

Fluid Structure Interaction of Involute Fuel Plates in the High Flux Isotope Reactor Using a Fully-coupled Numerical Approach

A Dissertation Presented for the
Doctor of Philosophy
Degree
The University of Tennessee, Knoxville

Franklin Guthrie Curtis

May 2018

© by Franklin Guthrie Curtis, 2018
All Rights Reserved.

Acknowledgments

I would like to begin by thanking God for without Him, I am nothing. My beautiful wife, Emi Curtis, who has been with me since the beginning of this *long* journey and has never wavered in her support of my pursuit of this degree and has been my helpmate for over 9 years. To my children Freeman, Rhodes and Abner who have continually prayed for me and have provided a welcomed relief at home.

I would like to thank my advisers, Dr. Kivanc Ekici and Dr. James D. Freels for their continued support and encouragement. Dr. Ekici has been instrumental in my understanding of CFD and his passion for academic pursuit is contagious. Dr. Freels has been an outstanding mentor at ORNL, introducing me to the research done at the Lab.

I would like to thank the members of my committee Dr. Rao V. Arimilli, Dr. Arthur E. Ruggles, and Dr. David H. Cook for taking the time to provide feedback on my work.

I would like to thank my colleagues at ORNL: Alan Stagg, Richard Cox, and David Sulfredge for welcoming me onto your team; Gary Giles for giving me the opportunity to begin my career while finishing my degree. My ‘old’ office mates who provided me good memories and stories to tell my wife: Isaac ‘Sweet Tea’ Bodey, Huang Huang, Adam ‘R. Travers’ Travis and Emily Clark.

Lastly, I would like to thank my parents, Barbara and Dale, for your support and love throughout the years. You constantly encouraged me to pursue my goals

and provided me with amazing opportunities to learn and grow. It is your love and teaching that has made me the man I am today.

This dissertation was funded in part by the Global Threat Reduction Initiative of the Department of Energy.

Quote

“The homemaker has the ultimate career. All other careers exist for one purpose only - and that is to support the ultimate career.”

C. S. Lewis

Abstract

This dissertation describes a fully-coupled (FC), finite-element (FE) based, algorithm for modeling and simulation of the fluid-structure interaction (FSI) of involute-shaped fuel plates used in research reactors; specifically the High Flux Isotope Reactor (HFIR) at the Oak Ridge National Laboratory (ORNL). Following a graded approach to code and model validation, a cylinder in cross-flow benchmark is used to establish flow physics as well as properly coupling the FSI phenomena with increasing complexity. As an interim step toward HFIR LEU fuel plate simulations, three experiments are used for validation. The first, performed by Smissaert, is used to envelope large plate deflections and understand the validity of various fluid boundary conditions for single plate comparisons. Continuing with Smissaert's data, a 5-plate simulation is presented showing the first-ever multi-plate simulation using this FC and FE approach. Second, a vibrating plate, presented by Liu et al., is simulated showing the same technique to encompass self-excited, periodic plate deflections. Lastly, an experiment for the conceptual Advanced Neutron Source Reactor (ANSR) using involute plates is utilized to validate the ability of this FC and FE algorithm to predict the deflections of the involute-shaped plates used in the HFIR. The method shown herein accurately captures the established 'S-shaped' deflection of the first mode of the involute plate providing guidance that researchers and designers can utilize in the forthcoming design of the next generation of low-enriched uranium (LEU) fuel plates for the HFIR. A 'Lessons Learned' section which describes external

routine coupling, geometry and meshing guidance, and solver settings used in the computational platform used to perform these FSI simulations is also provided.

Table of Contents

1	Introduction	1
1.1	Motivation	1
1.2	Background of HFIR	3
1.3	Overview of Previous Work	11
2	Governing Equations	18
2.1	Navier-Stokes: Fluid Domain	18
2.2	Structural Mechanics	25
2.3	Finite Element Analysis: Discretization	27
3	Cylinder in Cross-Flow	30
3.1	Stationary Cylinder	31
3.1.1	Domain and Boundary Conditions	32
3.1.2	Simulation and Results	34
3.1.3	Discussion	41
3.2	Oscillating Cylinder: Prescribed Displacement	43
3.2.1	Domain and Boundary Conditions	44
3.2.2	Simulation and Results	47
3.2.3	Discussion	49
3.3	Oscillating Cylinder: Free Displacement	49
3.3.1	Domain and Boundary Conditions	49
3.3.2	Simulation and Results	51

3.3.3	Discussion	59
3.4	Stationary Cylinder with Moving Tail	61
3.4.1	Domain and Boundary Conditions	61
3.4.2	Simulation and Results	62
3.4.3	Discussion	65
4	Flat and Curved Fuel Plate Simulations	66
4.1	Flat Plates	67
4.1.1	Smissaert Simulations: One-plate	67
4.2	Smissaert Simulations: Five-Plate	80
4.3	Liu Transient Experiments	85
4.4	Curved Plates	95
4.4.1	Advanced Neutron Source Reactor	95
4.4.2	HFIR	103
5	Lessons Learned	111
5.1	Calling External FORTRAN Functions	111
5.2	Geometry Creation and Meshing Techniques	114
5.3	Solver Settings	120
5.3.1	Segregated Vs. Fully-Coupled (Monolithic) Solvers	121
5.3.2	Iterative Vs. Direct Solvers	123
6	Conclusions and Recommendations	125
	List of References	127
	Appendix	140
A	Mesh Smoothing Techniques	141
B	Turek And Hron Additional Results	167

List of Tables

2.1	Constants used for the k - ϵ turbulence model for this dissertation. . .	25
3.1	Shedding frequencies of the stationary cylinder in cross-flow compared to experimental results.	38
3.2	Results for the 2-D simulation for the free displacement cylinder. . . .	56
3.3	Properties of the fluid and structure used for the FSI test cases. . . .	63
4.1	Properties of the PVC plate used by Smissaert in his experiment. . . .	68
4.2	Run times for each subsequent mesh performed on Mintaka at ORNL (8 nodes, 12 cores per node) for the simulation of the Smissaert [35] experiment.	81
4.3	Changes in flow area for each channel of the 5-plate simulation. . . .	85
4.4	Deflection frequencies for the 2-D Liu simulations.	87
4.5	Comparison of the deflection frequency and solution time for the 3-D case with fixed leading and trailing edges.	93
4.6	Frequency results for the 3-D simulation with only the 4 corners fixed similar to the experiment.	94
4.7	Inlet velocities for the ANS prototype and model simulations and experiments.	98
4.8	Properties used for the ANSR experiments.	99
4.9	Mechanical properties of the U-10Mo LEU fuel and aluminum used for the HFIR plate simulations.	107

A.1	Maximum time computed before error in mesh smoothing calculations for a maximum time-step of 0.1 s.	146
A.2	Maximum time computed before error in mesh smoothing calculations for a maximum time-step of 0.001 s.	147

List of Figures

1.1	Illustration of the HFIR core and the locations of the experiments. . .	5
1.2	Cutaway view of the HFIR fuel element.	6
1.3	Two involute curves with a base radius, r_b , and a base pitch, p_b , separating the two curves for the entire span.	8
1.4	Representative drawing of a single HFIR fuel plate with HEU fuel. . .	9
1.5	The HEU fuel profile for an inner fuel element.	9
1.6	The proposed LEU fuel profile for an inner fuel element.	10
1.7	‘S-shape’ curve predicted analytically and confirmed experimentally in the span-wise direction used in the HFIR SAR.	11
1.8	The ATR core with its clover-leaf fuel design.	12
1.9	Top-down view of the MIT core.	13
2.1	Arbitrary domain discretized into triangular finite elements.	27
3.1	Domain for the 2-D cylinder simulations.	33
3.2	Mesh comparison for the 2-D Cylinder.	34
3.3	Initial condition using the steady-state solver for $Re = 100$ for the 2-D cylinder simulations.	35
3.4	Set of time-domain solutions for $Re = 100$ for the stationary cylinder in cross-flow.	37
3.5	Time accurate C_l for the stationary cylinderfor the stationary cylinder in cross-flow.	39

3.6	FFT of the C_l for the stationary cylinder.	40
3.7	Strouhal number of the stationary cylinder simulations compared to the experimental results	41
3.8	C_l for the $Re = 150$ case with aliasing present due to large recording intervals.	42
3.9	Graphical Representation of the lock-on region for the cylinder in cross flow with prescribed displacement.	44
3.10	The cylinder lock-on simulations compared to the experiment by Koopman.	48
3.11	Visualization of the spring-mass-damper model used for the 2-D cylinder in cross-flow simulations.	50
3.12	Flow chart for the fully-coupled solver.	52
3.13	Flow chart for the segregated solver.	54
3.14	Flow chart for the one-way coupled solver.	55
3.15	Displacement and C_l of the fully coupled 2-D free displacement cylinder.	57
3.16	Solution of the fully coupled 3-D free displacement cylinder.	58
3.17	Incorrect displacement of the 2-D cylinder for the condition of $Re = 100$	60
3.18	Domain for the 2-D Stationary Cylinder with Moving Tail.	61
3.19	Results for the FSI2 test case.	64
3.20	2-D solution for the FSI2 case at 10s.	65
4.1	Deflection of a series of plates using periodic boundary conditions.	69
4.2	Bending of a series of plates with deflections of adjacent plates going in the opposite direction.	70
4.3	Single plate setup with full channels and wall boundary conditions.	72
4.4	Periodic boundary conditions with a plate offset of 5 mils.	73
4.5	The symmetry boundary increases and decreases the overall channel thickness between each plate when an offset is imposed on the plate location.	74

4.6	Leading-edge mesh for the single plate simulation with symmetric boundary conditions which simulated only half of the flow channel on each side of the plate for the Smitsaert experimental comparison. . .	75
4.7	Leading-edge mesh for the single plate simulation with wall boundary conditions which simulated the full flow channel on each side of the plate for the Smitsaert experimental comparison.	76
4.8	The pressure distribution for a 2-D simulation with inlet and out lengths of 2 inches.	77
4.9	The pressure distribution for a 2-D simulation with inlet and out lengths of 10 inches.	78
4.10	Deflection for a single plate simulation with symmetry boundary conditions.	79
4.11	Deflection for a single plate simulation with wall boundary conditions.	80
4.12	The mesh around the leading-edge of one of the plates used for the final mesh simulation with 815 680 total mesh elements for the five-plate Smitsaert simulation.	81
4.13	Deflection of the five-plate Smitsaert simulation for an inlet velocity of $1.23 \cdot M_c$ (approx 2.7 m/s).	82
4.14	Deflection along the centerline of the plates compared to the experimental deflections for the five-plate Smitsaert simulation.	83
4.15	2-D cut plane of the velocity (m/s) along the midplane of the plates at the leading-edge of the plates for the five-plate Smitsaert simulation.	84
4.16	2-D cut plane of the pressure (Pa) along the midplane of the plates at the leading-edge of the plates for the five-plate Smitsaert simulation.	84
4.17	The computational domain for the 2-D simulation of the Liu Experiment.	86
4.18	The deflection at the center of the plate for the 2-D Liu simulation with fixed leading- and trailing edges with an inlet velocity of 1.05 m/s	87
4.19	The deflection at the center of the plate for the 2-D Liu simulation with fixed leading- and trailing edges with an inlet velocity of 1.19 m/s	88

4.20	The deflection at the center of the plate for the 2-D Liu simulation with fixed leading- and trailing edges with an inlet velocity of 1.85 m/s	88
4.21	The deflection at the center of the plate for the 2-D Liu simulation with fixed leading- and trailing edges with an inlet velocity of 2.84 m/s	89
4.22	The deflection at the center of the plate for the 2-D Liu simulation with fixed leading- and trailing edges with an inlet velocity of 3.29 m/s	89
4.23	The deflection at the center of the plate for the 2-D Liu simulation with fixed leading- and trailing edges with an inlet velocity of 3.64 m/s	90
4.24	3-D domain for the transient Liu simulations with fixed leading and trailing edges of the plate.	91
4.25	Displacement for the 3-D Liu simulation with an inlet velocity of 1.85 m/s with the restricted time-step of 0.0215 s.	92
4.26	The time-steps used by the solver when there was no restriction on the maximum time-step for the 3-D Liu simulation with and inlet velocity of 1.85 m/s and fixed leading- and trailing-edges.	92
4.27	Displacement for the 3-D Liu simulation with an inlet velocity of 1.85 m/s with a free time-step and fixed leading- and trailing-edges. .	93
4.28	Displacement for the 3-D Liu simulation with free leading- and trailing- edges.	95
4.29	Progression of the 3-D displacement for an inlet velocity of 1.19 m/s for the 3-D Liu simulation with free leading- and trailing-edges. . . .	96
4.30	Proposed configuration of the ANSR core.	97
4.31	The mesh around the leading-edge of the single ANSR simulations with wall boundary conditions.	101
4.32	The leading-edge deflection of the aluminum ANSR simulation showing the ‘S-curve’ deflection with an inlet velocity of 17 m/s.	101
4.33	Deflections of the single plate ANSR simulations with wall boundary conditions.	102

4.34	The mesh around the leading-edge of the ANSR simulation for 2 plates with wall boundary conditions and an inlet velocity of 3.58 m/s. . . .	104
4.35	Leading-edge displacement of the 2 plate ANSR simulation at an inlet velocity of 3.58 m/s with the PVC plates.	104
4.36	Comparison of the leading-edge deflection of the HFIR fuel plate simulations using the properties of the PVC plate compared to the experiment by Swinson et al.	105
4.37	Mesh consisting of 958 649 elements for the inner fuel element of the HFIR with the inclusion of the LEU fuel.	106
4.38	Deflection of the HFIR IFE plate made of solid aluminum with a coolant velocity of 15 m/s.	108
4.39	Deflection of the HFIR IFE plate with the LEU fuel profile of U10Mo with a coolant velocity of 15 m/s.	109
4.40	Leading-edge deflection of a solid aluminum IFE plate compared to the IFE plate with the LEU fuel profile of U10Mo. The single plate simulation uses wall boundary conditions.	110
4.41	Difference in the deflection in the area of the plate where the LEU U10Mo fuel meat is located with a coolant velocity of 15 m/s.	110
5.1	The tree-like structure used in the COMSOL code to control the simulation.	112
5.2	Default generated mesh using free-tetrahedrals and boundary-layer meshes.	115
5.3	‘Smooth across removed control entities’ check box that is automatically turned on for most mesh generation settings and should be turned off for finer control.	117
5.4	Skewed mesh from the ‘smoothing’ of the mesh across a removed boundary.	118

5.5	Boundary layer mesh along the wall of a fluid domain added to a free triangular mesh.	119
5.6	Velocity along the wall for a turbulent flow solution comparing a free triangular mesh with and without boundary layer meshes added in. .	119
5.7	Mesh referencing and scaling settings to duplicate and refine/coarsen meshes.	120
5.8	Solver options in the COMSOL code.	122
A.1	Domain for the mesh smoothing example.	143
A.2	The mapped mesh used for the mesh smoothing comparisons.	144
A.3	The free mesh used for the mesh smoothing comparisons.	144
A.4	Linear boundary movement for the mesh smoothing example.	145
A.5	Quadratic boundary movement for the mesh smoothing example. . .	145
A.6	The deformed free mesh using the Hyperelastic smoothing technique at 0.9 s.	148
A.7	The deformed mapped mesh using the Hyperelastic smoothing technique at 0.9 s.	149
A.8	The deformed mapped mesh using the Laplace smoothing technique at 1.0 s.	150
A.9	The deformed mapped mesh using the Winslow smoothing technique at 1.0 s.	150
A.10	The deformed mapped mesh using the Hyperelastic smoothing technique at 0.565 s.	151
A.11	The deformed mapped mesh using the Hyperelastic smoothing technique at 0.75 s.	152
A.12	The deformed free mesh using the Hyperelastic smoothing technique at 0.75 s.	152
B.1	Results for the FSI3 test case.	168
B.2	Results for the Steel test case.	169

Nomenclature

Acronyms

1-D	One-dimensional
2-D	Two-dimensional
3-D	Three-dimensional
AEC	Atomic Energy Commission
ANSR	Advanced Neutron Source Reactor
AR	Aspect Ratio
ATR	Advanced Test Reactor
CAD	Computer Aided Design
DOE	Department of Energy
FC	Fully-coupled
FEA	Finite Element Analysis
FFT	Fast Fourier Transform
FSI	Fluid-Structure Interaction
GLS	Galerkin-Least Squares

GTRI	Global Threat Reduction Initiative
HEU	Highly-Enriched Uranium
HFIR	High Flux Isotope Reactor
HPRR	High-Performance Research Reactor
INL	Idaho National Laboratory
LBB	Ladyzhenskaya-Babuska-Brezzi
LEU	Low-Enriched Uranium
MIT-NRL	Massachusetts Institute of Technology Nuclear Reactor Laboratory
MURR	University of Missouri Research Reactor
NIST	National Institute of Standards and Technology
NNSA	National Nuclear Security Administration
ODE	Ordinary Differential Equation
ORNL	Oak Ridge National Laboratory
OSU	Oregon State University
RANS	Reynolds Averaged Navier-Stokes
RERTR	Reduced Enrichment for Research Test Reactors
SAR	Safety Analysis Report
SOF	Single Degree of Freedom
SSHTC	Steady State Heat Transfer Code
TRTR	Test, Research, and Training Reactors

WMDs Weapons of Mass Destruction

Greek Symbols

δ_{ij}	Kronecker Delta
ϵ_{ij}	Strain Tensor
λ	Coefficient of Bulk Viscosity, Lamé Constant
μ	Dynamic viscosity
∇	Gradient Operator
ν	Poissons ratio
ω	frequency
Φ	Dissipation Function
ρ_f	Density in the fluid domain
σ	Stress in the structural domain
τ'_{ij}	Viscous Stresses for a Newtonian Fluid
τ_{ij}	Stress Tensor
ε	Strain in the structural domain
δl_l	Deflection at the End of the Beam
ρ_f	Density of the Fluid Domain

Other Symbols

$\bar{U}$	Average Inlet Velocity
$\otimes$	Tensor Product

$\mathbf{F}_s$	Body forces in the structural domain
$\mathbf{q}$	Heat transfer to an element
$\mathbf{u}_s$	Location in the structural domain
$\mathbf{u}_f$	Velocity Field in the Fluid Domain
MW_{th}	Megawatt Thermal
a	Plate Thickness
b	Span of a Fuel Plate
C	Fourth Order Stiffness Tensor
c	Damping Coefficient
C_d	Coefficient of drag
C_l	Coefficient of lift
D	Characteristic Diameter
D_h	Hydraulic Diameter
E	Young's Modulus
e	Internal Energy per Unit Mass
E_t	Total Energy of a System
f	Frequency
f_n	Natural Frequency
g	Gravitational Constant, 9.81 m/s
h	Enthalpy

h	Flow Channel Thickness
I	Area Moment of Inertia
k	Thermal Conductivity, Spring Constant
l	Lift per unit length
m	Mass
M_c	Miller Critical Velocity
p	Pressure
p_b	Base Pitch, distance between concentric involute curves
Q	Heat Added to a System
r	Displacement of a Particle
r_b	Base Radius of Involute
Re	Reynolds number
St	Strouhal Number
T	Temperature
t	Time, Thickness of Fluid Channel for Calculation of D_h
U_∞	Free stream velocity
W	Heat Done on a System
w	Width of Fluid Channel for Calculation of D_h
$\mathbf{n}$	Normal Vector
U-10Mo	Uranium-10Molybdenum
y^+	Wall Offset in Viscous Units

Chapter 1

Introduction

1.1 Motivation

The High Flux Isotope Reactor (HFIR) at the Oak Ridge National Lab (ORNL) is a research reactor currently utilizing highly-enriched uranium (HEU) fuel. The U. S. Department of Energy's (DOE) National Nuclear Security Administration (NNSA) has funded research for all research reactors to convert their HEU fuel to low-enriched uranium (LEU) through the Global Threat Reduction Initiative (GTRI). The Convert program is one of the three pillars of the GTRI designed to reduce the use of HEU fuel which is capable of producing weapons of mass destruction (WMDs). Reducing and eliminating the use of HEU fuel by converting domestic and international civilian research reactors and isotope production facilities to LEU fuel will help to protect Americans and the rest of the world from nuclear and radiological terrorism [1]. In order to facilitate this conversion, the Reduced Enrichment for Research Test Reactors (RERTR) [2] and the National Organization of Test, Research, and Training Reactors (TRTR) [3] were created. Presently, over 40 research reactors have been converted from HEU to LEU fuel thanks to the support of the NNSA and RERTR. There are currently five major High-Performance Research Reactors (HPRRs) that still need to be converted in the United States: the Massachusetts Institute of Technology

Nuclear Reactor Laboratory (MIT-NRL), the University of Missouri Research Reactor (MURR), the National Institute of Standards and Technology (NIST) Research Reactor, the HFIR at ORNL and the Advanced Test Reactor (ATR) at Idaho National Laboratory (INL) [4].

There are several goals that the GTRI/RERTR program has defined for the replacement of HEU fuels in research reactors. Some of the key points of these goals are to maintain the current scientific missions of the reactors, ensure similar fuel lifetimes, ensure that conversion can be completed without major changes to the structure or equipment, and demonstrate that the reactor can be converted and operated safely after conversion. For the HFIR, the conversion must maintain the high neutron flux for the same cycle length (approximately 25 days) using existing structures, systems and components while incorporating the current fuel geometric envelope. Staff members at ORNL are currently performing safety calculations for the conversion.

The design of the current HEU core is based upon early research reactor experience, HFIR-specific experiments, and historical safety and design basis calculations. Today, it is not economically feasible to perform extensive experiments to support the design of the fuel and the use of computational models is becoming a necessity. Currently, the HFIR employs numerous safety calculations and codes, based upon empirical data, to assess each cycle of the reactor. Each code and its implementation for the HFIR is described in detail in the Safety Analysis Report (SAR) for the HFIR [5]. The two main codes used for the thermal-hydraulic analysis of the HFIR core and system are the Steady State Heat Transfer Code (SSHTC) [6] and a modified version of RELAP5 [7]; both of these codes are based on rather simplified one-dimensional flow theories. As the name implies, the SSHTC is a proprietary, steady-state code designed to analyze the core of the HFIR. Using experimental data and built-in safety margins, the SSHTC is used to verify the conditions of the core based upon inputs from a neutronics code called MCNP [8, 9]. RELAP5 is used to determine the transient response of the entire reactor system for events considered in the accident analysis of

the SAR. It is desired to supplement these codes with a more robust analysis of the HFIR core using a three-dimensional analysis that incorporates a variety of physics.

The computational tool chosen for the analysis of the HFIR core is COMSOL Multiphysics version 5.2a [10]. COMSOL is a finite-element analysis (FEA) commercial software package designed to incorporate a multitude of physics including fluid flow, heat transfer, structural mechanics, magnetism, and many others. The code also allows for the implementation of user defined physics not provided by the commercial developer. Incorporating the proper boundary conditions between each set of physics allows for the simulation and analysis of very complex systems such as that found in the core of the HFIR. Previously, analysis of the HFIR core using COMSOL has been limited to thermal-hydraulics [11–20]. The results of this analysis have also given rise to the simulation of the thermal-structure interaction in the HFIR performed by Jain et al. [21]. Chandler [22] incorporated reactor kinetics into the COMSOL code to model the reactions of the LEU fuel for the HFIR core.

The focus of this dissertation is to analyze the interaction of the coolant flow with the structural mechanics of the fuel plates. With the conversion to LEU fuel, the structure of the HFIR fuel plates will be altered and an in-depth analysis of the structural response is desired to ensure the safety of the reactor after conversion. Up until this work, a fully-coupled, numerical simulation of the fluid-structure interaction (FSI) phenomena has not been performed for the HFIR fuel plates. Not only can this work be applied to the conversion of the HFIR to LEU fuel, but it can be utilized in the safety analysis for the current HEU fuel design.

1.2 Background of HFIR

The HFIR was commissioned by the U. S. Atomic Energy Commission (AEC) in November of 1958 to meet the anticipated growing needs for transuranic isotopes such as plutonium and curium. In June of 1961, construction began on the HFIR site

with criticality achieved on August 25, 1965. By September of 1966, the HFIR was running at its designed power of $100 \text{ MW}_{\text{th}}$.

The HFIR was designed on the ‘flux trap’ principle in which an unfueled moderator region is surrounded by an annular region of fuel. The unfueled region, also known as an island, allows fast neutrons leaking from the fuel to be moderated and produces a region of very high thermal-neutron flux in the island. The thermalized neutrons are trapped within the reactor and can be used for isotope production. One significant isotope produced in the reactor is Californium-252 which is used for a variety of nuclear applications such as reactor start up, neutron activation analysis, and as a medical isotope used to treat some forms of cancer. HFIR is the only producer of this isotope in the western world.

In addition to isotope production within the flux trap, it is also possible to tap the high flux of neutrons outside of the fuel and inside the reflector by placing empty beam tubes into the reflector. This allows the neutrons to be ‘beamed’ into the experiments outside of the reactor shielding. Additionally, materials may be placed in holes inside the reflector for irradiation and later retrieval. A top-down view of the HFIR core and various experimental locations is provided in Figure 1.1.

In November of 1986, tests indicated that the HFIR vessel was being embrittled at a much faster rate than expected and the reactor was shut down for investigation. Up until this time, the HFIR was able to achieve a record of operation unsurpassed by any other nuclear reactor in the United States. After careful inspection and reevaluation of the reactor, the HFIR was restarted on May 18, 1990 at a lowered power of $85 \text{ MW}_{\text{th}}$. In 2007, the reactor was again shut down but this time to provide users with more experimental capacity. During its one-year shutdown, a variety of new instruments were installed as well as a cold neutron source [23]. The additional instrumentation and upgrading performed during this shutdown has allowed the HFIR to continue to attract leading researchers in the nuclear and related fields.

By design, the construction of the fuel core has some very interesting characteristics to facilitate the high performance needed to produce such a high neutron flux.

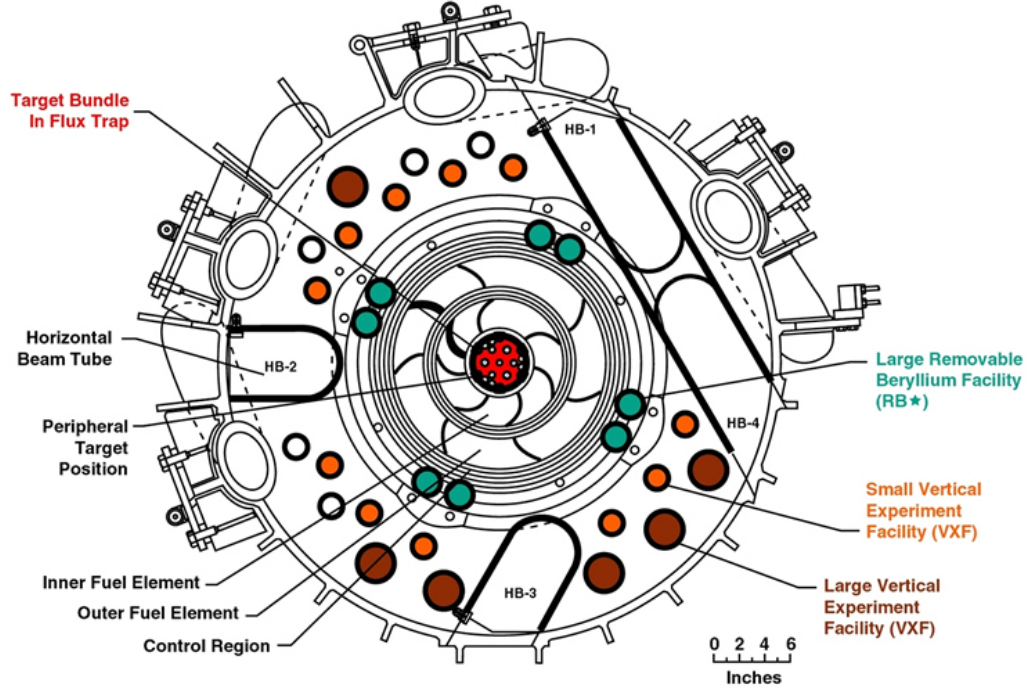


Figure 1.1: Illustration of the HFIR core and the locations of the experiments.

The core consists of two concentric fuel regions containing 171 curved fuel plates in the inner core and 369 fuel plates in the outer core [24]. A cutaway of the fuel core is provided in Figure 1.2. The fuel plates are 24 inches long with a 20 inch active fuel region in the middle of the plate. Each fuel plate has a nominal thickness of 50 mils (0.050 inches, 1.27 mm) with a coolant gap of 50 mils between each plate. Water is pumped through the core at a rate of 16,800 gallons per minute in order to maintain proper cooling for the reactor [5]. For clarity, the axial direction of the fuel plate refers to the flow direction while the span-wise direction refers to the length along the curved fuel plate, transverse of the flow direction.

Each fuel element follows an involute shape in order to maintain a constant coolant channel gap for the entirety of the fuel plate span. The involute is created by following the path a taut string travels when unwound from a curve. The HFIR fuel plate span is created by using the involute curve of a circle. In parametric form, the path is

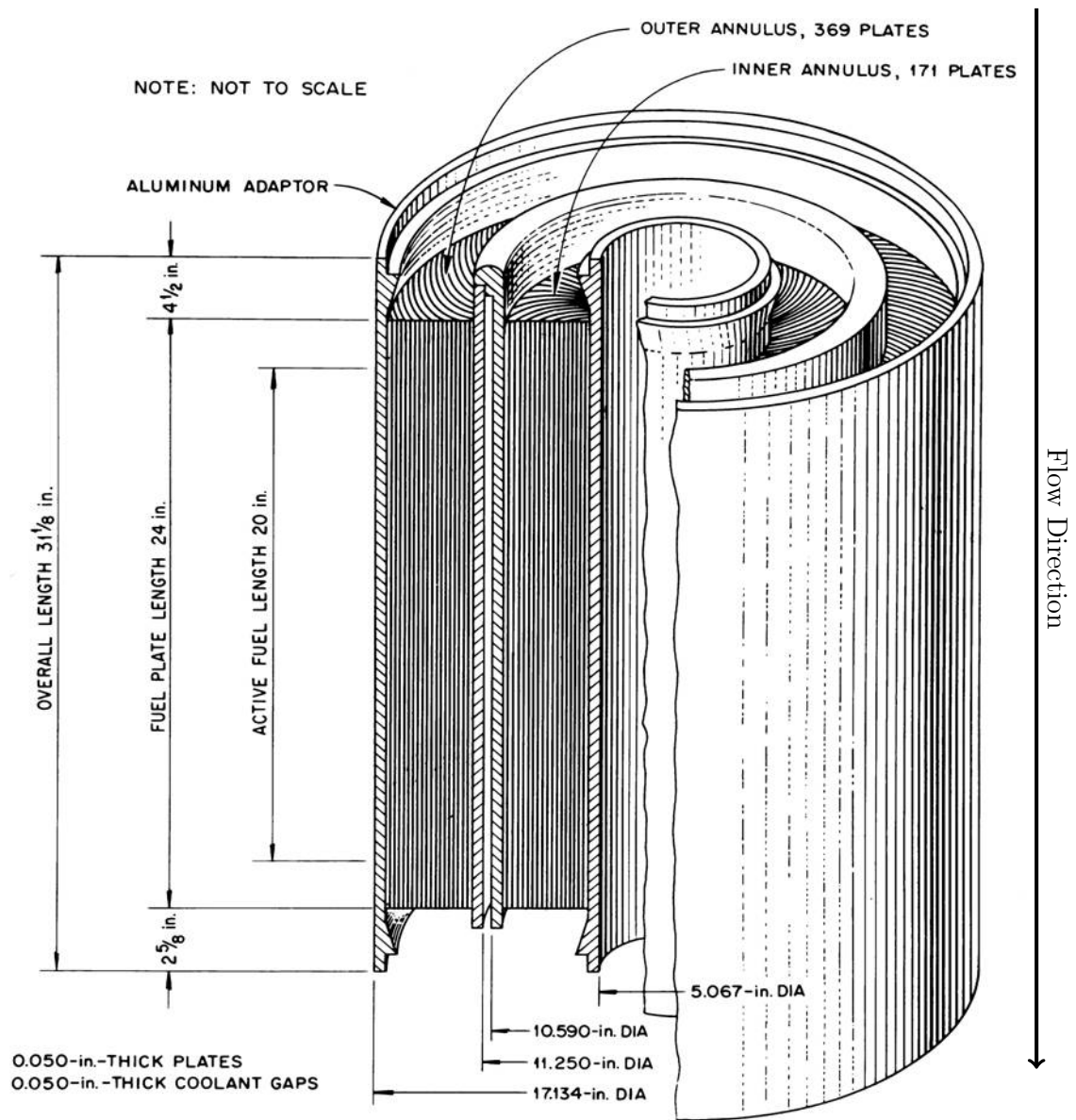


Figure 1.2: Cutaway view of the HFIR fuel element.

given by:

$$x = r_b (\cos \theta + \theta \sin \theta) \quad (1.1)$$

$$y = r_b (\sin \theta - \theta \cos \theta) \quad (1.2)$$

where r_b is the base radius of the involute circle. The involute geometry has been used extensively in the design and creation of gears as a means to reduce the wear and vibration that occurs when they mesh together [25]. Using the nomenclature from gear theory, the distance between each involute curve is denoted as the base pitch, p_b , which corresponds to the channel thickness of 50 mils for the HFIR. An example of two involute curves separated by a base pitch is provided in Figure 1.3. Each dashed line in Figure 1.3 is tangent to the base circle and is perpendicular to the involute curve.

For the HFIR, the inner fuel element has a base radius of 2.7215 inches and the outer fuel element has a base radius of 5.873 inches. The current fuel used in the HFIR core consists of 93% by weight Uranium-235 (U^{235}) in a dispersion-type fuel pressed between the aluminum cladding along with a filler material consisting of aluminum and a burnable poison of Boron [26]. A representation of a single fuel plate with HEU fuel is presented in Figure 1.4. This fuel profile provides a specific power density profile designed to provide the neutron flux desired within the core and to maintain a safety margin for the entirety of the fuel cycle. The fuel profile for HEU fuel is provided in Figure 1.5.

In order to convert the HFIR to LEU fuel, the internals of the fuel plates must be redesigned and evaluated to meet the requirements provided by the GTRI/RERTR. Initial designs have determined that the fuel will be changed from a powdered, dispersion-type fuel to a solid fuel type; the physical properties for the new LEU fuel are described in detail in the INL report by Burkes et al. [27]. The current proposed fuel contour for the LEU fuel is provided in Figure 1.6. The proposed LEU fuel profile will be used for the analysis of the HFIR plate in this dissertation.

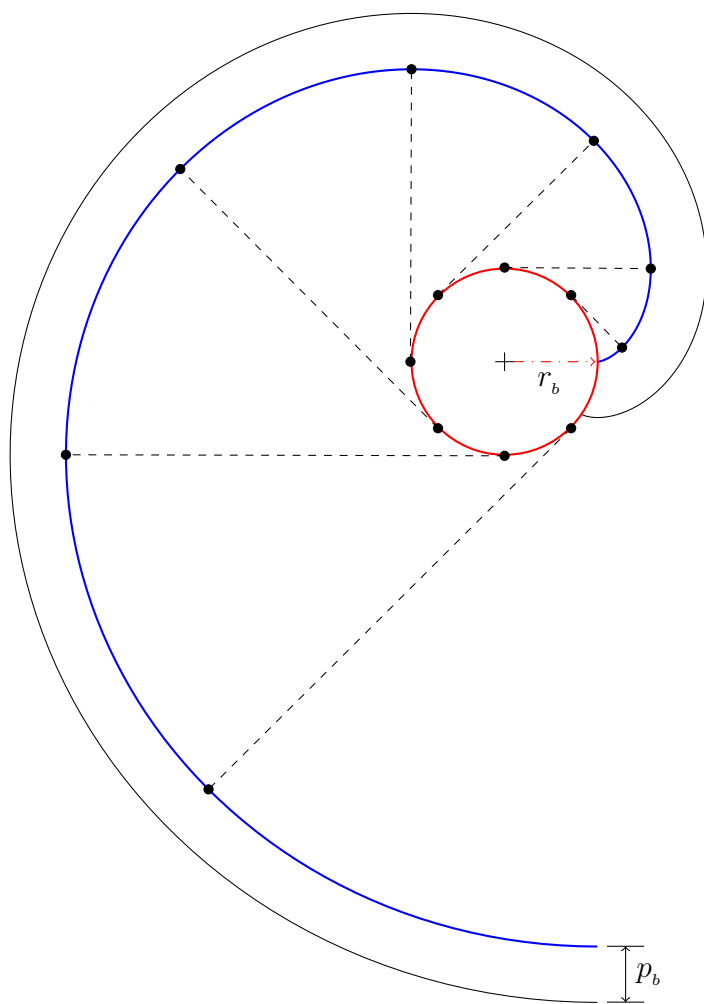


Figure 1.3: Two involute curves with a base radius, r_b , and a base pitch, p_b , separating the two curves for the entire span.

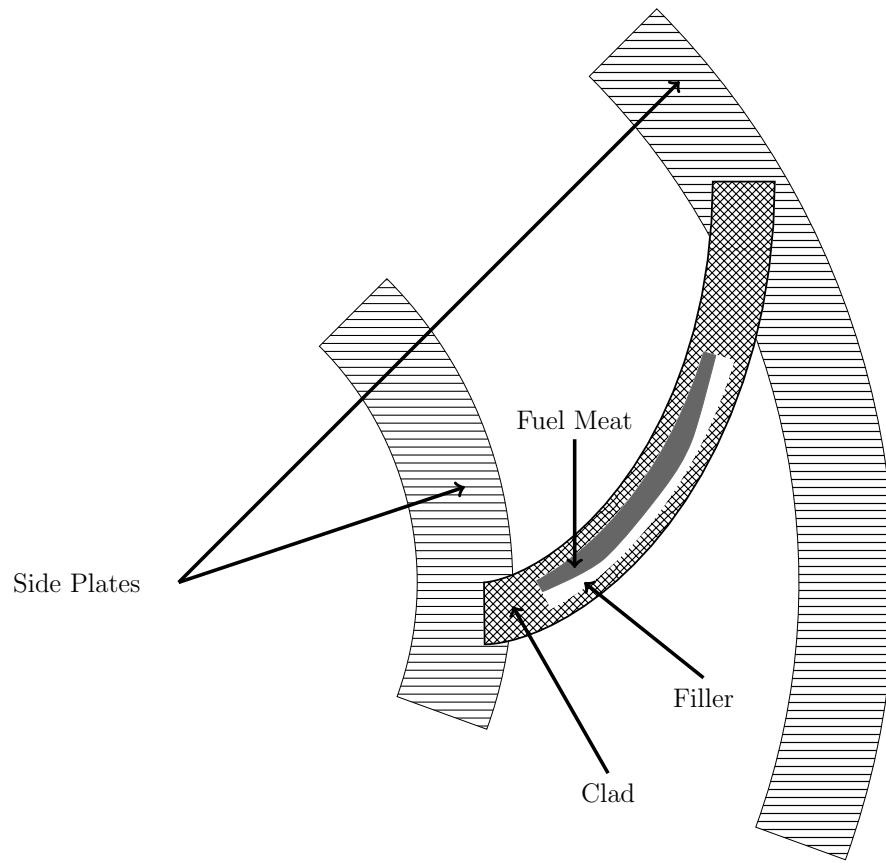


Figure 1.4: Representative drawing of a single HFIR fuel plate with HEU fuel. Note that the figure is not drawn to scale.

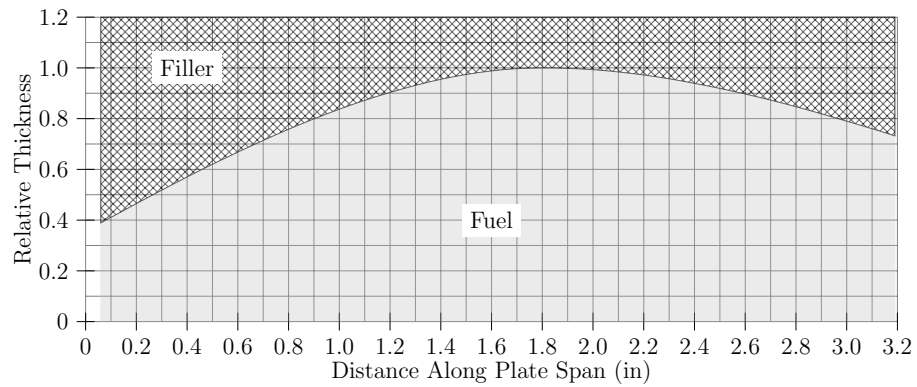


Figure 1.5: The HEU fuel profile for an inner fuel element.

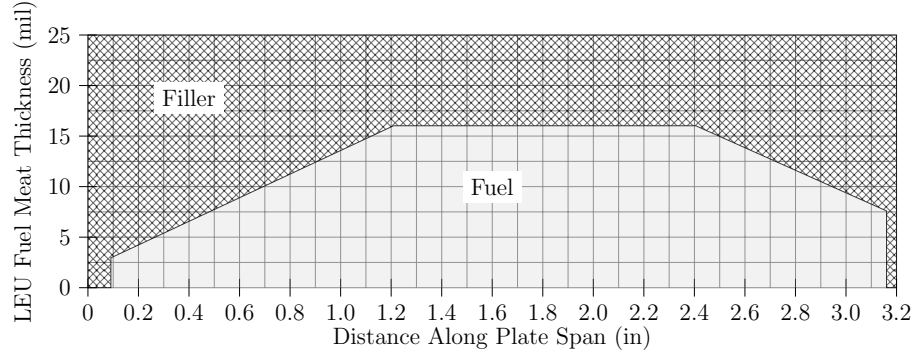


Figure 1.6: The proposed LEU fuel profile for an inner fuel element.

The guidelines for safe operation of the HFIR are found in the HFIR Safety Analysis Report (SAR) [5]. The SAR contains all of the information regarding assumptions used for the analysis of the HFIR core to ensure safety during reactor operation. There are several items addressed pertaining to the hydraulic deflections of the plates. The deflection along the span of the fuel plate was determined analytically and verified experimentally to create an ‘S-shape’ curve as shown in Figure 1.7. The critical flow velocity, calculated in the SAR using the assumptions developed by Gwaltney and Luttrell [28], are 49.8 m/s and 47.1 m/s for the inner and outer fuel plates, respectively. This is well above the current coolant flow of approximately 15.5 m/s.

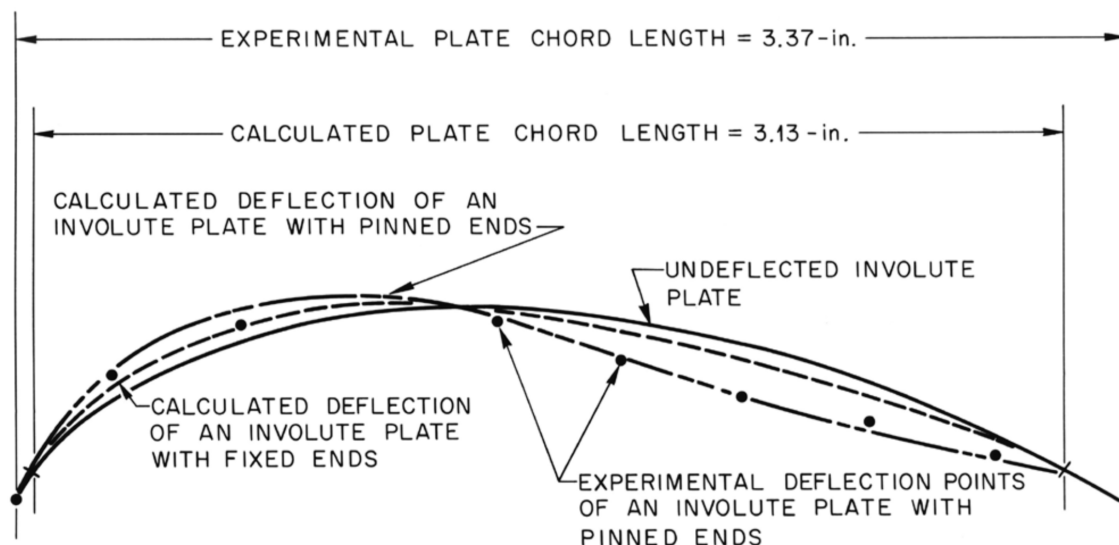


Figure 1.7: ‘S-shape’ curve predicted analytically and confirmed experimentally in the span-wise direction used in the HFIR SAR. Figure taken from the HFIR SAR.

1.3 Overview of Previous Work

The conversion of the HFIR from HEU fuel to LEU fuel has revealed the need to reevaluate the tools used to verify the safety of the reactor. Research reactors, by nature, are designed with specific scientific goals which often times require novel design elements. The HFIR, with its emphasis on a high neutron flux, utilizes involute fuel plates in two concentric circles to create its ‘flux trap.’ The Advanced Test Reactor at INL is designed to facilitate many different experiments as well as changes to the experiment during the fuel cycle [29] and a ‘clover-leaf’ fuel design has been implemented as seen in Figure 1.8. The reactor at the Massachusetts Institute of Technology (MIT) utilizes a grid pattern for experiment and fuel placement (Figure 1.9).

