>
<td>-</td>
<td>1.748e+00</td>
</tr>
<tr>
<td>1</td>
<td>256</td>
<td>1445</td>
<td>8.978e-03</td>
<td>2.73</td>
<td>4.621e-01</td>
</tr>
<tr>
<td>2</td>
<td>1024</td>
<td>5445</td>
<td>2.933e-03</td>
<td>1.61</td>
<td>1.261e-01</td>
</tr>
<tr>
<td>3</td>
<td>4096</td>
<td>21125</td>
<td>1.159e-03</td>
<td>1.34</td>
<td>3.659e-02</td>
</tr>
</tbody>
</table>

Figure C.4 shows the computed solution $\phi_h$ for the Y velocity component.
Figure C.4: A plot of the computed solution $\phi_h$ for the Y velocity component using MMS.

Figure C.5 shows the calculated norms for the pressure and Table C.3 shows the convergence rates.
Figure C.5: A plot of the calculated norms for the pressure component of the fluid formulation. The pressure component converges as expected.

Table C.3. Refinement table of the pressure component, converging at the expected rate for 1st order.

<table>
<thead>
<tr>
<th>cycle</th>
<th># cells</th>
<th># dofs</th>
<th>L2-error</th>
<th>H1-error</th>
<th>Linfty-error</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>64</td>
<td>405</td>
<td>3.899e-01</td>
<td>-</td>
<td>7.068e+00</td>
</tr>
<tr>
<td>1</td>
<td>256</td>
<td>1445</td>
<td>2.145e-01</td>
<td>0.86</td>
<td>3.835e+00</td>
</tr>
<tr>
<td>2</td>
<td>1024</td>
<td>5445</td>
<td>1.100e-01</td>
<td>0.96</td>
<td>1.957e+00</td>
</tr>
<tr>
<td>3</td>
<td>4096</td>
<td>21125</td>
<td>5.535e-02</td>
<td>0.99</td>
<td>9.840e-01</td>
</tr>
</tbody>
</table>

Figure C.6 shows the computed solution $\phi_h$ for the pressure component.
Figure C.6: A plot of the computed solution $\phi_h$ for the pressure component of the fluid formulation using MMS.

Figure C.7 shows the calculated norms for the x component of the pressure gradient and Table C.4 shows the convergence rates. The x component and y component terms were equivalent so only the x component terms will be shown.
Figure C.7: A plot of the calculated norms for the pressure gradient X component of the fluid formulation. The pressure gradient X component converges as expected.

**Table C.4.** Refinement table for the X component of the pressure gradient. The $L_2$ norm and $L_\infty$ norm converge as expected for 1st order.

<table>
<thead>
<tr>
<th>cycle</th>
<th># cells</th>
<th># dofs</th>
<th>L2-error</th>
<th>H1-error</th>
<th>Linfty-error</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>64</td>
<td>405</td>
<td>1.96e+00</td>
<td>-</td>
<td>4.085e+01</td>
</tr>
<tr>
<td>1</td>
<td>256</td>
<td>1445</td>
<td>1.266e+00</td>
<td>0.63</td>
<td>2.686e+01</td>
</tr>
<tr>
<td>2</td>
<td>1024</td>
<td>5445</td>
<td>6.77e-01</td>
<td>0.90</td>
<td>1.586e+01</td>
</tr>
<tr>
<td>3</td>
<td>4096</td>
<td>21125</td>
<td>3.458e-01</td>
<td>0.97</td>
<td>9.571e+00</td>
</tr>
</tbody>
</table>

Figure C.8 shows the computed solution $\phi_h$ for the X pressure gradient component.
The Taylor vortex problem is an analytical solution to the unsteady, incompressible Navier-Stokes equations in 2D. The solution is shown in equations (C.20)-(C.22):

\[
\begin{align*}
    u(x, y, t) &= -\cos(\pi x) \sin(\pi y)e^{-2\nu\pi^2 t}, \\
v(x, y, t) &= \sin(\pi x) \cos(\pi y)e^{-2\nu\pi^2 t}, \\
p(x, y, t) &= \frac{1}{4}(\cos(2\pi x) + \cos(2\pi y))e^{-4\nu\pi^2 t}.
\end{align*}
\]

The Taylor vortex problem is used a numerical test in Codina & Blasco (2000). The domain is \( \Omega = [0, 1] \times [0, 1] \) with a 31 x 31 nodal mesh. Dirichlet-type boundary conditions are imposed by the analytical solution of equations (C.20)-(C.22). The Navier-Stokes equations are numerically solved using equations (C.20)-(C.22) from \( t = 0 \) to \( t = 1 \) varying the time step size. Initially \( \theta = 1 \) and \( \beta = 0 \). The authors chose \( \nu = 0.1 \) as well. The \( L^2 \)-norm of the velocity is calculated. The results were reproduced and formulated in Deal.ii but with a 32 x 32 nodal mesh instead.
The base 10 log of the time step is plotted versus the base 10 log of the $L^2$-norm error. Both are shown in Figure C.9 using Q1 and Q2 elements. The values are not equivalent but the convergence rates, shown by the slopes, are equivalent at an order of 1.

C.3 Comparison of Projection Parameter

The projection method is applied to a cavity flow test case. For this problem the 2D domain is $[0,0]\times[1,1]$. With all velocity components = 0 except the x-velocity, $u_x = 1$ along the top of the domain and $\nabla p \cdot \hat{n} = 0$ as well. $\theta = 0$, $\delta t = 0.2h_k$, and $\beta = 0$. The stabilization parameter, $\alpha_k = \alpha = 1$ for one case, $\alpha_k = \alpha = 0.5$ for a second case and for a final case $\alpha_k$ is shown in equation (C.23):

$$\alpha_k = \left(\frac{4\nu}{\alpha_0 h_k^2} + \frac{2v_k}{h_k}\right)^{-\frac{1}{2}}.$$ 

(C.23)
\( \nu = 0.01, h_k \) is the element size, \( \alpha_0 = 1.0 \) is a given constant, and \( v_k \) is a representative velocity of that element at that current time step and set equal to \( \nu \). The last case is the only case where \( \alpha_k \) is dependent on the mesh. In Codina & Blasco (2000) \( \alpha_0 = 1/3 \) for bilinear elements. Results compare the x-velocity, y-velocity, and the pressure to the same problem produced using open foam and a multi-order, Galerkin Navier-Stokes code developed in deal.ii and are shown in Figure C.10.
Figure C.10: A plot comparing values of the projection parameter $\alpha$ for a driven cavity flow problem for the X and Y velocity components and pressure. When $\alpha = 0.1$ the solution is in close agreement with a multi-order simulation and an open foam simulation.
Bibliography


