Master BE/ME Thesis

Design of Small Scale Shockwave Generators for the ORCHID Setup

Tony Chew (42616362)

ENGG7280 Final Report

Due Date: 30 June 2015

Supervised by:

Dr. Ingo Jahn
The University of Queensland
School of Mechanical and Mining Engineering
St Lucia, 4072
Queensland, Australia

Professor Piero Colonna
TU Delft
Flight Performance and Propulsion
Kluyverweg 1, 2629 HS Delft
Zuid-Holland, Netherlands

Daily Supervisors:

Adam Joseph Head
TU Delft
PhD Candidate
Flight Performance and Propulsion

Carlo De Servi
TU Delft
Postdoctoral Researcher
Flight Performance and Propulsion

The University of Queensland
Faculty of Engineering, Architecture and IT

TU Delft
Delft University of Technology
Faculty of Aerospace Engineering
Abstract

This report focuses on the design and analysis of various subcomponents of the supersonic nozzle test section of the ORCHID setup. The primary concern is a study dedicated to the model support system which is crucial to the success of a series of gas dynamic experiments for high fidelity software validation. Standard design rules and empirical models are not directly suitable for the nozzle test section design, mainly due to the thermodynamic behaviour of the working fluid, which expand close to the critical point. The configuration of such a support system for the model typically consists of a sting and base support. The aim of the study is to design this model configuration and computationally investigate the influence of these non-ideal flow effects on the conceived model and support system. The model support must remain minimally intrusive to the results of the flow field whilst also being structurally sound. Such results will be useful for the ongoing development of realising the ORCHID setup.

A number of sub-goals to complete the objective included analysing non-ideal compressible flow simulations, designing the model and support system and conceiving a coupling tool between the fluid and structural domain. A field known as fluid-structure interaction was investigated to model the coupling tool, and it revealed that using a method known as the radial basis interpolation was recommended. This transferred values between the interfacing boundaries of the fluid and structural domain.

Several geometric models were investigated and a diamond model was chosen as it transfers the most stress onto the support system. Pertaining to the fluid setup, a frontal height of 9mm was selected for the diamond based off results and a review of literature concerning the supersonic blockage phenomenon. The Method of Characteristics was also used to generate the diverging nozzle profile. Concerning the structural setup, a low-carbon AISI 1010 steel was chosen. Two test cases were chosen which involved high-fidelity calculations. This required the coupling tool, where information was transferred to the FEA solver. The first case being an aligned case where the model was aligned to the nozzle and the second being a deflected case.

Results indicate that the effect of pressure was insignificant for the aligned case, with a maximum stress of 0.564MPa and deflections at the tip being 6.074 x 10^-4mm. For the deflected case, it was found that the influence of temperature played a major role in the structural integrity; an increase of 500% in stress from the results of pressure influence and 7% in maximum deflection at the tip. The main conclusions drawn are that the support system would not yield under steady state conditions but the deflections observed was considered significant enough such that it may hinder the results of the flow field. As a result, recommendations for future work include conceiving the automation tool to conduct a full two-way fluid-structure interaction process to analyse plant start-up.
Contents

Abstract .......................................................................................................................... 3

List of Figures .................................................................................................................. 7

List of Tables ................................................................................................................... 9

Acknowledgements ........................................................................................................ 10

List of Symbols and Abbreviations ................................................................................ 11

1. Introduction ................................................................................................................ 13
   1.1 Background & Motivation ..................................................................................... 13

2. Scope of Investigation ................................................................................................. 14
   2.1 Context .................................................................................................................. 14
   2.2 Research Questions .............................................................................................. 15
   2.3 Project Deliverables and Outcomes ..................................................................... 16

3. Literature Review ........................................................................................................ 17
   3.1 The ORCHID Test Rig ......................................................................................... 17
      3.1.1 Plant Outline .................................................................................................. 17
      3.1.2 Selection of the Working Fluid ...................................................................... 18
      3.1.3 Working Boundary Conditions ..................................................................... 19
      3.1.4 Nozzle Profile - Method of Characteristics .................................................... 19
   3.2 Shockwave Generators .......................................................................................... 21
      3.2.1 Normal and Oblique Shock Theory for Real Gases ......................................... 21
      3.2.2 Supersonic Blockage ..................................................................................... 23
   3.3 The Fluid-Structure Interaction Problem Definition .............................................. 25
      3.3.1 FSI Procedure Overview .............................................................................. 25
      3.3.2 Interpolation Methods .................................................................................. 27
      3.3.3 Radial Basis Functions for Interpolation between Non-Matching Meshes ....... 30