All of the five remaining HPRRs that have yet to be converted to LEU utilize an array of fuel plates. This use of flat fuel plates is in contrast to power reactors

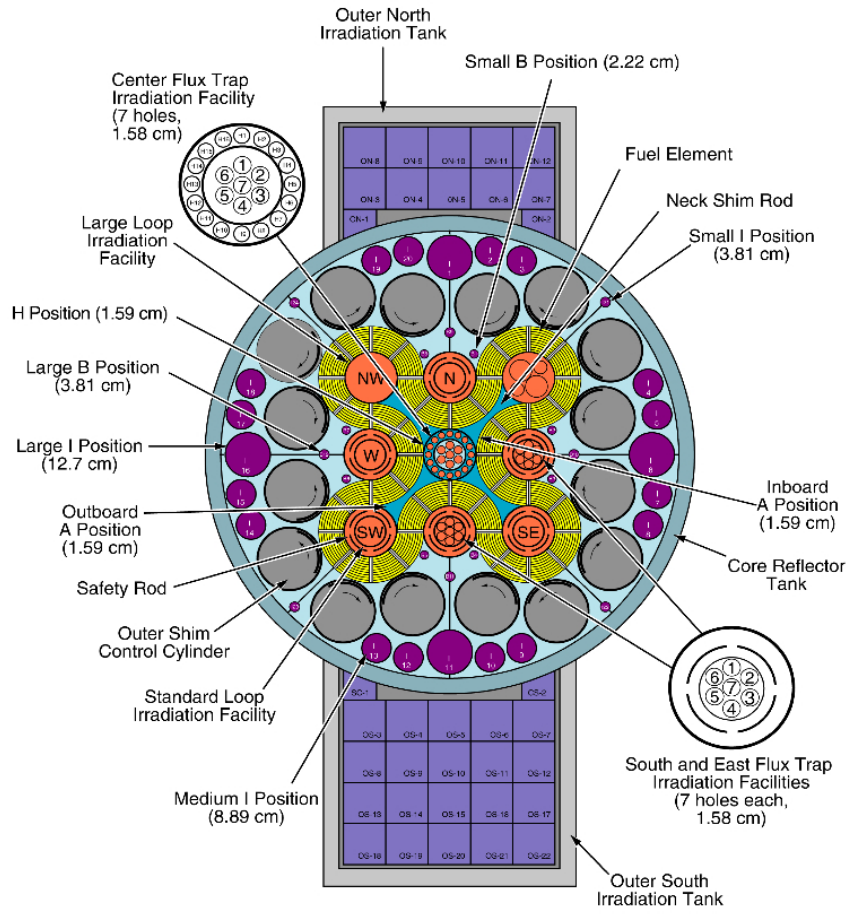


Figure 1.8: The ATR core with its clover-leaf fuel design [30].

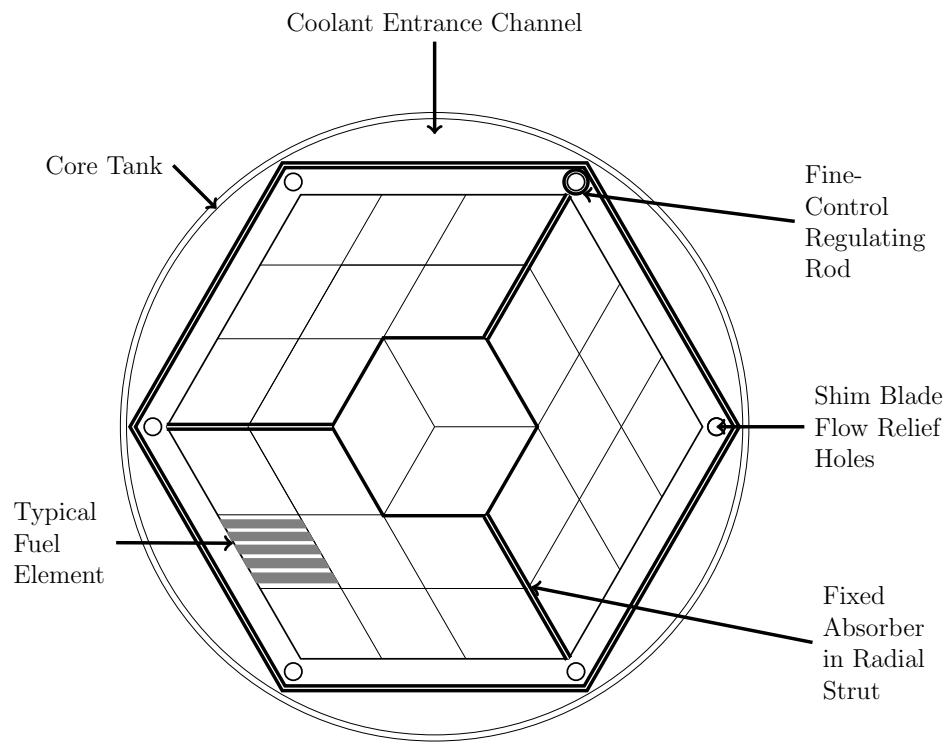


Figure 1.9: Top-down view of the MIT core [31]. The coolant flows into the page.

which typically utilize fuel rods and a significantly larger mass of uranium. Because the HPFRs use high aspect ratio flat plates, a significant effort has been made for quantifying the deflections caused by the coolant flow. One of the first experiments exploring the effects of the fluid-structure interaction of an array of fuel plates was performed at ORNL for preliminary research for curved fuel plates in 1948 [32]. Stromquist and Sisman were able to observe plate vibration with plates similar in thickness to the current HFIR design. Their experiments found that extreme spacing variation between plates (one channel was 15.9 mm while the adjacent was 3 mm) could produce buckling at the plate leading-edges. Under nominal plate spacing, the maximum deflection of the plates was found to be approximately 2 mils (0.05 mm).

Later, Miller developed a theoretical approach to determine the maximum velocity that a series of plates with fixed sides can sustain before collapse [33]. Miller's Critical Velocity, M_c , was based upon the assumption of incompressible, potential flow and it utilized an elastic wide-beam theory; Miller also assumed that the mass flow between all of the channels remained the same. Miller's approach determined the velocity in which the pressure difference between two plates is sufficient to maintain a deflection. The Miller Critical Velocity soon became the safety basis used in analyzing the critical flow velocity of parallel reactor fuel plates. For a flat plate with fixed edges, the Miller Critical Velocity is defined as

$$M_c = \left[\frac{15gEa^3h}{\rho_f b^4 (1 - \nu^2)} \right]^{1/2} \quad (1.3)$$

where g is the gravitational constant, E is the Young's Modulus of the plate, a is the plate thickness, h is the flow channel thickness, ρ_f is the density of the coolant, b is the width or span of the plate, and ν is the Poisson's ratio of the plate.

Experimental testing of the Miller Critical Velocity has been performed by many researchers [34–37]. These experimenters found that the Miller Critical Velocity was conservative, with flows tested to $2 \cdot M_c$ with no total plate collapse. In fact, Groninger and Kane [34] and Smissaert [35] found that vibration occurs at approximately

twice the Miller Critical Velocity. The only experiment in which total plate collapse occurred was performed by Ho et al. [38]. The collapse occurred at a velocity lower than M_c suggesting that this experiment may be an outlier.

In support of the conversion to LEU fuel, two facilities have been constructed to produce plate deflection data. A generic test plate experiment is being designed, constructed, and tested at the Oregon State University (OSU) that uses a flat plate, multi-channel design to measure both static and dynamic fuel plate deflections [39]. The OSU experiments will utilize strain gages to determine the deflections along the plate. Single plate experiments are being designed and tested at the University of Missouri that incorporate a Perspex flow channel allowing the use of a laser to measure the plate deflections [37].

Most of the experimental research has been focused on static deflection of perfectly flat fuel plates. A few experiments have been performed to investigate the deflections for curved fuel plates. Ha and Garland [40] conducted an experiment for a curved plate following a circular arc. Researchers at ORNL, in the design of the proposed Advanced Neutron Source Reactor (ANSR), conducted experiments using a series of involute shaped fuel plates [41]. A range of inlet velocities was considered and the experiments revealed a much smaller deflection for the curved plates than an equivalent flat plate.

The shortcomings and inherent conservative assumptions used for the Miller Critical Velocity led many researchers seek ways to improve the theory. Johansson [42], a colleague of Miller's, first expanded on the Miller Critical Velocity by including friction terms and the flow redistribution caused by the movement of plates. Kane [43] explored how deviations from the design thickness can affect the deflections. He found that large deviations, as expected, can lead to larger deflections. Further, Scavuzzo [44] included the entrance and exit length effects at the leading and trailing edge of the plate. Wambsganss Jr. [45] argued that the derivation first proposed by Miller can be improved upon by including second-order effects in the calculation of the critical velocity.

Following these efforts, researchers began to reevaluate the assumptions Miller made in his derivation of the critical flow velocity. Other analysis techniques were developed including a wave propagation/water-hammer approach [46], solving the series of plates as a system [47], and using Schlichting boundary layer theory [48]. Cekirge and Ural [49] used small deflection plate theory to rederive the critical flow velocity. Their modified formulation resulted in only a four percent difference from Miller’s original approach. In order to better capture the plate deflections, Pavone and Scarton [50] employed a fourth order structural model with laminar flow. Kerboua et al. [51] used a potential flow analysis of one plate to analyze a multi-plate channel and found that their method matched Miller’s Critical Velocity. Howard et al. [52] utilized a 1-D, semi-analytical model to obtain full-length plate deflections that compared well to the experiments provided by Kennedy et al. [53].

All efforts up to this point had assumed one dimensional, steady-state flow. A Galerkin approach was employed by Guo and Païdoussis [54] to analyze a two-dimensional plate with a three-dimensional flow field. By combining turbulent effects with a frequency analysis of thin rectangular plates, Kim and Davis [55] were able to obtain natural frequencies of a series of plates. Due to the added mass effect, they found that the natural frequencies of the plates in a fluid were lower than the plates in vacuo; they also found that by decreasing the channel gap, the natural frequencies would decrease. On the other hand, Cui et al. [56] used a wetting method to determine the natural frequency of the plates and found that the added fluid did not greatly affect the natural frequency of the plate. Using a stability criterion, Michelin and Llewellyn Smith [57] analyzed a series of plates to determine the stability of the system as it relates to flutter.

Until recently, the use of computer codes to simulate fluid-structure interactions between the fuel plates and the coolant flow has been too computationally expensive. Roth [58] simulated the fluid flow between the fuel plates but was not able to model plate deflections. Kennedy and Solbrekken [59] have used two separate codes, one for solving the fluid domain and the other one for solving the structural domain. Their

approach utilized time-dependent solvers for each code, which were loosely-coupled to each other. This approach resulted in an unstable solution process at higher Reynolds number simulations.

In this study, a fully-coupled approach has been adopted to eliminate stability issues found with the loosely-coupled approach. As will be shown in the later parts of this work, the resulting framework is robust and can model complex multiphysics phenomena inherent in narrow channel fuel plate designs. The modeling approach has been applied to problems with progressively increased complexity to establish guidelines. Validation studies were carried out for the prediction of static deflection in a series of flat plates, as well as involute plates that are currently used in the HFIR core. The method was also extended to the analysis of fluid-structure interactions of the LEU-based HFIR plates and the effects of the inclusion of the fuel meat in the model were investigated. Finally, time-accurate simulations for fluttering plates in an experimental setup were considered and the computed frequencies and vibration amplitudes were compared to available data. Based on those results, modeling guidelines were provided. The results and findings of this research have been presented at conferences [60] and have been published in an archival journal paper [61]. Currently, the work on involute plate FSI simulations is under consideration for publication as a journal article and the work on fluttering plates is under preparation for journal submission.

Chapter 2

Governing Equations

The goal of this dissertation is to develop a simulation technique that will allow the reproduction of the FSI phenomena that occurs within the HFIR fuel core. In order to numerically simulate this, a physical model must be chosen for the fluid domain as well as a model for the structural domain. For the simulation of the coolant flow, the Navier-Stokes equations will be used as the basis for simulation. The governing equations and how they will be used for the simulation will be demonstrated in Section 2.1. The structural domain will be represented by a linear structural model and will be described in Section 2.2. These equations are based upon continuum mechanics and often times cannot be solved directly; because of this, the governing equations must be discretized. For this dissertation, the Finite Element discretization will be used and is described in Section 2.3.

2.1 Navier-Stokes: Fluid Domain

In order to fully describe the fluid domain, the three laws of conservation must be met:

1. Conservation of mass
2. Conservation of momentum

3. Conservation of energy

These three laws, when implemented with the correct boundary conditions, can be used to describe the flow of a Newtonian fluid. The three unknown variables associated with the conservation laws are pressure p , velocity $\mathbf{u}_f$ and temperature T . All of the other variables that are associated with this such as density ρ_f , enthalpy h , viscosity μ and thermal conductivity k can be obtained assuming local thermodynamic equilibrium using only p and T .

The most general form of the laws of conservation is to assume a fluid that is compressible, that is the density of the fluid may change, and one that is changing with respect to time. These two assumptions produce the following conservation of mass equation

$$\frac{D\rho}{Dt} + \rho_f \nabla \cdot \mathbf{u}_f = 0 \quad (2.1)$$

In Eq. (2.1), $\frac{DQ}{Dt}$ is known as the substantial or material derivative and is given as

$$\frac{DQ}{Dt} = \frac{\partial Q}{\partial t} + (\mathbf{u}_f \cdot \nabla)Q \quad (2.2)$$

in which Q can represent any variable. The ∇ symbol describes the gradient operator and in Cartesian coordinates is defined as

$$\nabla = \hat{i} \frac{\partial}{\partial x} + \hat{j} \frac{\partial}{\partial y} + \hat{k} \frac{\partial}{\partial z} \quad (2.3)$$

If one assumes that the fluid is incompressible, for fluids such as water, Eq. (2.1) becomes

$$\nabla \cdot \mathbf{u}_f = 0 \quad (2.4)$$

This assumption will be used for the simulations performed in this dissertation as the working fluid will be water.

Moving on to the next conservation law, the conservation of momentum, one can begin with Newton's second law

$$\mathbf{F} = m\mathbf{a} \quad (2.5)$$

The general form of Newton's law for a fluid particle is

$$\rho_f \frac{D\mathbf{u}_f}{Dt} = \rho_f \mathbf{g} + \nabla \cdot \tau_{ij} \quad (2.6)$$

where $\mathbf{g}$ is the force due to gravity and τ_{ij} is the stress tensor.

In order to obtain the Navier-Stokes equations from Eq. (2.6), one must make the assumption of the Newtonian Fluid. In order for a fluid to be considered Newtonian, the fluid must be continuous and a linear relationship must exist between the stress tensor τ_{ij} and the strain rates ϵ_{ij} . The properties of the fluid must be independent of direction, that is, the fluid is isotropic. This assumption ensures that the deformation relationship is independent of the coordinate axes. Finally, the fluid must reduce to the hydrostatic pressure condition when the strain rates go to zero [62, 63]. Using these assumptions, the general form of the Navier-Stokes equations can be written as

$$\rho_f \frac{D\mathbf{u}_f}{Dt} = \rho_f \mathbf{g} - \nabla p + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right) + \delta_{ij} \lambda \operatorname{div} \mathbf{u}_f \right] \quad (2.7)$$

where δ_{ij} is the Kronecker delta ($\delta_{ij} = 1$ if $i = j$ and $\delta_{ij} = 0$ if $i \neq j$) and λ is known as the coefficient of bulk viscosity. The terms inside the bracket $[\dots]$ of Eq. (2.7) are the viscous stresses for a Newtonian fluid and will be represented by τ'_{ij} .

As stated earlier in the derivation, the working fluid is assumed to be incompressible such that $\operatorname{div} \mathbf{u}_f$ goes to zero due to the continuity equation. This assumption eliminates the coefficient of bulk viscosity. If the viscosity, μ , is not allowed to vary with temperature and pressure, Eq. (2.7) reduces to

$$\rho_f \frac{D\mathbf{u}_f}{Dt} = \rho_f \mathbf{g} - \nabla p + \mu \nabla^2 \mathbf{u}_f \quad (2.8)$$

The assumption that the flow is incompressible and that the viscosity does not change fully decouples the continuity and momentum equations from temperature. This allows the solution of Eqs. (2.4) and (2.8) without the need to solve for the temperature. The temperature can later be solved for separately from the fluid solution reducing the complexity of the problem.

The final conservation law, the Conservation of Energy, can be described by the first law of thermodynamics

$$dE_t = dQ + dW \quad (2.9)$$

where Q is the heat added and W is the work done on the system. The flowing particle is considered a moving system such that E_t includes the internal energy and all potential and kinetic energy of the system:

$$E_t = \rho_f \left(e + \frac{1}{2}U^2 - \mathbf{g} \cdot \mathbf{r} \right) \quad (2.10)$$

where e is the internal energy per unit mass and $\mathbf{r}$ is the displacement of the particle. Using an Eulerian point of view, Eq. (2.9) becomes

$$\frac{DE_t}{Dt} = \frac{DQ}{Dt} + \frac{DW}{Dt} \quad (2.11)$$

and applying this same point of view to Eq. (2.10) one gets

$$\frac{DE_t}{Dt} = \rho_f \left(\frac{De}{Dt} + U \frac{DU}{Dt} - \mathbf{g} \cdot \mathbf{u}_f \right) \quad (2.12)$$

Using Fourier's law to define the heat transfer to the element as $\mathbf{q} = -k\nabla T$ gives

$$\frac{DQ}{Dt} = -\text{div } \mathbf{q} = \text{div}(k\nabla T) \quad (2.13)$$

Similarly, the work done on each element is described by

$$\frac{DW}{Dt} = -\text{div } \mathbf{w} = \nabla \cdot (\mathbf{u}_f \cdot \tau_{ij}) \quad (2.14)$$

After careful rearrangement, the energy equation becomes

$$\rho_f \frac{Dh}{Dt} = \frac{Dp}{Dt} + \text{div}(k\nabla T) + \Phi \quad (2.15)$$

where h is the enthalpy defined as $h = e + \frac{e}{\rho}$ and Φ is known as the dissipation function and is defined as

$$\begin{aligned} \Phi = \tau'_{ij} \frac{\partial u_i}{\partial x_j} = \mu \left[2 \left(\frac{\partial u}{\partial x} \right)^2 + 2 \left(\frac{\partial v}{\partial y} \right)^2 + 2 \left(\frac{\partial w}{\partial z} \right)^2 + \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right)^2 + \right. \\ \left. \left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)^2 \right] + \lambda \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right)^2 \end{aligned} \quad (2.16)$$

Equations (2.4), (2.8) and (2.15) fully describe an incompressible, Newtonian fluid and will be used to model the fluid **within** a domain. Although these equations model the fluid within the boundaries of interest, they do not describe what the fluid is doing at the boundaries and thus the boundary conditions must be established in order to fully describe the fluid.

Fluid Domain Boundary Conditions

There are four types of boundary conditions that are represented in the present work: the wall, inlet, outlet and periodic flow conditions. These boundary conditions help to ensure that the problem remains physically accurate and bounds the problem to a domain that can be discretized for analysis.

The fluid along a wall must match what is traditionally called the no-slip wall condition. That is, the fluid velocity is equal to the wall velocity. This Dirichlet type boundary condition is defined as:

$$\mathbf{u}_{\text{fluid}} = \mathbf{u}_{\text{solid}} \quad (2.17)$$

For most cases, the velocity of the solid is zero such that Eq. (2.17) reduces to $\mathbf{u}_{\text{fluid}} = 0$ giving rise to the general no-slip condition term. This will not be the case when the FSI is implemented and the boundary condition will be described by Eq. (2.17).

The conservation of energy requires that there be either a Dirichlet or Neumann boundary condition for the temperature between the wall and the fluid. The Dirichlet boundary condition gives

$$T_{\text{fluid}} = T_{\text{wall}} \quad (2.18)$$

and the Neumann boundary condition gives

$$\left(k \frac{\partial T}{\partial n} \right)_{\text{fluid}} = q''_{\text{wall}} \quad (2.19)$$

The Dirichlet boundary condition ensures that there is not a temperature jump across the fluid-solid interface. The Neumann boundary condition ensures that the heat flux is transferred from the solid to the fluid without any losses.

The inlet conditions specify the inlet velocity and the temperature of the incoming fluid. The outlet condition is specified such that the pressure goes to zero and that there are no viscous stresses at the outlet such that

$$\left[\mu \left(\nabla \mathbf{u}_f + (\nabla \mathbf{u}_f)^T \right) \right] \cdot \mathbf{n} = 0 \quad (2.20)$$

where $\mathbf{n}$ is the vector normal to the outlet boundary.

Turbulence Model

Now that the governing equations have been established, it is worthwhile to examine the flow characteristics that occur at the conditions seen in the HFIR and other research reactors. In order to obtain sufficient cooling and to avoid incipient boiling, the flow rate between the fluid channels is quite high. The inlet flow rate for the HFIR, for example, is approximately 15 m/s through a channel gap of 50 mils. This high flow rate results in a Reynolds number of approximately 36,000 based upon

the hydraulic diameter, D_h , of the outer fuel element. The hydraulic diameter for a rectangular channel is defined by

$$D_h = \frac{2tw}{t + w} \quad (2.21)$$

where t represents the thickness of the channel and w represents width of the channel. Because the HFIR plates are of an involute shape, it is appropriate for the Reynolds number calculations to model the channel as a rectangular channel of thickness, t , and width, w . The arc length of the involute outer fuel element is 2.87 in (7.3 cm) and the inner fuel element is 3.189 in (8.1 cm) which can be attributed to the channel width. Because the channel width is much greater than the thickness, Eq. (2.21) can be reduced to twice the channel thickness such that:

$$D_h = 2t, \quad w \gg t \quad (2.22)$$

This high Reynolds number indicates that the flow between the channels is in the turbulent regime [64]. The unsteady nature of turbulence requires further manipulation of the continuity and Navier-Stokes equations (Eqs. (2.4) and (2.8)). For this dissertation, Reynolds Averaging will be applied to these equations [65]. The time-averaging equation, applied to the flow variable $f(\mathbf{x}, t)$, is defined by:

$$F_T(\mathbf{x}) = \lim_{T \rightarrow \infty} \frac{1}{T} \int_t^{t+T} f(\mathbf{x}, t) dt \quad (2.23)$$

In order to apply the time-averaging to the Navier-Stokes equations, it is best to put the convective term into the conservation form of

$$\rho_f \frac{D\mathbf{u}_f}{Dt} = \rho_f \frac{\partial \mathbf{u}_f}{\partial t} + \nabla \cdot (\rho_f \mathbf{u}_f \otimes \mathbf{u}_f) \quad (2.24)$$

The conservation form of the Navier-Stokes equations then becomes

$$\nabla \cdot (\rho \mathbf{u}_f) = 0 \quad (2.25)$$

$$\rho_f \frac{\partial \mathbf{u}_f}{\partial t} + \nabla \cdot (\rho_f \mathbf{u}_f \otimes \mathbf{u}_f) = -\nabla p + \mu \nabla^2 \mathbf{u}_f \quad (2.26)$$

where $\otimes$ is the tensor product.

Carrying out the Reynolds averaging on the conservation form of the Navier-Stokes equations results in fewer equations than unknowns and thus a multitude of closure models have been developed. For the present work, a modified version [66] of the standard two-equation k - ϵ model is used to model the turbulent flow [67]. The parameters for the k - ϵ turbulence model of this dissertation are provided in Table 2.1

Table 2.1: Constants used for the k - ϵ turbulence model for this dissertation.

Constant	Value
C_μ	0.09
$C_{\epsilon 1}$	1.44
$C_{\epsilon 2}$	1.92
σ_k	1.0
σ_ϵ	1.3

2.2 Structural Mechanics

The governing equations for the structural mechanics model follows the linear elastic model. The equation of motion is derived from Newton's Second law

$$\nabla \cdot \sigma + \mathbf{F}_s = \rho \frac{\partial^2 \mathbf{u}_s}{\partial t^2} \quad (2.27)$$

The strain-displacement equations are given by

$$\varepsilon = \frac{1}{2} \left[\nabla \mathbf{u}_s + (\nabla \mathbf{u}_s)^T \right] \quad (2.28)$$

The constitutive equation relates the unknown stresses and strains using Hooke's law for an elastic material

$$\sigma = C : \varepsilon \quad (2.29)$$

where C is the fourth order stiffness tensor. For an isotropic material, the tensor can be written in terms of the Lamé constants λ and μ

$$C_{ijkl} = \lambda \delta_{ij} \delta_{kl} + \mu (\delta_{ij} \delta_{jl} + \delta_{il} \delta_{jk}) \quad (2.30)$$

where

$$\lambda = \frac{E\nu}{(1+\nu)(1-2\nu)}, \quad \mu = \frac{E}{2(1+\nu)} \quad (2.31)$$

$\mathbf{A} : \mathbf{B} = A_{ij} B_{ij}$ is the inner product of two second order tensors. This formulation ensures that the material behaves as if in the linear elastic regime and assumes the material does not deform into the plastic regime.

Solid Domain Boundary Conditions

In order to fully describe the structural domain, the conditions at the boundary must be specified. For the present work, there are two structural boundaries: the fixed boundary and the fluid-solid boundary. The fixed boundary represents a boundary in which the displacement is held to zero such as the fixed end of a cantilevered beam and is represented by the following equation

$$\mathbf{u}_s = 0 \quad (2.32)$$

The fluid-solid interface boundary correlates the pressure and viscous stresses from the fluid to the stress in the structure:

$$\boldsymbol{\sigma} \cdot \mathbf{n} = \left[-p + \mu \left(\nabla \mathbf{u}_f + (\nabla \mathbf{u}_f)^T \right) \right] \cdot \mathbf{n} \quad (2.33)$$

The governing equations and boundary conditions are to be used to model the fluid-structure phenomena and the finite element approach used for discretization is described in the next section.

2.3 Finite Element Analysis: Discretization

The discretization method used for the governing equations in COMSOL is the Finite Element Analysis (FEA) method. This method breaks a domain up into a series of elements known as finite elements that can be either triangular or quadrilateral. Triangular elements are heavily utilized because of the ease in which complicated domains can be meshed as seen in Figure 2.1. As more triangular elements are included, the discretized domain begins to better represent the original domain.

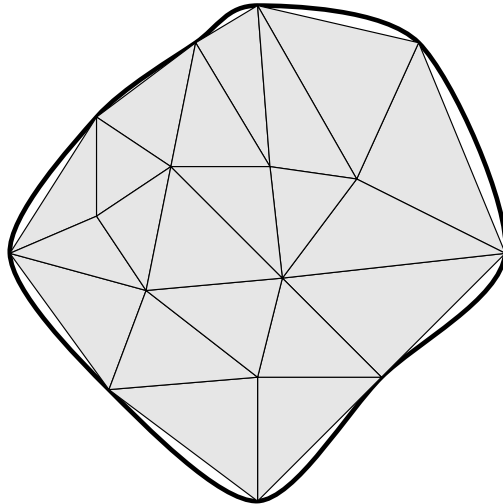


Figure 2.1: Arbitrary domain discretized into triangular finite elements.

The finite element model assumes that an equation can be approximated over a typical finite element denoted by the superscript e . Thus, for a quantity $U(x, y, z)$ the discretization assumes

$$U(x, y, z) \approx U^e(x, y, z) = \sum_{j=1}^n U_j^e \psi_j^e(x, y, z) \quad (2.34)$$

where $U^e(x, y, z)$ is the approximation of $U(x, y, z)$ over the element and U_j^e are the values of the function $U^e(x, y, z)$ at the nodes that make up the element. The approximation, or basis functions associated with the element are represented by ψ_j^e which, in most cases, are polynomials. As the polynomial order increases, the accuracy of the finite element discretization also increases and accordingly, so does the computational cost.

For fluid flow, the basis function for a triangular element is represented by the equation $Px + Py$ where Px is the basis function for the flow velocity and Py is the basis function for the pressure. For example, a triangular FEA representation of the fluid that uses a quadratic basis function for the flow variables and a linear basis function for the pressure variables would be denoted as $P2 + P1$. The Ladyzhenskaya-Babuska-Brezzi (LBB) condition requires that the velocity component basis functions be at least one order higher than the pressure basis function [68]. This condition can be bypassed by incorporating a smoothing technique known as the Galerkin-Least Squares (GLS) formulation which will allow the use of equal-order basis functions [69, 70]. The GLS formulation is utilized in the FEA discretization of COMSOL. The basis functions for the fluid domain in this dissertation will be $P1 + P1$; the $P1 + P1$ basis functions are accurate if a fine enough mesh is utilized [70].

For the structural domain, the FEA discretization will use a quadratic basis function for analysis. Because the structural domain is relatively small compared to the fluid domain for the majority of the simulations, the computational cost

associated with quadratic elements is justified. Some of the simulations will have large displacements in which a higher order basis function is beneficial.

Chapter 3

Cylinder in Cross-Flow

The cylinder in cross-flow is a fundamental fluids problem in which the physics allows for a stringent testing of the different aspects of a code. A stationary cylinder will naturally begin to shed vortices for Re greater than approximately 40 and will continue to shed into the turbulent regime. These shedding vortices are known as Kármán vortex streets and their occurrence is related to the Strouhal number defined as

$$St = fD/U \tag{3.1}$$

where f is the shedding frequency, D is the characteristic length, i.e., the cylinder diameter, and U is the free stream velocity. For flows with Re between 100 and 100,000, the non-dimensional Strouhal number is around 0.2 which means that the vortex shedding cycle is approximately the time it takes the free stream flow to move 5 cylinder diameters [63].

This phenomenon will occur for almost any bluff body for which $Re \geq 40$. The shedding vortices can cause significant vibrations in the structure. This vibration, if not carefully accounted for, can cause large oscillations and eventual failure of the structure such as that seen in 1940 with the Tacoma Narrows bridge [71]. This

shedding and subsequent oscillation has been studied at length experimentally [72–74] and numerically [75–82]. The progression for the modeling of the cylinder in cross-flow for this chapter will follow this outline:

3.1 - Stationary Cylinder

3.2 - Oscillating Cylinder: Prescribed Displacement

3.3 - Oscillating Cylinder: Free Displacement

3.4 - Stationary Cylinder with Moving Tail

This progression was chosen as a way to test the code incrementally. The complexity of both the flow solver and the structural solver increases with each section. Section 3.1 employs only the flow solver while holding the cylinder stationary and computing only the vortex shedding of the fluid. Sections 3.2 and 3.3 incorporate different moving cylinders: one type that follows a prescribed displacement and one that allows the cylinder to move freely in the transverse direction based upon the lift created by the shedding vortices, respectively. The last section, 3.4, incorporates a stationary cylinder with a long tail attached to the end. This simulation tests the ability of the mesh generator to handle large displacements of the tail caused by the shedding vortices.

3.1 Stationary Cylinder

The stationary cylinder in cross-flow is used to test and verify the capabilities of the flow solver. The solver should be able to capture the shedding of the vortices without any external manipulation from the user. The development of the Kármán vortex streets should occur naturally and progressively using a transient solver. Here, we consider a range of Reynolds numbers that result in laminar flow about a stationary cylinder.

3.1.1 Domain and Boundary Conditions

The 2-D domain for the stationary cylinder was chosen to be a large rectangular domain. The cylinder diameter is 1mm. The domain is 40 times the cylinder diameter in height and 40 times the cylinder diameter in length. In order to ensure enough resolution of the trailing vortices, the cylinder was offset in the flow direction with a larger section placed on the trailing edge of the domain. The top and bottom boundaries are taken to be periodic boundaries. The periodic boundary conditions are described by the following equations

$$\mathbf{u}_{\text{source}} = \mathbf{u}_{\text{destination}} \quad (3.2)$$

$$p_{\text{source}} = p_{\text{destination}} \quad (3.3)$$

where the source is the top boundary and the destination is the bottom boundary. The left boundary is an inlet with specified velocity based upon the Re number. The right boundary is a pressure boundary set to 0. The boundary around the cylinder is a no slip condition. Figure 3.1 provides the domain dimensions and boundary conditions used for the simulations. The working fluid for the simulation is taken to be incompressible water with the following constant properties at 293 K:

- Density: 1000 kg/m³
- Viscosity: 0.01 Pa s

Water was chosen because the working fluid for the majority of the simulations performed for analysis of the HFIR would be water.

A free triangular mesh was created for the simulation in order to handle the more complex geometry created by using a circular cylinder and a rectangular domain. This free triangular mesh posed a problem, which is made evident in Section 3.3, in which a non-uniform mesh is created at the trailing edge of the cylinder; this non-uniform mesh caused a spurious shift in the lift [83]. This non-uniformity was corrected by placing a horizontal line that divides the domain in half; this line is not used in any

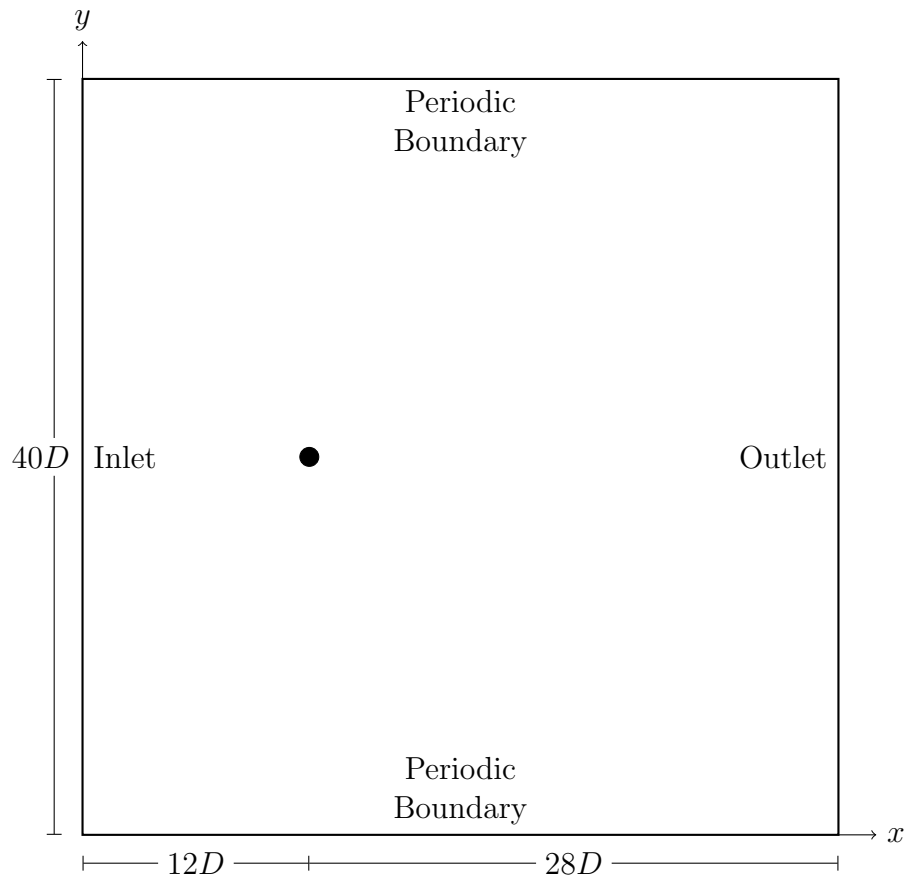


Figure 3.1: Domain for the 2-D cylinder simulations.

calculations but simply allows the creation of a symmetric mesh. The free mesh is created by first meshing the top half of the domain and then mirroring the mesh along the horizontal line giving a perfectly symmetric mesh and avoiding any discrepancies in results due to the grid asymmetry. Figure 3.2 provides the progression from the non-uniform mesh to the perfectly symmetric mesh. In order to capture the physics involved accurately, a boundary layer mesh was inserted with 8 layers on the boundary of the cylinder. The final mesh consisted of 29,558 mesh elements.

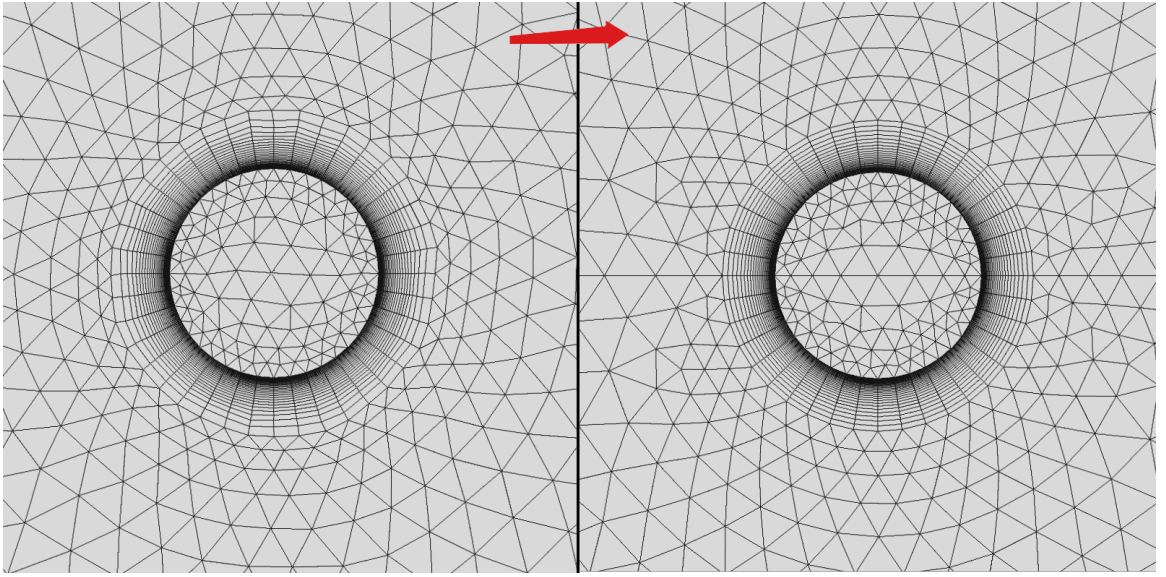


Figure 3.2: Mesh comparison for the 2-D Cylinder. Control of the mesh along the mid-plane of the cylinder ensures the lift around the cylinder is symmetric while using a free tetrahedral mesh.

3.1.2 Simulation and Results

The simulation for the stationary cylinder in cross-flow was performed using a fully implicit time domain solver. A steady-state solution was used as an initial condition

in order to reduce the computational costs by eliminating the start-up of the flow solver. The initial condition for the $Re = 100$ case is provided in Figure 3.3.

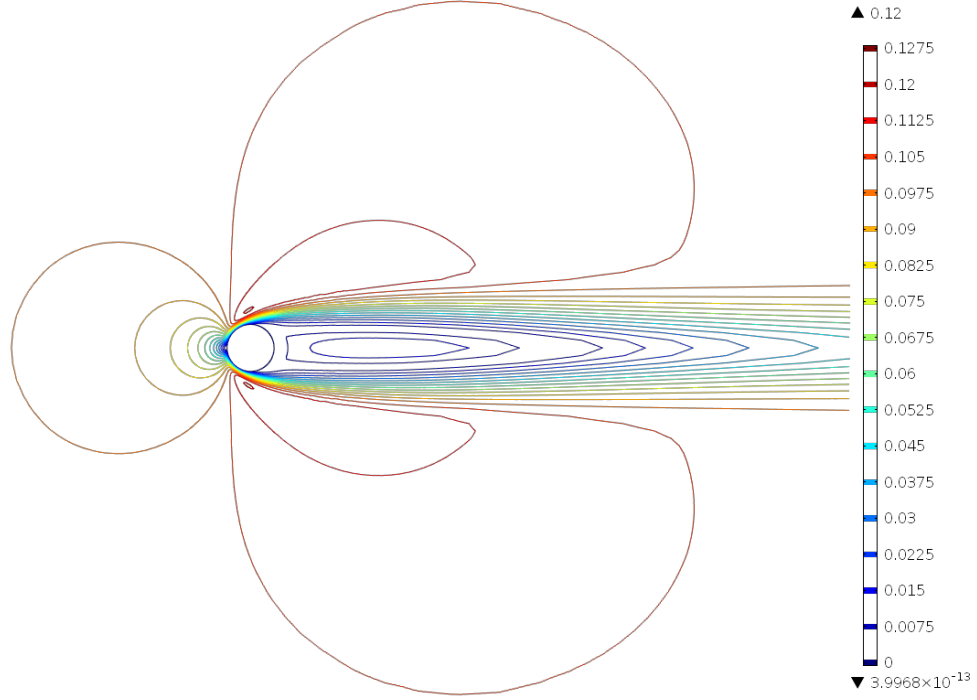


Figure 3.3: Initial condition using the steady-state solver for $Re = 100$ for the 2-D cylinder simulations. The working fluid is water and the velocity contours are represented in m/s.

The fully implicit solver is stable and lends itself to large time-step iterations [68]. When a steady-state solution with no periodic tendencies is desired, this greatly reduces the computational costs needed to obtain a steady-state solution. The cylinder in cross-flow, as stated earlier, should result in a time-accurate solution that converges into a periodic limit cycle. If the free-time step solver is implemented for the determination of the time-step used for the flow solver, certain physical phenomena will not be captured. For the cylinder in cross-flow with a steady-state initial condition, the time-step may be too large to allow the flow solver to capture

the instability that induces the vortex shedding. This phenomenon was seen with the present study in which the cylinder failed to produce vortex shedding.

For example, with $Re = 100$, the cylinder should shed within 10 physical seconds from the initial condition (here, physical seconds represents the time in the time domain as opposed to the computational time which is the interval it takes to run the code on a given machine and is dependent upon the number of cores and solving method used, i.e. direct or iterative method and is typically referred to as the ‘wall-clock time’). When the time-step was allowed to be set for any interval, the code would compute physical time-steps much too large to capture the vortex shedding. The solution would remain in steady-state and no shedding would be observed; the solution would be indistinguishable from the initial condition seen in Figure 3.3.

To correct this problem, a limit on the maximum time-step was imposed. This limit was taken to be $1/20$ of the approximate period of shedding for a particular Reynolds number. Again, using $Re = 100$ as an example and the other physical properties described above, the shedding frequency can be estimated from Eq. 3.1 with $St \approx 0.165$ to be approximately 16.5 Hz. Using the estimated frequency, a maximum time-step of approximately 0.001 51 s produces the desired vortex shedding. This method was employed for all of the simulations, allowing the code to properly capture the flow physics.

In order to verify the vortex shedding capabilities of the flow solver, a variety of Reynolds numbers were computed. Because the phenomenon is not observed for all Reynolds numbers, a simulation with $Re = 20$ was performed to ensure that the code could capture shedding as well as non-shedding cases. To establish a good basis, the following Reynolds numbers were considered: 20, 75, 100, 150 and 200. This range ensures laminar flow while allowing the analysis to remain two-dimensional. A sample of the solution obtained once the cylinder begins to shed vortices is provided in Figure 3.4 for $Re = 100$.

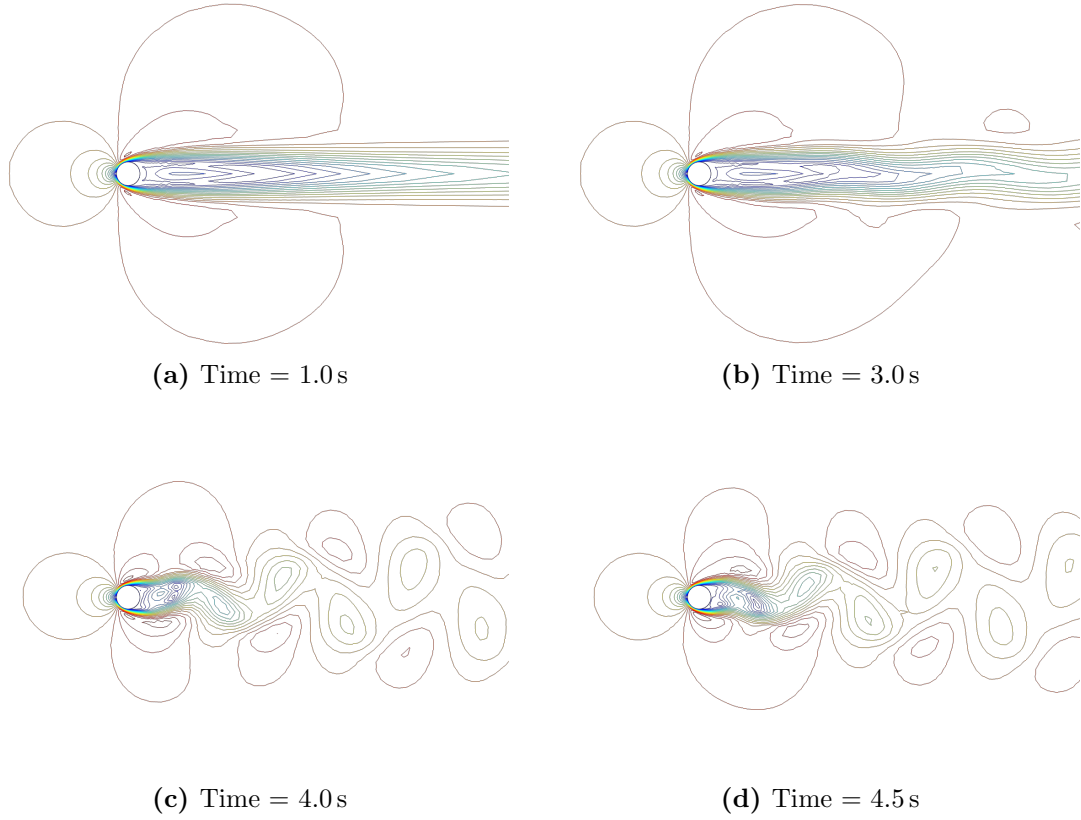


Figure 3.4: Set of time-domain solutions for $Re = 100$ for the stationary cylinder in cross-flow. The working fluid is water.

The fundamental way to measure the shedding frequency is to observe the coefficient of lift, C_l , defined by:

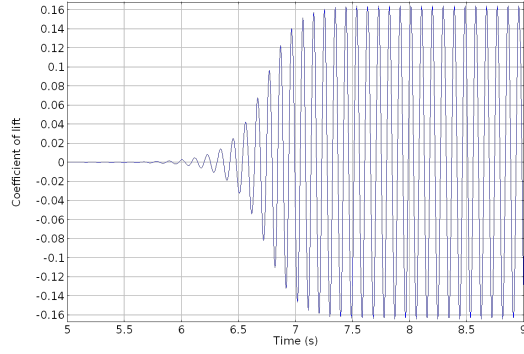
$$C_l = \frac{l}{1/2\rho_f U_\infty^2 D} \quad (3.4)$$

where l represents the lift on the cylinder per unit length, ρ_f represents the free-stream fluid density, U_∞ represents the free-stream fluid velocity and D is the cylinder diameter. Observation of the lift in the time domain shows that the vortex shedding has gone into a limit cycle (Figure 3.5). A Fast-Fourier Transform (FFT) [84] is employed to determine the dominant frequencies of the solution (Figure 3.6).