4. Software Overview ...................................................................................................... 34
   4.1 The Mesh Generator ............................................................................................. 34
   4.2 CFD Simulation Software ...................................................................................... 35

5. Fluid-Structure Interaction Setup ............................................................................... 36
   5.1 Justification of Chosen Interpolation Method ....................................................... 36
   5.2 Modeling Procedure .............................................................................................. 39

6. Fluid Model Setup ...................................................................................................... 42
   6.1 Nozzle Profile Generation ..................................................................................... 42
   6.2 Mesh Sensitivity Analysis ....................................................................................... 43
12.5 Appendix E - CFD Simulations for Sensitivity Analysis .......................................................... 89
12.6 Appendix F - Mesh Sensitivity Analysis Matlab Script Documentation ................................. 94
12.7 Appendix G - Supersonic Blockage and Profile Generation Matlab Script Documentation . 95
12.8 Appendix H - Model and Support System Dimensions ......................................................... 97
12.9 Appendix I - Layout of other conceived Model Support Systems.......................................... 99
12.10 Appendix J- FEA Model Setup ......................................................................................... 100
    12.10.1 Aligned Model ............................................................................................................... 100
    12.10.2 Deflected Model ........................................................................................................... 100
List of Figures

Figure 1 - Process flow diagram of ORCHID setup [1] .................................................................................. 13
Figure 2 - Representation of 3D de Laval nozzle [2] ..................................................................................... 14
Figure 3 - 2D outline of de Laval nozzle [2] .................................................................................................. 14
Figure 4 - Depiction of model insertion to generate oblique shocks and avoid bow shocks [2] ............... 15
Figure 5 - Schlieren experiments conducted under ambient air to observe shock wave angle [3] .... 15
Figure 6 - Streamline example [5] ............................................................................................................... 20
Figure 7 - Using the method of characteristics to compute values at point C [5] ........................................ 20
Figure 8 - The diverging section of a supersonic nozzle showing the reflected waves along the nozzle midline [5] ..................................................................................................................................................... 21
Figure 9 - Oblique shock wave [7] ............................................................................................................... 22
Figure 10 - Labeling of variables used to calculate supersonic blockage ................................................... 23
Figure 11 - Summary of blockage results for air based wind tunnels provided by Schueler [8] .......... 24
Figure 12 - Summary of blockage results for air based wind tunnels provided by Czysz [9] ............... 24
Figure 13 - Steps for conducting the Two-Way FSI Procedure [12] ......................................................... 26
Figure 14 - Interfacing of non-matching meshes [12] .................................................................................. 27
Figure 15 - Test case for interpolation testing of different methodologies [13] ........................................... 28
Figure 16 - Interpolation error vs no. of structural cells; Left: displacement, Right: pressure [13] .... 29
Figure 17 - Interpolation error vs CPU time; Left: displacement, Right: pressure [13] ........................... 29
Figure 18 - Consistent and Conservative plots over Exact Curve for interpolation of test pressures .. 37
Figure 19- RMS Error for increasing number of structure elements ...................................................... 38
Figure 20 - RMS error for increasing radius value .................................................................................... 39
Figure 21 - 1-Way FSI Procedure within the full 2-Way Procedure ......................................................... 40
Figure 22 - Conceived tool to carry out One-Way FSI Procedure ............................................................ 41
Figure 23 - Mach number and diverging nozzle section computed using the Method of Characteristics ..................................................................................................................................................................................... 43

Figure 24 - Pressure distribution and diverging nozzle section computed using the Method of Characteristics .............................................................................................................................................................................................................................................................. 43
Figure 25 - Test Geometry with defined boundary walls (Diamond model and sting included) .... 44
Figure 26 - Generation of the coarsest unstructured mesh ........................................................................ 44
Figure 27 - Aspect Ratio of mesh, Left: Finest mesh; Right: Coarsest mesh ............................................. 47
Figure 28 - Mach number between coarsest and finest mesh densities ................................................... 48
Figure 29 - Pressure distribution between coarsest and finest mesh densities ....................................... 48
Figure 30 - Residuals vs Iterations ............................................................................................................. 49
Figure 31 - Left: Total Computational Time, Right: Avg Computational Time per Iteration .......... 49
Figure 32 - Pressure Distribution along surface of model and sting ........................................ 50
Figure 33 - Close-Up of location of reflected shock along sting ........................................... 51
Figure 34 - Blockage ratio vs Mach for air conditions ......................................................... 52
Figure 35 - Blockage ratio vs Mach at tip of model ............................................................ 53
Figure 36 - Plots of theoretical ratios against x position. Left: nozzle area ratio; Right: pressure loss ratio .............................................................. 54
Figure 37 - Blockage validation check - at 9mm ................................................................. 55
Figure 38 - Blockage Validation check - at 14mm ............................................................... 56
Figure 39 - 3D View of model and support system layout within de Laval nozzle ................ 59
Figure 40 - Fluid Domain (Nozzle) of aligned case ............................................................. 60
Figure 41 - Structural Domain (Model and support system) of aligned case ...................... 60
Figure 42 - Fluid Domain (nozzle) of deflected case .......................................................... 61
Figure 43 - Structural Domain (model and sting) of deflected case ................................. 61
Figure 44 - 3D Simulation to test validity of simulation cases ........................................... 62
Figure 45 - Mach number and static pressure results of aligned model ............................. 63
Figure 46 - Mach and Pressure flow at tip of diamond ....................................................... 64
Figure 47 - Interpolation Results for the aligned case ......................................................... 65
Figure 48 - Structural Results of aligned model ................................................................. 66
Figure 49 - Mach number, pressure and temperature of deflected case ............................. 68
Figure 50 - Temperature and pressure at tip of diamond .................................................... 68
Figure 51 - Structural results of deflected case, effect of pressure ...................................... 73
Figure 52 - Structural results of deflected case, effect of pressure and temperature .......... 74
Figure 53 - Aspect Ratio of test meshes for Sensitivity Analysis ........................................ 87
Figure 54 - Skewness of test meshes for Sensitivity Analysis ............................................. 89
Figure 55 - Mach Number of CFD simulations conducted for sensitivity analysis ............. 91
Figure 56 - Static pressures of CFD Simulations conducted for sensitivity analysis .......... 93
Figure 57 - Visual representation of applying interpolated structural pressures onto model domain ........................................................................................................ 100
Figure 58 - Setup of aligned model, showing the mesh, loadings and constraints ................ 100
Figure 59 - Setup of deflected model, showing the mesh, loadings with temperature included and constraints .................................................................................................. 100
Figure 60 - Pressure distribution shown in Nastran, confirming pressure results applied from CFD simulation ......................................................................................... 101
List of Tables

Table 1 - Working parameters for selected fluids for the nozzle test section .............................................. 18
Table 2 - Working parameters for selected fluids for the turbine test section .................................................. 18
Table 3 - Subcritical boundary conditions of nozzle ......................................................................................... 19
Table 4 - Interpolation schemes used by Beckert [13] ..................................................................................... 28
Table 5 - Overview of chosen programs ............................................................................................................ 34
Table 6 - Steps for the generation of unstructured meshes in UMG2 ............................................................... 34
Table 7 - RMS Error obtained with the RBF interpolation test case ................................................................. 37
Table 8 - Summary of meshes used with test densities ..................................................................................... 44
Table 9 - Fluid Domain and Global Conditions ............................................................................................... 46
Table 10 - Boundary Conditions ..................................................................................................................... 46
Table 11 - Material properties of 1010 steel ....................................................................................................... 57
Table 12 - Von Mises Stress Criteria ................................................................................................................ 58
Table 13 - Summary of oblique shock wave angle results ............................................................................... 64
Table 14 - RMS Errors calculated for aligned case ........................................................................................... 65
Table 15 - Structural results for aligned case .................................................................................................... 65
Table 16 - Pressure comparison between top and bottom surface of model and support system .................. 69
Table 17 - Temperature comparison between top and bottom surface of model and support system .......... 71
Table 18 - RMS Error for pressure interpolation ............................................................................................... 72
Table 19 - RMS Error for temperature interpolation ......................................................................................... 72
Table 20 - Structural results for deflected case ................................................................................................. 75
Table 21 - Summary of proposed future recommendations ........................................................................... 78
Table 22 - Configuration setup for UMG2 to generate unstructured meshes .................................................. 81
Table 23 - Commands needed to generate unstructured meshes in UMG2 ..................................................... 81
Table 24 - Collection of outputted solution files from SU2 for post processing ............................................ 82
Acknowledgements

The work put together in this report would not have been possible but for the help, motivation and support of so many people. I would like to express my gratitude to each of the following who has contributed in some way to the completion of my thesis.