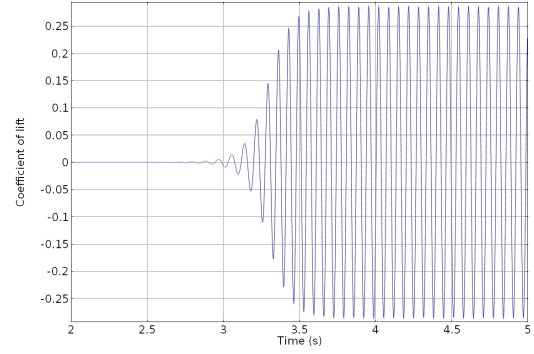
The results of Figure 3.6 are used to obtain the shedding frequency for each case. Williamson [85] performed experiments for the cylinder in cross-flow and presented his results in graphical form. For a direct comparison of our solution with the experimental data, the open source program called Engauge Digitizer [86] was used to extract the data from the paper. The extracted experimental data is compared to the simulations in Table 3.1 and Figure 3.7.

Table 3.1: Shedding frequencies of the stationary cylinder in cross-flow compared to experimental results of Williamson [85].

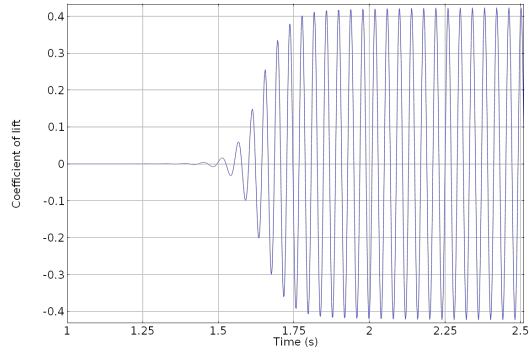
Re	Experimental		Numerical	
	St	Freq, Hz	St	Freq, Hz
20	0.0000	0.00	0.0000	0.00
75	0.1424	10.70	0.1398	10.50
100	0.1600	16.03	0.1517	15.20
150	0.1800	27.05	0.1642	24.68
200	0.1825	36.57	0.1723	34.52



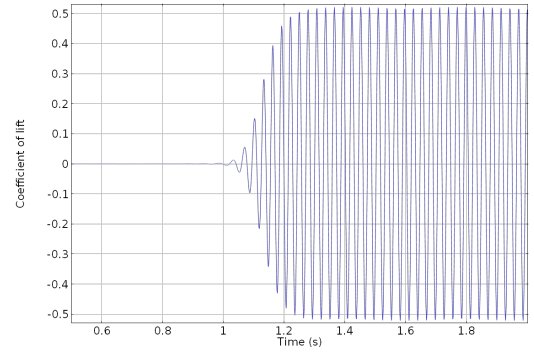
(a) $Re = 75$



(b) $Re = 100$

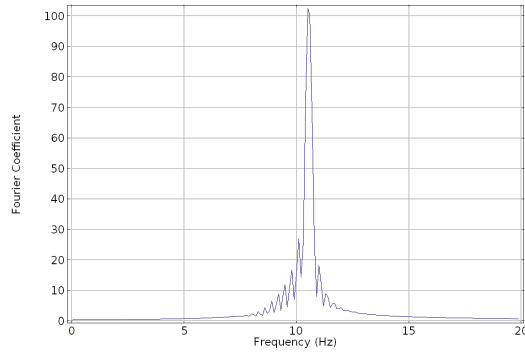


(c) $Re = 150$

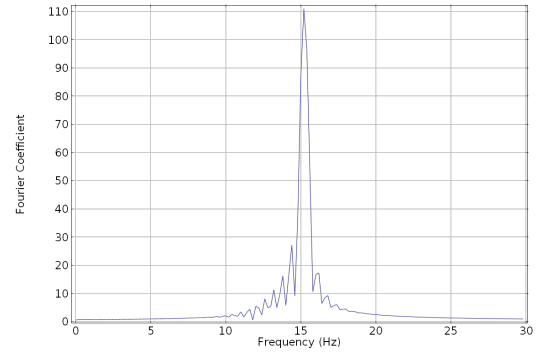


(d) $Re = 200$

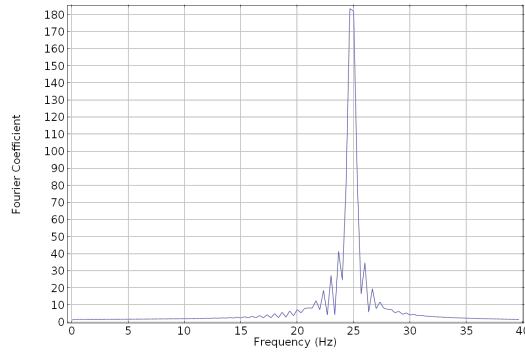
Figure 3.5: Time accurate C_l for the stationary cylinder in cross-flow. The working fluid is water.



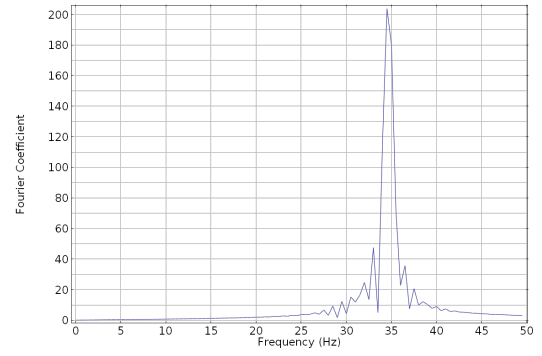
(a) $Re = 75$



(b) $Re = 100$



(c) $Re = 150$



(d) $Re = 200$

Figure 3.6: FFT of the C_l for the stationary cylinder. The working fluid is water.

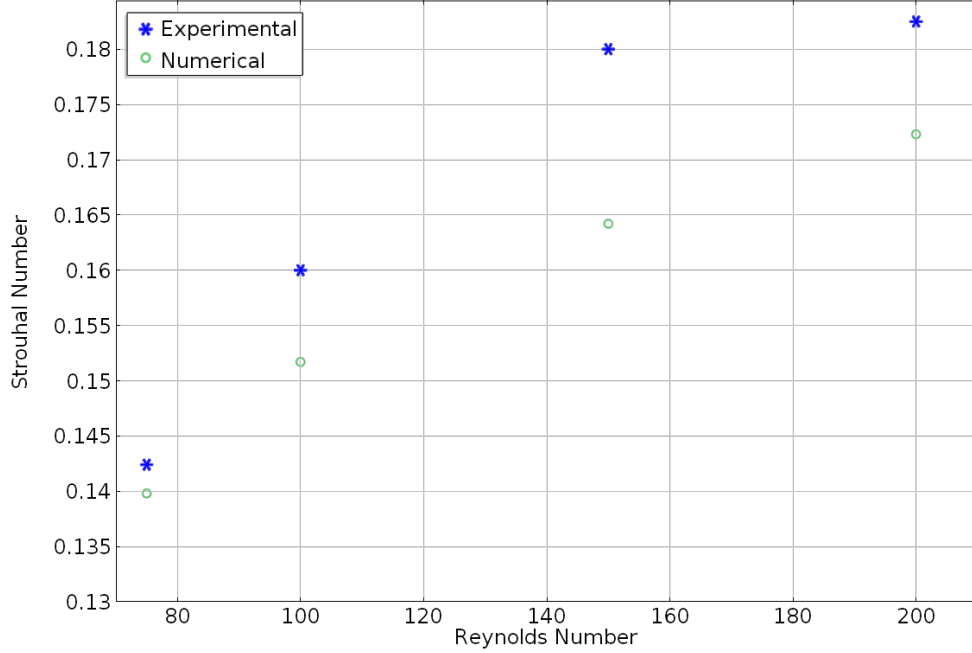


Figure 3.7: Strouhal number of the stationary cylinder simulations compared to the experimental results by Williamson [85].

3.1.3 Discussion

The stationary cylinder in cross-flow provides a context to test real-world instabilities with a numerical code. The simulations show that the code is capable of reproducing this naturally occurring phenomena when careful consideration of the numerical tools is employed. By limiting the transient solver time-step to $1/20$ the time it takes to shed one vortex, the code is able to capture the expected Kármán vortex streets close to the experimentally obtained Strouhal number.

Because the small time-step could result in large data files, it was originally proposed to record the data at intervals larger than the $1/20$ time-step but smaller than the shedding interval. This resulted in very manageable file sizes, but it was quickly discovered that the interval, as with all digital recording, could induce unwanted aliasing [87]. This aliasing is evident as a smaller frequency on top of the main frequency as seen in Figure 3.8. In order to alleviate the aliasing, the

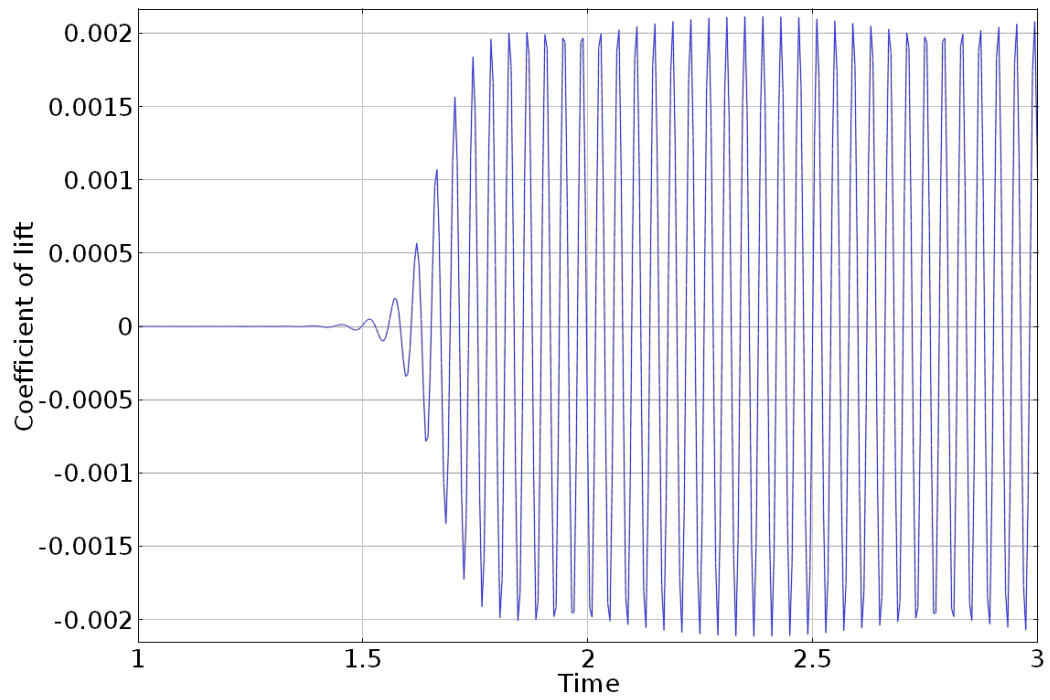


Figure 3.8: C_l for the $Re = 150$ case with aliasing present due to large recording intervals. Limiting the maximum time-step alleviates the aliasing seen in this example.

sampling rate was configured to be the same as the maximum time-step allowed for each simulation. This change eliminated the aliasing problem from the solution and aliasing effects were considered for all other time-dependent solutions.

The code has proven to effectively capture the shedding vortices for a 2-D cylinder in cross-flow. Table 3.1 shows that the numerical simulation provides reasonable results compared to the experiment. Using the stationary cylinder as a proving ground and stepping stone for analysis, it is now appropriate to move on to more complicated cases.

3.2 Oscillating Cylinder: Prescribed Displacement

The next logical step in testing the FSI capabilities of the code is to allow the cylinder to move. In this section, the cylinder movement will follow a prescribed periodic displacement that is not affected by the flow of the fluid and its subsequent shedding vortices. This will allow the testing of both the mesh deformation capability and the fluid solver for a moving cylinder. The cylinder will be allowed to oscillate in the transverse direction of the free-stream flow velocity. This case was also considered experimentally by Koopman [73] for Reynolds numbers of 100 and 200.

The vortex streets that are shed from the oscillating cylinder can fall into two categories: lock-on and non lock-on. In the lock-on region, the vortices are shed at the same frequency of the oscillating cylinder. When one investigates the coefficient of lift, there is only one principle frequency present. For the non lock-on case, the cylinder sheds vortices at its ‘natural’ frequency based upon the Strouhal number for the flow conditions. This will result in two principle frequencies being present in the coefficient of lift calculations (the cylinder oscillation and the ‘natural’ vortex shedding). A representation of the cylinder lock-on and non lock-on regions is presented in Figure 3.9.

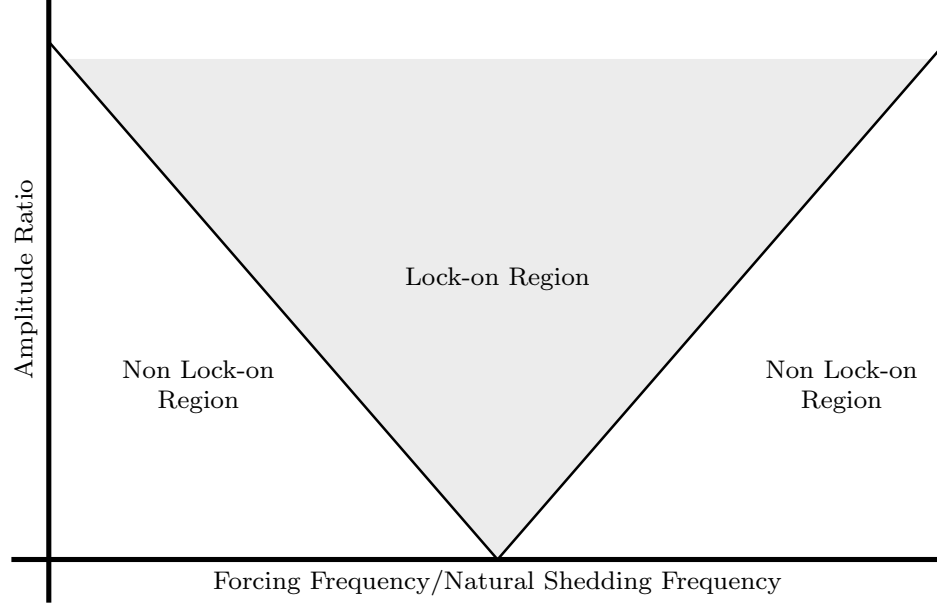


Figure 3.9: Graphical Representation of the lock-on region for the cylinder in cross flow with prescribed displacement.

3.2.1 Domain and Boundary Conditions

The domain for the oscillating cylinder with prescribed displacement is the same as that used for the stationary cylinder. The only boundary condition that is different from the one used for the stationary cylinder is the solid moving boundary around the cylinder. The boundary will now follow the general wall boundary condition described in Eq. (2.17) such that the fluid velocity is equal to the velocity of the wall such that $\mathbf{u}_{\text{fluid}} = \mathbf{u}_{\text{solid}}$.

The cylinder will follow a periodic sinusoidal motion normal to the free-stream velocity. The maximum amplitude and frequency will vary depending on the particular case being examined during the simulation. The equation of motion for the cylinder is given by:

$$\hat{j} \cdot \mathbf{u}_{\text{solid}} = h \sin(\omega t) \quad (3.5)$$

where $\hat{j}$ is the unit vector in the direction transverse to the inlet flow, h is the maximum cylinder amplitude and ω is the oscillation frequency. The baseline mesh for this case is the same as that used in Section 3.1 for the stationary cylinder.

Now that movement is to take place during the simulation of the flow around the cylinder, it is necessary to allow the mesh to deform and move with the cylinder. In COMSOL, there are 3 different types of mesh smoothing techniques:

1. Laplace
2. Winslow
3. Hyperelastic

The Laplace technique, first described by Buell and Bush [88] for a structured mesh and refined by Field [89] for a free triangular mesh, applies the Laplace operator on the computational coordinates (ξ, η) in the physical space (X, Y) . When the Laplace equation (3.6) is satisfied, a smooth distribution of the computational coordinates is present.

$$\begin{aligned}\nabla^2 \xi &= \frac{\partial^2 \xi}{\partial x^2} + \frac{\partial^2 \xi}{\partial y^2} = 0 \\ \nabla^2 \eta &= \frac{\partial^2 \eta}{\partial x^2} + \frac{\partial^2 \eta}{\partial y^2} = 0\end{aligned}\tag{3.6}$$

The Laplace equations should satisfy the max-min conditions such that parameters inside the domain will not exceed the boundary of the domain which ensures that the grid lines will not cross resulting in inverted meshes. In Eq. (3.6), the known variables are (ξ, η) and the unknown variables are (x, y) . Applying the Laplacian of Eq. (3.6) to the computational space results in the Laplace smoothing equation:

$$\begin{aligned}\nabla^2 x &= \frac{\partial^2 x}{\partial \xi^2} + \frac{\partial^2 x}{\partial \eta^2} = 0 \\ \nabla^2 y &= \frac{\partial^2 y}{\partial \xi^2} + \frac{\partial^2 y}{\partial \eta^2} = 0\end{aligned}\tag{3.7}$$

Although Eq. (3.7) satisfies the max-min condition for a structured mesh, this is not always the case for an unstructured mesh where some of the elements may be inverted, resulting in negative cell volumes.

To overcome the limitations of the Laplace smoothing technique, Winslow [90] proposed a transformation resulting in the following set of equations:

$$\begin{aligned}
\alpha \frac{\partial^2 x}{\partial \xi^2} - 2\beta \frac{\partial^2 x}{\partial \xi \partial \eta} + \gamma \frac{\partial^2 x}{\partial \eta^2} &= 0 \\
\alpha \frac{\partial^2 y}{\partial \xi^2} - 2\beta \frac{\partial^2 y}{\partial \xi \partial \eta} + \gamma \frac{\partial^2 y}{\partial \eta^2} &= 0 \\
\alpha &= \left(\frac{\partial x}{\partial \eta} \right)^2 + \left(\frac{\partial y}{\partial \eta} \right)^2 \\
\beta &= \frac{\partial x}{\partial \xi} \frac{\partial x}{\partial \eta} + \frac{\partial y}{\partial \xi} \frac{\partial y}{\partial \eta} \\
\gamma &= \left(\frac{\partial x}{\partial \xi} \right)^2 + \left(\frac{\partial y}{\partial \xi} \right)^2
\end{aligned} \tag{3.8}$$

This set of equations, though more complex, can greatly improve the accuracy and may alleviate the problems of inverted meshes found using the Laplace smoothing equations [91].

The last mesh smoothing technique available in the code is a hyperelastic technique inspired by neo-Hookean materials [92–94]. The technique searches for the minimum of the mesh deformation energy in which the energy is given by:

$$W = \int_{\Omega} \frac{\mu}{2} (I_1 - 3) + \frac{\kappa}{2} (J - 1)^2 dV \tag{3.9}$$

where μ is the artificial shear modulus and κ is the artificial bulk modulus. The invariants J and I_1 are given by:

$$\begin{aligned}
J &= \det(\nabla_x x) \\
I_1 &= J^{-2/3} \text{tr} \left((\nabla_x x)^T \nabla_x x \right)
\end{aligned}$$

In the above definitions, $\det(A)$ represents the determinant of the matrix A and $\text{tr}(A)$ represents the trace of the matrix A . The gradient operator denoted with the subscript ‘ X ’ represents the gradient with respect to the physical coordinates (X, Y) .

Each of the three mesh smoothing techniques have their benefits and trade-offs. The fastest and least computationally expensive technique would be the Laplace smoothing. The most robust of the three techniques is the hyperelastic smoothing technique. Because of the large mesh displacements possible in the upcoming simulations, the Laplace smoothing technique will not be used. In order to retain some accuracy, to prevent inverted meshes, and to obtain solutions in a reasonable amount of time, the Winslow smoothing technique will be used for all of the moving mesh simulations of this dissertation. Further analysis of the three mesh smoothing techniques is explored in Appendix A.

3.2.2 Simulation and Results

The issues seen with the stationary cylinder and the use of the implicit solvers is not observed with the moving cylinder because the simulation does not have to rely on a natural instability to begin to shed. Although any time-stepping scheme may be used, the issue of aliasing could still be present such that the restriction on the time-step of $1/20$ of the predicted frequency will still be used. There will be 35 cases run for the prescribed cylinder motion with a Reynolds number of 100. The cases were chosen in order to span across the lock-on and non lock-on regions. The frequency ratio, f/f_n , spans from 0.8 to 1.2 where f_n is the natural vortex shedding frequency and the amplitude ratio, h/D , spans from 0.05 to 0.35. The results are presented in Figure 3.10 where the plus (+) symbols represent the case in which lock-on occurs and the cross (×) symbols represent the case in which no lock-on occurs.

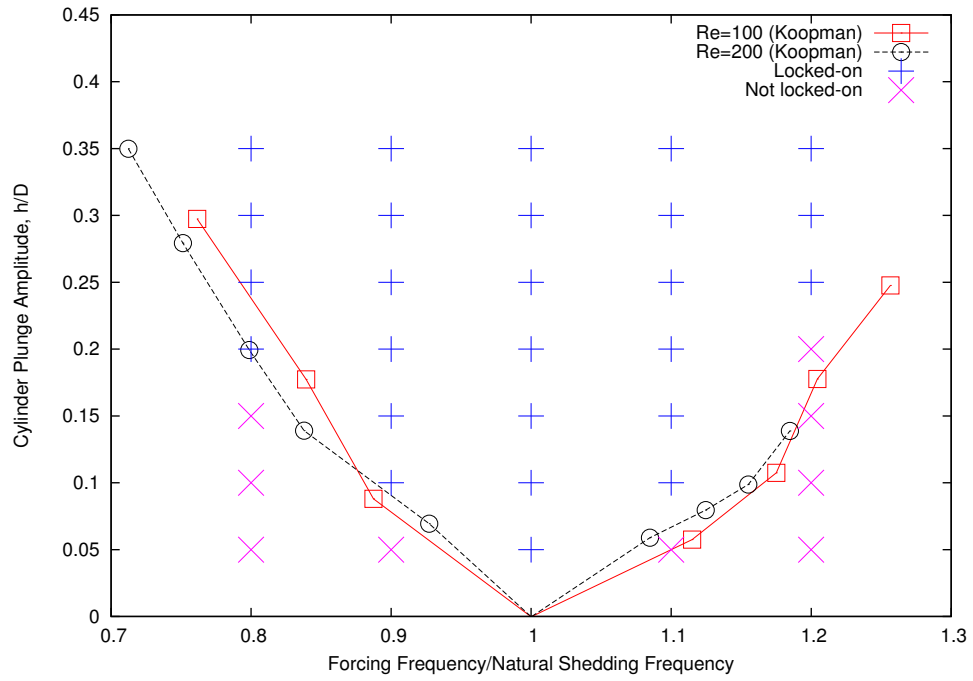


Figure 3.10: The cylinder lock-on simulations compared to the experiment by Koopman. The plus symbols (+) represent cases in which lock-on was achieved and the cross symbols (x) represent the cases in which no lock-on was present.

3.2.3 Discussion

The oscillating cylinder with prescribed displacement follows the experimental setup quite well. There is one simulation that does not follow the experimental results presented by Koopman at a plunge amplitude of 0.2 and a frequency ratio of 1.2. This ‘extreme’ case is an outlier in all of the other lock-on cases, and based upon the success of the other simulations does not negate the solver. For instance, the simulations around the ‘fork’ of the experimental results follow the lock-on/non lock-on progression as expected. The next step in establishing the validity of the FSI code is to allow the cylinder to move freely based upon the lift created by the shedding vortices.

3.3 Oscillating Cylinder: Free Displacement

Following the progression described previously, the cylinder will now be allowed to move freely based upon the lift created by the shedding vortices. This oscillation will be restricted to movement only in the transverse direction to the free-stream inlet velocity. The movement for the 2-D cylinder will be described as a spring-mass-damper system. The cylinder with free displacement lends itself to a full FSI study in 3-D utilizing the coupling of the Navier-Stokes equations with the structural mechanics equations. This progression was previously done in other simulations [77].

3.3.1 Domain and Boundary Conditions

As with the previous two sections, the domain for the free oscillating cylinder is a rectangle with the same dimensions as presented in Figure 3.1. The inlet, outlet and periodic boundary conditions remain unchanged from the previous two sections. In this section however, the cylinder is displaced not based upon a prescribed motion known *a priori* but based upon the periodic lift created by the oscillating vortices. For the 2-D simulation, the cylinder displacement is determined from the equation of

motion for a spring-mass-damper system:

$$m\ddot{x} + c\dot{x} + kx = f(t) \quad (3.10)$$

where the forcing function, $f(t)$, is determined by the lift on the cylinder. Figure 3.11 provides a visual representation of the mass-spring-damper assumption used for the 2-D simulation. The spring constant, k , is determined using beam theory for a

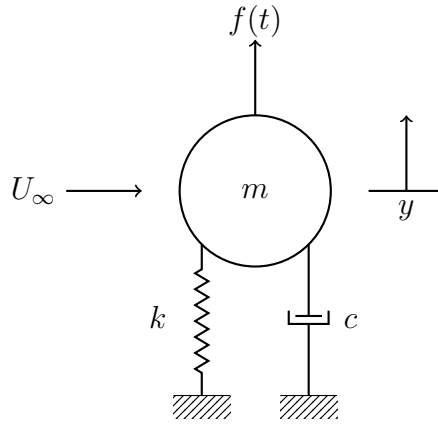


Figure 3.11: Visualization of the spring-mass-damper model used for the 2-D cylinder in cross-flow simulations.

cantilevered beam:

$$\delta_l = \frac{FL^3}{3EI} \quad (3.11)$$

where δ_l is the deflection at the end of the beam, F is the force at the end of the beam, L is the length of the beam, E is the Modulus of Elasticity of the beam and I is the area moment of inertia of the beam [95]. Rearranging Eq. (3.11) such that $F/\delta_l = k$ yields the effective stiffness of the beam:

$$k_{\text{effective}} = F/\delta_l = \frac{3EI}{L^3} \quad (3.12)$$

Assuming that there is no internal damping yields $c = 0$ and the mass is assumed to be the total mass of the cylinder such that $m = \rho_{\text{cyl}} \cdot V_{\text{cyl}}$. Equation (3.10) is a single degree of freedom (SOF) ordinary differential equation (ODE) which will be solved using an ODE solver.

The 3-D case is the first test of the coupling of the structural mechanics and fluid mechanics to form a Fluid-Structure Interaction problem. The 3-D domain has an identical setup to the 2-D domain except that the domain is ‘swept’ so that a cylinder is created. The boundary conditions are the same for the fluid domain except that the new walls created by the ‘sweeping’ of the domain are held as slip boundaries such that

$$\mathbf{u}_f \cdot \mathbf{n} = 0 \quad (3.13)$$

$$\mathbf{K} - (\mathbf{K} \cdot \mathbf{n}) \mathbf{n} = 0 \quad (3.14)$$

where $\mathbf{K} = \left[\mu \left(\nabla \mathbf{u}_f + (\nabla \mathbf{u}_f)^T \right) \right] \mathbf{n}$. During the sweep, 4 cuts are made to create a 3-D simulation. This limited number of cuts is utilized in an effort to speed up computational times; this is also made possible because of the low Reynolds number of the flow and the slip boundary conditions along the two specified walls of the domain. This results in a mesh consisting of 118 232 points.

3.3.2 Simulation and Results

The displacement of the cylinder should match the frequency of the shedding vortices. For completeness, two simulations will be performed to ensure that both are excited at the natural shedding frequency. The simulations will use inlet conditions such that the Reynolds number is 100 and 200. The working fluid is again chosen to be water with the properties previously described in Section 3.1. The cylinder properties, taken to be steel, are as follows: $\rho = 7800 \text{ kg/m}^3$, $E = 70 \text{ GPa}$, $\nu = 0.35$, and $L = 0.01 \text{ m}$. These parameters result in a cylinder with an area moment of inertia

of $I = 4.909 \times 10^{-14} \text{ m}^4$ and an effective spring stiffness of $k_{\text{effective}} = 10\,308 \text{ N/m}$. As with the previous two sections, a steady-state solution is used as the initial condition for the transient simulation.

The FSI problem (in both 2-D and 3-D) can be solved numerically in three different ways: fully-coupled, segregated, and one-way coupled. The fully-coupled method, as the name implies, solves for all of the physics (fluid and structure) in the same matrix. This method is the most expensive yet most robust solution technique. The solver iterates on the fully-coupled matrix until the convergence criterion is met. Figure 3.12 provides a flow chart for the visualization of the fully-coupled solver.

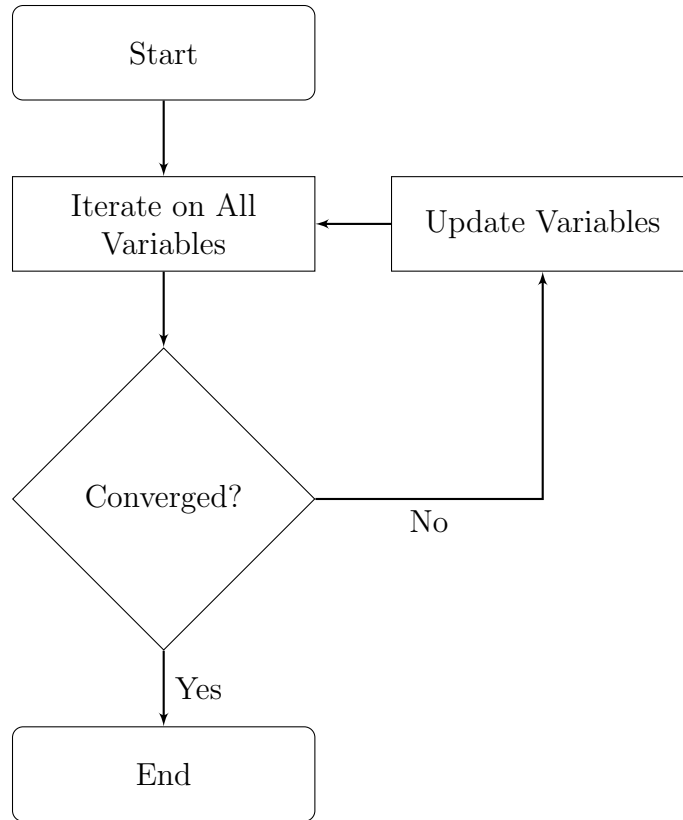


Figure 3.12: Flow chart for the fully-coupled solver.

The segregated solver allows the dependent variables to be solved independently of each other. For the FSI problem, the most logical segregation is the separation of the fluid mechanics and the solid mechanics. First, the fluid variables are iterated upon until a stopping criteria is met (typically a set number of iterations), then the fluid solution is used to establish the boundary conditions for the structural model. The solver then computes the structural variables. The overall convergence is checked; if the convergence criteria is not met, the boundaries for the fluid domain are updated and the process continues. Figure 3.13 shows the progression of the segregated solver. One of the strongest benefits of the segregated solver is the smaller memory requirements needed. Because the structural mechanics is solved second, the structure lags by one outer iteration and, depending upon how coupled the fluid and structure are, this could cause stability issues.

The one-way coupled solution technique fully decouples the fluid and structural variables. First, the fluid domain is fully solved for using the initial conditions of the structural domain for its boundary conditions. Once the fluid variables are fully converged, they are used to establish the boundary conditions for the structural mechanics solver. The structural variables are computed without updating the fluid solution and the process is completed. The flow chart for the one-way coupled solver is provided in Figure 3.14. This solution technique is the fastest of the three solvers discussed. If the fluid and structure are not strongly coupled, then this technique works very well. For example, a very stiff structure in which the flow induced displacement is very small and does not greatly affect the characteristics of the flow is a very good candidate for the one-way coupled solver.

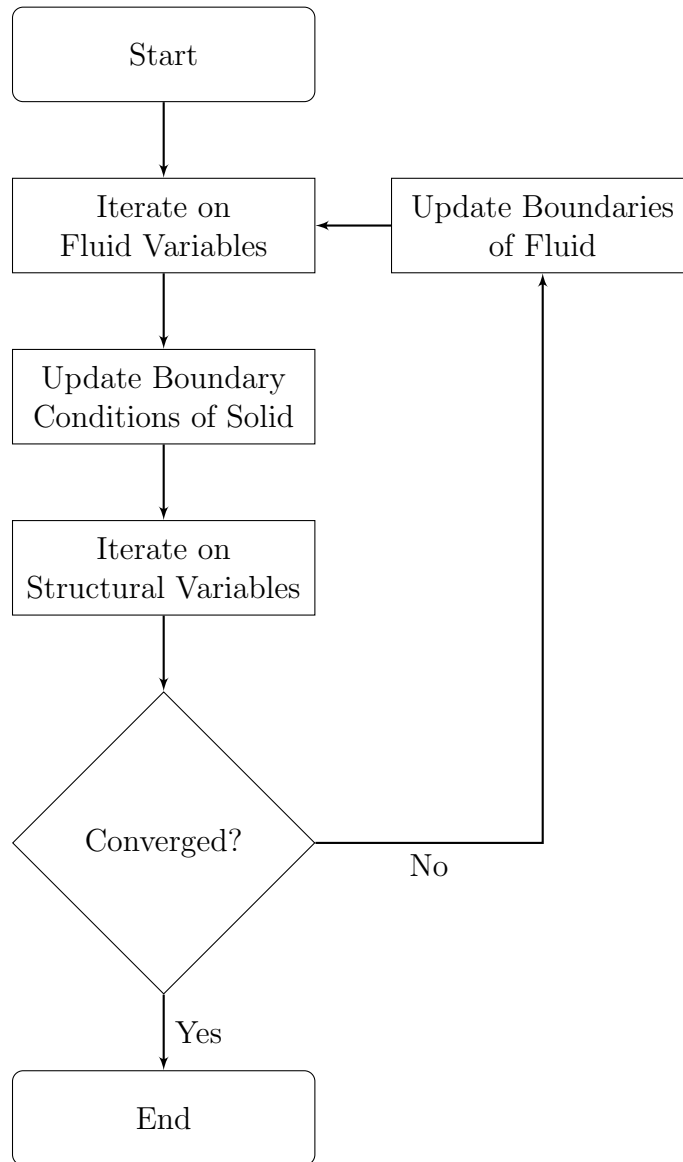


Figure 3.13: Flow chart for the segregated solver.

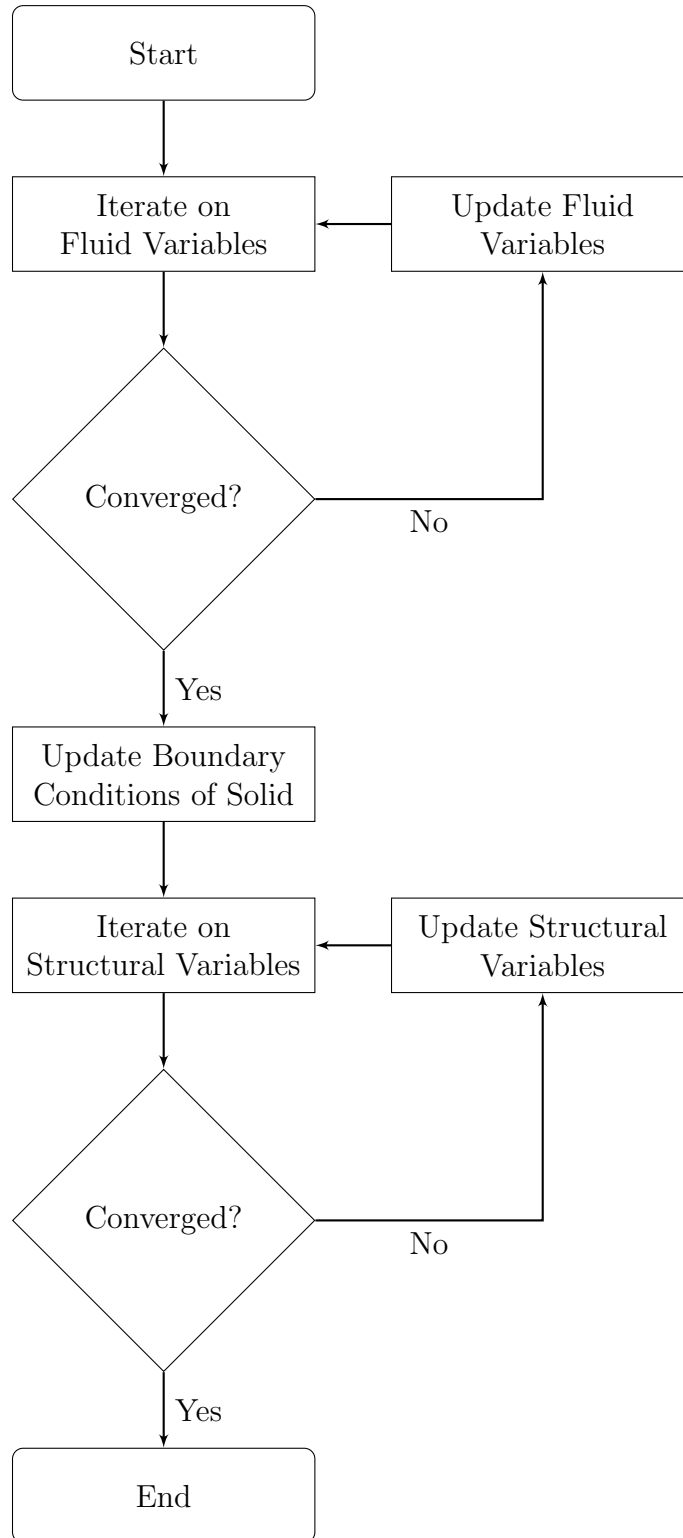


Figure 3.14: Flow chart for the one-way coupled solver.

2-D Case

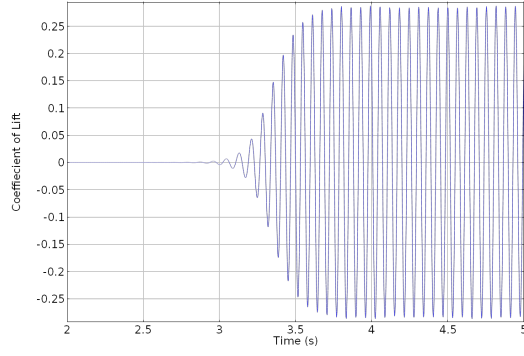
For the 2-D case, all three solvers will be explored. For the transient cases, the fully-coupled and segregated solvers work similarly in that for each time-step, the solver is run until convergence before the next time-step is computed. For the one-way coupled solver, the transient fluid solution is computed and then the transient structural solution is computed using the previously computed fluid solution as the boundary condition at each time-step. The one-way coupled solver solves the fluid solution to completion and then uses the fluid solution to obtain the transient structural solution. The results for the first case, the fully-coupled solver, are provided in Figure 3.15. The frequency of the coefficient of lift and displacement as well as the solution time are provided in Table 3.2. A solution was not obtained for the segregated solver because of instabilities created with the segregation process.

Table 3.2: Results for the 2-D simulation for the free displacement cylinder. The frequencies were obtained using the FFT. The simulations were performed on a 12 core machine with 192GB RAM.

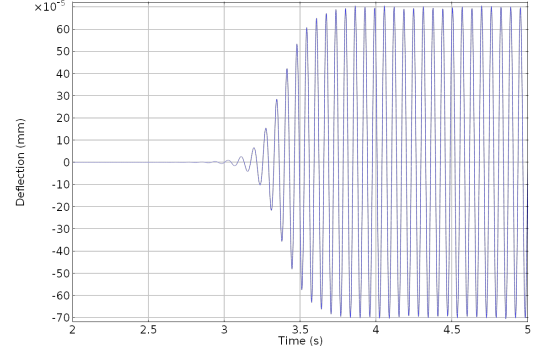
Re		C_l , Hz	Displacement, Hz	Max. Disp., mm	Time, s
100	Fully-Coupled	15.67	15.67	69.5×10^{-5}	10 107
	Segregated	—	—	—	—
	One-Way	15.2	15.2	5.8×10^{-3}	890
200	Fully-Coupled	37.5	37.5	13.9×10^{-5}	6888
	Segregated	—	—	—	—
	One-Way	34.2	34.2	1.0×10^{-3}	746

3-D Case

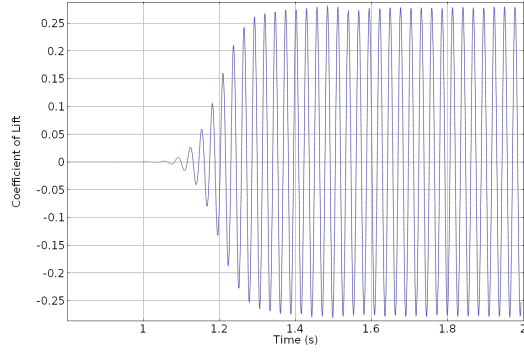
The 3-D simulation, as previously described, is a simple extrusion of the 2-D setup. This simulation is run using the fully-coupled solver in order to retain the interaction between both the cylinder and the fluid. The cylinder is modeled as a cantilevered



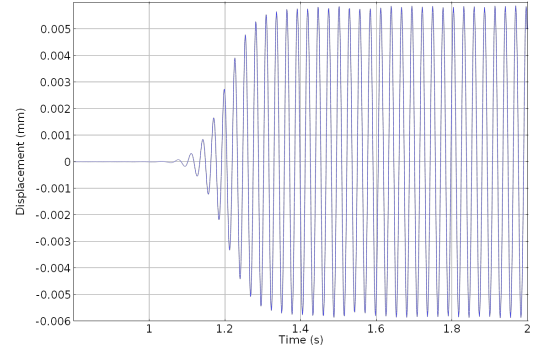
(a) $Re = 100 C_l$



(b) $Re = 100$ Displacement



(c) $Re = 200 C_l$



(d) $Re = 200$ Displacement

Figure 3.15: Displacement and C_l of the fully coupled 2-D free displacement cylinder. The working fluid is water.

beam with a fixed boundary on one side and a free boundary on the other. The results for the coefficient of lift and cylinder displacement are provided in Figure 3.16.

The fully-coupled, 3-D simulation, as expected, took a much longer time to complete than the 2-D cases. The cases took 68,743 and 46,303 seconds for the Re 100 and 200 cases, respectively. The oscillating frequency and displacement frequency matched for both cases at 15 Hz for $Re = 100$ and 34 Hz for $Re = 200$.

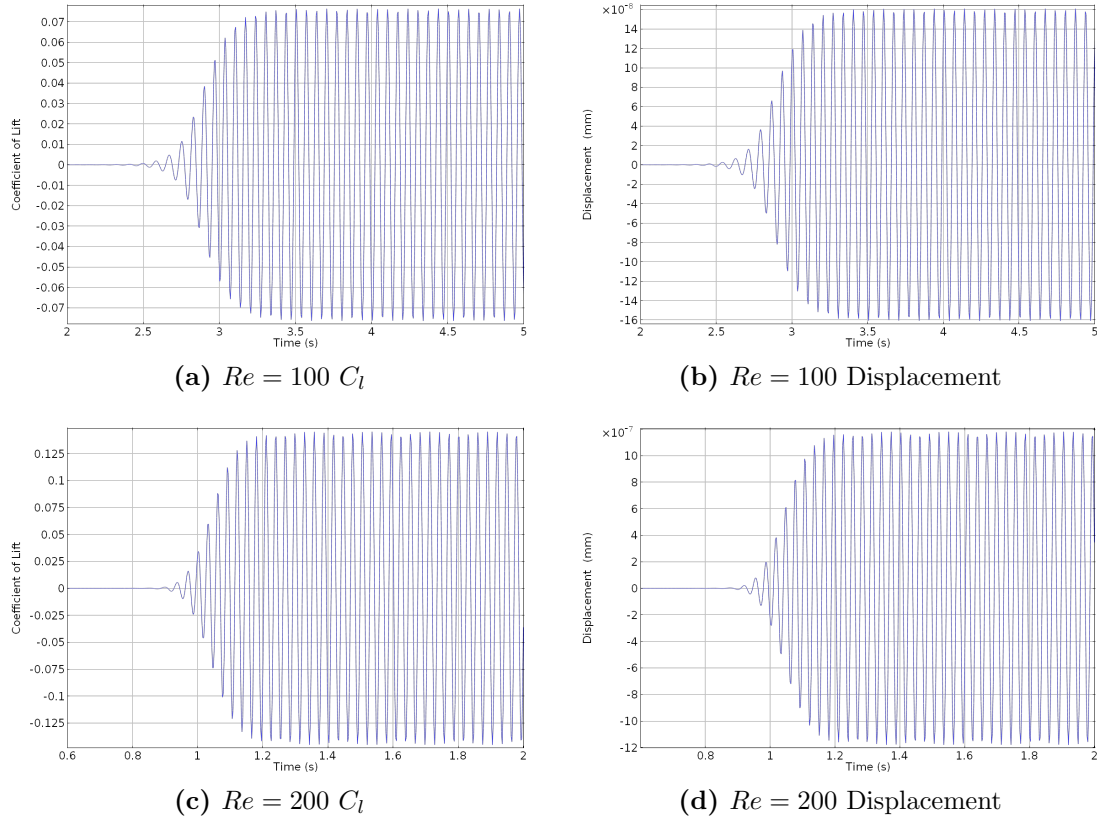


Figure 3.16: Solution of the fully coupled 3-D free displacement cylinder. The results are taken at the free end of the cylinder.

3.3.3 Discussion

The oscillating cylinder with free displacement provides insight into the capabilities of the FSI solver to capture the physics of the phenomenon. The exploration of the 3 solving techniques available in the code for the 2-D case allows for an assessment of the applicability of the methods for the FSI cases. Looking at the results in Table 3.2, it can be seen that the fastest solution can be obtained using the one-way coupling. While the one-way coupled solver produced oscillating frequencies similar to the fully-coupled solver, the maximum deflections were larger than the fully-coupled case. The segregated solver was not able to capture the physics correctly resulting in an unstable solution in which the case would diverge.

The 2-D, free displacement case revealed the need for the perfectly symmetric mesh described in Section 3.1. The need for this new meshing technique was discovered by observing the displacement of the cylinder once it reached its limit cycle. Assuming a perfectly round cylinder, the displacement should oscillate around the original position of the cylinder but when the mesh was not symmetric, the cylinder would initially drift up and begin to oscillate about another position which can be seen in Figure 3.17.

The 3-D case, which included the linear-elastic structural model, was able to capture the physics as well. For each case, the maximum displacement was smaller than the 2-D cases. This can be attributed to the assumptions made for the spring-mass-damper system: all of the mass of the cylinder is at the end, increasing its momentum and there is no internal damping to decrease the displacement for a given input force. Taking these assumptions into account, the displacements obtained from the simulations are concluded to be reasonable.

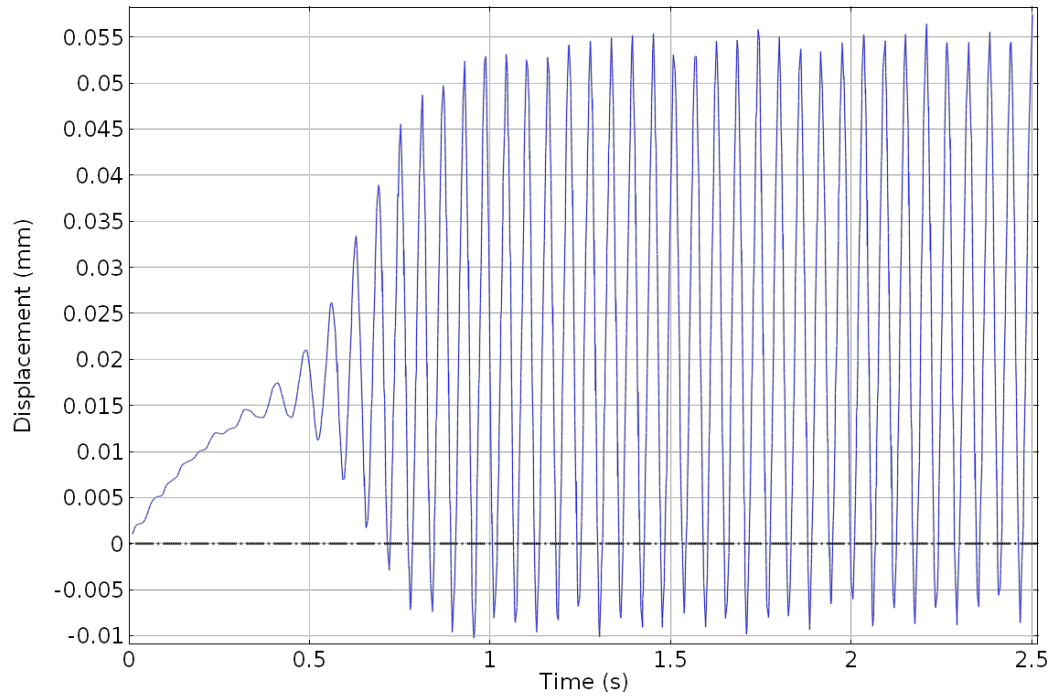


Figure 3.17: Incorrect displacement of the 2-D cylinder for the condition of $Re = 100$. The dashed line represents the initial position and the displacement should oscillate about it. Controlling the free mesh around the mid-plane of the cylinder corrects this.

3.4 Stationary Cylinder with Moving Tail

The last section for the cylinder in cross-flow will incorporate a stationary cylinder with a moving tail. This numerical test, proposed by Turek and Hron [80], allows the testing of large displacement FSI models using a 2-D setup. The published paper for the numerical test case includes a multitude of results for three different laminar cases that demonstrate the effectiveness of both the fluid and structural solver. The cylinder, as seen in the previous sections, will begin to shed vortices under the ideal conditions; these vortices will then create changes in pressure that will cause the cylinder tail to deform. It is this deformation, in conjunction with the vortex shedding, that is the main focus of this section.

3.4.1 Domain and Boundary Conditions

The stationary cylinder with moving tail is placed in a rectangular domain as proposed by Turek and Hron and the domain is presented in Figure 3.18. The domain has the

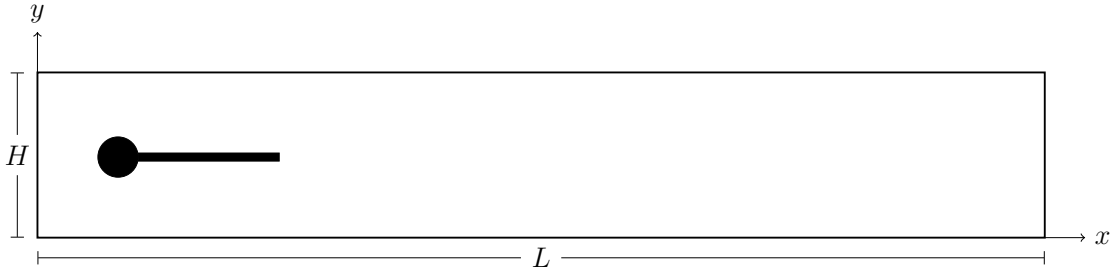


Figure 3.18: Domain for the 2-D Stationary Cylinder with Moving Tail.

following dimensions: The length of the channel, L , is 2.5 m and the height, H , is 0.41 m. The cylinder has a radius, r , of 0.05 m and its center is positioned at $(0.2, 0.2)$ m measured from the bottom left corner of the domain. The bar has a length, l , of 0.35 m and height, h , of 0.02 m with the bottom right corner placed at

(0.6, 0.19) m and it is rigidly attached to the cylinder. The boundary conditions are as follows:

- Inlet: $u(0, y) = 1.5\bar{U} \frac{y(H-y)}{(\frac{H}{2})^2}$
- Outlet: Pressure, $P = 0$
- Wall: No-slip, including the fluid-structure interfaces between the fluid and the stationary cylinder and tail

where $\bar{U}$ is the average velocity across the inlet.

The cylinder is held fixed and no structural computations are performed for the cylinder. As the cylinder begins to shed vortices, the tail will begin to experience a pressure differential which will result in deformation of the tail. This deformation is calculated using the structural mechanics equations in a similar manner to the 3-D cylinder in cross-flow.

3.4.2 Simulation and Results

The simulation of the cylinder with moving tail includes four cases, three of which were performed and presented by Turek and Hron (FSI1, FSI2, and FSI3). The first three simulations will have the following fluid and structural properties based upon the cases presented in Turek and Hron’s paper: The Reynolds number of 20 is run to ensure that the code will not begin to artificially shed vortices for conditions below $Re \approx 40$ [63]. The final simulation, labeled ‘Steel,’ was performed to observe the characteristics of a material similar to that in the HFIR; this material was not used in the simulations of Turek and Hron [80].

Each run was performed using the implicit transient solver with a maximum time-step of 5×10^{-3} s for a total of 10 s. The physics were modeled as fully-coupled. Unlike all of the previous cylinder in cross-flow simulations, the initial condition for the transient solver was not the steady-state solution for the corresponding Reynolds number; instead the initial conditions were all set to zero. In order to overcome any

Table 3.3: Properties of the fluid and structure used for the FSI test cases.

Parameter	FSI1	FSI2	FSI3	Steel
ρ^s , density of solid [10^3 kg/m ³]	1.0	10.0	1.0	7.80
ν^s , Poissons Ratio of solid	0.4	0.4	0.4	0.29
μ^s , shear modulus of solid [10^6 kg/(ms ²)]	0.5	0.5	2.0	81 400.00
ρ^f , density of fluid [10^3 kg/m ³]	1.0	1.0	1.0	1.00
ν^f , dynamic viscosity of fluid [10^{-3} m ² /s]	1.0	1.0	1.0	1.00
$\bar{U}$, average inlet velocity [m/s]	0.2	1.0	2.0	2.00
Re , Reynolds number	20.0	100.0	200.0	200.00

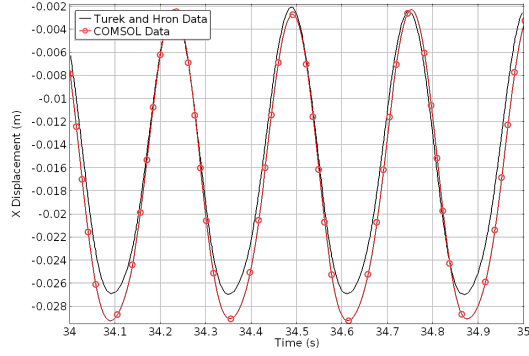
instabilities, the inlet velocity was increased from zero using a smoothing function with the following parameters:

$$u(t, 0, y) = \begin{cases} u(0, y) \frac{1 - \cos(\frac{\pi}{2}t)}{2} & \text{if } t < 2.0 \\ u(0, y) & \text{otherwise} \end{cases} \quad (3.15)$$

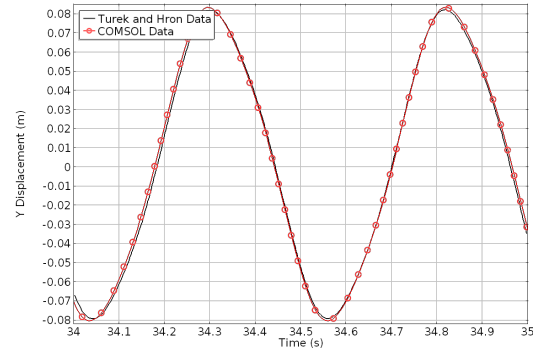
The use of the smoothing function also demonstrates the ability of the code to capture the transients that can happen during start-up of the HFIR.

Turek and Hron present four parameters for comparison of their data: x and y displacement at the end of the tail and the lift and drag around the entire cylinder and tail. As expected, FSI1 did not create any vortex shedding or subsequent tail oscillations. FSI2 and FSI3 produced very comparable results to Turek and Hron and the results for FSI2 are presented in Figure 3.19. The data is presented well after the cylinder has gone into a limit cycle. The results for FSI3 and the Steel case are provided in Appendix B for completeness.

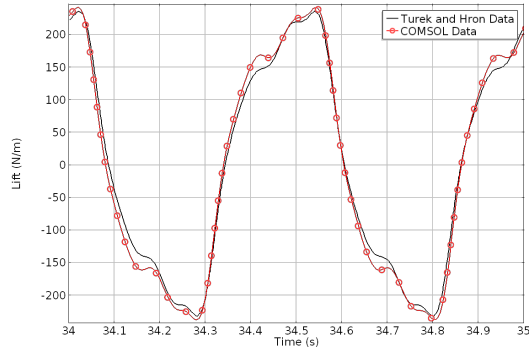
Observation of Figure 3.19 will reveal that the results presented exceed the computational time domain performed for the simulation. This discrepancy results from the fact that Turek and Hron presented their results for this time. Because the current simulation had reached a limit cycle, it was not necessary to run the simulation for the extended period.



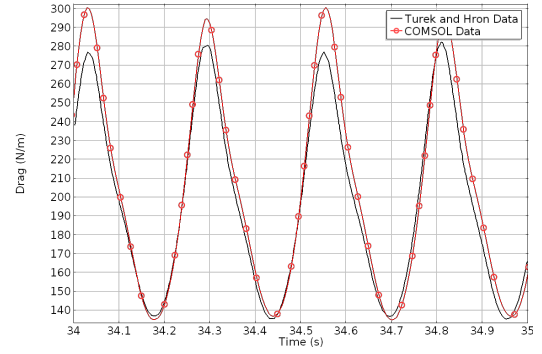
(a) X displacement



(b) Y displacement



(c) Lift around cylinder and tail



(d) Drag around cylinder and tail

Figure 3.19: Results for the FSI2 test case. The results (red lines with circles) are compared to the data provided by Turek and Hron in black.

3.4.3 Discussion

The cylinder with tail provides a context to explore large deflections produced by the fluid. In particular, the FSI2 case produces results in which the deflection at the tail is approximately 8 times larger than the thickness of the plate. This large deflection tests the code's ability to resolve the large stresses in the linear-structural model, to adapt the mesh as the structure deforms, and to capture the change in fluid characteristics, especially around the end of the tail. The final solution of FSI2 demonstrating these attributes is provided in Figure 3.20.

The monolithic solver was required to obtain the solutions provided in this section. This need is consistent with the experience of Turek and Hron who also used a monolithic simulation technique. The simulation matched the test cases provided by Turek and Hron very well and also produced deflections for a much stiffer plate as seen in the Steel case. Using all of the cylinder in cross-flow cases as benchmark simulations for the FSI capabilities of the code, it is now appropriate to explore experiments and cases related to the stacked plate configuration of the HFIR.

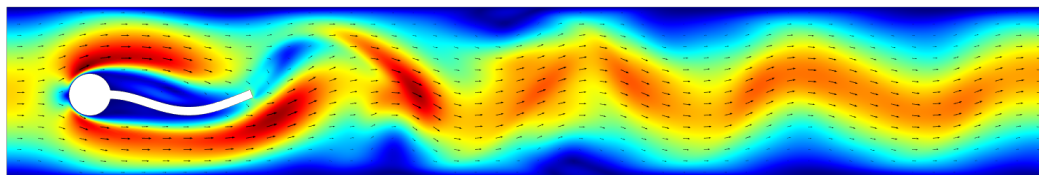


Figure 3.20: 2-D solution for the FSI2 case at 10s, $Re = 100$.

Chapter 4

Flat and Curved Fuel Plate Simulations

The ultimate goal of this dissertation is to effectively simulate the fluid-structure interaction that occurs in the HFIR core; this ability will then be used to calculate the deflection of the proposed LEU fuel. To date, there have been no experiments performed that include fuel in the plates. In order to gain confidence in the deflection predictions, a number of unfueled experiments are used as code benchmarks. The following chapter provides the benchmarks cases as well as the HFIR analysis.

Following a similar progression to the cylinder in cross-flow, a ‘simple’ setup of flat plates will be used to explore the suitable setup of these complex FSI problems. The first set of simulations will attempt to capture the static deflection of the flat plates of Smissaert [35]. This is a good case as Smissaert utilized plastic plates in order to obtain large deflections, much larger than what is expected to be seen in the HFIR core. The next step is to simulate transient deflections for flat plates. This was done experimentally by Liu et al. [96]. Finally, experiments were performed for the Advanced Neutron Source Reactor in the 1990’s using curved fuel plates [36, 41, 97–103]. The experimenters have deflection data for an involute ANSR fuel plate as well as deflections for a single HFIR-shaped fuel plate. Once good agreement with the

ANSR experiments has been achieved, the proposed LEU fuel will be inserted into the plates and the deflection will be predicted.

4.1 Flat Plates

Acknowledgment: This section is derived in part from an article published in *Nuclear Science and Engineering* , available online:

<http://www.tandfonline.com/doi/full/10.1080/00295639.2017.1379304>.

In this section, simulation techniques will be explored in order to better understand what is required to correctly capture the tightly coupled FSI physics seen with large deflection experiments. Various boundary conditions and the consequences of those assumptions will be demonstrated and reported. The knowledge gained from the static deflection simulations will provide insight into best practices for the simulation of a dynamically moving plate.

4.1.1 Smitsaert Simulations: One-plate

Georges S. Smitsaert performed a series of flat plate experiments at the Pennsylvania State University in 1968 in order to better quantify the Miller Critical Velocity [35]. Smitsaert constructed a hydraulic loop with a test section consisting of a series of parallel flat plates made out of PVC plastic; his experiment utilized pressure taps and strain gauges to measure the pressure drop and plate deflections, respectively. All of the plates were 0.0625 inch thick, 4.5 inches wide, and 45.0 inches long. The test section was designed to allow multiple plate configurations of 5, 9, and 15 plates. The 5 plate/6 channel case with a channel thickness of 0.250 inches was chosen for the validation case for the FSI code. The physical properties of the PVC plates as modeled in this study are provided in Table 4.1. Smitsaert chose to use plastic plates to observe larger deflections and to induce vibrations at lower mass flow rates. It was realized that there are discrepancies between the stated plate thickness in Smitsaert's

Table 4.1: Properties of the PVC plate used by Smitsaert in his experiment [35].

Property	English Values	Metric Values
Width	4.500 in	0.1143 m
Thickness	0.058 ± 0.002 in	1.473 ± 0.051 mm
Length	45.000 in	1.143 m
Modulus of Elasticity	4.8×10^5 lb _f /in ²	3.310×10^9 Pa
Poisson's ratio	0.25	0.25
Density	90.4 lb _m /ft ³	1448.1 kg/m ³

written description and what is provided in a table in his paper. The plate thickness in the table provides a value of 0.058 ± 0.002 in suggesting that this is the “as built” thickness of the plates. As such, a thickness of 0.058 in will be used in the simulations presented herein.

Several configurations for the simulation of a flat plate are considered for the Smitsaert case. In order to control the size of the simulation, a single flat plate is the obvious choice. As with all numerical simulations, the boundary conditions play a vital role in the convergence and accuracy of a solution. A variety of boundary conditions above and below the plates will be considered for simulation.

Domain and Boundary Conditions

Because the plates are parallel, a logical one-plate simulation would be to utilize periodic boundary conditions incorporating flow channels half the size of the full channel on both sides of the plate. The half-channel with periodic boundary conditions makes the assumption that all of the plates deflect in the same direction for an infinite number of plates (see Figure 4.1). Experiments have shown that the parallel plates deflect in opposite directions [34, 35] as illustrated in Figure 4.2. Due to this configuration being nonphysical, the simulations do not converge, and the convergence rate approaches an asymptote. Two other boundary conditions have

proven to work much better in reproducing accurate deflections for a single plate case: **symmetry and wall**.

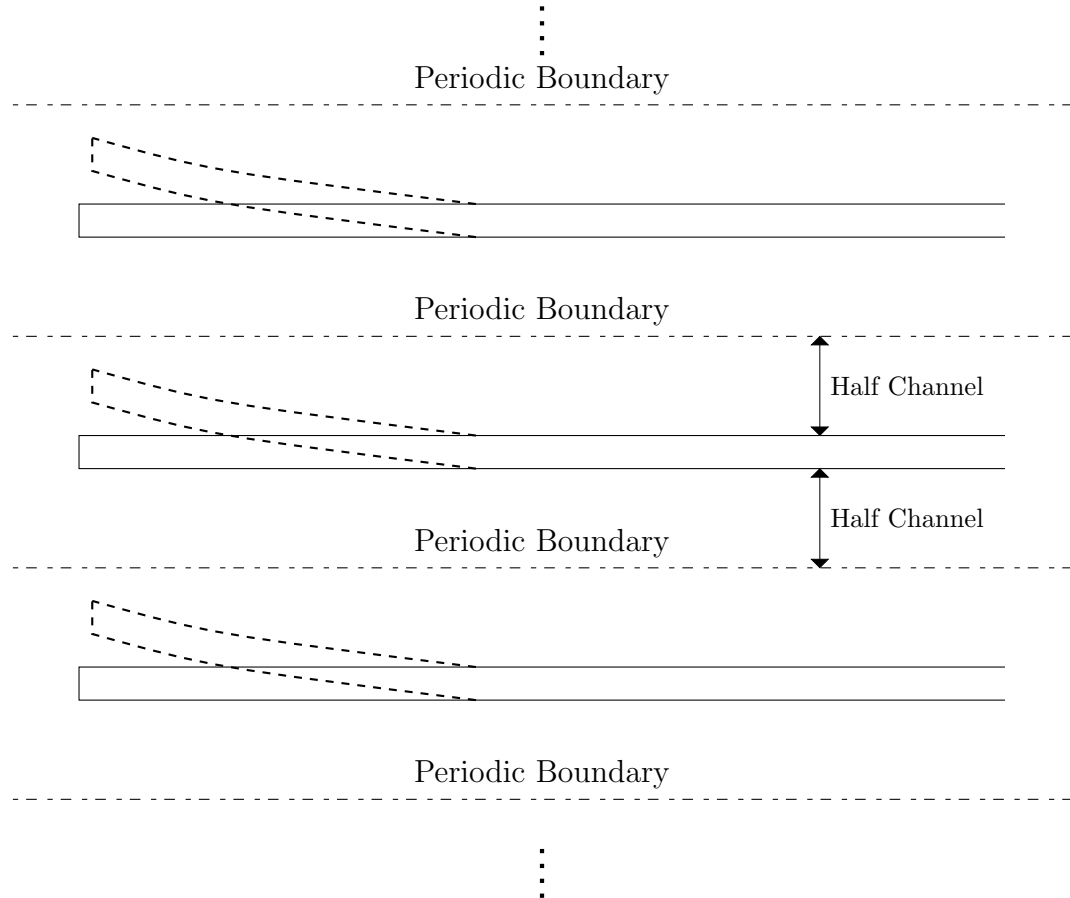


Figure 4.1: Deflection of a series of plates using periodic boundary conditions. The periodic boundary condition, applied to a single plate, assumes that all of the fuel plates deflect in the same direction.

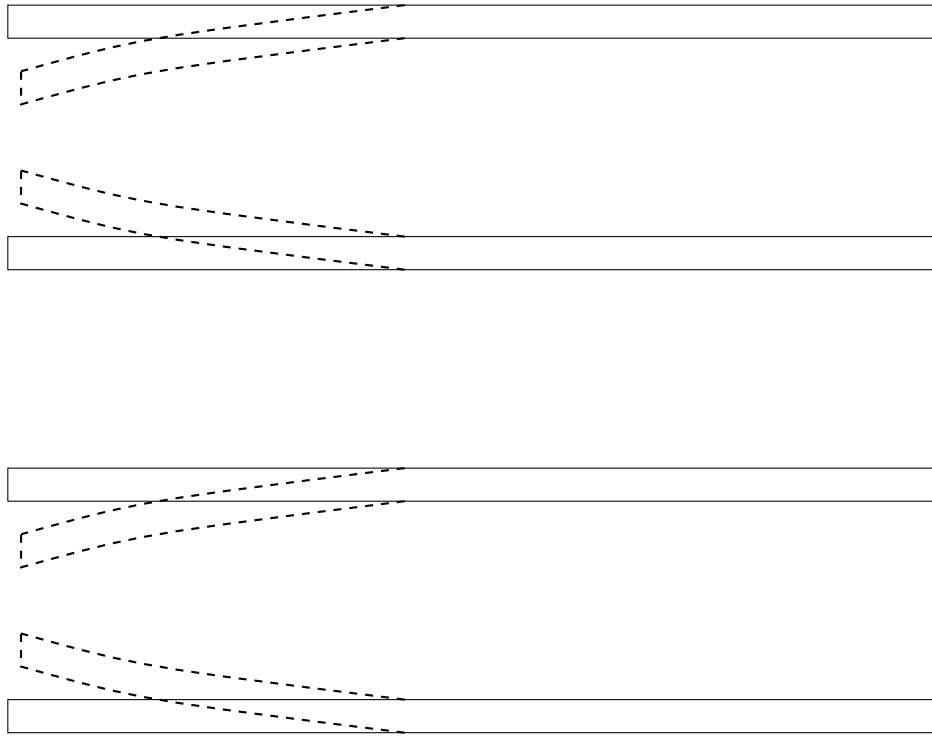


Figure 4.2: Bending of a series of plates with deflections of adjacent plates going in the opposite direction which is a more physical assumption [34].

The alternating small and large channels predicted by experiments due to the deflections can be replicated by utilizing a half channel setup similar to Figure 4.1 except the boundaries are now symmetric. The symmetric boundary imposes no velocity penetration and vanishing shear stress and is described by the following equations:

$$\mathbf{u}_f \cdot \mathbf{n} = 0 \quad (4.1)$$

$$\left(-p + \mu \left(\nabla \mathbf{u}_f + (\nabla \mathbf{u}_f)^T\right)\right) \mathbf{n} = 0 \quad (4.2)$$

where $\mathbf{n}$ is the normal vector to the boundary. Whereas the periodic boundary condition assumes the plates all deflect in the same direction, the symmetry boundary condition assumes that each plate deflects in the opposite direction of the adjacent plate in a similar fashion to Figure 4.2.

The wall boundary conditions take a different approach to solving the series of parallel plates. For this configuration, a full channel is simulated on each side of the plate with wall boundary conditions at the outside of the channels demonstrated in Figure 4.3. This configuration has proven to be the most stable for the one-plate models because of the wall function (no-slip) boundary conditions.

The remaining fluid boundary conditions are the inlet and outlet conditions and the boundary between the fluid and the moving plate. The inlet conditions are similar to those imposed for the cylinder in cross-flow utilizing a uniform velocity inlet condition. The outlet condition is similarly set as a 0 pressure outlet. The side walls, where the plates are held, are set to wall boundary conditions satisfying the wall function.

Spacing

Deviations from ideal channel thickness play a major role in the amount of deflection observed for the plates. As Kane [43] showed in his theoretical work, the larger the deviations, the larger the leading-edge deflections. The experiments performed by

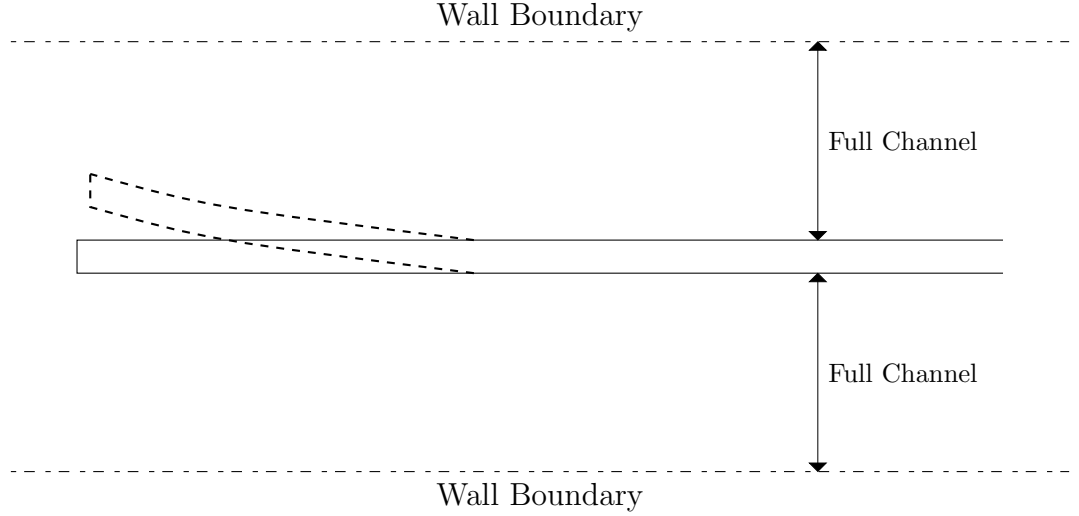


Figure 4.3: Single plate setup with full channels and wall boundary conditions.

Stromquist and Sisman [32] showed that large deviations produced instabilities in the plates and the extreme cases resulted in plate buckling. The experiment by Kennedy et al. [37] was designed with a single plate and two different sized flow channels (80 mils and 100 mils) to ensure a predictable deflection utilizing the knowledge that the deviations create a pressure differential that produces a deflection. The most stringent manufacturing techniques can never guarantee a perfect geometry thus providing a context to include some deviations when simulating flat plates.

Numerical simulations, particularly ones with uniform meshes on each side of a plate, could fail to produce deflections if the channel sizes are the same. In fact, a fully coupled solution begins to show instabilities and issues with convergence if the channels are the same size. In order to circumvent this problem, a small deviation from the ideal geometry is introduced which provides an initial pressure difference between the plates to induce the deflection. These deviations do not have to be large in order to produce deflections.

For the periodic boundary case, introducing an offset or deviation from the ideal geometry does not alleviate the convergence issues previously observed for these

boundary conditions. The periodic condition, on a simplified level, repeats the geometry over and over. Using the Smisaert case as an example with channel thicknesses of 0.250 in, if the plate offset is 5 mils, then the upper half channel becomes 120 mils and the lower half channel becomes 130 mils. Although the simulation channels have different sizes for this case, the periodic boundary condition keeps the total channel thickness of 250 mils or 0.250 in; this concept is demonstrated in Figure 4.4. Both the nonphysical plate deflections in the same direction and the uniform channels make the use of periodic boundary conditions inappropriate for single plate simulations.

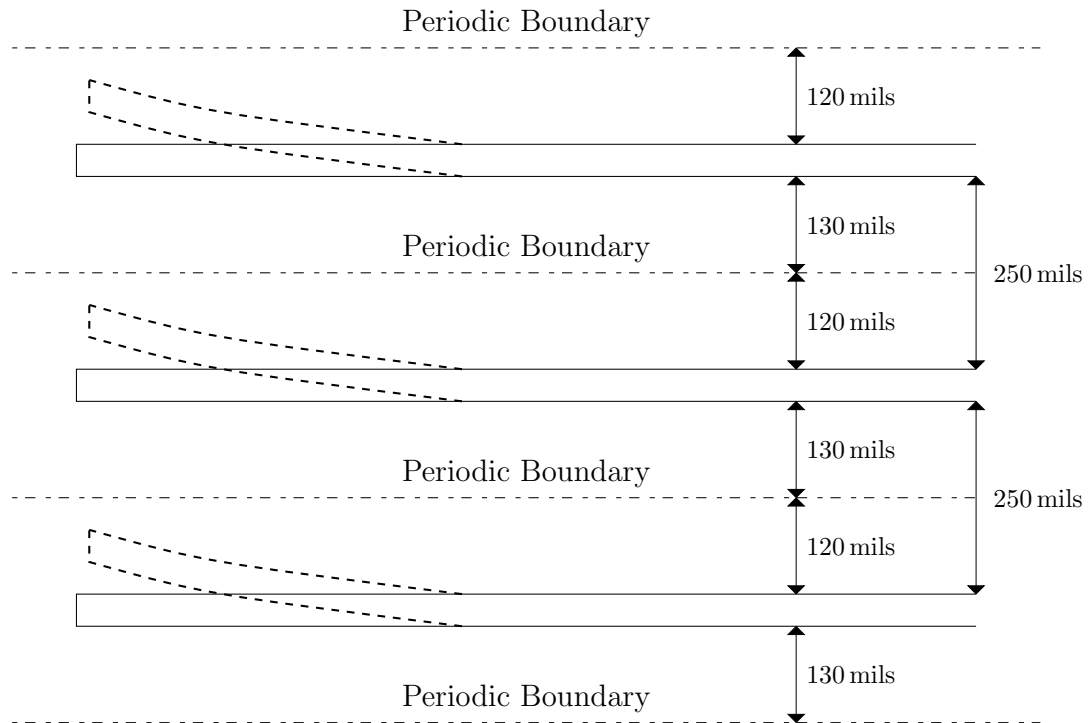


Figure 4.4: Periodic boundary conditions with a plate offset of 5 mils. The periodic boundary condition maintains the original channel thickness of 250 mils between each plate which does not produce any stable deflections.

Unlike the periodic boundary condition, the symmetric boundary condition preserves the offset between adjacent plates. Figure 4.5 shows how the offset is preserved. By preserving the offset, the deflection caused by the leading-edge deviation described by Kane can be observed and the stability of convergence is greatly improved.

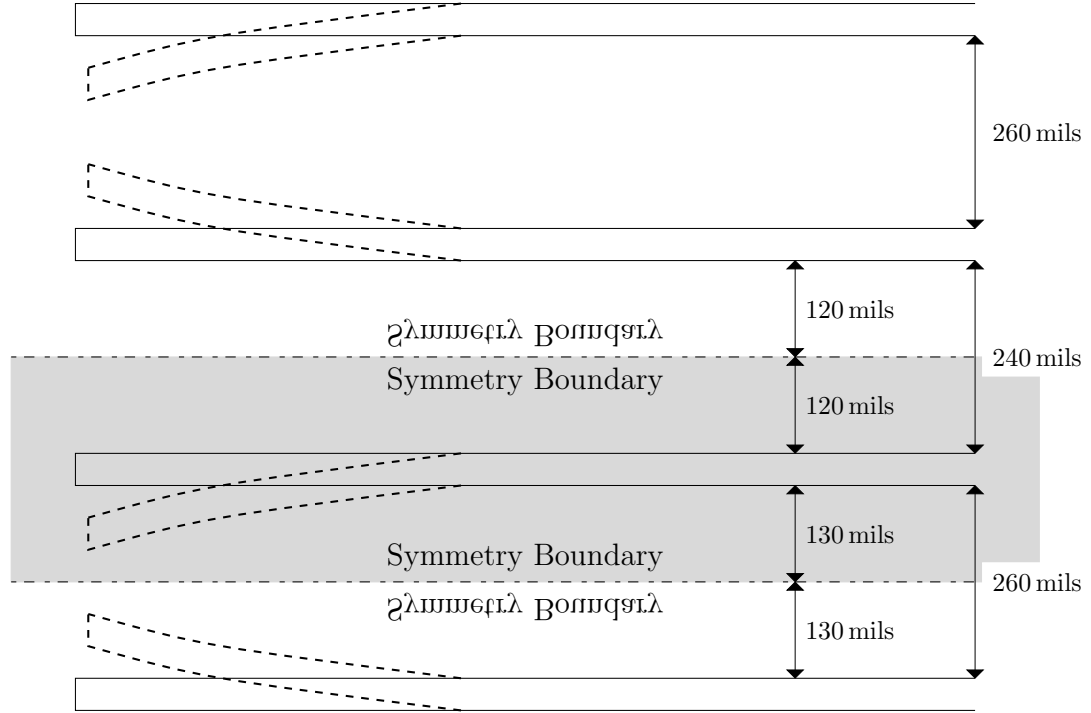


Figure 4.5: The symmetry boundary increases and decreases the overall channel thickness between each plate when an offset is imposed on the plate location. In this case, the offset is 5 mils and the computational domain is in the grey box.

The offset of the single plate with wall boundary conditions is also necessary to obtain deflections and maintain stability. Without this deviation, the simulation behaves similarly to the periodic boundary condition and a converged solution is not obtained.

Simulation and Results

The large aspect ratio of the Smitsaert experimental setup (180 length to channel thickness ratio) presents a difficult computational problem when it comes to balancing computational time, size and stability. The long, thin flow channel requires a large number of elements in order to properly capture the flow field. A quadrilateral mesh is utilized to capture the flow field while reducing the mesh requirements compared to a triangular mesh. The final mesh consists of 234 120 elements for the symmetric boundary condition and 493 320 elements for the wall boundary condition simulations. The leading-edge mesh is shown in Figures 4.6 and 4.7 for the symmetric and wall boundary conditions, respectively.

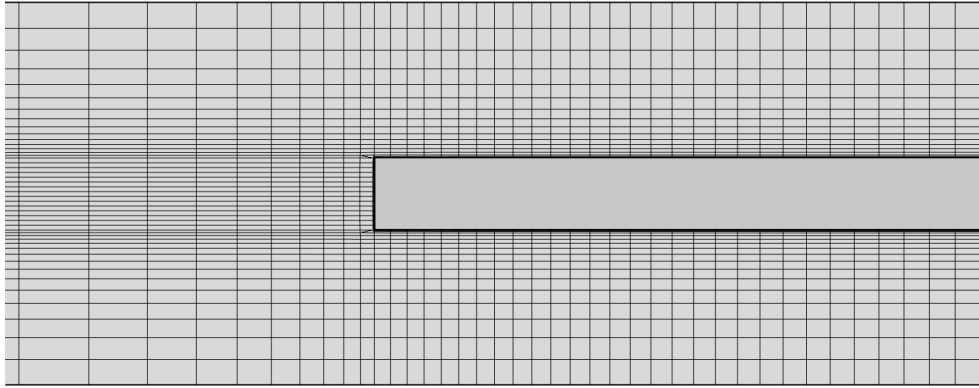


Figure 4.6: Leading-edge mesh for the single plate simulation with symmetric boundary conditions which simulated only half of the flow channel on each side of the plate for the Smitsaert experimental comparison.

The inlet and outlet portions of the test section were not clearly defined in Smitsaert's original work. Therefore, in this work, the lengths were chosen by comparing the pressure profiles of a simulation with an inlet and outlet length of

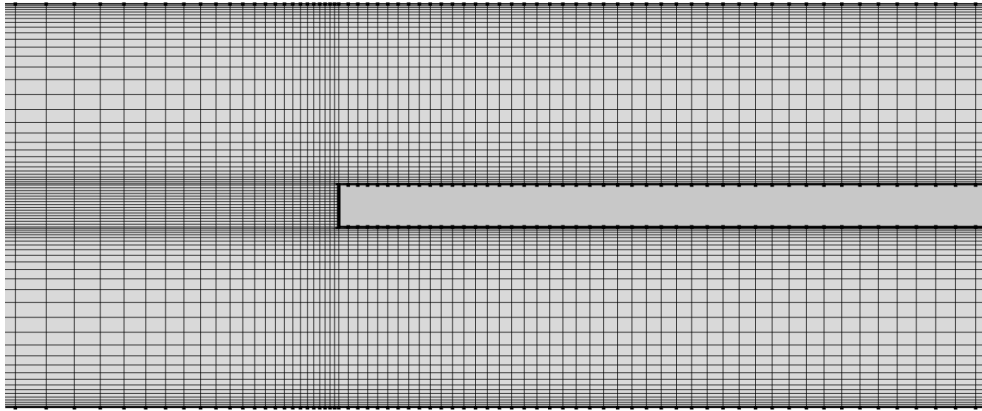


Figure 4.7: Leading-edge mesh for the single plate simulation with wall boundary conditions which simulated the full flow channel on each side of the plate for the Smitsaert experimental comparison.

2 inches and 10 inches for 2-D flow neglecting the deflection of the plates. This exercise resulted in the following comparison of the pressure distribution for the full channel with wall boundaries in Figures 4.8 and 4.9. After comparing the pressure distribution for the two inlet lengths, it was determined that a length of 10 inches was the appropriate inlet and outlet length.

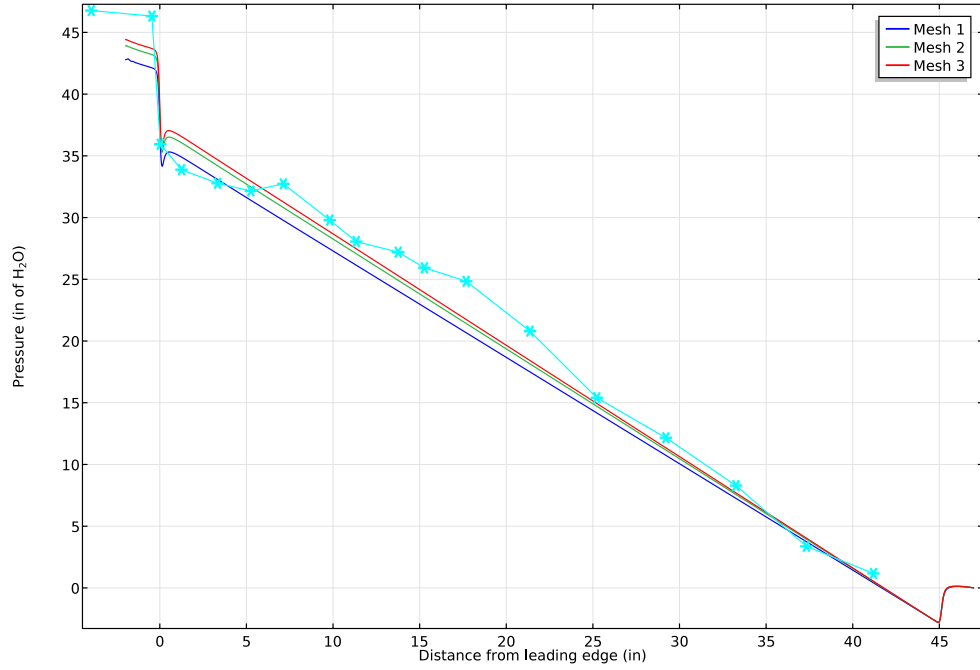


Figure 4.8: The pressure distribution for a 2-D simulation with inlet and out lengths of 2 inches. The 2-D simulation had an inlet velocity of $1.23 \cdot M_c$ (approx 2.7 m/s) and showed that an inlet plenum of 2 inches is not sufficient to match the experimental pressure drop. Mesh 1: 17 284 elements Mesh 2: 67 412 elements Mesh 3: 262 920 elements

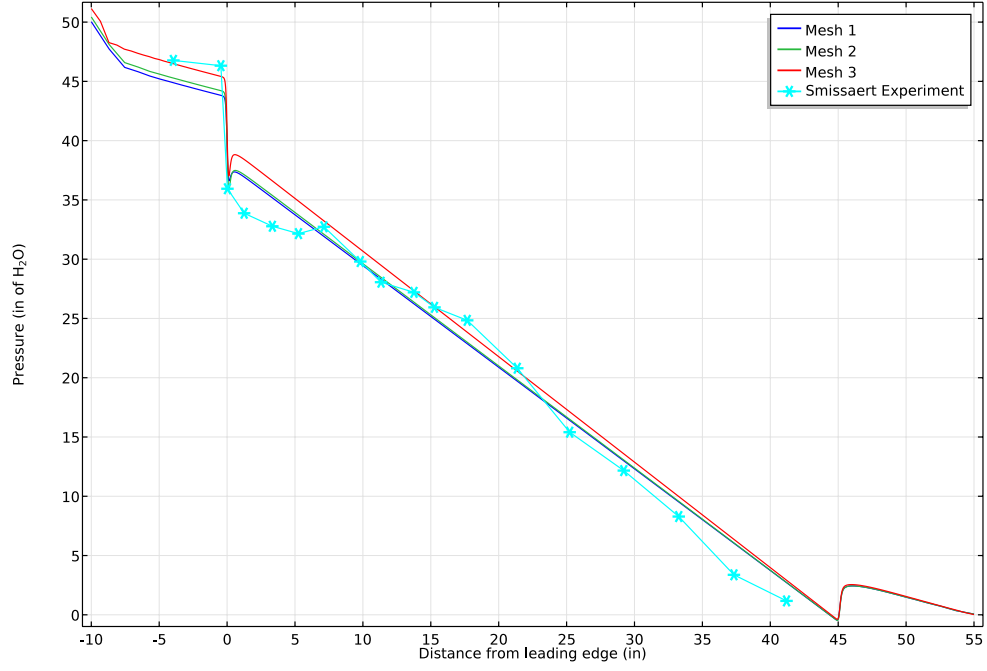


Figure 4.9: The pressure distribution for a 2-D simulation with inlet and out lengths of 10 inches. The 2-D simulation had an inlet velocity of $1.23 \cdot M_c$ (approx 2.7 m/s) and showed that an inlet plenum of 10 inches is appropriate to match the experimental pressure drop. Mesh 1: 17 284 elements Mesh 2: 67 412 elements Mesh 3: 262 920 elements

Displacement

Deflection calculations for a fully converged solution with symmetry boundary conditions are provided in Figure 4.10. The simulations were not able to capture all of the mode shapes reported by Smissaert however it is worth noting that there are mode shapes at the ends of the simulated plates where there were no deflections measured in the experiment. Figure 4.11 provides the deflection for the case in which the full and half channels were simulated with wall boundary conditions. These results show good agreement with the leading-edge deflection where the largest reduction in flow occurs for the half channel simulation.

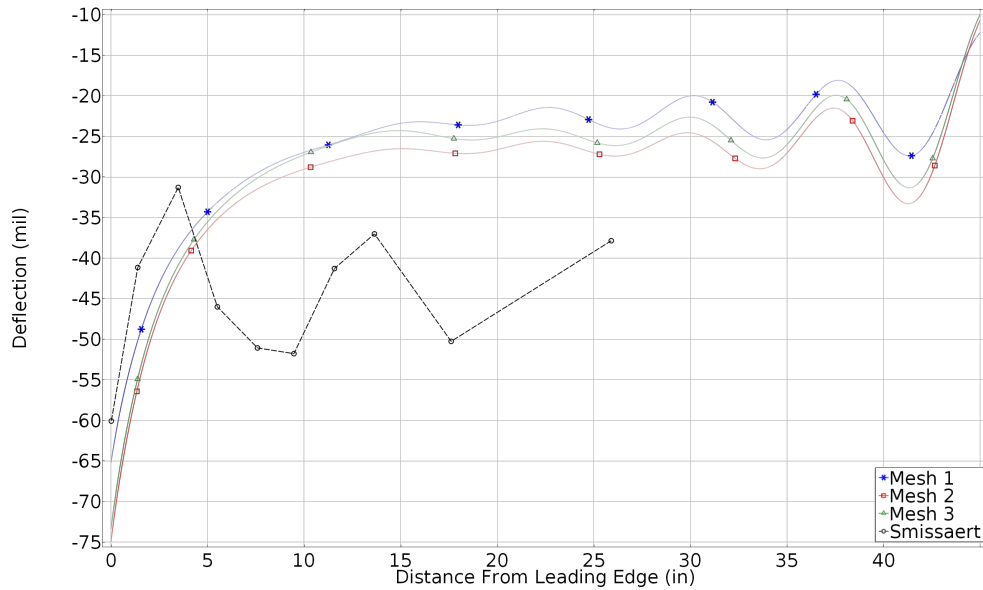


Figure 4.10: Deflection for a single plate simulation with symmetry boundary conditions. The inlet velocity was set to $1.23 \cdot M_c$ (approx 2.7 m/s) with a plate offset of 2.5 mil. Mesh 1: 31 684 elements Mesh 2: 125 012 elements Mesh 3: 493 320 elements

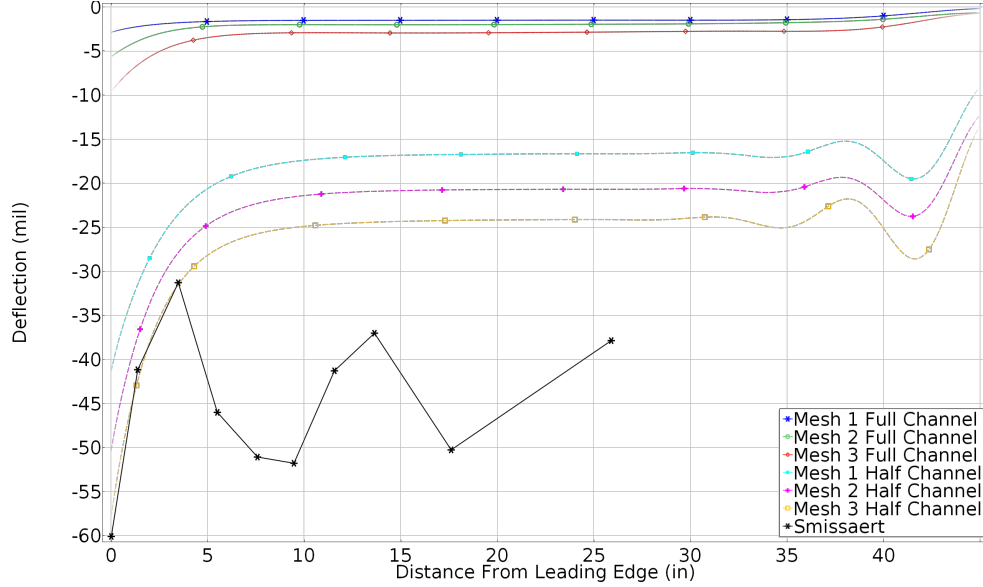


Figure 4.11: Deflection for a single plate simulation with wall boundary conditions. The inlet velocity was set to $1.23 \cdot M_c$ (approx 2.7 m/s) with a plate offset of 2.5 mil. Mesh 1: 31 684 elements Mesh 2: 125 012 elements Mesh 3: 493 320 elements

4.2 Smissaert Simulations: Five-Plate

The 5-plate simulation was constructed to model the Smissaert experiment as described in the paper. All five plates were included utilizing the linear structural model with fixed edges along both sides of each plate. The side walls and top and bottom walls were modeled as fixed wall boundaries with an inlet and outlet section 10 inches long.