Within the staff at UQ, I would like to thank Dr. Ingo Jahn for his guidance as my primary UQ supervisor. I would also like to thank Professor David Mee and Laura Bainbridge, the industry placements coordinator, for providing the means to allow me to conduct my thesis abroad.

Here at TU Delft, among the propulsion and power group, I would like to thank Carlo De Servi and assistant professor Matteo Pini for providing their assistance regarding fluid-structure interaction and fluid mechanics topics. I also express my gratitude to PhD candidates Salvatore Vitale for providing assistance with SU2 and associate professor Antonio Ghidoni for providing assistance with the mesh generator UMG2. Among the professors within the aerospace faculty, I express my thanks to Dr. Alexander van Zuijlen for providing his invaluable guidance regarding fluid-structure interaction and assistant professor Roeland De Brueker for his information on structural analysis within wind tunnels.

I would like to express my most sincere gratitude towards PhD candidate, Adam Head who served as my daily supervisor. His technical and motivational guidance has been invaluable and the work completed in this report would not be at all possible without his patience, enthusiasm and passion for his work.

My family has always been a constant source of inspiration throughout my life, I acknowledge them for providing me the necessary courage and support to conduct my thesis abroad. Much of the credit also goes to my colleagues within the power and propulsion group in room 6.01, who made me feel welcome as I started my thesis in Delft. Acknowledgements goes to Nando van Arnhem, Thomas Schuwer, Tom Stokkermans, Sander Doppenburg, Jian Hao Wei, Marc Boorsma, Mario Verhagen, Evert Windels, Roel de Koning, Marten Sol, Jan Banan, Filip Martens, Siddharth Iyer, Ton Hettema and Thibault Crepin.

Finally I would like to thank Professor Piero Colonna for being my TU Delft primary supervisor who was able to make this project go ahead.
# List of Symbols and Abbreviations

## Latin Characters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>Elastic Modulus</td>
<td>GPa</td>
</tr>
<tr>
<td>S</td>
<td>Yield strength</td>
<td>MPa</td>
</tr>
<tr>
<td>$S_u$</td>
<td>Ultimate tensile strength</td>
<td>MPa</td>
</tr>
<tr>
<td>v</td>
<td>Velocity</td>
<td>m/s</td>
</tr>
<tr>
<td>h</td>
<td>Enthalpy</td>
<td>J/kg</td>
</tr>
<tr>
<td>t</td>
<td>Temperature</td>
<td>°C</td>
</tr>
<tr>
<td>p</td>
<td>Pressure</td>
<td>kPa</td>
</tr>
<tr>
<td>$p_o$</td>
<td>Stagnation pressure</td>
<td>kPa</td>
</tr>
<tr>
<td>$c_p$</td>
<td>Specific heat capacity at constant pressure</td>
<td>J/kg/K</td>
</tr>
<tr>
<td>N</td>
<td>Number of nodes</td>
<td></td>
</tr>
<tr>
<td>M</td>
<td>Mach Number</td>
<td></td>
</tr>
<tr>
<td>u</td>
<td>Displacement</td>
<td>mm</td>
</tr>
</tbody>
</table>

## Greek Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\gamma$</td>
<td>Ratio of specific heats</td>
<td></td>
</tr>
<tr>
<td>$\beta$</td>
<td>Oblique shock wave angle</td>
<td>°</td>
</tr>
<tr>
<td>$\theta$</td>
<td>Model angle</td>
<td>°</td>
</tr>
<tr>
<td>$\sigma_e$</td>
<td>Equivalent stress</td>
<td>MPa</td>
</tr>
<tr>
<td>$\sigma_1, \sigma_2, \sigma_3$</td>
<td>Principal stress directions</td>
<td>MPa</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Poisson's ratio</td>
<td></td>
</tr>
<tr>
<td>p</td>
<td>Density</td>
<td>kg/m³</td>
</tr>
</tbody>
</table>

## Subscript

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>f</td>
<td>Fluid domain</td>
</tr>
<tr>
<td>s</td>
<td>Structural domain</td>
</tr>
<tr>
<td>fs</td>
<td>Fluid domain to structural domain</td>
</tr>
</tbody>
</table>
## Abbreviations

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FEA</td>
<td>Finite Element Analysis</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>FSI</td>
<td>Fluid-Structure Interaction</td>
</tr>
<tr>
<td>NICF</td>
<td>Non-Ideal Compressible Flow</td>
</tr>
<tr>
<td>RMS</td>
<td>Root Mean Square error</td>
</tr>
<tr>
<td>MoC</td>
<td>Method of Characteristics</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Averaged Navier-Stokes</td>
</tr>
<tr>
<td>RBF</td>
<td>Radial Basis Function</td>
</tr>
</tbody>
</table>
1. Introduction