The simulation was performed using a series of decreasing mesh sizes with the final mesh consisting of 815 680 mesh elements. The final mesh around the leading-edge of one of the plates is provided in Figure 4.12. The final mesh was also checked to satisfy the requirements for the $k-\epsilon$ turbulence model with $y^+ = 11.06$ on all walls. The initial condition for the course mesh transient solution was taken from a steady-state solution of only the flow variables. The simulations were performed on an 8 node cluster with 12 cores per node and Table 4.2 provides the run times for each mesh.

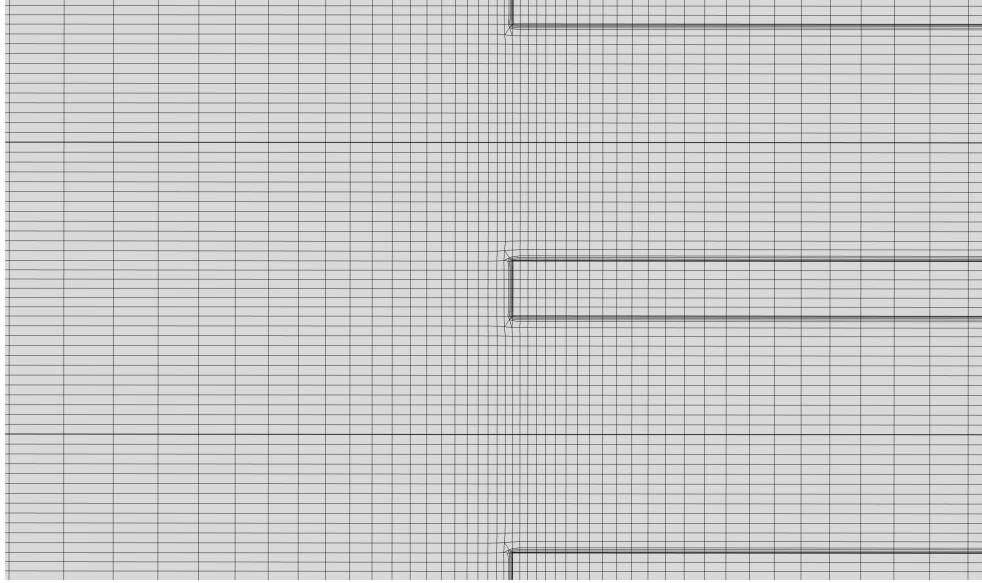


Figure 4.12: The mesh around the leading-edge of one of the plates used for the final mesh simulation with 815 680 total mesh elements for the five-plate Smissaert simulation.

As with the single plate simulations, a monolithic approach with a direct solver was utilized.

Table 4.2: Run times for each subsequent mesh performed on Mintaka at ORNL (8 nodes, 12 cores per node) for the simulation of the Smissaert [35] experiment.

Num of Elements	Deg of Freedom	Physical Time (s)	Wall-Clock Time (s)	Time-steps
89 080	573 048	Steady-state	1096	Steady-state
89 080	573 048	5.0	53 831	140
115 640	1 060 635	0.01	38 945	108
815 680	8 712 387	0.168	2 246 400 (26 days)	220

For the five plate simulations, the use of initial plate deviations was not needed, and all of the flow channels were 0.250 inch thick. The deflections were induced by the non-moving walls above and below the first and last plate. The total deflections for each plate are provided in Figure 4.13. According to Groninger and Kane [34],

adjacent plates always deflect in opposite directions and the three middle plates, Plates 2, 3 and 4, follow this trend. The top and bottom plate, Plates 1 and 5 respectively, do not seem to follow this trend because of their interactions with the non-moving top and bottom walls.

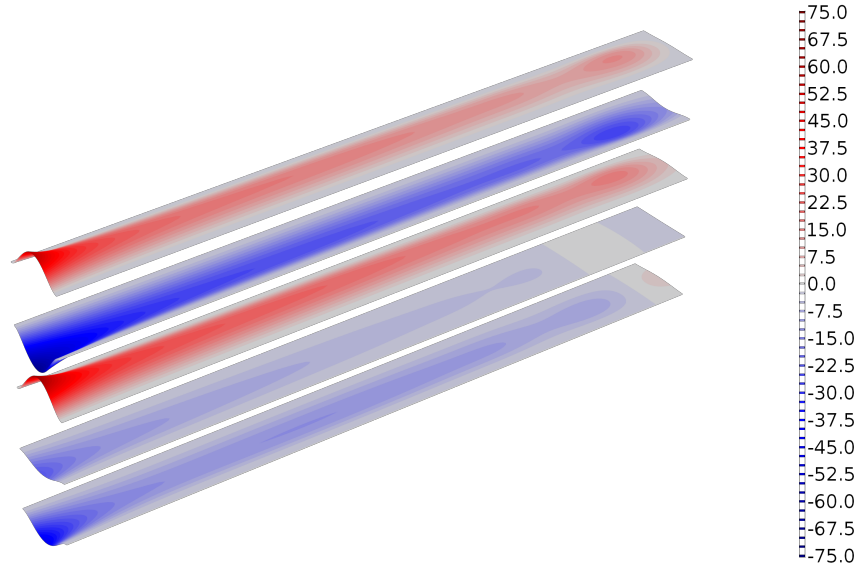


Figure 4.13: Deflection of the five-plate Smitsaert simulation for an inlet velocity of $1.23 \cdot M_c$ (approx 2.7 m/s). For visualization, the distance between each plate has been increased from 250 mil to 400 mil and the deflection is exaggerated by 25 times. The deflection is presented in mils.

Comparison of the deflections of the plates with the experiment is provided in Figure 4.14 and shows good agreement for the leading-edge deflection amplitudes. Plate 3, the middle plate, matches well with the large leading-edge deflection provided by Smitsaert. In his paper, Smitsaert also provided three deflection points for a plate adjacent to the middle plate, either Plate 2 or 4 and this is also provided in Figure 4.14. The leading-edge deflection of Plate 2 of the simulation again matches quite well with the reported deflection of one of the plates adjacent to the middle plate.

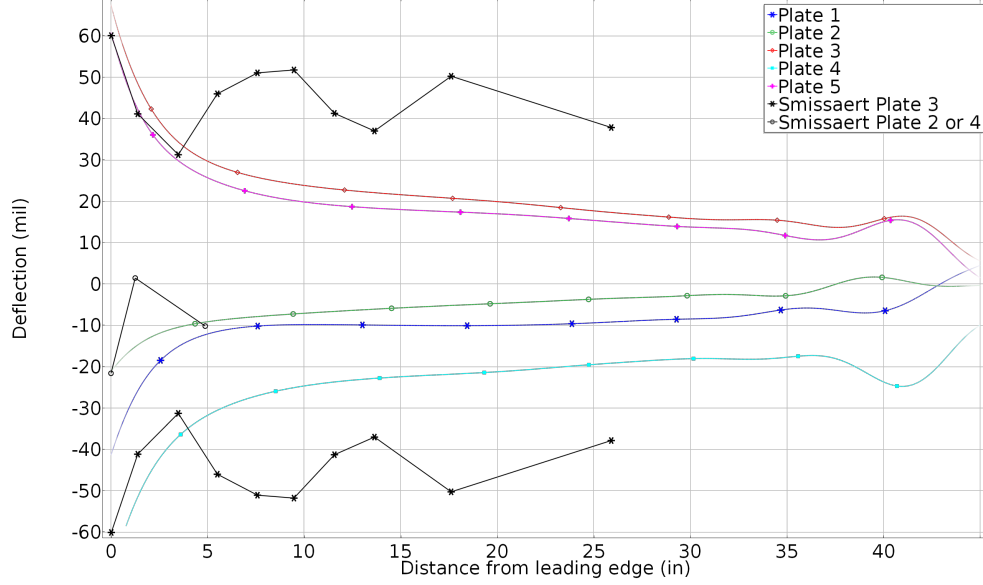


Figure 4.14: Deflection along the centerline of the plates compared to the experimental deflections for the five-plate Smissaert simulation. The inlet velocity was set to $1.23 \cdot M_c$ (approx 2.7 m/s).

It is desired to couple this solution with a thermal-hydraulics solution for the HFIR core. For this application, the reduction in flow that results from the deflections at the leading-edge of the plates could have an effect on the ability of the flow to effectively remove heat from the reactor. Figures 4.15 and 4.16 provide the velocity and pressure at the midplane of the leading-edges of the plates. As can be seen in these two figures, the change in channel size results in a redistribution of the flow around the plates which is not fully captured by the Miller Critical Velocity.

The Steady-State Heat Transfer Code (SSHTC) has been used for many years at the HFIR to ensure the safety of the reactor before startup. This code includes a provision for the reduction in flow rate from the deflection of the HFIR plates. This area reduction can be calculated from these simulations and is presented in Table 4.3. The changes in area for the Smissaert cases are quite large but this should not be the case for the stiffer HFIR plates because they are curved and made out of

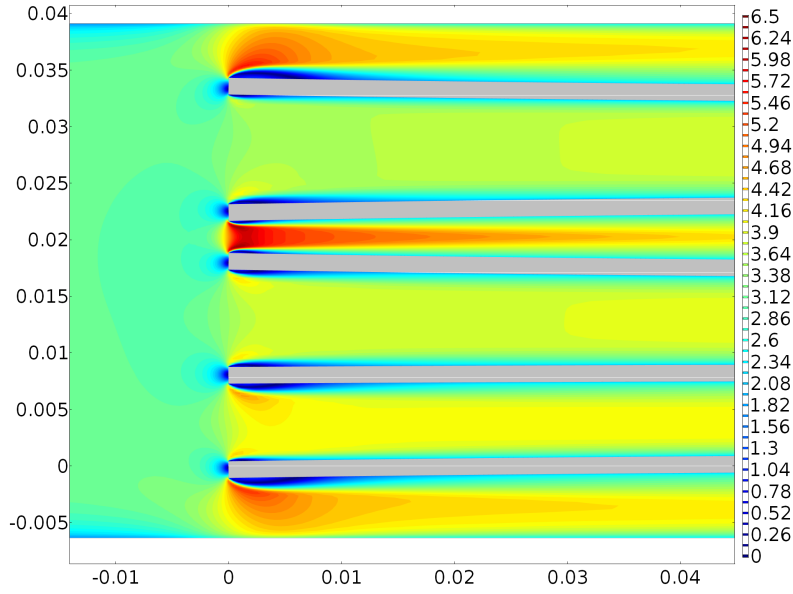


Figure 4.15: 2-D cut plane of the velocity (m/s) along the midplane of the plates at the leading-edge of the plates for the five-plate Smissaert simulation. The inlet is 10 inches from the leading-edge of the plate and set to $1.23 \cdot M_c$ (approx 2.7 m/s).

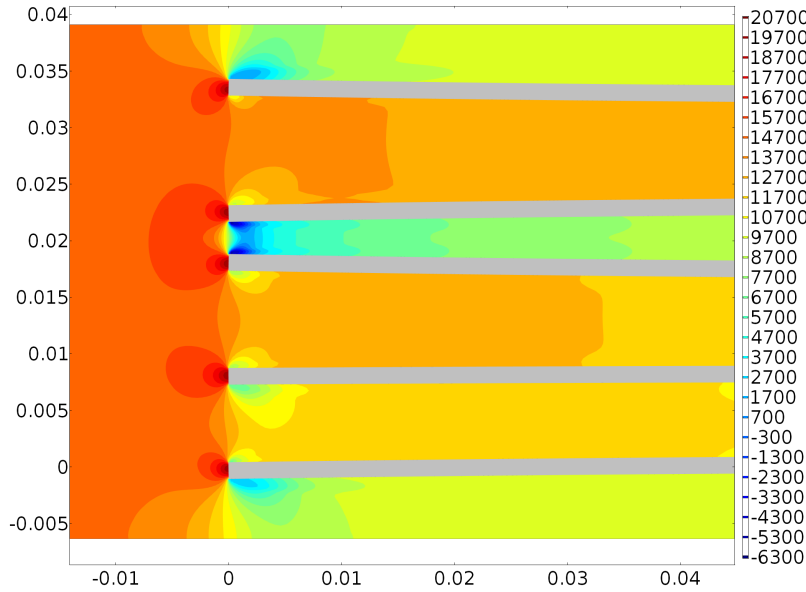


Figure 4.16: 2-D cut plane of the pressure (Pa) along the midplane of the plates at the leading-edge of the plates for the five-plate Smissaert simulation. The inlet is 10 inches from the leading-edge of the plate and set to $1.23 \cdot M_c$ (approx 2.7 m/s).

aluminum. Flow reduction knowledge can be used to better predict the performance of the thermal-hydraulics of the reactor during its operation.

Table 4.3: Changes in flow area for each channel of the 5-plate simulation.

Channel	New Area (in ²)	Percent Change in area
1	1.0400	-7.5600
2	1.1775	4.6682
3	1.2878	14.4736
4	0.8536	-24.1211
5	1.3936	23.8795
6	0.9974	-11.3402

4.3 Liu Transient Experiments

Liu et al. [96] designed and conducted an experiment to observe the self-excited, periodic vibrations of a flat plate. In order to ensure the periodic displacement, a 1 mm thick plate was held at the four corners creating what they termed a ‘four-square’ plate. The thin plate was 850 mm long by 76 mm wide and was placed in a test section 1250 mm long by 80 mm wide and 35 mm thick. The displacement was measured through the plexiglass housing using a laser in the middle of the plate.. A series of numerical simulations will be performed to explore the ability to capture vibration of the plates; 2-D and 3-D simulations with various boundary conditions will be explored.

2-D Simulation

As with the Smissaert work, an exploration of the boundary conditions and how they affect the solution was conducted to obtain the best way to simulate the transient case. The transient cases take much longer to run because of the need to capture many cycles of deflection, so an initial case utilizing a 2-D geometry was created. The plate

was held fixed at the leading and trailing edges. Initially, the mesh smoothing was applied to the entire domain adding to the size and complexity of the model solved. In order to reduce computation time and to prevent inverted meshes during the mesh smoothing calculations, the fluid domain was split with the inlet and outlet sections held fixed. Figure 4.17 shows how the domain was split up.



Figure 4.17: The computational domain for the 2-D simulation of the Liu Experiment. The purple highlight shows the part of the domain in which the mesh smoothing is applied. Outside of this domain, the mesh does not move and the mesh smoothing is not calculated.

The 2-D simulation proved to be very stable and ran very quickly on a 12 core desktop machine. The fast running and stable nature of the simulations allowed for an exploration of all of the cases presented by Liu. Unlike the static deflection simulations run for the Smits experiments, no offset was needed; the plate was simulated in the middle of the channel. An initial condition was established using the stationary flow solver and used for the fully-coupled FSI transient solver. Using a laser, Liu measured the deflection in the middle of the plate and the results of the simulations compared to the experimental results are provided in Table 4.4. The deflection in the middle of each plate for the range of inlet velocities are provided in Figures 4.18 to 4.23.

Although the 2-D simulations exhibited self-excitation, they did not match the published vibrational frequencies. The maximum measured displacement at the middle of the plate was also much smaller than the experimental results by over an order of magnitude. This is to be expected as the fixed leading-edges stiffen the

Table 4.4: Deflection frequencies for the 2-D Liu simulations. *The deflection plot provided in the paper does not contain sufficient resolution to obtain the frequency of the plate and thus the maximum time-step was chosen based on an estimated frequency of 1.75 Hz.

Inlet Velocity (m/s)	Max Time-step (s)	Experimental Frequency (Hz)	Simulation Frequency (Hz)	Solution Time (s)
1.05	2.1758	0.1149	0.125	50 694
1.19	0.0855	0.2924	0.375	81 017
1.85	0.0215	1.1620	0.651	35 617
2.84	0.0144	0.000*	—	125 045
3.29	0.0132	1.9000	—	166 194
3.64	0.0119	2.1000	1.138	36 211

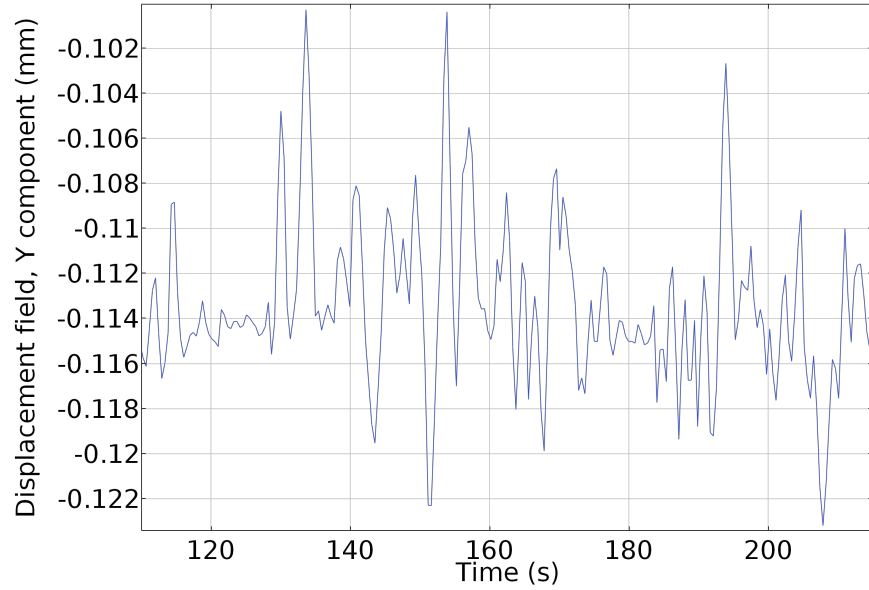


Figure 4.18: The deflection at the center of the plate for the 2-D Liu simulation with fixed leading- and trailing edges with an inlet velocity of 1.05 m/s

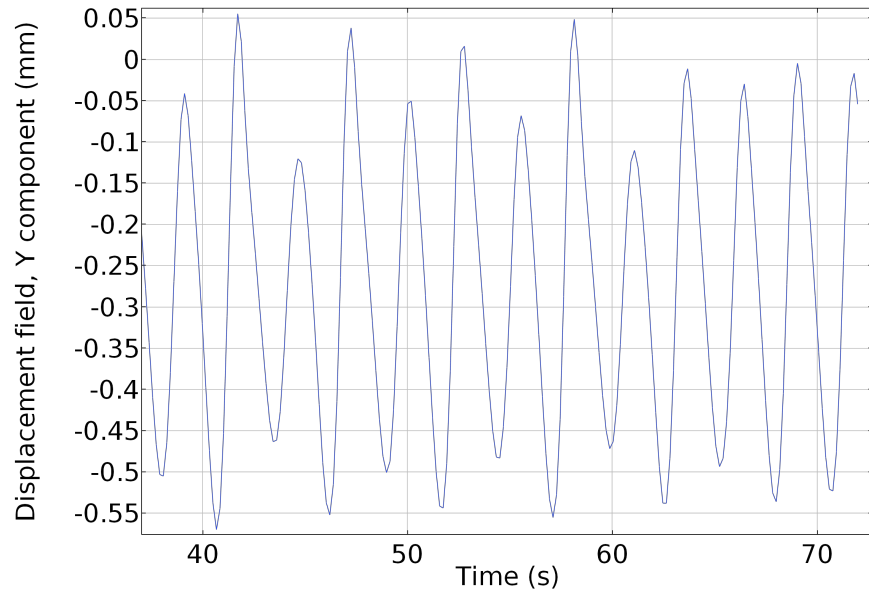


Figure 4.19: The deflection at the center of the plate for the 2-D Liu simulation with fixed leading- and trailing edges with an inlet velocity of 1.19 m/s

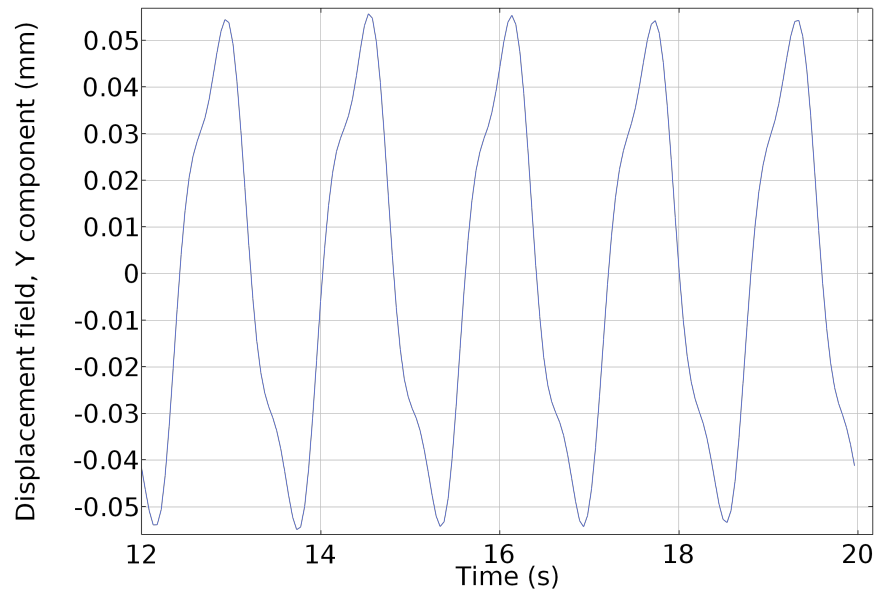


Figure 4.20: The deflection at the center of the plate for the 2-D Liu simulation with fixed leading- and trailing edges with an inlet velocity of 1.85 m/s

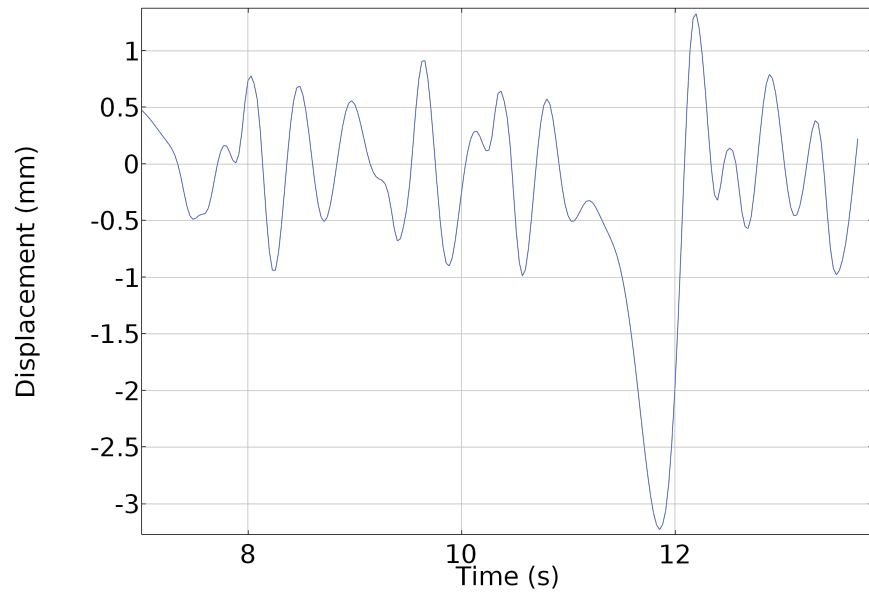


Figure 4.21: The deflection at the center of the plate for the 2-D Liu simulation with fixed leading- and trailing edges with an inlet velocity of 2.84 m/s

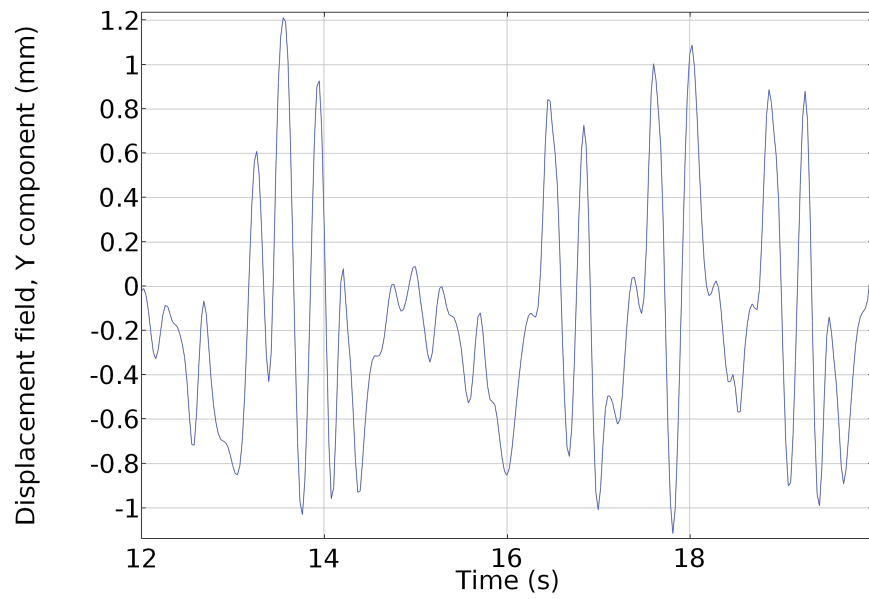


Figure 4.22: The deflection at the center of the plate for the 2-D Liu simulation with fixed leading- and trailing edges with an inlet velocity of 3.29 m/s

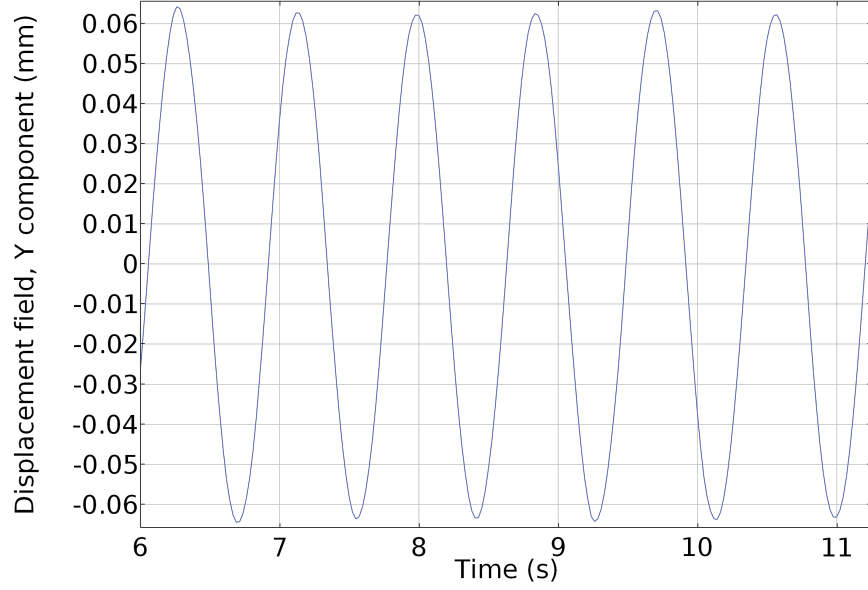


Figure 4.23: The deflection at the center of the plate for the 2-D Liu simulation with fixed leading- and trailing edges with an inlet velocity of 3.64 m/s

plate quite a bit leading to smaller deflections and higher natural frequencies. Some of the deflections also exhibited somewhat random behavior which (Figures 4.18, 4.21 and 4.22) was seen in the deflection data provided by Liu et al. The domain was then extended to include the 3-D effects next.

3-D Fixed Leading-Edge Simulation

As a first step, the domain was extended in the span-wise direction to simulate the entire domain in 3-D while holding the entire leading and trailing edges fixed. The sides were free to move in the z-direction only and no fluid was exchanged between the sides of the plates and the domain wall. Figure 4.24 provides an example of the computational domain. The reduced mesh smoothing domain that was used for the 2-D case was also employed for the 3-D cases.

A coarse mesh of 58 248 elements was used to determine the relevance of the fixed leading-edge case. After refining the mesh to 288 880 elements, the maximum

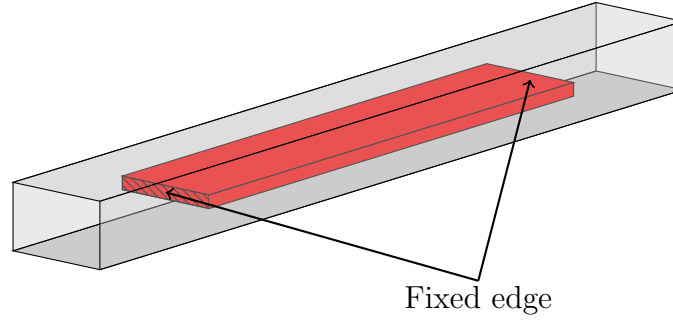


Figure 4.24: 3-D domain for the transient Liu simulations with fixed leading and trailing edges of the plate.

displacement in the middle of the plate was closer to the experimental results while the frequency was still incorrect. Figure 4.25 shows the displacement for the inlet velocity of 1.85 m/s .

These transient runs were taking about $350\,000 \text{ s}$ wall-clock time when the maximum time-step was held fixed similar to the cylinder in cross-flow. In order to determine if the restriction on the time-step was necessary for these calculations, the time-step was allowed to freely change and actually took longer to calculate due to the code taking more Newton iterations per time-step. Figure 4.26 shows how the time-step changed during the free time-step simulation and Figure 4.27 shows the displacement of the plate using the free time-step. As can be seen when comparing Figures 4.25 and 4.27, the unrestricted time-step results in an unstable solution with an almost random deflection. In fact, it was impossible to extract the displacement frequency from the solution using the free time-step. The free time-step solver used a time-step of 0.11671 s for the majority of the calculations with one instance of a time-step of 0.23342 s . Although the free-time step study required fewer time-steps, it required a significant amount of Newton iterations to converge each time-step thus taking longer to complete.

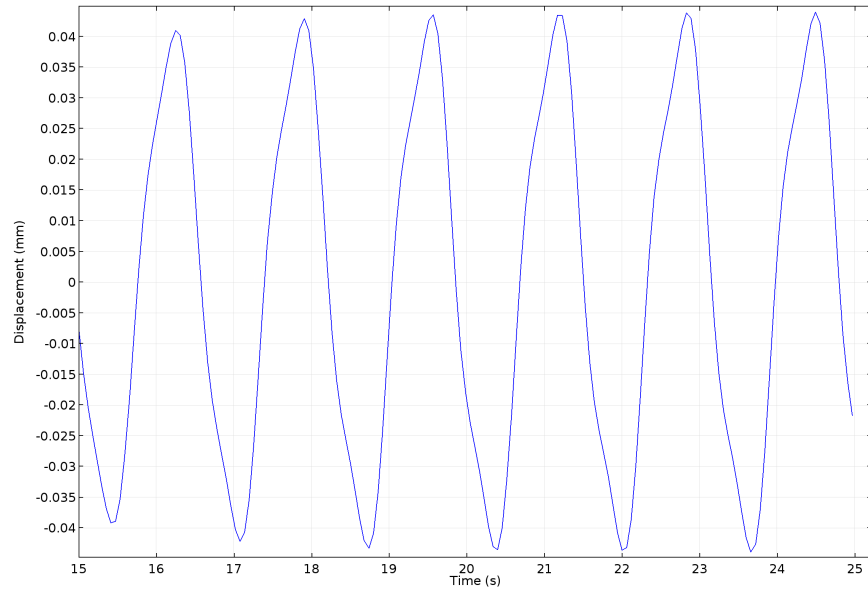


Figure 4.25: Displacement for the 3-D Liu simulation with an inlet velocity of 1.85 m/s with the restricted time-step of 0.0215 s.

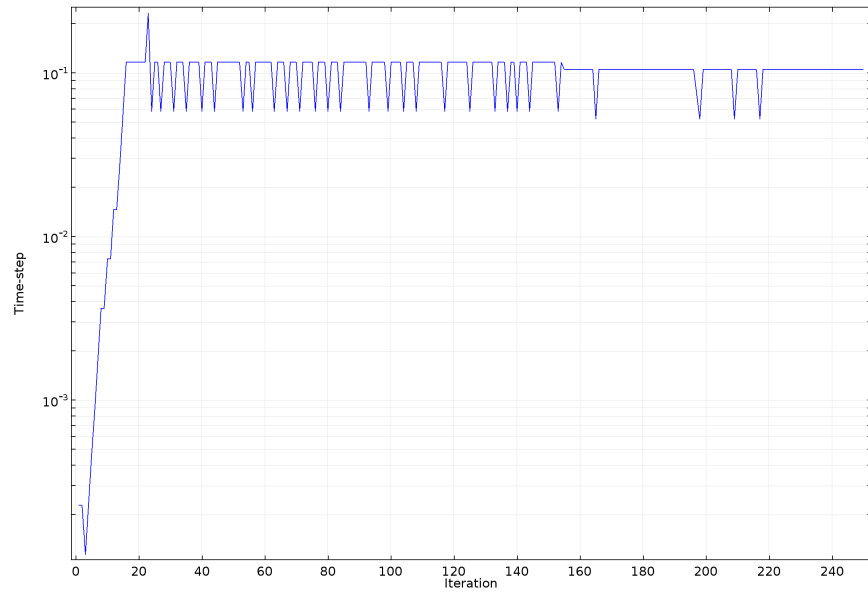


Figure 4.26: The time-steps used by the solver when there was no restriction on the maximum time-step for the 3-D Liu simulation with an inlet velocity of 1.85 m/s and fixed leading- and trailing-edges.

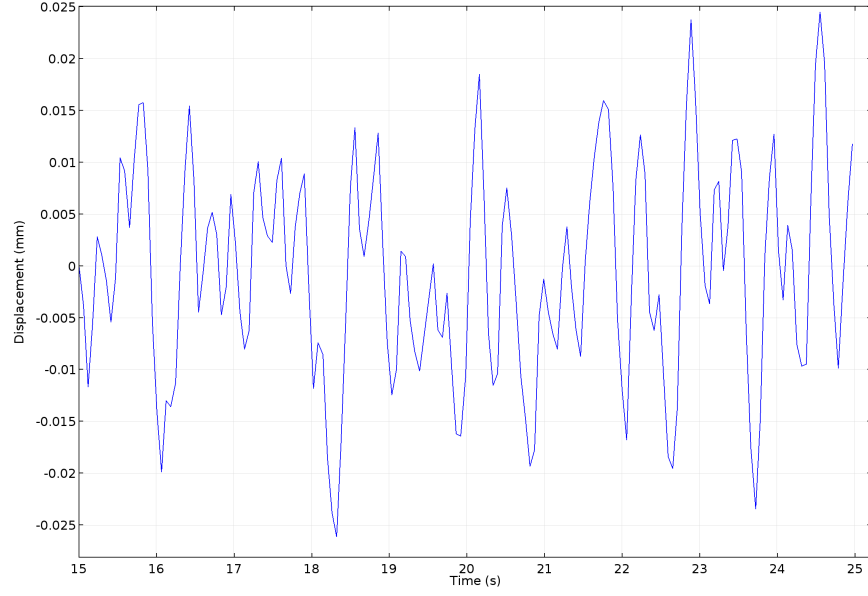


Figure 4.27: Displacement for the 3-D Liu simulation with an inlet velocity of 1.85 m/s with a free time-step and fixed leading- and trailing-edges.

Table 4.5: Comparison of the deflection frequency and solution time for the 3-D case with fixed leading and trailing edges.

Inlet Velocity: 1.85 m/s	Max Time-step (s)	Experimental Frequency (Hz)	Simulation Frequency (Hz)	Solution Time (s)
Restricted max. time-step	0.0215	1.1620	0.601	354 208
Free max. time-step	0.1167	1.1620	—	483 274

3-D Fixed Corners Simulation

As expected, the simulation holding the entire leading and trailing edges fixed provided incorrect results. In order to simulate the experiment more accurately, a final case with only the four corners held fixed and the leading and trailing edges allowed to freely move was created. A mesh consisting of 58 248 elements was used to determine the appropriate maximum time-step. Although this mesh satisfies the $k-\epsilon$ y^+ values on all of the walls, an even finer mesh with 288 880 elements was used and evaluated. This solution was obtained running on the betty cluster with 64 cores for approximately 3 months. The displacement of the middle of the plate is provided in Figure 4.28 and a progression of the 3-D displacement of the plate is provided in Figure 4.29

This final mesh solution with 288 880 elements provided the closest prediction of the displacement and frequency presented by Liu et al. The experimenters attempted to keep water from mixing between the bottom and top of the plate by placing the plate as close to the side walls as possible. Even though they were successful in doing this, it is likely fluid was exchanged between the bottom and top of the plate and this exchange of momentum likely altered the damping effects of the fluid and allowed larger displacements and higher frequencies than if the plates were truly water-tight. The experimenters only observed the displacement and frequency from one location on the plate and Figure 4.29 clearly shows the maximum deflection may not be in that location. Another drawback to the laser measurement methods is its susceptibility to errors from imperfections in the plexiglass housing.

Table 4.6: Frequency results for the 3-D simulation with only the 4 corners fixed similar to the experiment.

Inlet Velocity: 1.19 m/s	Max Time-step (s)	Experimental Frequency (Hz)	Simulation Frequency (Hz)	Solution Time (s)
58 248 elements	0.1384	0.2924	0.172	813 645
58 248 elements	0.0433	0.2924	0.287	882 441
288 880 elements	0.0433	0.2924	0.312	7 884 000

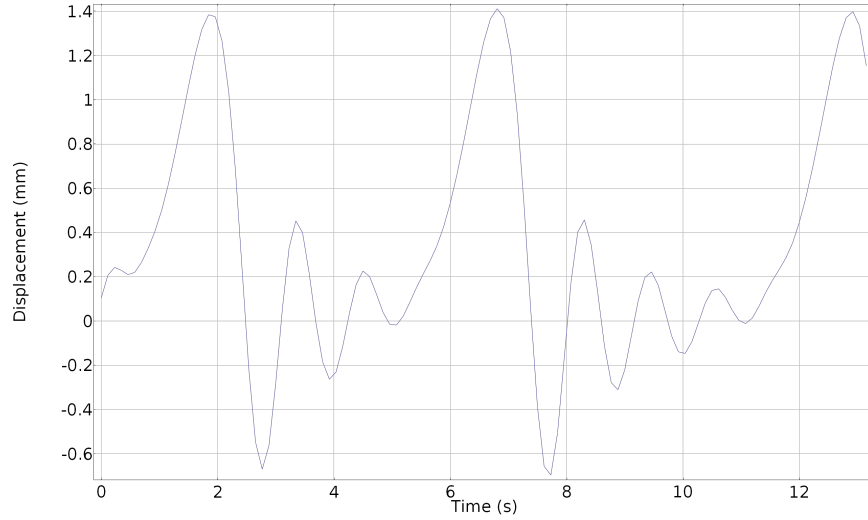


Figure 4.28: Displacement for the 3-D Liu simulation with free leading- and trailing-edges. The simulation has an inlet velocity of 1.19 m/s with a final mesh of 288 880 elements and a maximum time-step of 0.043 25 s which is 1/80th the predicted period.

4.4 Curved Plates

Now that appropriate simulation techniques have been established with verification using flat plate simulations, analysis of curved plates can be conducted. During the literary search, the majority of analyses, both experimental and numerical, has been dedicated to flat plates. As stated earlier, the large design effort for the ANSR included deflection measurements of representative curved, involute plates which is used as validation of the deflection of the ANSR and HFIR plates.

4.4.1 Advanced Neutron Source Reactor

The Advanced Neutron Source Reactor (ANSR) was a proposed high-performance, heavy-water research reactor at ORNL. The reactor was to consist of involute fuel plates in a similar shape to the HFIR with cooling flow rates at approximately 25 m/s. The reactor design was to meet or exceed the neutron flux characteristics of the HFIR

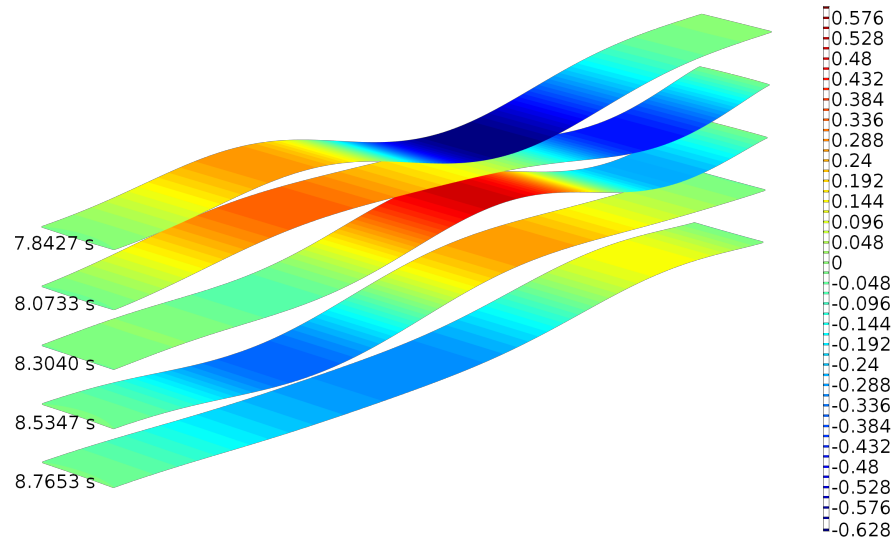


Figure 4.29: Progression of the 3-D displacement for an inlet velocity of 1.19 m/s for the 3-D Liu simulation with free leading- and trailing-edges.

making it the highest neutron flux reactor in the world. Due to the extremely high flow rates, extensive analysis was done to ensure the stability and integrity of the fuel plate structure in the reactor [36, 41, 97–100, 102, 103].

The ANSR was designed to have two fuel cores similar to the HFIR except the cores in the ANSR were to be stacked one after the other instead of the concentric configuration of the HFIR. The conceptual core design is provided in Figure 4.30.

The flow through the core is from the bottom to top of Figure 4.30 and the nomenclature of upper element and lower element will be used to describe the larger and smaller cores, respectively. The upper element consists of 432 plates with an inner diameter of 175 mm and an outer diameter of 235 mm; the lower element consists of 252 plates with an inner diameter of 102 mm and an outer diameter of 168 mm. Each fuel element is 527 mm long with 507 mm being fueled.

During the design process for the ANSR, many experiments were performed in order to establish a safety basis for the reactor operations and to better quantify

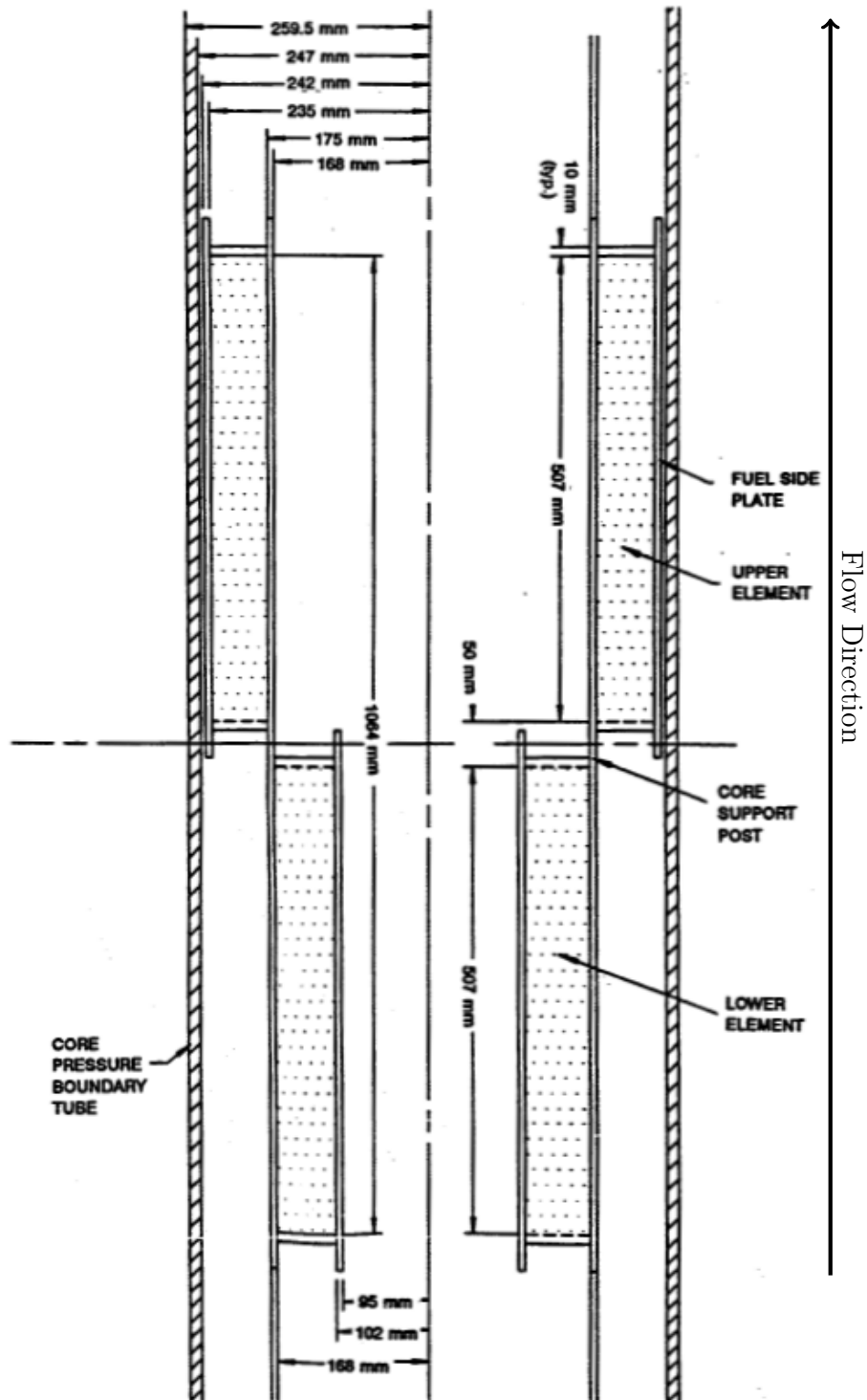


Figure 4.30: Proposed configuration of the ANSR core [102].

the design requirements. Flow tests measuring thermal characteristics of the fuel plates were performed that were designed to measure the heat transfer capabilities of the design [101, 104–106]. Another set of experiments were performed to test the deflection caused by the pressure differentials from the coolant flow [36, 41, 97–100, 102, 103].

The emphasis of this section will be on the experiments performed to establish the deflection characteristics of the plates. The experimental setup consisted of five “fuel” plates made out of PVC plastic. The experiment was run on plates with the involute shape of the upper element with six full flow channels around the plates. A dimensional analysis was performed by the researchers in order to predict the leading-edge deflection for a series of aluminum plates. The analysis, which is described by Swinson et al. [41], assumes that the Poisson’s ratio of each material is approximately the same and thus the measured deflection of the PVC plate (referred to as the model) equals the deflection of the aluminum plate (referred to as the prototype), leading to the following equations:

$$\delta_p(\text{prototype deflection}) = \delta_m(\text{model deflection}) \quad (4.3)$$

$$V_p = V_m \sqrt{E_p/E_m} \quad (4.4)$$

Using Eq. (4.4), Table 4.7 presents the inlet velocities for both the prototype and models along with the Re of each inlet velocity. The Reynolds number in Table 4.7 is

Table 4.7: Inlet velocities for the ANS prototype and model simulations and experiments.

V_m , m/s	Re_m	V_p , m/s	Re_p
3.58	9093	17.35	44 069
5.18	13 157	25.09	63 729
6.65	16 891	32.24	81 890
8.32	21 133	40.32	102 413
10.03	25 476	48.63	123 520

calculated using the assumption that the hydraulic diameter, d_h , can be set to twice the channel thickness for a channel whose width is much greater than its thickness. For the upper element, the channel width, also the arclength of the involute curve, is approximately 71.2 mm long while the channel thickness is 1.27 mm; thus, Re is determined from twice the channel thickness of 1.27 mm.

The deflection of the plates was measured using strain gauges located at 5 evenly spaced locations along the length of the plates. The deflection was measured at the leading and trailing edges and also at the 1/4, 1/2, and 3/4 locations along the plate. An inlet section of 527 mm was used. The outlet section length was not specified. Table 4.8 provides the physical properties for both the PVC and aluminum plates used for the analysis. The density of the PVC plate was not specified. Therefore, an average of the density range found in Titow [107] was used for these simulations.

Table 4.8: Properties used for the ANSR experiments.

Material	E , GPa	ρ , kg/m ³	ν
PVC	2.937	1350	0.35
6061 Aluminum	69	2700	0.33

In order to model the plates as closely as possible to the specified design criteria, SolidWorks [108], a CAD software, was used to create the involute geometry used for the ANSR plates. The equations and background of the involute shape are discussed in Section 1.2. The geometry was then imported into COMSOL for the analysis.

One Plate: Simulation and Results

In order to understand the physical phenomena and to incrementally increase the complexity of the involute problem, a single-plate, two-channel model is used as the first step for the simulation of the ANSR. The Smissaert one plate simulation utilizing periodic boundary conditions does not create a large enough pressure difference so

that the FSI simulation results in little or no deflection and this proved to be true for the involute shaped plates as well. This small deflection was shown to actually create an instability in the code which would prevent convergence to a stable solution. In order to work around this problem, wall boundary conditions are used in place of the periodic conditions while also utilizing the full channel thickness for the fluid domain. This technique, as shown previously, produces accurate leading-edge deflections, as well as, provides faster running times. This is attributed to a smaller computational domain needed for one plate. Furthermore, a more efficient meshing technique can also be established, and also contributes to a faster-running simulation.

Inlet deviations play a large role in the size of deflections seen at the inlet [43]. A deviation of 5 mil was used on the concave side of the channel in order to create a pressure differential across the plate. This deviation from the nominal as-designed distance was chosen because the allowed manufacturing deviations for the HFIR are approximately 6 mils as stated in the SAR [5].

All of the flows evaluated for the simulation, as related to those performed in the experiment and presented in Table 4.7, are turbulent and thus the k - ϵ turbulent model described earlier was used for all simulations. To avoid the use of artificial dissipation to stabilize the flow, the steady-state solver was run using a pseudo-time stepping scheme.

The ANSR experiments were performed using plastic plates at a lower speed than what the ANSR would experience. The researchers used Eq. (4.4) to determine the velocity that would cause similar deflections in the aluminum plates. Single plate simulations were performed for both plates made of different materials and their respective inlet velocities. The final mesh consisted of 270 264 elements and a close-up view of the leading-edge mesh is provided in Figure 4.31. The leading-edge of the fuel plate produced the s-shaped deflection in Figure 4.32. This unusual deflection was predicted by Sartory [98] and has been confirmed by analyzing the eigen modes of the involute plate with the first mode showing the s-shaped deflection. The deflections for the aluminum plate are provided in Figure 4.33.

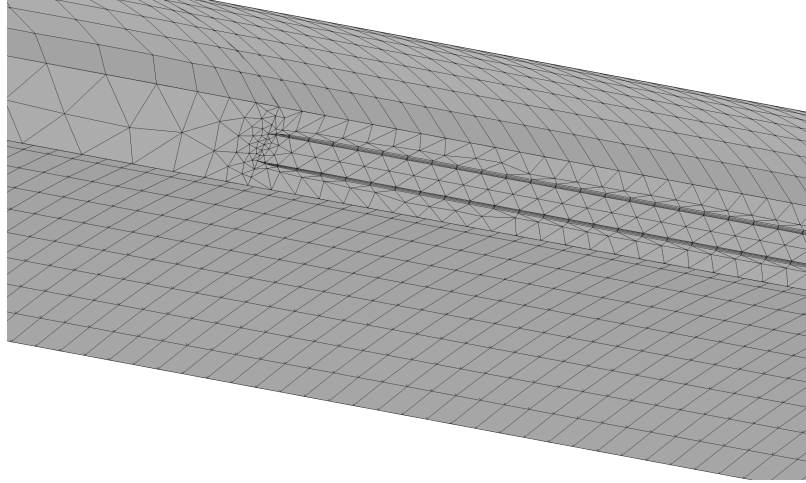


Figure 4.31: The mesh around the leading-edge of the single ANSR simulations with wall boundary conditions.

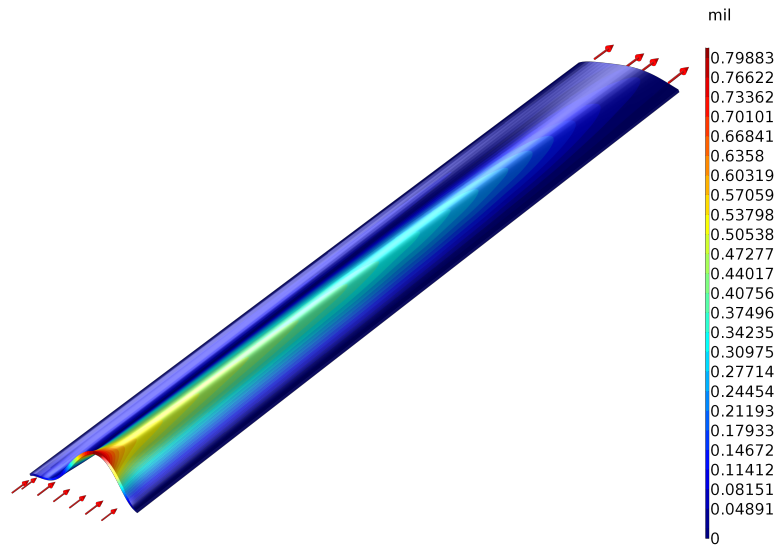


Figure 4.32: The leading-edge deflection of the aluminum ANSR simulation showing the ‘S-curve’ deflection with an inlet velocity of 17 m/s. The simulation consisted of a single ANSR plate with wall boundary conditions.

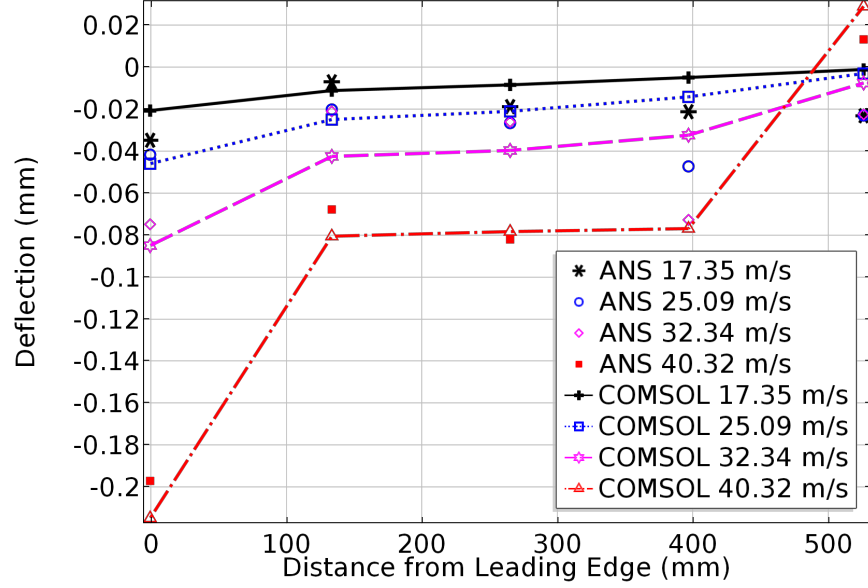


Figure 4.33: Deflections of the single plate ANSR simulations with wall boundary conditions. The simulations are compared to the deflections measured by the experiment performed by Swinson et al. [41]

As can be seen, the agreement between the current simulations and the experimental data is very good. The simulations of the ANSR show that the fully-coupled approach works very well to reproduce the deflections in the experiment. The deflections, particularly at the leading-edge of the plate, match very well with the experiment. The ability of the code to accurately capture the leading-edge deflection at the high coolant speed of 40 m/s provides confidence that the deflections can be accurately predicted for the HFIR which incorporates coolant speeds closer to 15 m/s. It must also be noted that for the highest flow speed (40.32 m/s) the experiments indicate “snapping” at the trailing edge, which was also predicted by the numerical simulations. With the established confidence, predictions for the HFIR fuel plate are presented next.

Two Plate: Preliminary Simulation and Results

In addition to the 1 plate simulations presented above, a 2 plate simulation was performed to verify that the interaction between the involute plates behaves the same as what was observed with the Smitsaert simulations. The PVC plate was used with an inlet velocity of 3.58 m/s which translates to 17.35 m/s for an equivalent aluminum plate. The plates were placed 50 mil apart and as with the Smitsaert case, the offset needed for the single plate deflection was not necessary. The mesh for the simulation was coarser than usual as this was a demonstration of the ability of the solver to capture the interaction between the two plates and consisted of 153 495 elements; the leading-edge mesh is provided in Figure 4.34. The simulation was run using the steady-state solver and the leading-edge deflection is provided in Figure 4.35.

The deflections presented in the simulations for the 2 plate case, although they have a coarser mesh, provide similar results to the simulation of the single plate. The maximum deflection of this case is approximately 0.0167 mm while the single plate case resulted in a maximum deflection of 0.023 mm. Previous studies have shown that refining the mesh typically results in larger deflections, moving toward a mesh-converged solution which indicates that the deflections of this simulation should show similar results.

4.4.2 HFIR

Because the experimenters for the ANSR deflection measurements had a test stand capable of incorporating involute fuel plates, a series of tests for the HFIR inner fuel element (IFE) were conducted. Instead of the 5 plate setup used for the ANSR, the experimenters only set up a one-plate experiment and only measured deflections at the leading-edge of the plate. Simulations for the plate with the properties of the PVC plate were used again as validation and deflection of an aluminum plate, with and without LEU fuel, are predicted.

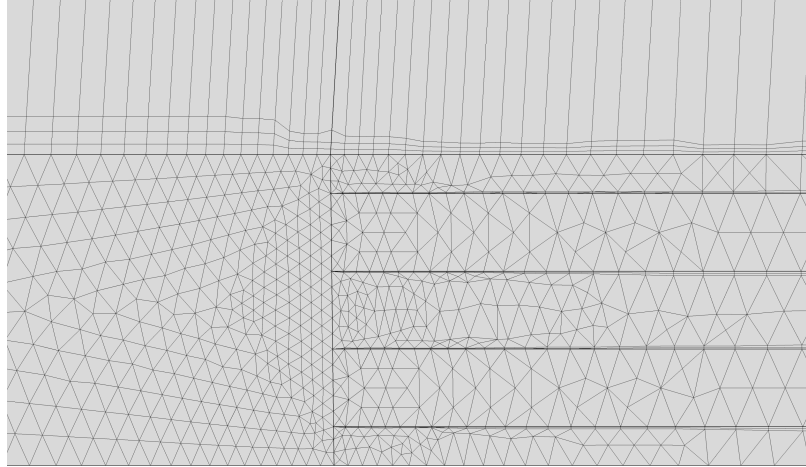


Figure 4.34: The mesh around the leading-edge of the ANSR simulation for 2 plates with wall boundary conditions and an inlet velocity of 3.58 m/s.

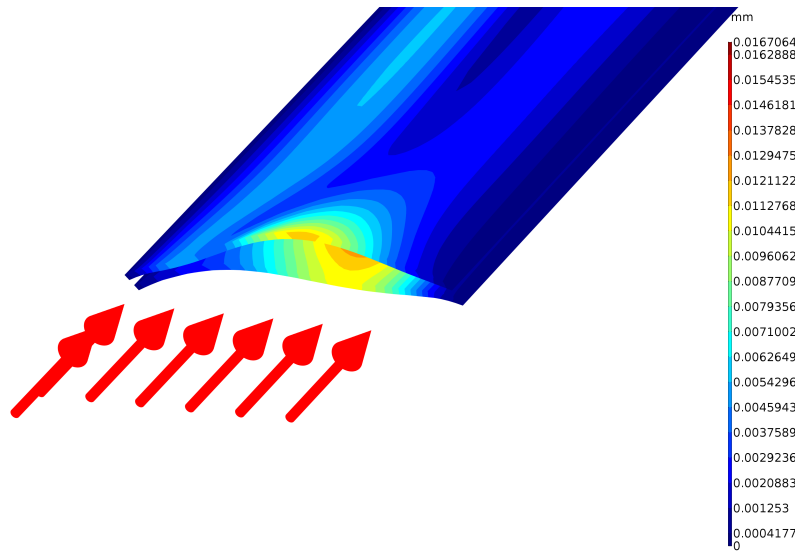


Figure 4.35: Leading-edge displacement of the 2 plate ANSR simulation at an inlet velocity of 3.58 m/s with the PVC plates. The displacement has been exaggerated by 250 times in order to show the deflection of the plates. The plates exhibit the alternating deflections observed by Groninger and Kane [34].

The simulation of the PVC plate consisted of a homogeneous plate with a similar mesh structure as that used for the comparison of the ANSR plates. The simulations were performed using the steady-state solver with the inlet velocities reported by the experimenters. The results of the simulations, compared to the experimental results, are provided in Figure 4.36. These results show acceptable agreement with the leading-edge deflections predicted using COMSOL which is consistent with previous simulations of for the Smissaert and ANSR experiments.

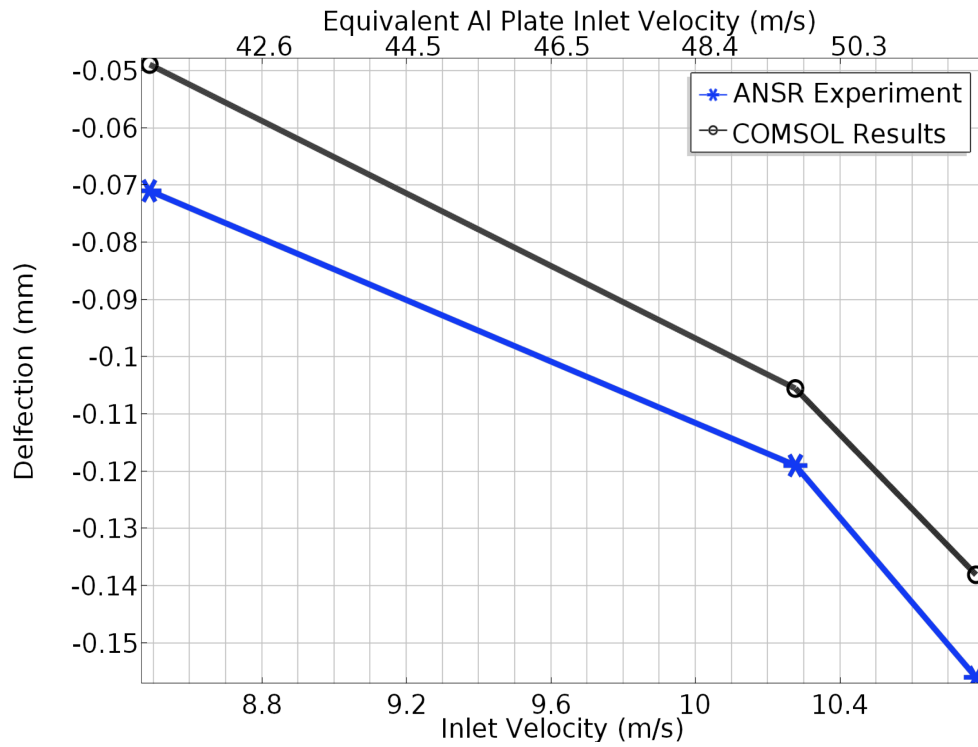


Figure 4.36: Comparison of the leading-edge deflection of the HFIR fuel plate simulations using the properties of the PVC plate compared to the experiment by Swinson et al. [41]. The equivalent inlet velocities for an aluminum plate are provided at the top of the figure for reference.

As stated a number of times in this dissertation, it is desired to use COMSOL to predict the deflections of the HFIR fuel plates. Using the LEU fuel profile in Figure 1.6 with the properties of the proposed Uranium-10Molybdenum (U-10Mo)

deflection predictions will be calculated and compared to a solid aluminum plate. The U-10Mo fuel consists of low-enriched uranium fuel mixed with 10% by molybdenum to form a solid fuel [27]. In comparison, the current HEU fuel is a dispersion-type fuel meaning it is a compressed powder. To create the fuel plates, two flat, aluminum plates with the fuel profiles removed, are rolled together with the fuel inside. The rolled, flat plates are then formed into the involute shape.

Inclusion of the LEU fuel profile introduces complexity into the creation of the mesh for the simulations. In the past, a swept mesh could be used in order to generate efficient meshes as seen with the flat plate comparisons. This can also be done with the involute plates if there is no fuel meat inside of the plates. In order to accommodate the fuel meat, a mapped mesh was created for the flow region and then converted into tetrahedrals. A free mesh was then generated for the aluminum plate and fuel meat. The mesh, which consists of 958 649 elements and is presented in Figure 4.37. This mesh was used for the simulation of both the solid aluminum and LEU-fueled plates in order to compare simulations without differences in mesh.

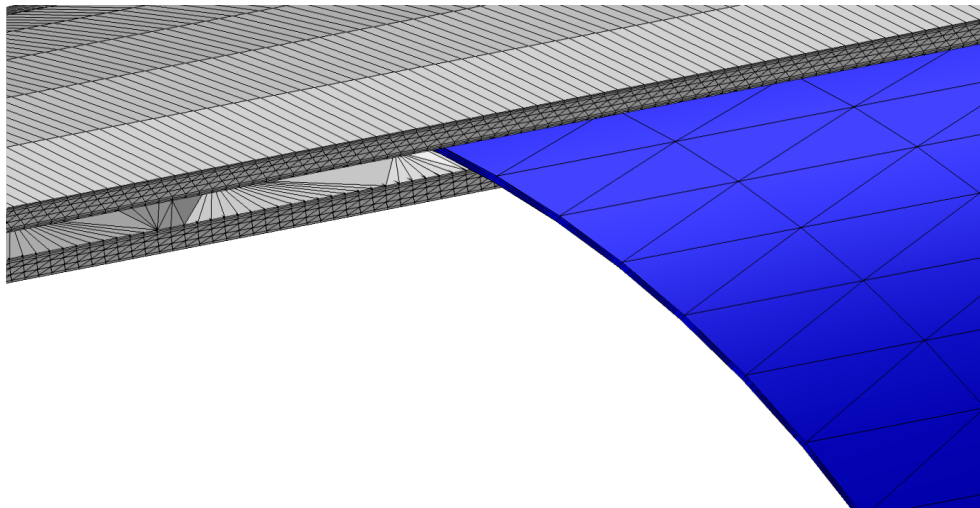


Figure 4.37: Mesh consisting of 958 649 elements for the inner fuel element of the HFIR with the inclusion of the LEU fuel. The grey area is the fluid domain and the LEU fuel meat is highlighted in blue.

The simulations were performed beginning with a coarse mesh of 322 163 and using the previous solutions of the coarser meshes to end with a refined mesh of 958 649 which satisfies the y^+ requirements on all wall boundaries. The all-aluminum and LEU simulations were performed with the same meshes but the mechanical properties of the fuel domain were changed to reflect either the aluminum or U-10Mo fuel properties. The mechanical properties of both the aluminum and U-10Mo used for the simulations are provided in Table 4.9.

Table 4.9: Mechanical properties of the U-10Mo LEU fuel [27] and aluminum used for the HFIR plate simulations.

Property	Aluminum	U-10Mo Fuel
Modulus of Elasticity	69 GPa	67.75 GPa
Poisson's ratio	0.33	0.35
Density	2700 kg/m ³	17 140 kg/m ³

Simulation and Results

The simulations for the HFIR IFE with LEU fuel were done to closely match the conditions seen in the HFIR core. As such, the inlet velocity was set to a nominal velocity of 15 m/s. The simulations of both the fueled and unfueled plates were performed on a cluster with 84 cores using the steady-state, fully-coupled solver. The total deflections of the unfueled and U-10Mo plates are provided in Figures 4.38 and 4.39, respectively. The maximum displacement of the plates was found to be 0.023 752 mm and 0.023 661 mm which translates to 0.935 118 mil and 0.931 535 mil for the solid aluminum and U-10Mo fueled plates, respectively. There is a slight decrease in maximum deflection for the U-10Mo fueled plate that may be attributed to the difference in the Poisson's ratio between the materials. The difference is so small that it is most likely unmeasurable with currently available techniques.

The comparison of the leading-edge deflections is provided in Figure 4.40. As expected, the results show the ‘S-shape’ as well as the slightly smaller deflection of the fuel plates. Comparison of the difference in deflection for the entire plates is provided in Figure 4.41. This figure shows the biggest difference in deflection occurs at the leading-edge, in particular where the largest part of the ‘S-shape’ occurs.

Comparison of the HFIR IFE fuel plate with and without the LEU fuel meat included shows that the maximum deflection differs in the fourth decimal place of the deflection. Because the difference in deflection of the fuel plate with and without the LEU fuel meat is small, it shows that inclusion of the fuel meat in deflection calculations is not necessary. Further adding to this is the fact that inclusion of the fuel meat greatly increases the complexity of the mesh creation and neglecting the fuel meat would allow for a better optimized mesh for the geometry of the involute plates.

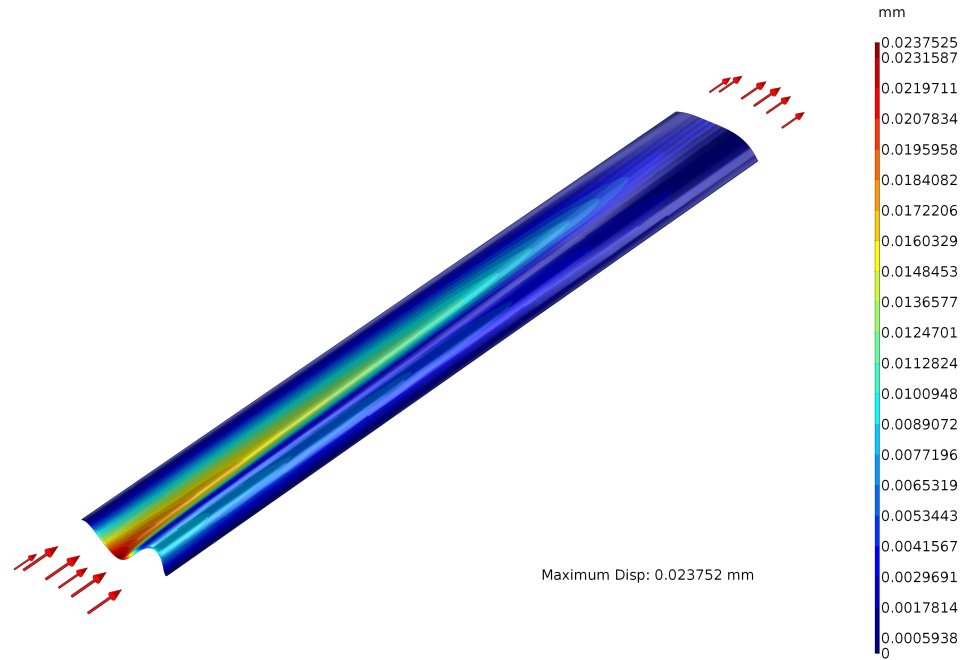


Figure 4.38: Deflection of the HFIR IFE plate made of solid aluminum with a coolant velocity of 15 m/s. The single plate simulation uses wall boundary conditions.

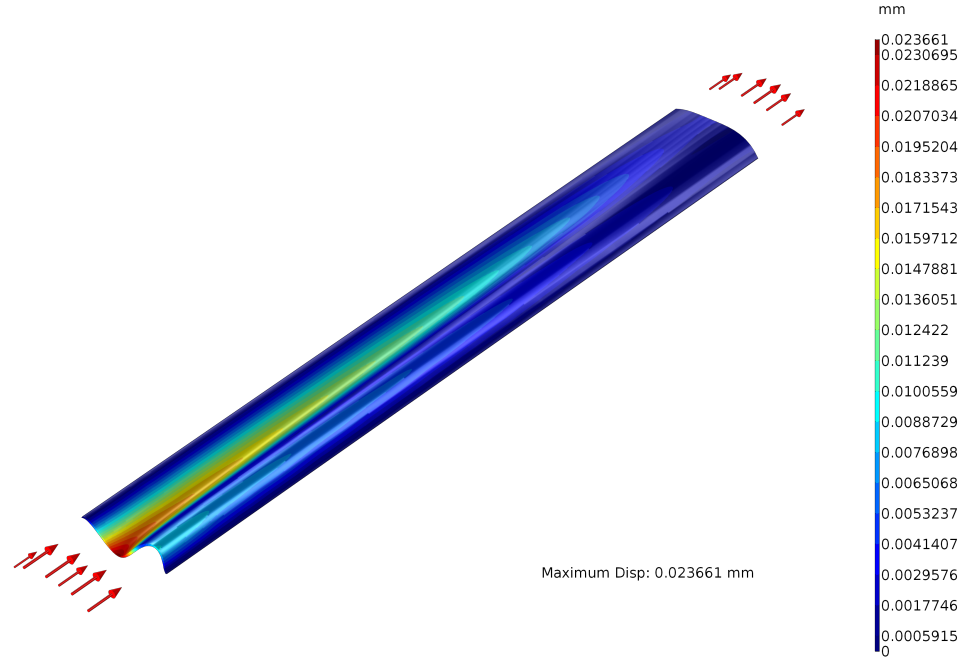


Figure 4.39: Deflection of the HFIR IFE plate with the LEU fuel profile of U10Mo with a coolant velocity of 15 m/s. The single plate simulation uses wall boundary conditions.

The simulations of the HFIR plate show a maximum deflection of approximately 1 mil at the leading-edge of the plate. This deflection is well within the tolerances for plate spacing within the SAR and reduction in flow is minimal. Assuming adjacent plates deflect in the opposite directions as seen with the Smissaert and 2 plate ANSR simulations, the largest increase and decrease in distance between plates is only 2 mil which is still within the manufacturing tolerances of the SAR. A significant amount of time has been spent learning the best practices for using COMSOL to effectively simulate the fluid-structure interaction of multiple involute plates and the next chapter provides the lesson learned while using the commercial code in order to provide insight for future work using COMSOL for FSI problems.

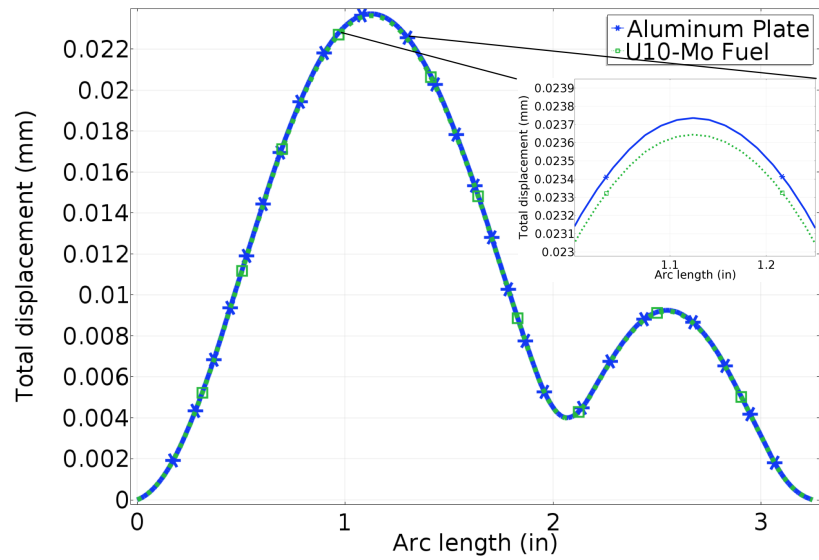


Figure 4.40: Leading-edge deflection of a solid aluminum IFE plate compared to the IFE plate with the LEU fuel profile of U10Mo. The single plate simulation uses wall boundary conditions. The simulation has an inlet velocity of 15 m/s and uses wall boundary conditions.

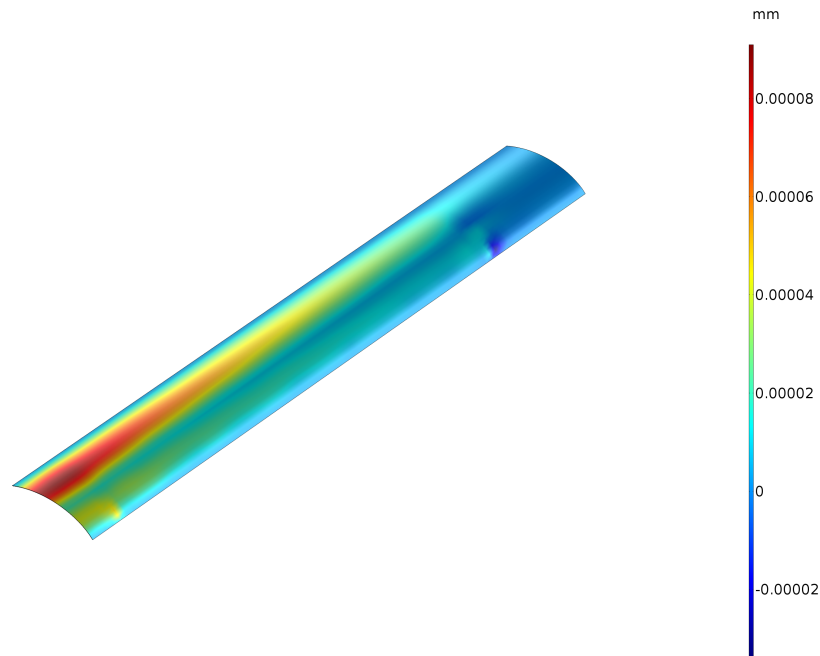


Figure 4.41: Difference in the deflection in the area of the plate where the LEU U10Mo fuel meat is located with a coolant velocity of 15 m/s.

Chapter 5

Lessons Learned

COMSOL Multiphysics is a very powerful platform for analyzing very complex systems with a wide variety of physics. In order to facilitate this ability, the code allows for a wide range of options in order to obtain stable and realistic solutions. This chapter is going to lay out the best practices found for simulating large deflection FSI problems with COMSOL. The COMSOL code splits up the simulation using a tree-like structure shown in Figure 5.1 guiding the user through the steps of the simulation starting with global definitions, moving onto geometry creation, mesh generation, solver settings, and finally post-processing.

5.1 Calling External FORTRAN Functions

COMSOL allows for the specification of global variables and functions to be used by the code. The functions can be specified using built-in functions such as analytic, interpolated from a table, various ramping techniques as well as external functions (c++, etc) and MATLAB functions. Previously, the NIST steam tables were implemented using FORTRAN source code. COMSOL does not allow a direct link for a FORTRAN code but will allow the use of a MATLAB function directly implemented into the code. It is possible, with some extra work, to compile a FORTRAN code in MATLAB and then be returned a MATLAB function. The command in MATLAB

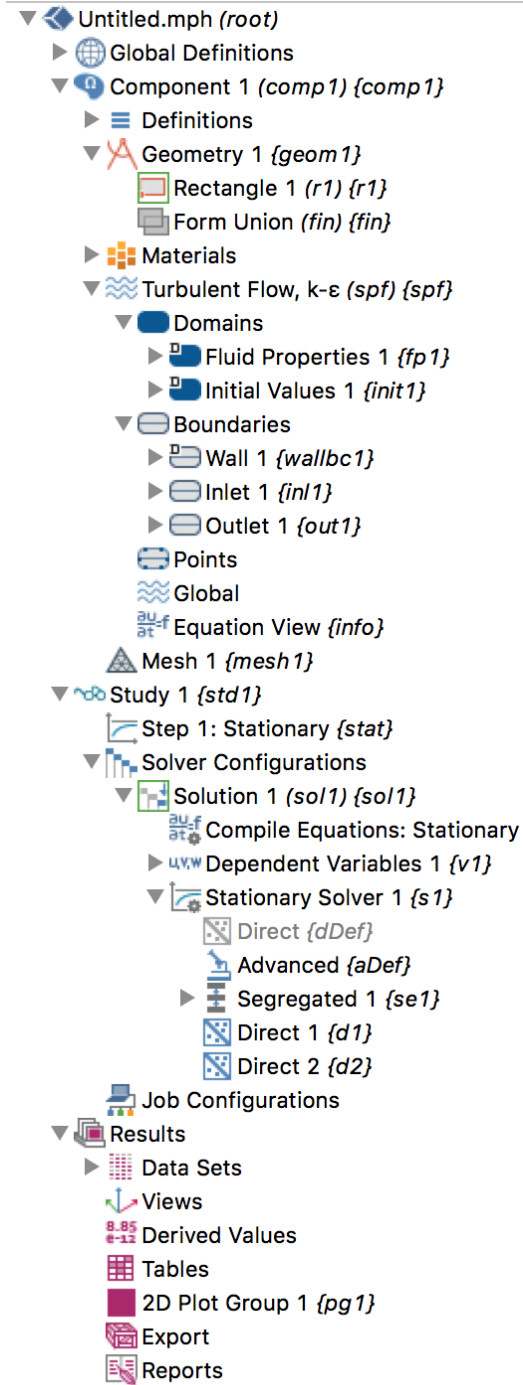


Figure 5.1: The tree-like structure used in the COMSOL code to control the simulation.

is called “mex” and takes a file ending with the extension “.F” and returns a file, actually a MATLAB function, with the extension “.mexa64.” This file can then be run inside MATLAB just as any other function is run: using input arguments to return some output arguments in the form $a = \text{func}(x,y)$.

The NIST FORTRAN file was successfully compiled inside MATLAB resulting in a .mexa64 file. The compiled code has been compared to the steam tables available in Thermodynamics: An Engineering Approach [109] with great accuracy. This code has been implemented into the COMOSL GUI by creating a MATLAB function that calls the mex-function. This implementation was made possible after several discussions with the COMSOL support group. It was revealed that COMSOL sends its variables as vector arrays of 1xN while the mex-function was created to only accept a scalar variable that is a 1x1 variable. In using the NIST code implemented in MATLAB, the mex-function was called as follows:

```
output = input(press,temp)
```

Where output is the desired output from the mex-function (density, thermal conductivity, etc) and input is the mex-function. The mex-function only accepts one value for press and temp where COMSOL gives an array of variables. Using this knowledge allowed the creation of the following MATLAB function to call the mex-function:

```
function output = COMSOL_function(press,temp)
for ii = 1:length(press)
    output(ii) = mex_function(press(ii),temp(ii))
end
return
```

The above MATLAB function has allowed the implementation of the NIST steam tables into COMSOL. Unfortunately, this passing of the vector and the subsequent for-loop increased the computation time for a very small problem by orders of magnitude.

It is much better to create a look-up table with the thermodynamic properties covering the temperatures and pressures seen in the analysis and use the interpolation function built into COMSOL to obtain the correct properties.

5.2 Geometry Creation and Meshing Techniques

In order to properly capture the physics being simulated, the geometry created using the code must represent the actual setup of the experiment or physical system being analyzed. The first step is to use the geometry builder in COMSOL. For simple geometries, such as the Smitsaert flat plate experiments, the built-in geometry creation tools are sufficient to generate the geometry. As the complexity of the geometry increases, the CAD Import Module can be used and was used for the implementation of the involute-curved plates in the HFIR and the ANSR experiments.

A note of caution: when using the CAD Import Module, a license is checked out for the entirety of the simulations and once the CAD Import Module is utilized in a model, it is very difficult to release that license without recreating the geometry and losing all of the solutions accompanying the geometry. If one thinks a geometry can be created without the CAD Import Module, then that is the best route to pursue if CAD Import Module licenses are scarce.

The geometry can also be used to better control the mesh generation. Although COMSOL has automatic mesh generation capabilities, the code attempts to use tetrahedral meshes along with boundary layer meshes to capture the physics. For the high-aspect ratio geometries seen in the research reactors, the default generated meshes tended to contain small mesh elements which drove up the computational cost of the analysis (Figure 5.2). Using what COMSOL calls the mapped mesh (which looks very much like a structured mesh) allows for careful control of the mesh size in the areas of interest and large gradients.

Using the Smitsaert simulation as an example, the mapped mesh was created by splitting the computational domain into smaller subdomains and using the newly

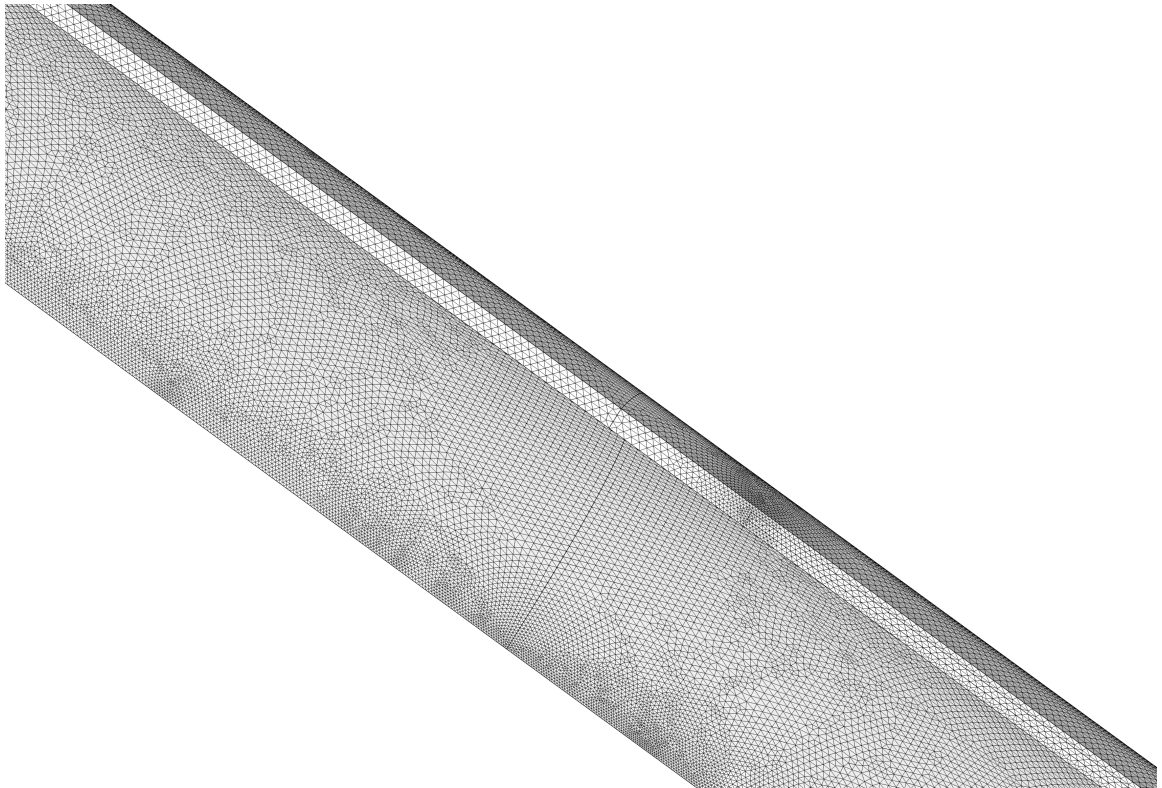


Figure 5.2: Default generated mesh using free-tetrahedrons and boundary-layer meshes.

created boundaries to control the mesh size. In order to avoid computational issues, the subdomain boundaries were removed by using the Virtual Geometry Features of the code and specifying the subdomain boundaries as ‘Mesh Control Edges and Faces.’

Once the geometry is created, it is now time to begin the mesh generation. For the majority of the flat plate FSI simulations, a mapped mesh was created on one of the side plate faces and the mesh was then swept to generate the 3-D geometry. On the face where the mapped mesh is being created, a distribution element is used to control the aspect ratio of the mesh.

By default, COMSOL assumes there is a combination of free triangular/tetrahedral mesh and mapped meshes that need to be properly combined in order to reduce computational errors. When using the Virtual Geometries setting in the geometry creation, COMSOL attempts to smooth the mesh transition across the boundary being removed. This setting called ‘Smooth across removed control entities’ is checked by default for most mesh generation settings (Figure 5.3) and needs to be unchecked in order to avoid skewed meshes, especially for mapped meshes such as in Figure 5.4. Not only does this negate the effort of manually controlling the mesh, this smoothing step also adds to the computation time required to generate the mesh by the code.

Once the mesh has been sufficiently generated on one face of the geometry, it is most efficient to sweep the mesh across the domain. For the reactor plate geometries, the mesh is swept across the span of the plate, perpendicular to the coolant flow direction. Similar to the mapped mesh on the boundary, the swept mesh can be controlled with the distribution node allowing for precise control over the number of elements created by the mesh generation.

Depending on the geometry and mesh generation, it is often necessary to refine the mesh around the wall boundaries of the domain in order to accurately capture the boundary layer velocity profile. In order to facilitate the creation of the refined mesh along the walls, COMSOL has a meshing technique called ‘Boundary Layer Meshing’ that allows fine control over the mesh close to the wall without the need to create

Mapped

 Build Selected  Build All

Label:

▼ **Boundary Selection**

Geometric entity level:

Selection:

 Active

5
31
52
82
109 (meshed and removed)
110 (meshed and removed)
111 (meshed and removed)



▼ **Control Entities**

☒ Smooth across removed control entities

Number of iterations:

Maximum element depth to process:

► **Advanced Settings**

Figure 5.3: ‘Smooth across removed control entities’ check box that is automatically turned on for most mesh generation settings and should be turned off for finer control.

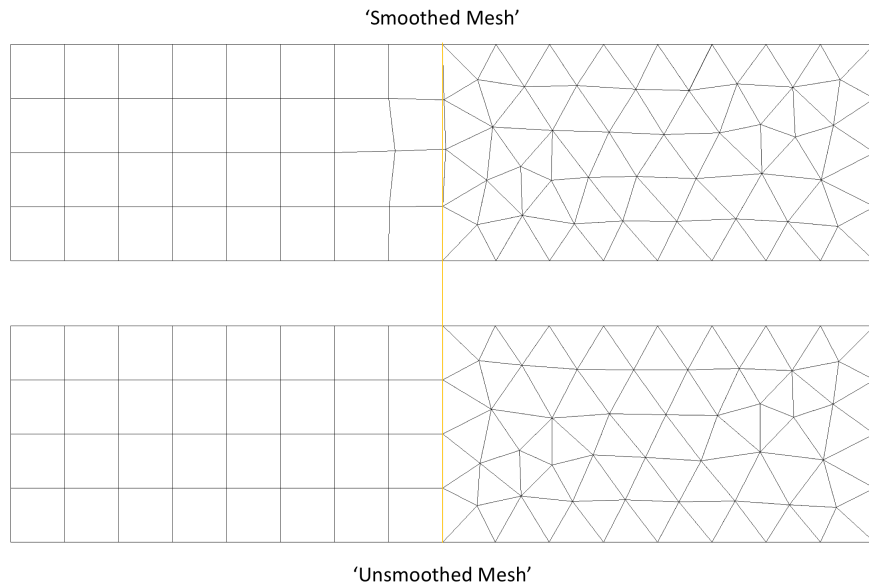


Figure 5.4: Skewed mesh from the ‘smoothing’ of the mesh across a removed boundary.

virtual geometry elements. This boundary layer mesh can be created for both 2-D and 3-D simulations, but the boundary layers can only be created on an existing 3-D mesh when used on a 3-D domain. The boundary layer mesh can also be added to both free tetrahedral/triangular and mapped meshes. Care needs to be taken in order to avoid skewing the meshes when the boundary layer mesh is added in. Figures 5.5 and 5.6 demonstrate what a boundary layer mesh looks like as well the improvement of the velocity profile near the wall.