1.1 Background & Motivation

At the Delft University of Technology, within the power and propulsion group, the design of a hybrid test facility termed the Organic Rankine Cycle Hybrid Integrated Device (ORCHID) is underway. It is a continuous closed-circuit supersonic facility, whose purpose is to study the fundamentals of non-ideal gas dynamics and to evaluate turbomachinery performance. The development of this test facility will enable scientists to better understand the challenges posed when designing for small scale renewable energy generators; with its main motivation to spear-head development of new highly supersonic ORC turbines [1].

Head et al. [1] provides a simple process flow diagram of the ORCHID design shown in Figure 1. Within the two dotted rectangles are the two main test sections: a supersonic nozzle to perform experiments on Non-Ideal Compressible Flows (NICF) and a mini-ORC turbine for performance testing and evaluation. This project will focus and assist with the ongoing development of the supersonic nozzle test section as it belongs to stage one of the ORCHID project.

![Process flow diagram of ORCHID setup](image)

*Figure 1 - Process flow diagram of ORCHID setup [1]*
2. Scope of Investigation

2.1 Context

A nozzle test section has been designed [2] consisting of a settling chamber, axisymmetric contraction, a 2D converging-diverging nozzle section and testing channel outlined in Figure 2 and Figure 3.

![Figure 2 - Representation of 3D de Laval nozzle [2]](image)

Due to reasons associated to continuous operation, input power limitations and fluid choice [1], the throat area chosen is 20x10mm, the resulting converging-diverging section is 75mm in length followed by a 50x20mm test section. The diverging section profile is designed by utilising the method of characteristics.

![Figure 3 - 2D outline of de Laval nozzle [2]](image)

Small geometrical models will be injected into the diverging section to induce responses in compressibility effects and must be sufficiently designed to avoid bow shocks and supersonic blockage, shown in Figure 4. In addition, the support system to hold the model in place is yet to be conceived and must structurally withstand the oncoming flow.
In the past, Schlieren experiments [3] (out of scope) have been conducted under ideal air conditions with the geometries shown in Figure 5 to gain experimental data such as the shock wave angle and Mach number. However, similar experiments have not yet been conducted with vapours under NICF conditions. Therefore, conducting CFD simulations with vapours that exhibit NICF behaviour is useful for experiments to be performed in the future. This can subsequently help validate in-house codes, leading to further optimisation of high-speed ORC turbomachinery designs.

**2.2 Research Questions**

Simulations concerning the flow behaviour of these NICF within the nozzle test section are interesting as a first step for the design of such a component and its installations (model support system). In addition, the small geometrical models to be injected must be sufficiently designed to ensure supersonic flow and avoid bow shocks. Typically, under ideal gas assumptions at ambient conditions with air as the working fluid, inlet conditions for a trans-supersonic wind tunnel setup are low temperature and high pressure. However, depending on the working fluid chosen, and as it reaches critical point conditions, the majority of the expansion can be said to be highly influenced by the thermodynamics of the flow [2]. The compressibility factor, being the molar volume ratios between a gas to an ideal gas, would be low. This results in the flow being non-ideal for the initial part of the expansion.
In addition, there is also a lack of understanding concerning how the model support system structurally reacts to the oncoming NICF. Therefore, the force loadings acting on the model support that comes from the fluid flow must also be investigated to ensure a working design. As a result of these statements, it can be summarised that problems due to force loadings of pressure and temperature from NICF and the nature of the small-scale slender models affect the design and configuration of the model and support system.

Thus, some research questions and problems involving the design of the model and support system have been identified as the motivation behind this project:

- Can a model support system be designed such that it will be structurally sound for small-scale organic rankine cycle wind tunnels?
- What is the optimal model configuration with which to perform non-ideal gas experiments?
- How can the influence of the flow domain be coupled to influence the results of the structural domain?

2.3 Project Deliverables and Outcomes

In an effort to answer the research questions posed, the project deliverables have been identified and in order of completion these are to:

- Generate 2D high fidelity, steady state, non-ideal compressible flow simulations;
- Design a suitable model and support system that remains minimally intrusive to the results of the flow field;
- Write a tool that couples results from the CFD analysis to an FEA solver;
- Use this tool to investigate structurally how the designed model support system will react to the oncoming flow field during steady state analysis.