Figure 5.6 provides the velocity profile along the wall for a $k-\epsilon$ turbulent solution for a mesh with and without the boundary layer. The $k-\epsilon$ turbulent model will not produce a solution in which the velocity along the wall is zero, but the mesh with the boundary layer does produce a solution with a velocity ‘closer’ to zero. Proper y^+ values must be obtained in order to have confidence in the solution. An over prediction of the velocity along the wall could produce an increased heat transfer coefficient which would over predict the heat removed by the cooling fluid of a reactor.

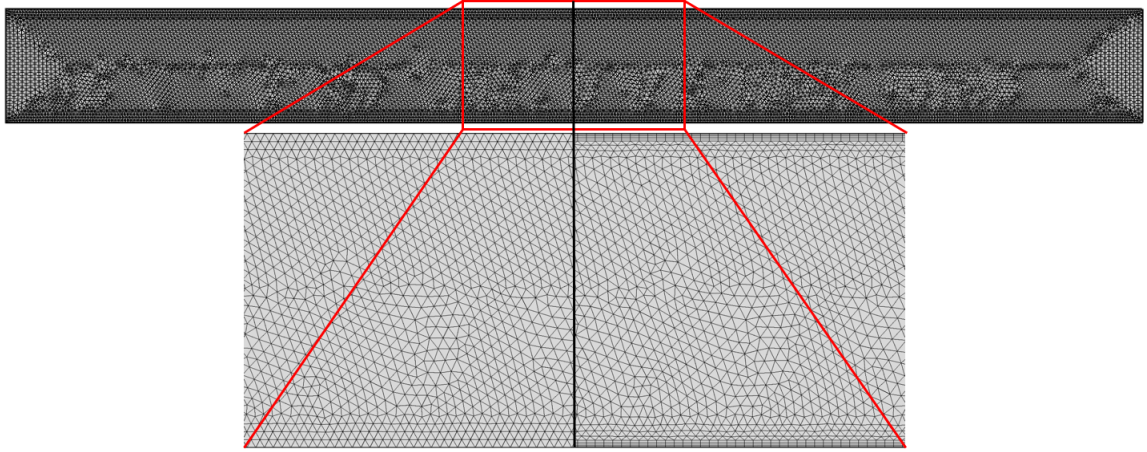


Figure 5.5: Boundary layer mesh along the wall of a fluid domain added to a free triangular mesh. The mesh on the left side is the original triangular mesh and the mesh on the right side has the boundary layer mesh added in.

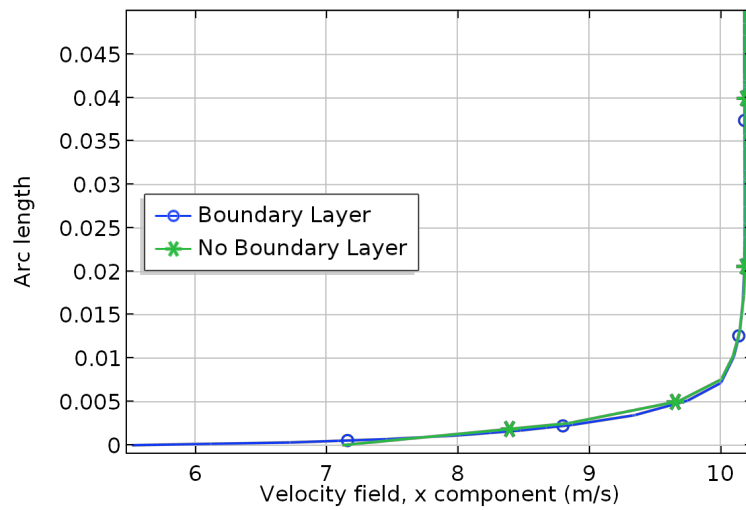


Figure 5.6: Velocity along the wall for a turbulent flow solution comparing a free triangular mesh with and without boundary layer meshes added in. The boundary layer mesh satisfies the y^+ conditions for the $k-\epsilon$ turbulence model.

It is often necessary to refine the mesh in order to produce coarse meshes for initial solution creation and perform mesh refinement studies. In order to avoid creating a new mesh for each refinement in the study, a well-defined mesh can be copied, and scaling can be applied to produce meshes of varying densities. The refinement can go in either direction and is controlled by using the ‘Reference Mesh’ and then selecting ‘Scale.’ The scaling can be controlled on the entire geometry or on specific domains of the geometry and a demonstration is provided in Figure 5.7

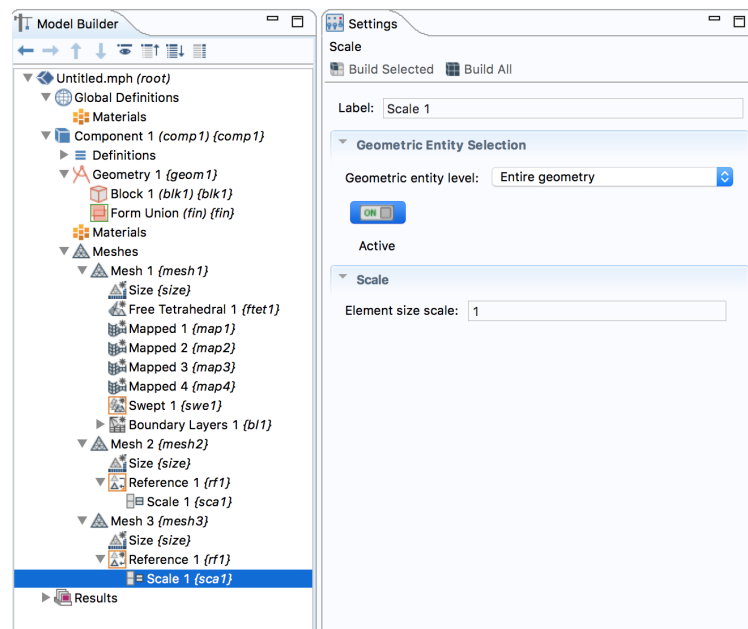


Figure 5.7: Mesh referencing and scaling settings to duplicate and refine/coarsen meshes.

5.3 Solver Settings

COMSOL provides a variety of solvers that can be utilized in order to obtain solutions for varying physics and problem sizes. Even with a well posed simulation, convergence to the correct solution can be difficult if the proper solver settings are not employed.

This section will lay out the best practices for obtaining solutions in COMSOL for the large-deflection, tightly-coupled simulations presented in this dissertation.

5.3.1 Segregated Vs. Fully-Coupled (Monolithic) Solvers

One of the greatest strengths of the COMSOL software is its ability to couple and solve a variety of multiphysics problems. Because of this, COMSOL also allows users to solve the physics using either a segregated or monolithic (COMSOL calls this fully-coupled) approach. This is chosen in the settings for the solver shown in Figure 5.8. By choosing the ‘Fully Coupled’ solver, COMSOL places all of the computational model into one matrix including all relevant physics and boundary conditions, including those coupling the physics. The segregated solver allows for the computation of the physics incorporated into the model to be solved independently and the coupling is performed after each segregated step is computed. For instance, a turbulent flow solution can be computed using the fully-coupled solver in which all of the flow variables (U , p , k , and ϵ) are solved simultaneously or the solution can be segregated where the flow variables (U , p) and the turbulence variables (k , ϵ) are calculated separately.

The segregated solver is well suited for loosely coupled multiphysics problems such as incompressible conjugate heat transfer. By using the segregated solver, the memory requirements can be drastically reduced as each segregated step has a reduction in the number of degrees of freedom and can provide the ability to solve large problems on a RAM-limited system. The segregated approach can also be beneficial for very stiff problems to provide an initial condition for a fully-coupled solver.

COMSOL allows for an unlimited number of segregated solvers to be used and even allows splitting up all variables. The stronger the coupling between variables, the less advisable it is to separate them into separate segregated solvers. Taking the turbulent flow calculation as an example, COMSOL would allow a segregated solver for each variable U , p , k , and ϵ resulting in 4 segregated solvers. This would

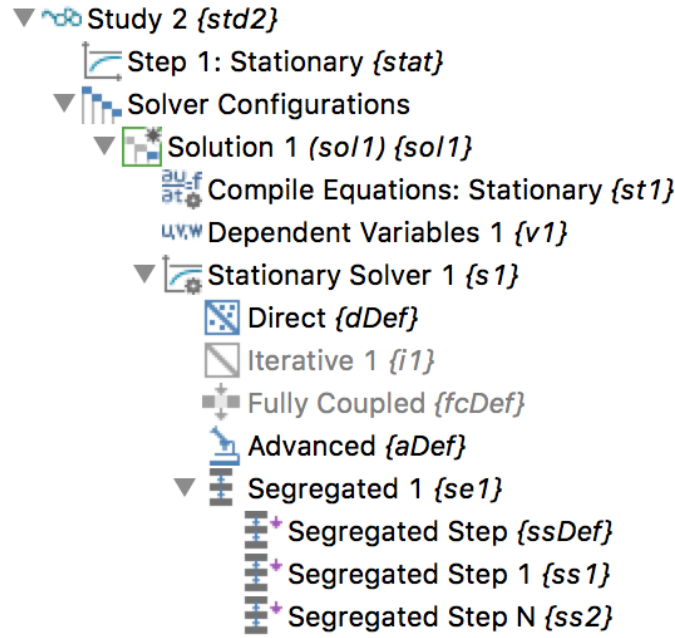


Figure 5.8: Solver options in the COMSOL code.

most likely be very unstable as the fundamental flow variables U and p are strongly coupled.

For FSI problems, COMSOL has an additional segregated solver they call the ‘One-way coupled’ solver. This solution technique solves for the flow variables to convergence and then uses the flow solution as the boundary conditions to solve the structural mechanics displacements. The flow solution is not updated. This solution technique is applicable to very loosely coupled problems where the displacement of the structure does not affect the flow pattern significantly. Once a solution is obtained using this technique, it is advisable to run a coupled solution, even with a coarse mesh, to ensure this approach is appropriate. Being a much less computationally demanding technique, this approach could be used for an engineering estimate of the FSI in certain applications.

As discussed earlier in the dissertation, for large deflection FSI problems, the fluid and structure are tightly coupled. Using the segregated solver provides an unstable solution and a fully coupled solver must be used. This is a much more

robust solver technique that provides the stability needed to obtain large deflection FSI calculations [80, 110]. Unfortunately, the fully coupled approach requires a large amount of memory and can result in long computational times.

5.3.2 Iterative Vs. Direct Solvers

COMSOL also provides a wide variety of solvers for researchers to use in order to make the best use of the computing resources available. There are a number of iterative and direct solvers provided in the code:

- Iterative Solvers
 - ◊ GMRES
 - ◊ FGMRES
 - ◊ Conjugate Gradients
 - ◊ BiCGStab
- Direct Solvers
 - ◊ MUMPS
 - ◊ PARDISO
 - ◊ SPOOLES

The iterative solvers provide a great deal of benefit when used in COMSOL. One of the largest benefits is the significant reduction in memory required compared to a direct solver. If memory is a concern, it is best to attempt to use the iterative solvers if possible. The iterative solvers are also very well suited for massively parallel calculations and speed-up should be observed well beyond what is achievable with direct solvers. As the matrix becomes stiffer, the iterative solvers begin to have more difficulty in obtaining a solution. This was seen with the strongly coupled FSI problems encountered for this dissertation. By nature, the iterative solvers allow for great control and thus have many ‘knobs’ to tune in order to obtain a stable solution; this can take quite some time to master.

The direct solvers provide a means to add stability to the solution technique. There are much fewer options with the direct solver which allows it to be generically applied to a much larger range of problems with very little tweaking. The direct solver proved to be the most stable for the large deflection FSI calculations. The direct solver has drawbacks because of its large memory requirements and the nature of the direct solver does not lend well to massively parallel architectures. Currently, the only direct solver that can be run using distributed memory is the MUMPS solver.

Chapter 6

Conclusions and Recommendations

Multiphysics is part of every engineering analysis. How important it is to couple those physics drives the numerical analysis that needs to be performed. In the past, the limited computational speed and memory sizes of computers required the analyst to find ways to simulate the physics separately. Although this works well for most analyses, this dissertation shows that there are situations where this is not sufficient. As computers have become more powerful, the ability to include multiple physics has become possible with many researchers beginning to combine various codes not designed to handle multiphysics problems. Developing and using computer codes designed with multiphysics in mind creates a powerful tool that can be used to provide insight into areas where experimental research is either not possible or prohibitively expensive.

The conversion of the High Flux Isotope Reactor to LEU fuel provides a great opportunity to explore the possibilities of using a fully-coupled multiphysics code to predict the performance of the reactor. In order to ensure safe operations of the reactor, accurate analysis of the various physics involved in cooling the reactor core is needed and experiments (especially those with LEU fuel) are expensive. In this dissertation, it has been shown that deflections of both flat and curved fuel plates can be accurately predicted using fully-coupled FSI techniques. It may be possible to

include more physics in reactor core simulations using larger computational resources than those used for this dissertation (1000's of cores). As an example, one can couple a thermal analysis to the existing FSI framework to account for thermal-fluid-structural effects, and investigate the impact of flow area reduction on the ability of the fluid to remove heat in an efficient way. Incorporating more physics in the simulations would improve accuracy since the structural deformations are not only driven by the pressures and stresses induced by the fluid but also by the thermal effects.

Furthermore, including more plates in the analysis would lead to a better understanding of the interaction between the plates. This would allow the simulation of 10-20 plates at the same time and would remove the 'end wall' effects, which was also observed with the 5-plate Smitsaert simulations.

Finally, the application of a fully-coupled FSI simulation is not limited to only research-type nuclear reactors. Large deflection FSI is also observed in many engineering applications. The techniques utilized in this dissertation can be used as a building block for researchers and designers to improve upon their understanding of how FSI affects their areas of interest.

List of References

- [1] “Global Threat Reduction Initiative.” National Nuclear Security Administration, April 2014. <<http://nnsa.energy.gov/aboutus/ourprograms/dnn/gtri>>.
- [2] “Reduced Enrichment for Research and Test Reactors (RERTR) [Nonproliferation].” Argonne National Laboratory, April 2014. <<http://www.rertr.anl.gov>>.
- [3] “TRTR Home Page.” The National Organization of Test, Research, and Training Reactors, April 2014. <<http://www.trtr.org>>.
- [4] “Domestic U.S. Reactor Conversions: Fact Sheet.” National Nuclear Security Administration, May 2014. <<http://nnsa.energy.gov/mediaroom/factsheets/reactorconversion>>.
- [5] “High Flux Isotope Reactor Safety Analysis Report.” Tech. Rep. ORNL/HFIR/SAR/2344, ORNL, 2013.
- [6] McLain, H. A. “HFIR Fuel Element Steady State Heat Transfer Analysis: Revised Version.” Tech. Rep. ORNL/TM/1904, Oak Ridge National Laboratory, 1967.
- [7] Morris, D. G. and Wendel, M. W. “High Flux Isotope Reactor System RELAP5 Input Model.” Tech. Rep. ORNL/TM-11647, Oak Ridge National Laboratory, 1993.

- [8] Renfro, D., Chandler, D., et al. “Preliminary Evaluation of Alternate Designs For HFIR Low-enriched Uranium Fuel.” Tech. Rep. ORNL/TM2014/154, ORNL, 2014.
- [9] “X-5 Monte Carlo Team, MCNP–A General Monte Carlo N-Particle Transport Code.” Version 5, LA-CP-03-0245, Los Alamos National Laboratory, April 2003. [Software].
- [10] “COMSOL.” Version 5.2a, 2016. [Software].
- [11] Tschaepe, L., Ruggles, A. E., et al. “Evaluation of HFIR LEU Fuel Using the COMSOL Multiphysics Platform.” Tech. Rep. ORNL/TM-2008/188, Oak Ridge National Laboratory, 2008.
- [12] Freels, J. D., Bodey, I. T., et al. “Two-Dimensional Thermal Hydraulic Analysis and Benchmark in Support of HFIR LEU Conversion using COMSOL.” Tech. Rep. ORNL/TM-2010/018, ORNL, 2010.
- [13] Renfro, D. G., Chandler, D., et al. “Low-Enriched Uranium Fuel Conversion Activities for the High Flux Isotope Reactor, Annual Report for FY 2011.” Tech. Rep. ORNL/TM-2011/507, ORNL, 2011.
- [14] Bodey, I. T., Arimilli, R. V., et al. “Complex Geometry Creation and Turbulent Conjugate Heat Transfer Modeling.” COMSOL Conference: Boston, 2011.
- [15] Freels, J. D. and Jain, P. K. “Multiphysics Simulations of the Complex 3D Geometry of the High Flux Isotope Reactor Fuel Elements Using COMSOL.” COMSOL Conference: Boston, 2011.
- [16] Khane, V. B., Jain, P. K., et al. “COMSOL Simulations for Steady State Thermal Hydraulics Analyses of ORNL’s High Flux Isotope Reactor.” COMSOL Conference: Boston, 2012.

- [17] Khane, V. B., Jain, P. K., et al. “Development of CFD Models to Support LEU Conversion of ORNL’s High Flux Isotope Reactor.” ANS Winter Meeting, 2012.
- [18] Travis, A. R., Ekici, K., et al. “Simulating HFIR Core Thermal Hydraulics Using 3D-2D Model Coupling.” COMSOL Conference: Boston, 2013.
- [19] Wang, D., Jain, P. K., et al. “Application of COMSOL Pipe Flow Module To Develop a High Flux Isotope Reactor System Loop Model.” COMSOL Conference: Boston, 2013.
- [20] Travis, A. R. *Simulating High Flux Isotope Reactor Core Thermal-Hydraulics via Interdimensional Model Coupling*. Master’s thesis, University of Tennessee, 2014.
- [21] Jain, P. K., Freels, J. D., et al. “3D COMSOL Simulations for Thermal Deflection of HFIR Fuel Plate in the Cheverton-Kelley Experiments.” Tech. Rep. ORNL/TM-2012/138, ORNL, 2012.
- [22] Chandler, D. *Spatially-Dependent Reactor Kinetics and Supporting Physics Validation Studies at the High Flux Isotope Reactor*. Ph.D. thesis, University of Tennessee, Knoxville, 2011.
- [23] “The High Flux Isotope Reactor at Oak Ridge National Lab.” Neutron Sciences, March 2014. <<http://neutrons.ornl.gov/facilities/HFIR/history/>>.
- [24] Xoubi, N. and Primm III, R. T. “Modeling of the High Flux Isotope Reactor Cycle 400.” Tech. Rep. ORNL/TM-2004/251, ORNL, 2005.
- [25] Juvinall, R. C. *Fundamentals of Machine Component Design*. John Wiley & Sons, Ltd, 1983.

- [26] Ilas, G. and Primm III, R. T. “Low Enriched Uranium Fuel Design with Two-Dimensional Grading for the High Flux Isotope Reactor.” Tech. Rep. ORNL/TM-2010/318, ORNL, 2010.
- [27] Burkes, D. E., Mickum, G. S., et al. “Thermophysical Properties of U-10Mo Alloy.” Tech. Rep. INL/EXT-10-19373, Idaho National Lab, 2010.
- [28] Gwaltney, R. and Luttrell, C. “Critical Flow Velocities for Involute Parallel Fuel Plate Assemblies.” In “Trans. Am. Nucl. Soc.; (United States),” vol. 57. Jan. 1988.
- [29] Chang, G. S., Ambrosek, R. G., et al. “Advanced Test Reactor LEU Fuel Conversion Feasibility Study.” Tech. Rep. INL/CON-05-00921, Idaho National Lab, 2005.
- [30] “Irradiation Capabilities at the Idaho National Laboratory.” Idaho National Lab, May 2013. <<https://atrnsof.inl.gov>>.
- [31] “MIT Nuclear Reactor Laboratory: Reactor.” The MIT Nuclear Reactor Laboratory, May 2014. <http://web.mit.edu/nrl/www/reactor/core_description.htm>.
- [32] Stromquist, W. K. and Sisman, O. “High Flux Reactor Fuel Assemblies Vibration and Water Flow.” Tech. Rep. ORNL-50, ORNL, 1948.
- [33] Miller, D. R. “Critical Flow Velocities for Collapse of Reactor Parallel-Plate Fuel Assemblies.” Tech. Rep. KAPL-1954, Knolls Atomic Power Lab, 1960.
- [34] Groninger, R. D. and Kane, J. J. “Flow Induced Deflections of Parallel Flat Plates.” *Nuclear Science and Engineering*, 16, pp. 218–226, 1963.
- [35] Smissaert, G. S. “Static and Dynamic Hydroelastic Instabilities in MTR-type Fuel Elements Part 1. Introduction and Experimental Investigation.” *Nuclear Engineering and Design*, 7, pp. 535–546, 1968.

- [36] Swinson, W. F., Battiste, R. L., et al. “Fuel Plate Stability Experiments and Analysis for the Advanced Neutron Source.” Tech. Rep. ORNL/TM-12353, ORNL, 1993.
- [37] Kennedy, J. C., Jesse, C. J., et al. “Fluid-Structure Interaction Modeling and Experimental Benchmarking.” Tech. Rep. TDR-MU-CALC-201210-005A, University of Missouri, 2012.
- [38] Ho, M., Hong, G., et al. “Experimental Investigation of Flow-induced Vibration in a Parallel Plate Reactor Fuel Assembly.” In “15th Australasian Fluid Mechanics Conference,” pp. 13–17. 2004.
- [39] Marcum, W. R. and Woods, B. G. “Predicting the Onset of Dynamic Instability of a Cylindrical Plate Under Axial Flow Conditions.” *Nuclear Engineering and Design*, 250, pp. 81–100, 2012.
- [40] Ha, T. and Garland, W. J. “Hydraulic Study of Turbulent Flow in MTR-type Nuclear Fuel Assembly.” *Nuclear Engineering and Design*, 236, pp. 975–984, 2006.
- [41] Swinson, W. F., Battiste, R. L., et al. “Fuel-Plate Stability Experiments and Analysis for the Advanced Neutron Source.” *Symposium on Flow-Induced Vibration and Noise*, 5, pp. 133–143, 1992.
- [42] Johansson, E. B. “Hydraulic Instability of Reactor Parallel-Plate Fuel Assemblies.” Tech. rep., General Electric Company, 1959.
- [43] Kane, J. J. “The Effect of Inlet Spacing Deviations on the Flow-Induced Deflections of Flat Plates.” *Nuclear Science and Engineering*, 15, pp. 305–308, 1962.
- [44] Scavuzzo, R. J. “Hydraulic Instability of Flat Parallel-Plate Assemblies.” *Nuclear Science and Engineering*, 21, pp. 463–472, 1965.

- [45] Wambsganss Jr., M. W. “Second Order Effects as Related to Critical Coolant Flow Velocities and Reactor Parallel Plate Fuel Assemblies.” *Nuclear Engineering and Design*, 5, pp. 268–276, 1967.
- [46] Wick, R. S. “Hydro-elastic Behavior of Multiple-plate Fuel-assemblies - I: Pressure Wave Propagation.” *Journal of Nuclear Energy*, 23, pp. 387–405, 1969.
- [47] Wick, R. “Hydro-elastic Behavior of Multiple-Plate Fuel-assemblies - II: Hydrostatic Divergence.” *Journal of Nuclear Energy*, 23 (7), pp. 407 – 418, 1969.
- [48] Kim, Y. T. and Scarton, H. A. “Flow Induced Bending of Rectangular Plates.” *Journal of Applied Mechanics*, 44, pp. 207–212, 1977.
- [49] Cekirge, H. M. and Ural, E. “Critical Coolant Flow Velocities in Reactors Having Parallel Fuel Plates.” *Comp. and Math with Appls.*, 4, pp. 153–156, 1978.
- [50] Pavone, S. J. and Scarton, H. A. “Laminar Flow Induced Deflections of Stacked Plates.” *Nuclear Engineering and Design*, 74 (1), pp. 79–89, 1983.
- [51] Kerboua, Y., Lakis, A. A., et al. “Modelling of Plates Subjected to Flowing Fluid Under Various Boundary Conditions.” *Engineering Applications of Computational Fluid Mechanics*, 2, pp. 525–539, 2008.
- [52] Howard, T., Marcum, W., et al. “A Novel Approach to Modeling Plate Deformations in Fluid-structure Interactions.” *Nuclear Engineering and Design*, 293, pp. 1 – 15, 2015.
- [53] Kennedy, J. C., Jesse, C. J., et al. “Experimental Investigation of Deflection of Flat Aluminum Plates Under Variable Velocity Parallel Flow.” Tech. rep., University of Missouri, 2014.

- [54] Guo, C. and Païdoussis, M. “Analysis of Hydroelastic Instabilities of Rectangular Parallel-Plate Assemblies.” *Journal of Pressure Vessel Technology*, 122 (4), pp. 502–508, 2000.
- [55] Kim, G. and Davis, D. “Hydrodynamic Instabilities in Flat-plate-type Fuel Assemblies.” *Nuclear Engineering and Design*, 158 (1), pp. 1–17, 1995.
- [56] Cui, Z.-D., Tang, Y.-Q., et al. “Flow-induced Vibration and Stability of an Element Model for Parallel-plate Fuel Assemblies.” *Nuclear Engineering and Design*, 238, pp. 1629–1635, 2008.
- [57] Michelin, S. and Llewellyn Smith, S. G. “Linear Stability Analysis of Coupled Parallel Flexible Plates in an Axial Flow.” *Journal of Fluids and Structures*, 25, pp. 1136–1157, 2009.
- [58] Roth, G. D. *CFD Analysis of Pressure Differentials in a Plate-Type Fuel Assembly*. Master’s thesis, Oregon State University, 2012.
- [59] Kennedy, J. C. and Solbrekken, G. L. “Coupled Fluid Structure Interaction (FSI) Modelling of Parallel Plate Assemblies.” Proceedings of the ASME 2011 International Mechanical Engineering Congress & Exposition, 2011.
- [60] Curtis, F. G., Ekici, K., et al. “Fluid-Structure Interaction Modeling of High-Aspect Ratio Nuclear Fuel Plates Using COMSOL.” COMSOL Conference: Boston, 2013.
- [61] Curtis, F. G., Freels, J. D., et al. “Predicting Large Deflections of Multiplate Fuel Elements Using a Monolithic FSI Approach.” *Nuclear Science and Engineering*, 189 (1), pp. 82–92, 2018.
- [62] Stokes, G. G. “On the Theories of Internal Friction of Fluids in Motion.” *Transactions of the Cambridge Philosophical Society*, 8, pp. 287–305, 1865.
- [63] White, F. M. *Viscous Fluid Flow*. McGraw-Hill Book Company, 3rd ed., 2006.

- [64] White, F. M. *Fluid Mechanics*. McGraw-Hill Book Company, 2nd ed., 1986.
- [65] Reynolds, O. “On the Dynamical Theory of Incompressible Viscous Fluids and the Determination of the Criterion.” *Philosophical Transactions of the Royal Society of London. A*, 186, pp. 123–164, 1895.
- [66] Kuzmin, D., Mierka, O., et al. “On the Implementation of the $\kappa\text{-}\varepsilon$ Turbulence Model in Incompressible Flow Solvers Based on a Finite Element Discretisation.” *International Journal of Computing Science and Mathematics*, 1 (2), pp. 193–206, 2007.
- [67] Launder, B. and Sharma, B. “Application of the Energy-Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disc.” *Letters in Heat and Mass Transfer*, 1 (2), pp. 131 – 137, 1974.
- [68] Reddy, J. N. and Gartling, D. K. *The Finite Element Method in Heat Transfer and Fluid Dynamics*. CRC Press, Third ed., 2010.
- [69] Hughes, T. J., Franca, L. P., et al. “A New Finite Element Formulation for Computational Fluid Dynamics: VIII. The Galerkin/least-squares Method for Advective-diffusive Equations.” *Computer Methods in Applied Mechanics and Engineering*, 73 (2), pp. 173 – 189, 1989.
- [70] Tezduyar, T., Mittal, S., et al. “Incompressible Flow Computations With Stabilized Bilinear and Linear Equal-order-interpolation Velocity-pressure Elements.” *Computer Methods in Applied Mechanics and Engineering*, 95 (2), pp. 221 – 242, 1992.
- [71] Billah, K. and Scanlan, R. H. “Resonance, Tacoma Narrows Bridge Failure, and Undergraduate Physics Textbooks.” *American Journal of Physics*, 51, pp. 118–124, 1991.
- [72] Roshko, A. “On the Development of Turbulent Wakes from Vortex Streets.” Tech. Rep. # 1191, National Advisory Committee for Aeronautics, 1954.

- [73] Koopman, G. H. “The Vortex Wakes of Vibrating Cylinders at Low Reynolds Numbers.” *Journal of Fluid Mechanics*, 28, pp. 501–512, 1967.
- [74] Norberg, C. “An Experimental Investigation of the Flow Around a Circular Cylinder: Influence of Aspect Ratio.” *Journal of Fluid Mechanics*, 258, pp. 287–316, January 1994.
- [75] Schäfer, M., Turek, S., et al. *Benchmark Computations of Laminar Flow Around a Cylinder*. Springer, 1996.
- [76] Blackburn, H. M. and Henderson, R. D. “A Study of Two-dimensional Flow Past an Oscillating Cylinder.” *Journal of Fluid Mechanics*, 385, pp. 255–286, 1999.
- [77] Evangelinos, C. and Karniadakis, G. E. “Dynamics and Flow Structures in the Turbulent Wake of Rigid and Flexible Cylinders Subject to Vortex-induced Vibrations.” *Journal of Fluid Mechanics*, 400, pp. 91–124, December 1999.
- [78] McMullen, M. S. *The Application of Non-linear Frequency Domain Methods to the Euler and Navier-Stokes Equations*. Ph.D. thesis, Stanford University, 2003.
- [79] Pedro, G., Suleman, A., et al. “A Numerical Study of the Propulsive Efficiency of a Flapping Hydrofoil.” *International Journal for Numerical Methods in Fluids*, 42 (5), pp. 493–526, 2003.
- [80] Turek, S. and Hron, J. *Proposal for Numerical Benchmarking of Fluid-structure Interaction Between an Elastic Object and Laminar Incompressible Flow*. Springer, 2006.
- [81] Spiker, M. A. *Development of an Efficient Design Method for Non-synchronous Vibrations*. Ph.D. thesis, Duke University, 2008.

- [82] Placzek, A., Sigrist, J.-F., et al. “Numerical Simulation of an Oscillating Cylinder in a Cross-flow at Low Reynolds Number: Forced and Free Oscillations.” *Computers and Fluids*, 38 (1), pp. 80 – 100, 2009.
- [83] Curtis, F. G., Ekici, K., et al. “FSI for Coolant Flow in Research-type Nuclear Reactors.” COMSOL Conference: Boston, 2011.
- [84] Cochran, W., Cooley, J. W., et al. “What is the fast Fourier transform?” *Proceedings of the IEEE*, 55 (10), pp. 1664–1674, 1967.
- [85] Williamson, C. H. K. “Defining the Universal and Continuous Strouhal-Reynolds Number Relationship for the Laminar Vortex Shedding of a Circular Cylinder.” *Physics of Fluids*, 1988.
- [86] “Engauge Digitizer.” Version 2.12, 2012. [Open Source Software].
- [87] Figliola, R. S. and Beasley, D. E. *Theory and Design for Mechanical Measurements*. J. Wiley & Sons, 3rd ed., 2000.
- [88] Buell, W. R. and Bush, B. A. “Mesh Generation – A Survey.” *American Society of Mechanical Engineers Publication*, 95 (1), pp. 332–338, 1973.
- [89] Field, D. A. “Laplacian Smoothing and Delaunay Triangulations.” *Communications in Applied Numerical Methods*, 4 (6), pp. 709–712, 1988.
- [90] Winslow, A. M. “Numerical Solution of the Quasilinear Poisson Equation in a Nonuniform Triangle Mesh.” *Journal of Computational Physics*, 1 (2), pp. 149 – 172, 1966.
- [91] Karman, S. L. “Virtual Control Volumes for Two-Dimensional Unstructured Elliptic Smoothing.” In “Proceedings of the 19th International Meshing Roundtable,” pp. 121–142. Sandia National Laboratory, Chattanooga, TN, 2010.

- [92] Askes, H., Kuhl, E., et al. “An ALE Formulation Based on Spatial And Material Settings of Continuum Mechanics. Part 2: Classification and Applications.” *Computer Methods in Applied Mechanics and Engineering*, 193 (39-41), pp. 4223 – 4245, 2004. The Arbitrary Lagrangian-Eulerian Formulation.
- [93] Mosler, J. and Ortiz, M. “On the Numerical Implementation of Variational Arbitrary LagrangianEulerian (VALE) Formulations.” *International Journal for Numerical Methods in Engineering*, 67 (9), pp. 1272–1289, 2006.
- [94] Persson, P.-O., Peraire, J., et al. “A High Order Discontinuous Galerkin Method For Fluid-Structure Interaction.” In “18th AIAA Computational Fluid Dynamics Conference,” 2007.
- [95] Gere, J. M. and Timoshenko, S. P. *Mechanics of Materials*. PWS-Kent Publishing Company, 3rd ed., 1990.
- [96] Liu, L., Lu, D., et al. “Large-amplitude and narrow-band vibration phenomenon of a foursquare fix-supported flexible plate in a rigid narrow channel.” *Nuclear Engineering and Design*, 241 (8), pp. 2874–2880, 2011.
- [97] Sartory, W. K. “Analysis of Hydraulic Instabilities of ANS Involute Fuel Plates.” Tech. Rep. ORNL/TM-11580, ORNL, 1991.
- [98] Sartory, W. K. “Nonlinear Analysis of Hydraulic Buckling Instability of ANS Involute Fuel Plates.” Tech. Rep. ORNL/TM-12319, ORNL, 1993.
- [99] Swinson, W. F., Battiste, R. L., et al. “Follow-up Fuel Plate Stability Experiments and Analysis for the Advanced Neutron Source.” Tech. Rep. ORNL/TM-12629, ORNL, 1993.
- [100] Swinson, W. F., Luttrell, C. R., et al. “An Examination of the Elastic Structural Response of the Advanced Neutron Source Fuel Plates.” Tech. Rep. ORNL/TM-12712, ORNL, 1994.

- [101] Yoder, G. L., Dixon, J. R., et al. “Steady-State Thermal-Hydraulic Design and Analysis of the Advanced Neutron Source Reactor.” Tech. Rep. ORNL/TM-12398, ORNL, May 1994.
- [102] Luttrell, C. R. “Finite Element Analysis of Advanced Neutron Source Fuel Plates.” Tech. Rep. ORNL/TM-4600, ORNL, 1995.
- [103] Swinson, W., Battiste, R., et al. “An Experimental Investigation of the Structural Response of Reactor Fuel Plates.” *Experimental Mechanics*, 35 (3), pp. 212–215, 1995.
- [104] Siman-Tov, M., Gambill, W., et al. “Thermal Hydraulic Correlations for the Advanced Neutron Source Reactor Fuel Element Design and Analysis.” Proceeding 1991 ASME Winter Annual Meeting, Atlanta, December 1991.
- [105] Felde, D. K. “The Advanced Neutron Source Thermal Hydraulic Test Loop.” Proceedings 8th Power Plant Dynamics, Control and Testing Symposium, Knoxville, TN, May 1992.
- [106] Siman-Tov, M., Felde, D. K., et al. “FY 1995 Progress Report on the ANS Thermal-Hydraulic Test Loop Operation and Results.” Tech. Rep. ORNL/TM-12972, ORNL, Jul 1997.
- [107] Titow, W. V. *PVC Technology*. Elsevier Applied Science, Fourth ed., 1984.
- [108] “SolidWorks.” Version 12.2, 2012. [Software].
- [109] Çengel, Y. A. and Boles, M. A. *Thermodynamics: An Engineering Approach*. McGraw-Hill Book Company, 5th ed., 2006.
- [110] Heil, M., Hazel, A. L., et al. “Solvers for Large-displacement Fluid-structure Interaction Problems: Segregated Versus Monolithic Approaches.” *Computational Mechanics*, 43 (1), pp. 91–101, 2008.

Appendix

Appendix A

Mesh Smoothing Techniques

The simulation of a moving computational domain requires some method to allow the mesh to follow the deformation. As the structure deforms into the fluid domain, the existing mesh is no longer valid for computation. Because remeshing the entire domain at every time-step or iteration would be too expensive, it is best to ‘smooth’ the mesh to match the new boundaries created by the moving structure. As described in Section 3.2, there are three smoothing techniques available in COMSOL: Laplace, Winslow, and Hyperelastic. In this Appendix, an in depth demonstration of the strengths and weaknesses of each technique will be presented.

Governing Equations

The Laplace technique, first described by Buell and Bush [88] for a structured mesh and refined by Field [89] for a free triangular mesh, applies the Laplace operator on the computational coordinates (ξ, η) in the physical space (X, Y) . When the Laplace equation (A.1) is satisfied, a smooth distribution of the computational coordinates is present.

$$\begin{aligned}\nabla^2 \xi &= \frac{\partial^2 \xi}{\partial x^2} + \frac{\partial^2 \xi}{\partial y^2} = 0 \\ \nabla^2 \eta &= \frac{\partial^2 \eta}{\partial x^2} + \frac{\partial^2 \eta}{\partial y^2} = 0\end{aligned}\tag{A.1}$$

The Laplace equations should satisfy the max-min conditions such that parameters inside the domain will not exceed the boundary of the domain which ensures that the grid lines will not cross resulting in inverted meshes. In Eq. (A.1), the known variables are (ξ, η) and the unknown variables are (x, y) . Applying the Laplacian of Eq. (A.1) to the computational space results in the Laplace smoothing equation:

$$\begin{aligned}\nabla^2 x &= \frac{\partial^2 x}{\partial \xi^2} + \frac{\partial^2 x}{\partial \eta^2} = 0 \\ \nabla^2 y &= \frac{\partial^2 y}{\partial \xi^2} + \frac{\partial^2 y}{\partial \eta^2} = 0\end{aligned}\tag{A.2}$$

Although Eq. (A.2) satisfies the max-min condition for a structured mesh, this is not always the case for an unstructured mesh where some of the elements may be inverted, resulting in negative cell volumes.

To overcome the limitations of the Laplace smoothing technique, Winslow [90] proposed a transformation resulting in the following set of equations:

$$\begin{aligned}\alpha \frac{\partial^2 x}{\partial \xi^2} - 2\beta \frac{\partial^2 x}{\partial \xi \partial \eta} + \gamma \frac{\partial^2 x}{\partial \eta^2} &= 0 \\ \alpha \frac{\partial^2 y}{\partial \xi^2} - 2\beta \frac{\partial^2 y}{\partial \xi \partial \eta} + \gamma \frac{\partial^2 y}{\partial \eta^2} &= 0 \\ \alpha &= \left(\frac{\partial x}{\partial \eta} \right)^2 + \left(\frac{\partial y}{\partial \eta} \right)^2 \\ \beta &= \frac{\partial x}{\partial \xi} \frac{\partial x}{\partial \eta} + \frac{\partial y}{\partial \xi} \frac{\partial y}{\partial \eta} \\ \gamma &= \left(\frac{\partial x}{\partial \xi} \right)^2 + \left(\frac{\partial y}{\partial \xi} \right)^2\end{aligned}\tag{A.3}$$

This set of equations, though more complex, can greatly improve the accuracy and may alleviate the problems of inverted meshes found using the Laplace smoothing equations [91].

The last mesh smoothing technique available in the code is a hyperelastic technique inspired by neo-Hookean materials [92]. The technique searches for the minimum of

the mesh deformation energy in which the energy is given by:

$$W = \int_{\Omega} \frac{\mu}{2} (I_1 - 3) + \frac{\kappa}{2} (J - 1)^2 dV \quad (\text{A.4})$$

where μ is the artificial sheer modulus and κ is the artificial bulk modulus. The invariants J and I_1 are given by:

$$J = \det(\nabla x^x)$$

$$I_1 = J^{-2/3} \text{tr} \left((\nabla x^x)^T \nabla x^x \right)$$

Domain

The domain for the mesh smoothing example is a 2-D rectangle with an aspect ratio of 10 (Figure A.1). The left side of the domain will be held fixed and the right side of the domain will be prescribed with three different deformations. Each deformation is

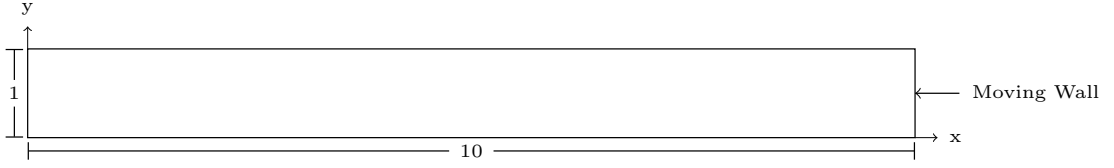


Figure A.1: Domain for the mesh smoothing example.

run using the transient solver and will run until the right boundary meets the left or the mesh smoothing crashes. Both a mapped (structured) and triangular (free) mesh will be evaluated for each mesh smoothing technique. The mapped mesh consists of 1000 elements (Figure A.2) and the free mesh consists of 2610 elements (Figure A.3).

At the left boundary, the mesh is held fixed in both the x and y directions. At the top and bottom boundaries, the mesh is held fixed in the y direction but has no

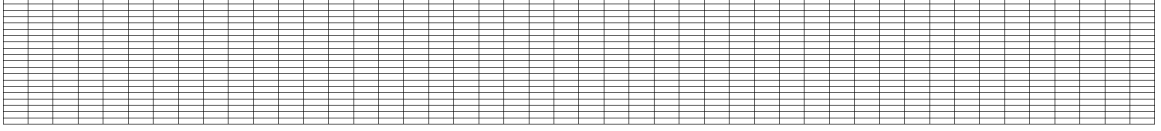


Figure A.2: The mapped mesh used for the mesh smoothing comparisons consisting of 1000 elements.

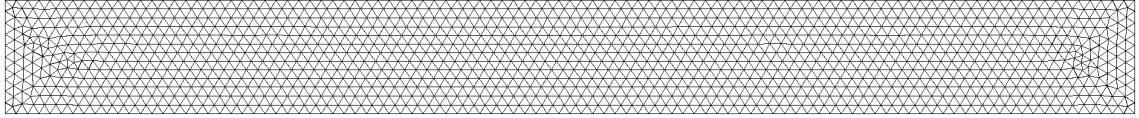


Figure A.3: The free mesh used for the mesh smoothing comparisons consisting of 2610 elements.

restriction in the x direction. The right boundary is deformed by the following set of equations:

1. $x = -t \cdot \text{AR}$
2. $x = -t \cdot y \cdot \text{AR}$
3. $x = -t \cdot (1 - y) \cdot y \cdot \text{AR}$

where AR is the aspect ratio of the domain. The first equation moves the entire right boundary to the left as a function of time. The second equation creates a linear boundary movement with the bottom right of the domain fixed as seen in Figure A.4. Figure A.5 demonstrates the third equation which creates a quadratic bulge into the mesh domain. The simulations are run for 1 s or until the mesh smoothing scheme breaks down.

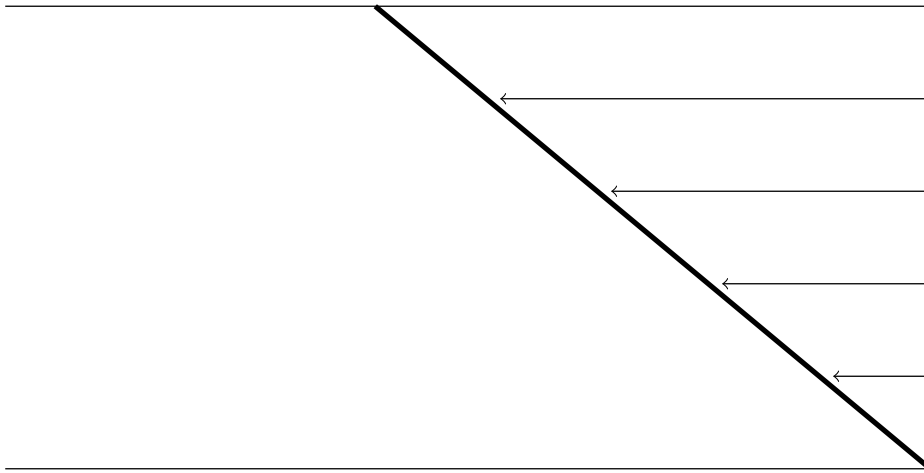


Figure A.4: Linear boundary movement for the mesh smoothing example.

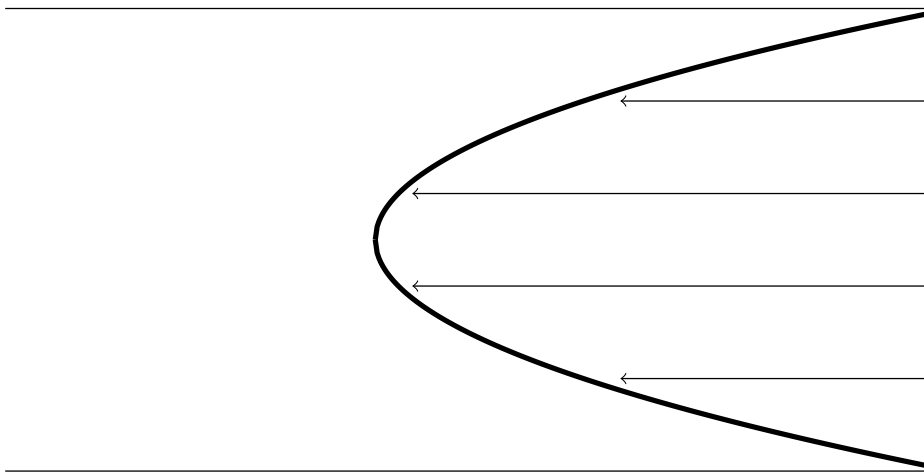


Figure A.5: Quadratic boundary movement for the mesh smoothing example.