These objectives have been devised to complement the overarching goals and research purposes of the ORCHID design. The results obtained will aim to highlight any influences the model support system may have on compressible flow behaviour of non-ideal gases within the nozzle test section.
3. Literature Review

This section provides the literature study to gain insight on the topics related to this project. A thorough explanation on the chosen working fluid and boundary conditions of the supersonic nozzle is provided in Section 3.1 along with the theory used to define the diverging nozzle profile section. The theory behind forming relationships between normal and oblique shocks is given next in Section 3.2.1 with detailed insight to the differences between using real gas relations as opposed to the ideal gas law. Supersonic blockage is also investigated with results from literature included to discuss the phenomenon. This is used to design the model configuration and is provided in Section 3.2.2. A final review is conducted that studies the fluid-structure interaction field that relates the fluid domain to the structural domain and is shown in Section 3.3. Interpolation methods are discussed and conclusions found in literature are considered for further use in this project.

3.1 The ORCHID Test Rig

3.1.1 Plant Outline

Head et al [1], describes the working process behind the ORCHID in more detail. The schematic, provided in Figure 1, consists of the main components of the test facility comprising of the two main test sections, the pump, evaporator, regenerator and condenser. Not shown in the figure are auxiliary components such as the buffer tanks, valves, instrumentation or filters. Concerning the nozzle test section, technical limits of the lab dictate that the maximum thermal power input be 400kW and the constraint on the minimum exit Mach number be at least 2. Mach numbers of at least 2 resemble typical operating conditions downstream of ORC turbines currently under investigation in the propulsion and power group. To achieve this Mach number the pressure ratio across the nozzle must be appropriately chosen, with a higher pressure ratio resulting in a greater achievable Mach number, suggesting a high pressure at the nozzle inlet (evaporator outlet). As the inlet pressure is increased, the vapour arrives closer to the critical point conditions, and thus the expansion is influenced to a larger extent by non-ideal effects. These effects can be differentiated by a series of traditional measurements and more advanced LASER diagnostics techniques [1].

A throat area of 200mm² corresponding to a throat height and width of 10mm and 20mm respectively has also been sized as a balance between the influence of the boundary layer and power input required. For a more specific enquiry into the working conditions of the ORCHID plant not related to this project, Head et al [1] provides detailed explanations behind the sizing and design parameters of each component, a more detailed schematic layout of the overall ORCHID, and a step by step plant process operation.
3.1.2 Selection of the Working Fluid

Many organic fluids were considered for use as the working fluid in the nozzle. Head et al [1], conducts a thermodynamic cycle analysis for different candidate fluids to ensure compatibility of both test sections in one integrated balance of plant. A working criteria for the fluid selection was the thermal power required at the evaporator leading to the inlet of the nozzle (evaporator thermal load).

The table below summarises the most suitable working fluids by calculating their operating conditions for the nozzle test section. All calculated fluid properties and thermodynamic quantities came from the thermodynamic library FluidProp [1].

Table 1 - Working parameters for selected fluids for the nozzle test section

<table>
<thead>
<tr>
<th>Fluid (nozzle)</th>
<th>Variable</th>
<th>MM</th>
<th>MDM</th>
<th>PP2</th>
<th>PP90</th>
</tr>
</thead>
<tbody>
<tr>
<td>Evaporator Thermal Load</td>
<td>$Q_{\text{Evaporator}}$ [kW]</td>
<td>425</td>
<td>291</td>
<td>302</td>
<td>235</td>
</tr>
<tr>
<td>Regenerator Thermal Load</td>
<td>$Q_{\text{Evaporator}}$ [kW]</td>
<td>339</td>
<td>284</td>
<td>254</td>
<td>254</td>
</tr>
<tr>
<td>Condenser Thermal Load</td>
<td>$Q_{\text{Evaporator}}$ [kW]</td>
<td>436</td>
<td>298</td>
<td>309</td>
<td>240</td>
</tr>
<tr>
<td>Pump Power</td>
<td>$P_{\text{power}}$ [kW]</td>
<td>11</td>
<td>7</td>
<td>7</td>
<td>5</td>
</tr>
<tr>
<td>Maximum Temperature</td>
<td>$T_{\text{max}}$ [°C]</td>
<td>254</td>
<td>299</td>
<td>219</td>
<td>241</td>
</tr>
<tr>
<td>Maximum Pressure</td>
<td>$P_{\text{max}}$ [bar]</td>
<td>21.4</td>
<td>15.6</td>
<td>22.7</td>
<td>18.5</td>
</tr>
<tr>
<td>Minimum Pressure</td>
<td>$P_{\text{min}}$ [bar]</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Condensing Temperature</td>
<td>$T_{\text{condensing}}$ [°C]</td>
<td>100</td>
<td>153</td>
<td>75</td>
<td>100</td>
</tr>
<tr>
<td>Cycle Mass Flow</td>
<td>$m$ [kg/s]</td>
<td>1.9</td>
<td>1.5</td>
<td>3</td>
<td>2.6</td>
</tr>
</tbody>
</table>

Table 2 - Working parameters for selected fluids for the turbine test section

<table>
<thead>
<tr>
<th>Fluid (turbine)</th>
<th>Variable</th>
<th>MM</th>
<th>MDM</th>
<th>PP2</th>
<th>PP90</th>
</tr>
</thead>
<tbody>
<tr>
<td>Evaporator Thermal Load</td>
<td>$Q_{\text{Evaporator}}$ [kW]</td>
<td>41.0</td>
<td>51.3</td>
<td>37.7</td>
<td>46</td>
</tr>
<tr>
<td>Regenerator Thermal Load</td>
<td>$Q_{\text{Evaporator}}$ [kW]</td>
<td>43.3</td>
<td>63.3</td>
<td>60.9</td>
<td>76.6</td>
</tr>
<tr>
<td>Condenser Thermal Load</td>
<td>$Q_{\text{Evaporator}}$ [kW]</td>
<td>31.8</td>
<td>42.1</td>
<td>28.3</td>
<td>36.9</td>
</tr>
<tr>
<td>Pump Power</td>
<td>$P_{\text{power}}$ [kW]</td>
<td>0.9</td>
<td>0.9</td>
<td>0.8</td>
<td>1.0</td>
</tr>
<tr>
<td>Maximum Temperature</td>
<td>$T_{\text{max}}$ [°C]</td>
<td>300.0</td>
<td>320.0</td>
<td>320.0</td>
<td>320.0</td>
</tr>
<tr>
<td>Maximum Pressure</td>
<td>$P_{\text{max}}$ [bar]</td>
<td>22.0</td>
<td>14.0</td>
<td>22.0</td>
<td>20.0</td>
</tr>
<tr>
<td>Condensing Pressure</td>
<td>$P_{\text{min}}$ [bar]</td>
<td>0.3</td>
<td>0.3</td>
<td>0.43</td>
<td>0.3</td>
</tr>
<tr>
<td>Condensing Temperature</td>
<td>$T_{\text{condensing}}$ [°C]</td>
<td>63</td>
<td>112</td>
<td>50.0</td>
<td>87.5</td>
</tr>
<tr>
<td>Cycle efficiency</td>
<td>$\eta_{\text{cycle}}$</td>
<td>22.3</td>
<td>17.7</td>
<td>24.5</td>
<td>19.6</td>
</tr>
</tbody>
</table>
Comparing the results from Table 2 indicate that PP2 seems to be the best candidate as the fluid is able to withstand the highest max temperature and pressure. However this fluid was disregarded due to significant disadvantages of cost and uncertainties with thermodynamic properties and thermal stability.

As a result, the linear siloxane MM fluid was deemed more suitable, with a similar performance compared to PP2 for cycle efficiency and MM having required the largest thermal load for supersonic flow in the nozzle. It is also worth noting that to avoid future adaptation complications of the ORCHID, the design of the plant can be easily adapted in the future to also be compatible with other working fluids such as D₄, MDM, PP2 and PP90.

3.1.3 Working Boundary Conditions

The operating conditions pertaining to the nozzle for the working fluid MM are populated below in Table 3. As the chosen working fluid for the ORCHID design phase, these conditions will be used to label the boundary conditions for the subsequent CFD simulations to be conducted throughout this project. With only the necessary conditions listed here, full tables and specifications on the operating conditions of other components can be found in [1].