Simulation and Results

The mapped and free mesh domains were computed using the implicit, transient direct solver in COMSOL using each mesh smoothing technique. The maximum computation times for each mesh and smoothing technique are presented in Tables A.1 and A.2. A simulation that runs for the entire 1 s goes to the other side of the domain. Although the solver may run for the entire time, it does not mean that there are no inverted meshes and this is demonstrated in the **Inversion** part of Tables A.1 and A.2.

The mesh inversion was calculated from the calculation of the minimum element area; when the minimum element area is less than zero, then an inverted mesh is present in the computational domain. Although an inverted mesh can be present in the solution, the solver may still be able to run after the inversion. As expected, the uniform mesh movement results in no inverted mesh for the entire computation. The other two cases show inverted meshes before the code breaks down. With a smaller maximum time-step, and thus smaller changes between each mesh smoothing calculation, it can be seen that both mesh inversion can be reduced and mesh smoothing can be improved.

Table A.1: Maximum time computed before error in mesh smoothing calculations for a maximum time-step of 0.1 s.

		$x = -t$		$x = -t \cdot y$		$x = -t \cdot y(1 - y)$	
		Mapped	Free	Mapped	Free	Mapped	Free
Run time	Laplace	1.000 s	1.000 s	1.000 s	1.000 s	1.000 s	1.000 s
	Winslow	1.000 s	1.000 s	0.156 s	0.217 s	0.518 s	0.274 s
	Hyperelastic	1.000 s	1.000 s	0.256 s	0.236 s	0.278 s	0.280 s
Inversion	Laplace	1.000 s	1.000 s	0.042 s	0.037 s	0.057 s	0.055 s
	Winslow	1.000 s	1.000 s	0.105 s	0.095 s	0.043 s	0.043 s
	Hyperelastic	1.000 s	1.000 s	0.140 s	0.204 s	0.217 s	0.243 s

Table A.2: Maximum time computed before error in mesh smoothing calculations for a maximum time-step of 0.001 s.

		$x = -t$		$x = -t \cdot y$		$x = -t \cdot y(1 - y)$	
		Mapped	Free	Mapped	Free	Mapped	Free
Run time	Laplace	1.000 s	1.000 s	1.000 s	1.000 s	1.000 s	1.000 s
	Winslow	1.000 s	1.000 s	1.000 s	1.000 s	1.000 s	0.998 s
	Hyperelastic	1.000 s	1.000 s	0.565 s	1.000 s	0.929 s	0.899 s
Inversion	Laplace	1.000 s	1.000 s	0.042 s	0.037 s	0.057 s	0.055 s
	Winslow	1.000 s	1.000 s	0.136 s	0.135 s	0.043 s	0.044 s
	Hyperelastic	1.000 s	1.000 s	0.146 s	0.188 s	0.728 s	0.602 s

Results for $x = -t \cdot \mathbf{AR}$

Moving the mesh uniformly from one side of the domain to the other is the easiest of the three movements to solve. Both maximum time-steps and all mesh smoothing techniques were able to calculate the deformed meshes for the entire length of the domain.

This movement highlights the benefits of the mapped mesh when a smoothing technique is applied. When looking at the extreme mesh deformations in Figure A.6 for the free mesh solution at 0.9 s, it can be seen that the deformed mesh does not maintain the original form of the free mesh. Although the mesh is smooth, close to the left and right boundaries the mesh begins to exhibit some wiggles that should not be there. The horizontal lines of the mesh should be straight in the middle of the mesh domain and close observation shows that this is not the case. The mapped mesh on the other hand does not have these issues as seen in Figure A.7. Other than the quality of the deformed meshes, there were no other issues or differences in the mesh smoothing techniques.

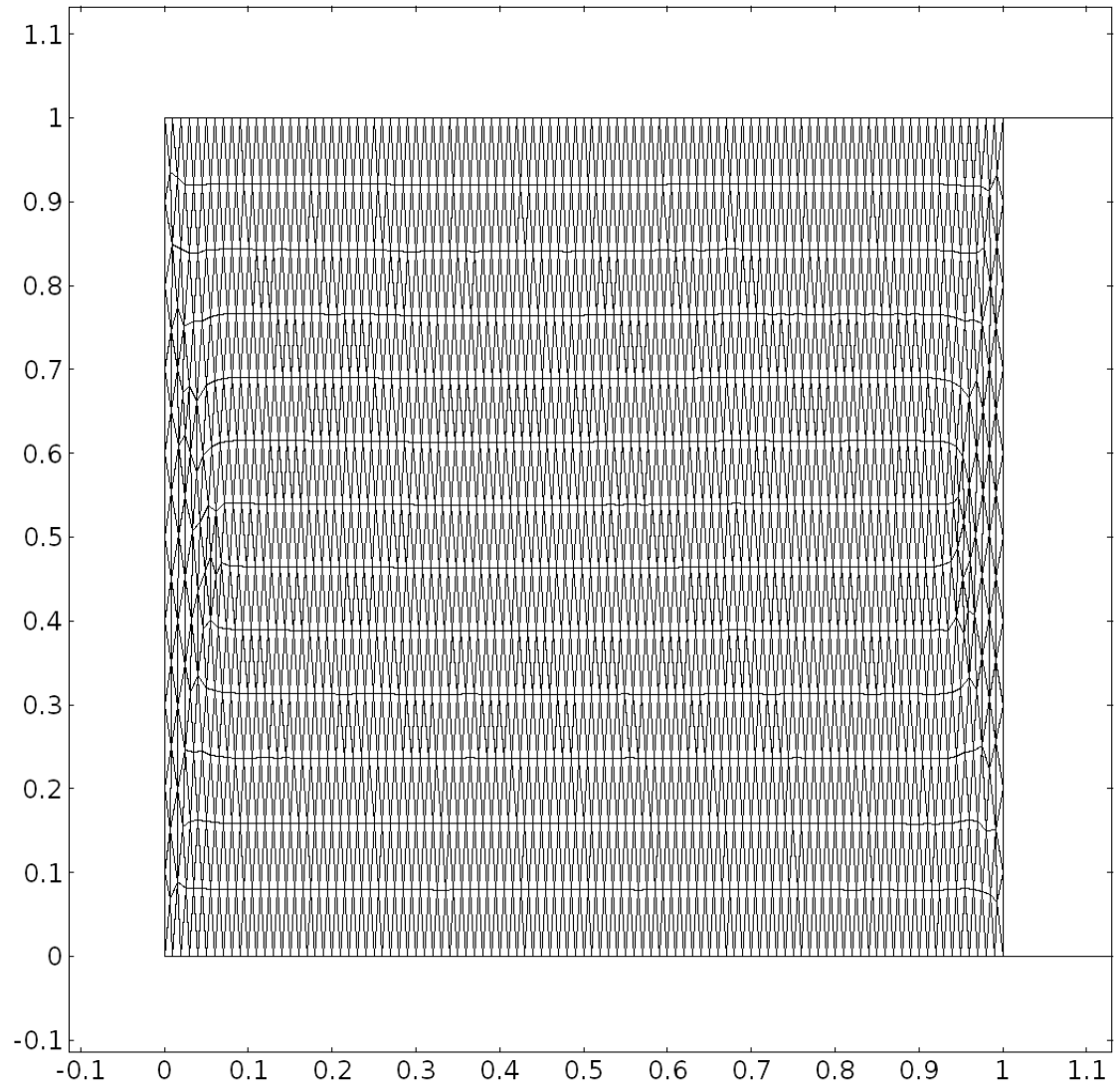


Figure A.6: The deformed free mesh using the Hyperelastic smoothing technique at 0.9s.

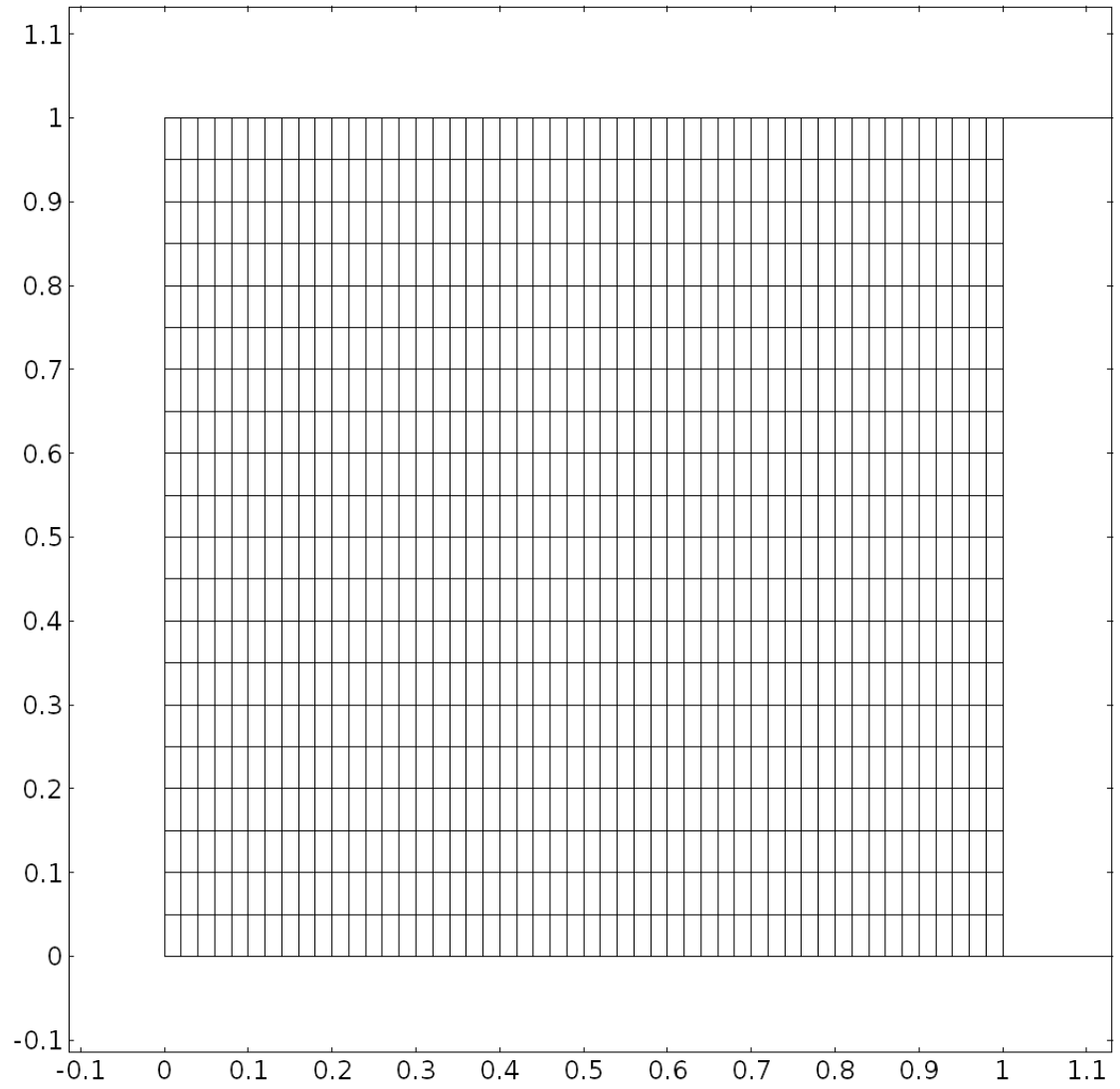


Figure A.7: The deformed mapped mesh using the Hyperelastic smoothing technique at 0.9s.

Results for $x = -t \cdot y \cdot \mathbf{AR}$

The linear mesh deformation begins to introduce some difficulties in the ability of the solver to properly smooth the mesh over the entire domain. Although the Laplace smoothing technique is able to completely run for the full 1 s, both free and mapped meshes resulted in inverted meshes very early into the computation. The final solution for the Laplace smoothing technique with the small maximum time-step is presented in Figure A.8.

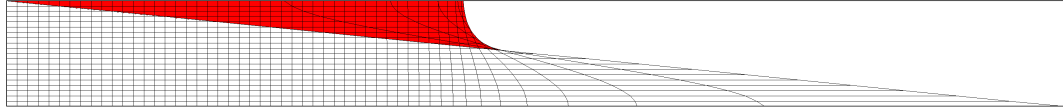


Figure A.8: The deformed mapped mesh using the Laplace smoothing technique at 1.0s. The inverted meshes are highlighted in red.

As the robustness of the mesh smoothing scheme increases, the number of inverted meshes decreases. Figure A.9 shows the final results at 1.0s for the Winslow smoothing. Note that the mesh follows the linear profile much better as well as having fewer inverted meshes.

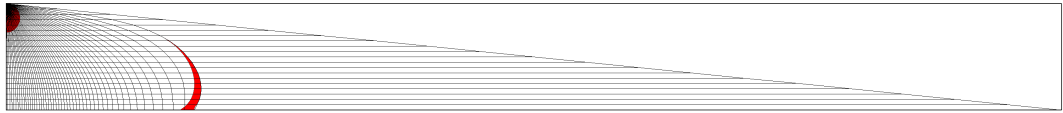


Figure A.9: The deformed mapped mesh using the Winslow smoothing technique at 1.0s. The inverted meshes are highlighted in red.

As one would expect, the Hyperelastic mesh smoothing scheme provides the best protection against inverted elements. This protection comes at a cost, one of which is the difficulty in obtaining a solution for the domain. This can be seen in the fact that the code had to stop at 0.565 s for the mapped mesh case. Figure A.10 shows the final calculated solution for the mapped mesh with much fewer inverted meshes than the previous two techniques.

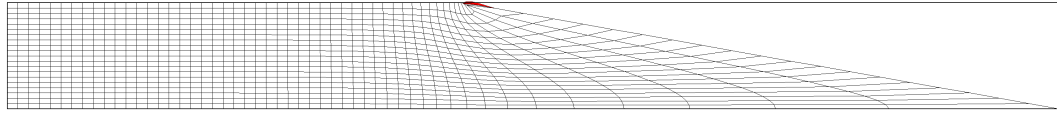


Figure A.10: The deformed mapped mesh using the Hyperelastic smoothing technique at 0.565 s. The inverted meshes are highlighted in red.

Results for $x = t \cdot (1 - y) \cdot y \cdot \mathbf{AR}$

The quadratic case is the only mesh deformation that induces a curved edge for the mesh smoothing to follow. Each mesh smoothing technique ran to or close to completion with mesh inversion similar to the linear deformation. Again, the smaller time-stepping allowed for a more accurate smoothing in which the solver was able to nearly complete all of the runs. The solution for the mapped mesh and free mesh for the Hyperelastic case at 0.75 s is provided in Figures A.11 and A.12. Although it is difficult to see in the figures, the mesh is inverted at the middle of the bulge in each mesh. The quality of the mesh continues to degrade as the simulation progresses to the final time-step. Each mesh, mesh smoothing technique, and deformation are presented in the Additional Plots Section below for the intervals 0.25 s, 0.50 s, 0.75 s and 1.00 s.

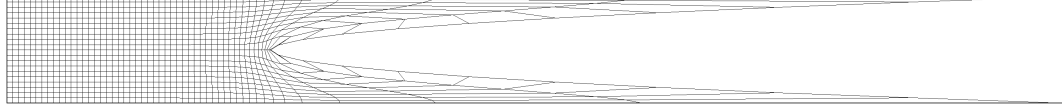


Figure A.11: The deformed mapped mesh using the Hyperelastic smoothing technique at 0.75 s. The inverted meshes are highlighted in red.



Figure A.12: The deformed free mesh using the Hyperelastic smoothing technique at 0.75 s. The inverted meshes are highlighted in red.

Discussion

This demonstration shows some of the differences between each mesh smoothing technique. More important than the ability of the code to complete the transient problem is the ability of the mesh smoothing technique to avoid inverted meshes. An inverted mesh element greatly decreases the stability of a fluid-flow problem and typically results in a diversion of the residual as expected. The resistance to inversion is a better metric to judge the quality of each mesh smoothing technique in the present study over the ability to complete the run.

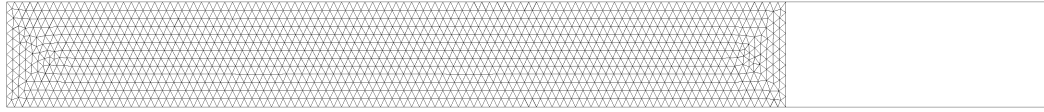
Although the Laplace smoothing is very stable, resulting in solutions obtained for the full deformation, the early onset of mesh inversion makes the accuracy of large deflections questionable. The ease of deformation computation make the Laplace smoothing technique applicable for small deflections. The ease of computation also lends itself to production-type models in which speed of attaining results is more important than overall accuracy.

The Winslow smoothing technique, which builds upon the Laplace smoothing, falls into the middle of the pack as far as resistance to inversion and stability is concerned. The resistance to inversion is better in the linear case but unexpectedly worse for the quadratic bulge case.

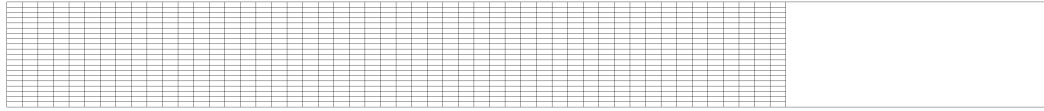
Additional Plots

In this section, each deflection is presented for time intervals of 0.25 s, 0.50 s, 0.75 s and 1.00 s for each mesh smoothing technique. For each time interval, the mapped and free mesh are presented together in order to demonstrate the differences that the mesh smoothing technique has on each mesh type.

Laplace Smoothing: $x = -t \cdot \mathbf{AR}$

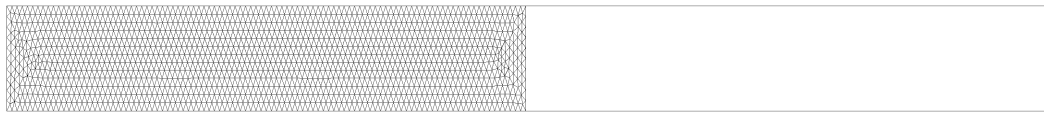


(a) Free Mesh

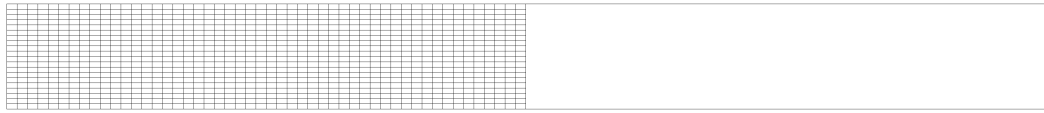


(b) Mapped Mesh

Figure A.13: Time = 0.25 s



(a) Free Mesh



(b) Mapped Mesh

Figure A.14: Time = 0.50 s

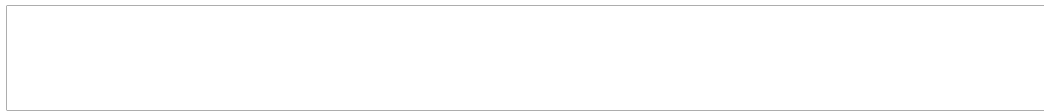


(a) Free Mesh

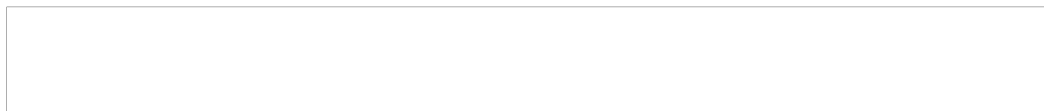


(b) Mapped Mesh

Figure A.15: Time = 0.75 s

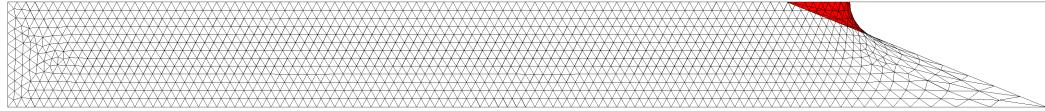


(a) Free Mesh

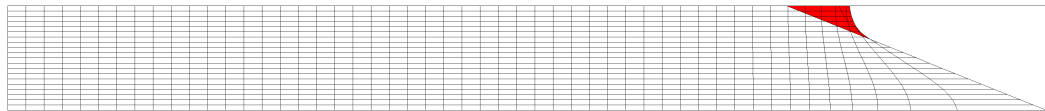


(b) Mapped Mesh

Figure A.16: Time = 1.00 s

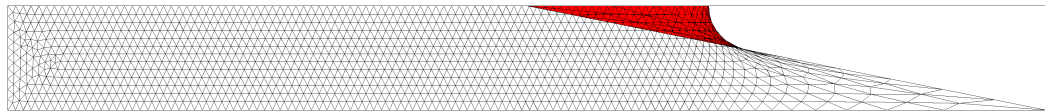


(a) Free Mesh



(b) Mapped Mesh

Figure A.17: Time = 0.25 s

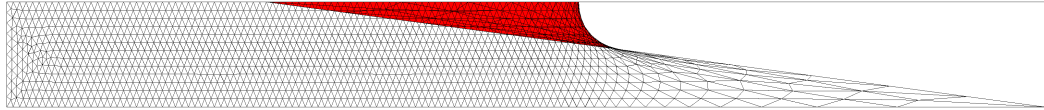


(a) Free Mesh

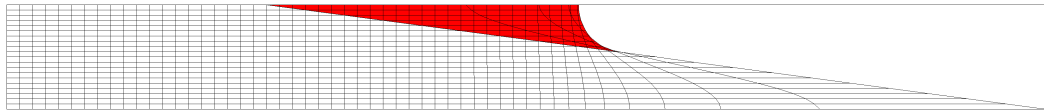


(b) Mapped Mesh

Figure A.18: Time = 0.50 s

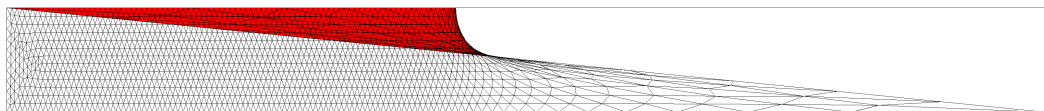


(a) Free Mesh

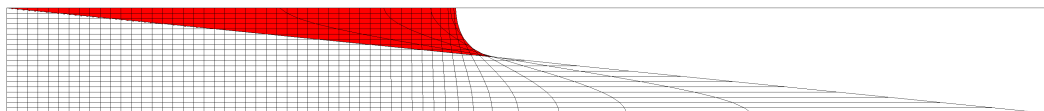


(b) Mapped Mesh

Figure A.19: Time = 0.75 s

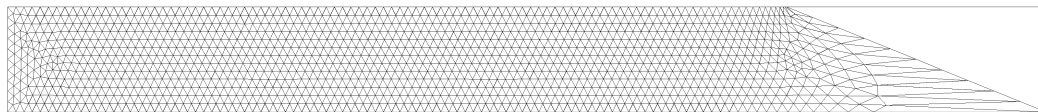


(a) Free Mesh

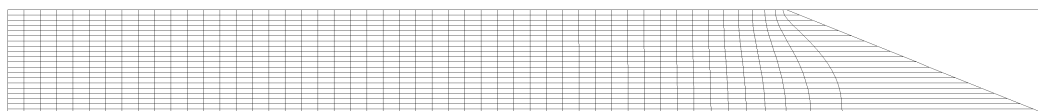


(b) Mapped Mesh

Figure A.20: Time = 1.00 s

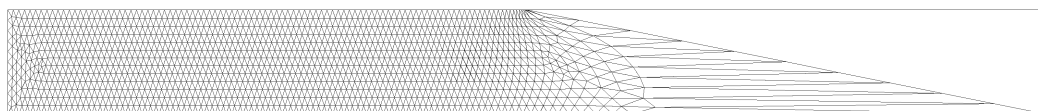


(a) Free Mesh

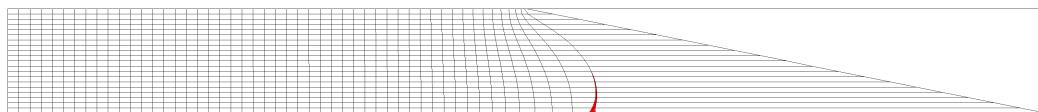


(b) Mapped Mesh

Figure A.21: Time = 0.25 s

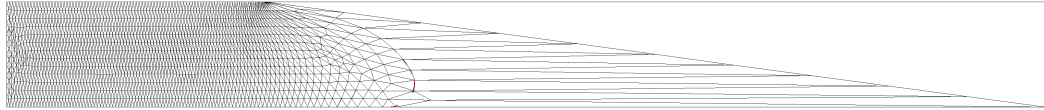


(a) Free Mesh

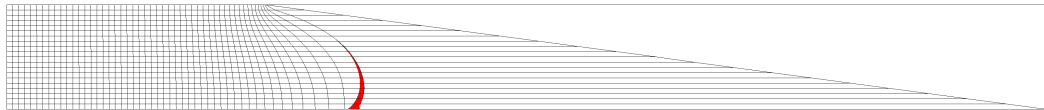


(b) Mapped Mesh

Figure A.22: Time = 0.50 s

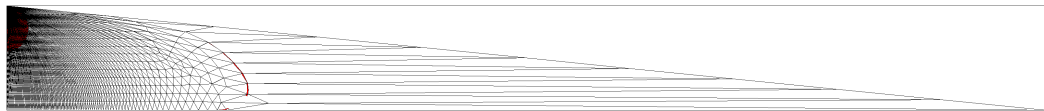


(a) Free Mesh

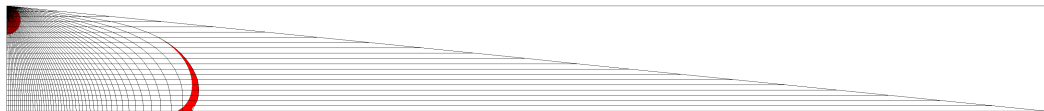


(b) Mapped Mesh

Figure A.23: Time = 0.75 s

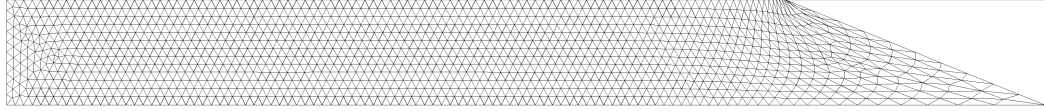


(a) Free Mesh

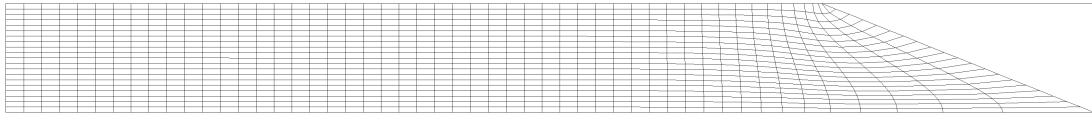


(b) Mapped Mesh

Figure A.24: Time = 1.00 s

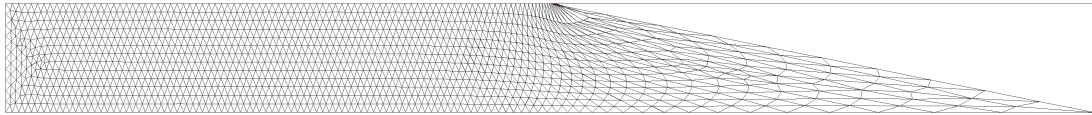


(a) Free Mesh

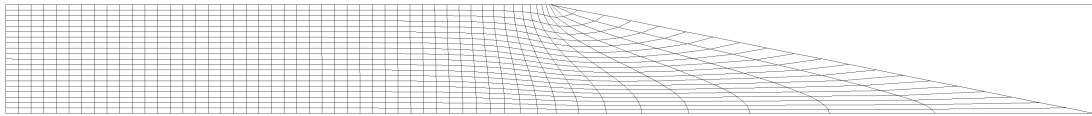


(b) Mapped Mesh

Figure A.25: Time = 0.25 s

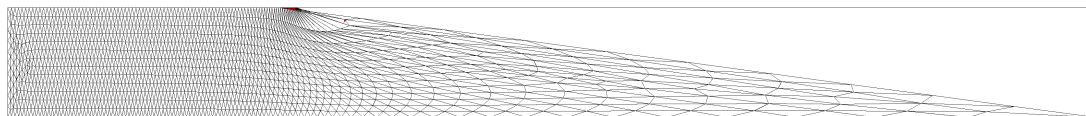


(a) Free Mesh



(b) Mapped Mesh

Figure A.26: Time = 0.50 s

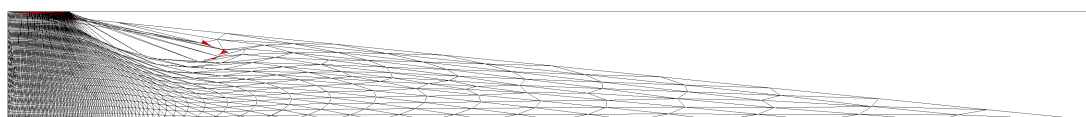


(a) Free Mesh

Failed Mesh
No Solution

(b) Mapped Mesh

Figure A.27: Time = 0.75 s



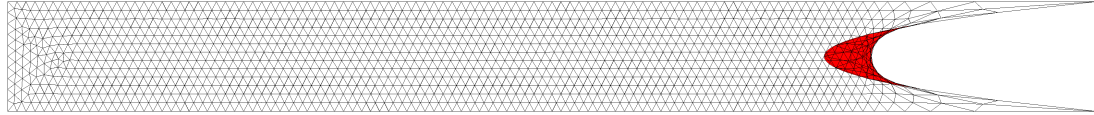
(a) Free Mesh

Failed Mesh
No Solution

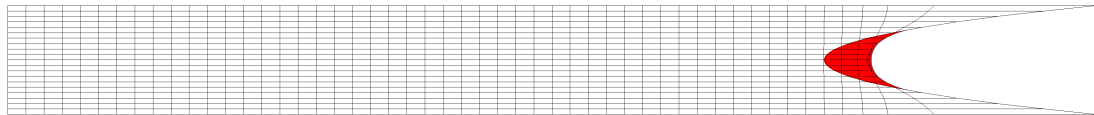
(b) Mapped Mesh

Figure A.28: Time = 1.00 s

Laplace Smoothing: $x = -t \cdot (1 - y) \cdot y \cdot \mathbf{AR}$

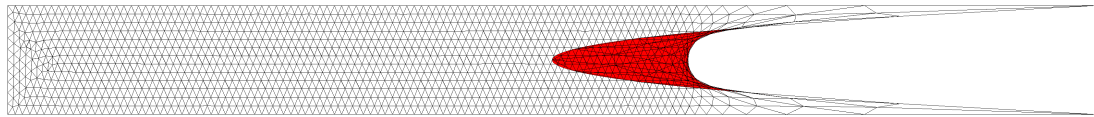


(a) Free Mesh

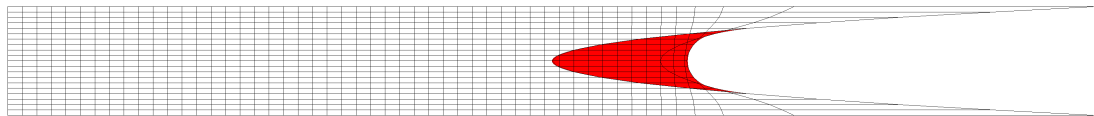


(b) Mapped Mesh

Figure A.29: Time = 0.25 s

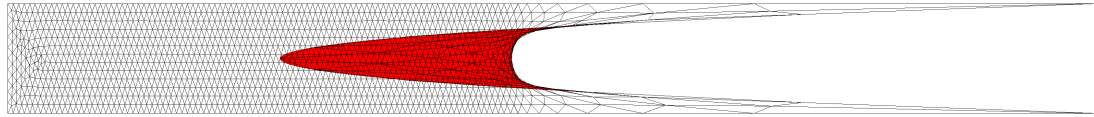


(a) Free Mesh

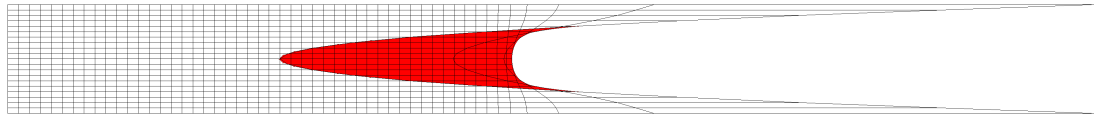


(b) Mapped Mesh

Figure A.30: Time = 0.50 s



(a) Free Mesh

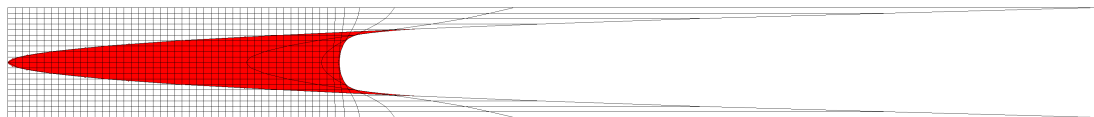


(b) Mapped Mesh

Figure A.31: Time = 0.75 s



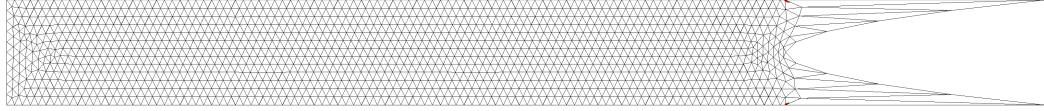
(a) Free Mesh



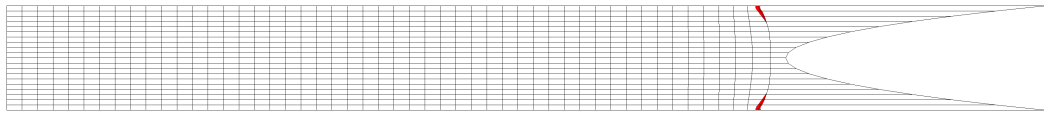
(b) Mapped Mesh

Figure A.32: Time = 1.00 s

Winslow Smoothing: $x = -t \cdot (1 - y) \cdot y \cdot \mathbf{AR}$

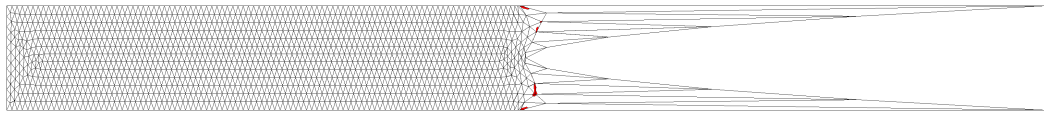


(a) Free Mesh

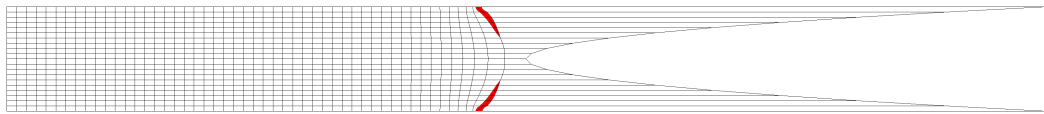


(b) Mapped Mesh

Figure A.33: Time = 0.25 s

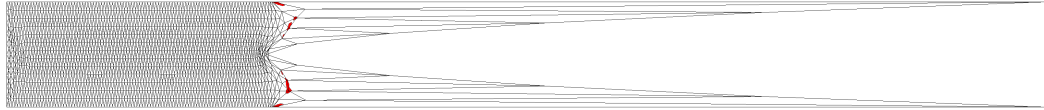


(a) Free Mesh

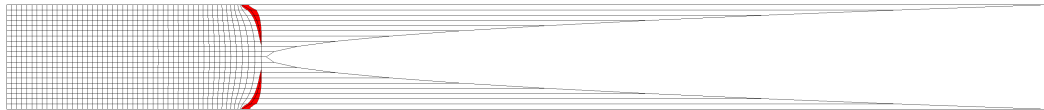


(b) Mapped Mesh

Figure A.34: Time = 0.50 s



(a) Free Mesh



(b) Mapped Mesh

Figure A.35: Time = 0.75 s



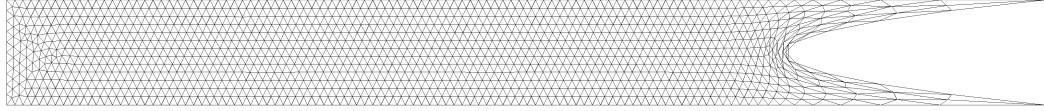
(a) Free Mesh



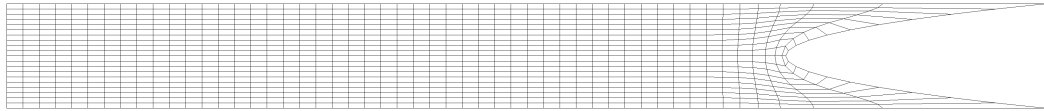
(b) Mapped Mesh

Figure A.36: Time = 1.00 s

Hyperelastic Smoothing: $x = -t \cdot (1 - y) \cdot y \cdot \mathbf{AR}$

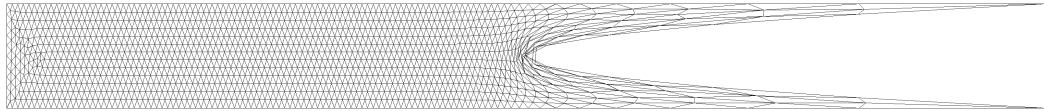


(a) Free Mesh

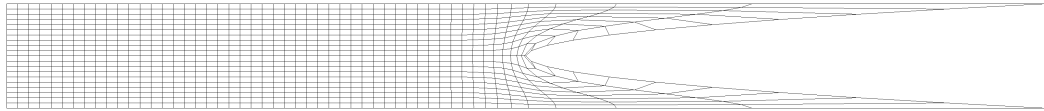


(b) Mapped Mesh

Figure A.37: Time = 0.25 s



(a) Free Mesh

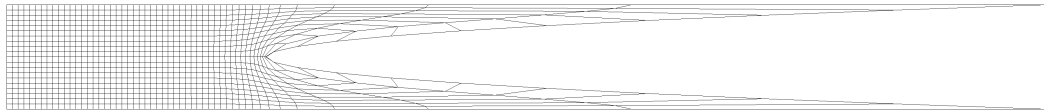


(b) Mapped Mesh

Figure A.38: Time = 0.50 s

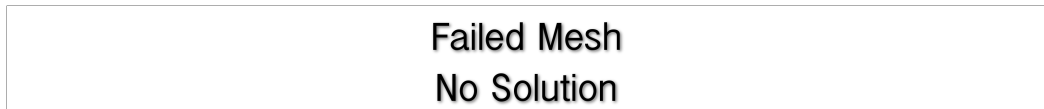


(a) Free Mesh



(b) Mapped Mesh

Figure A.39: Time = 0.75 s



(a) Free Mesh



(b) Mapped Mesh

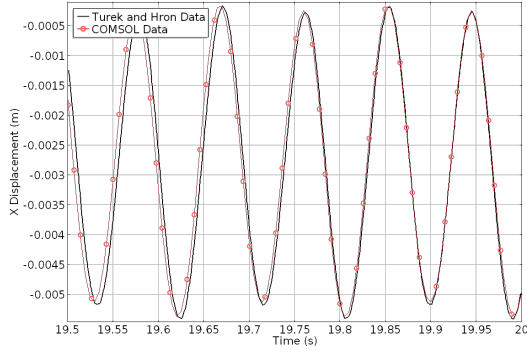
Figure A.40: Time = 1.00 s

Appendix B

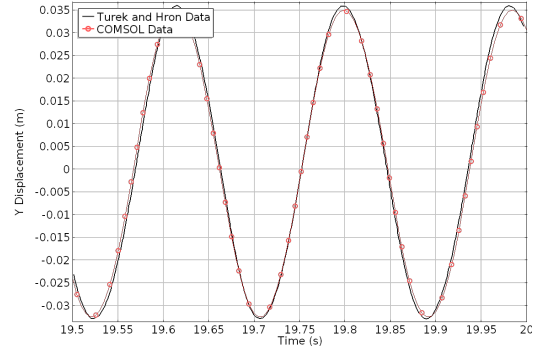
Turek And Hron Additional Results

Two other cases were run using the cylinder in cross-flow with the tail. The case for FSI3 is provided in Figure B.1 and compared to Turek and Hron [80]. The steel case, which is much stiffer than FSI2 or FSI3, is provided in Figure B.2. This case was not run by Turek and Hron [80] and there is no comparison to be made.

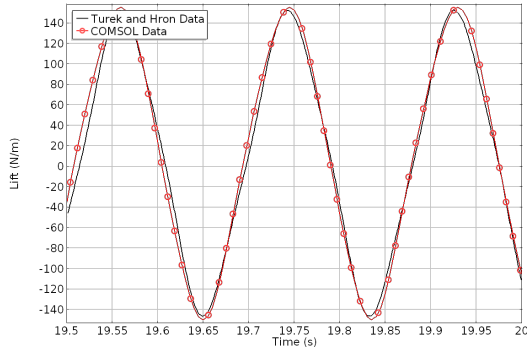
Although the Steel case is significantly stiffer, it is worth noting that the structural mechanics solver was able to capture displacements in the X- and Y-directions. The displacements are about 6 orders of magnitude smaller than the similar FSI3 case.



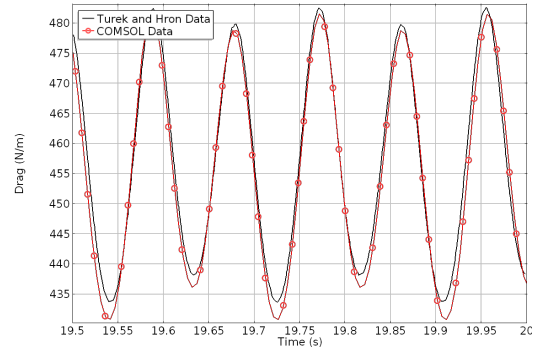
(a) X displacement



(b) Y displacement

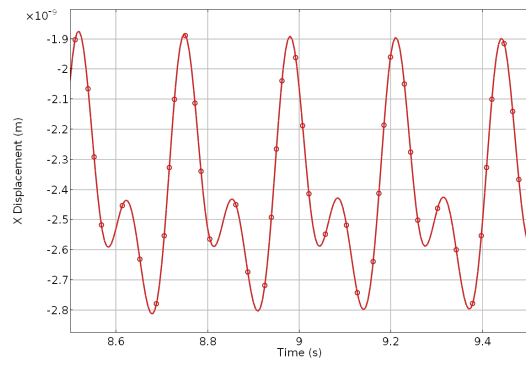


(c) Lift around cylinder and tail

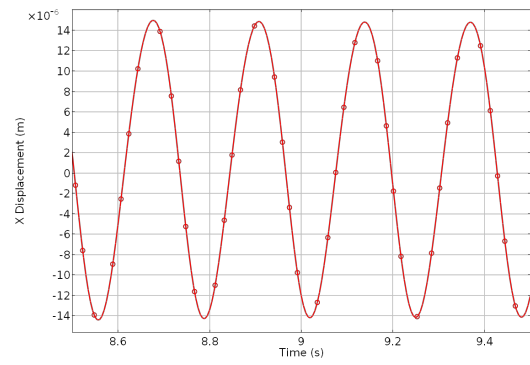


(d) Drag around cylinder and tail

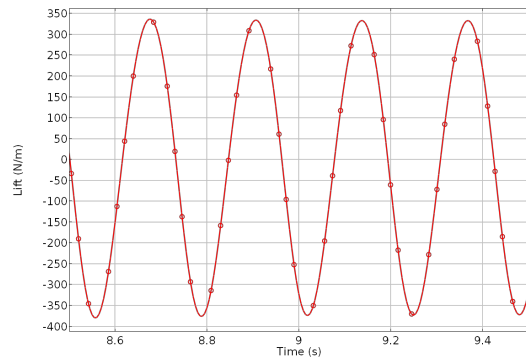
Figure B.1: Results for the FSI3 test case. The results (red lines with circles) are compared to the data provided by Turek and Hron in black.



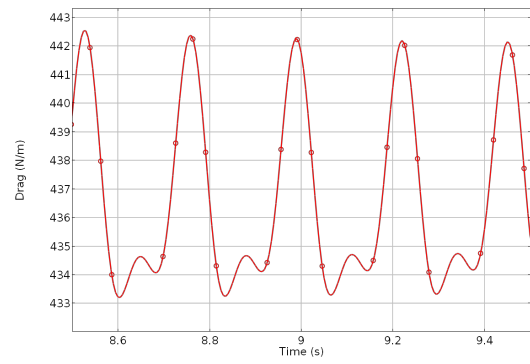
(a) X displacement



(b) Y displacement



(c) Lift around cylinder and tail



(d) Drag around cylinder and tail

Figure B.2: Results for the Steel test case.

Vita

Franklin Curtis was born in Nashville, TN, to parents Barbara and Dale Curtis. After graduating from high school in Middle Tennessee, Franklin began attending the University of Tennessee, Knoxville, going on to earn Bachelors and Masters of Science degrees in Aerospace Engineering. Under the guidance of Dr. Kivanc Ekici at the University of Tennessee and Dr. James D. Freels at the Oak Ridge National Laboratory (ORNL), Franklin began researching Fluid-Structure Interaction at the High Flux Isotope Reactor at ORNL in pursuit of a Doctor of Philosophy degree in Mechanical Engineering. During the course of his studies, Franklin and his wife Emi, had three children: Freeman, Rhodes and Abner. Franklin is now a staff member at ORNL focusing on high-Mach number rotating flows as well as thermal hydraulics.