Table 3 - Subcritical boundary conditions of nozzle

<table>
<thead>
<tr>
<th>Nozzle</th>
<th>Parameter</th>
<th>MM Boundary Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inlet (Evaporator Outlet/Nozzle Inlet)</strong></td>
<td>Pressure P₁ (Bar)</td>
<td>18.4</td>
</tr>
<tr>
<td></td>
<td>Temperature T₁ (K)</td>
<td>527</td>
</tr>
<tr>
<td></td>
<td>Volumetric Flow V₁ (m/s)</td>
<td>424</td>
</tr>
<tr>
<td><strong>Outlet (Nozzle Outlet/Nozzle Inlet)</strong></td>
<td>Pressure P₃ (Bar)</td>
<td>2.1</td>
</tr>
<tr>
<td></td>
<td>Temperature T₃ (K)</td>
<td>197</td>
</tr>
<tr>
<td></td>
<td>Density ρ₃ (kg/m³)</td>
<td>9.3</td>
</tr>
<tr>
<td></td>
<td>Velocity U₃ (m/s)</td>
<td>298.4</td>
</tr>
<tr>
<td></td>
<td>Area ratio (A²/A₁)</td>
<td>0.295</td>
</tr>
<tr>
<td></td>
<td>Mach Number (M₃)</td>
<td>2.0</td>
</tr>
</tbody>
</table>

3.1.4 Nozzle Profile - Method of Characteristics

To design for the nozzle profile, a technique known as the method of characteristics can be utilised. This is a numerical technique that uses partial differential equations to calculate isentropic portions within supersonic flows [4]. These supersonic flows belong to a hyperbolic class of partial differential equations and there exists characteristic lines describing the Mach waves.
Figure 6 shows an example of a characteristic Mach wave, denoted as $s$ in supersonic flow with an axis $n$ normal to it. The two characteristic lines $\eta$ and $\xi$ are inclined at an angle $\mu$ to the streamline. It can be shown that for both Mach lines:

\[
\frac{\partial}{\partial n} (v - \theta) = 0; \quad v - \theta = R
\]

\[
\frac{\partial}{\partial n} (v + \theta) = 0; \quad v + \theta = Q
\]

These compatibility relations say that $Q$ and $R$ are invariant and can be used to model the profile of the nozzle. The positive sign is used for expansion and the negative for compression. By taking known thermodynamic and flow quantities along the midline of the nozzle axis for one dimensional analysis, the diverging section can be modeled. For example, by considering a curve AB shown in Figure 7, values at point C can be calculated using the compatibility relations \[6\]. Solving for values at point C gives:

\[
v_C = \frac{1}{2} (Q + R)
\]

\[
\theta_C = \frac{1}{2} (Q - R)
\]

This process is continued to calculate the waves downstream of the throat shown in Figure 8. In general, the diverging section should be designed with smooth curved walls by approximating the curved section by short straight segments, hence the more straight segments by increasing the number of points along a starting curve, the more accurate the method.

The strength and reflection of the wave along the midline of the nozzle remains an expansion wave or a compression wave. Therefore, $\theta$ remains constant and hence, its angle of incidence must equal the reflection angle as shown in figure 8. The diverging section of the de Laval nozzle will be modeled using this method and is described in Section 6.1.
3.2 Shockwave Generators

3.2.1 Normal and Oblique Shock Theory for Real Gases

Conditions across surfaces of discontinuity such as oblique or normal shocks are governed by the integral conservation equations [7]. For flows involving steady shock waves, the governing equations are given as:

$$h_1 + \frac{u_1^2}{2} = h_2 + \frac{u_2^2}{2}$$  \hspace{1cm} (4)

where subscript 1 refers to conditions upstream of the normal shock and subscript 2 being conditions downstream. If conditions upstream of the shock such as pressure, density, enthalpy and velocity are known or specified, relations for the downstream conditions can be expressed.

Considering a gas which is both thermally and calorically perfect, the state relationship

$$h = c_p T = \frac{\gamma p}{\gamma - 1 \rho}$$  \hspace{1cm} (5)

may be used. The well known Rankine-Hugoniot relation is expressed where ratios between the upstream and downstream conditions can be calculated. Of particular interest is the loss in stagnation pressure across the normal shock given as:

$$\frac{p_{02}}{p_{01}} = \left[1 + \frac{2\gamma}{\gamma + 1}(M_1^2 - 1)\right]^{-\frac{\gamma}{\gamma - 1}} - \left[\frac{2 + (\gamma - 1)M_1^2}{(\gamma - 1)M_1^2}\right]^{-\frac{\gamma}{\gamma - 1}}$$  \hspace{1cm} (6)

This relation will be used for Section 3.2.2 to investigate supersonic blockage. Under oblique shock conditions, where a component of a shock wave is turned tangential to a normal shock $V_r$, conservation equations are also used by observing the velocity normal to the oblique shock $V_{nr}$, shown in Figure 9.
Under ideal conditions, the flow is conserved similarly when:

\[ V_{t1} = V_{t2} \quad (7) \]

\[ h_1 + \frac{V_{n1}^2}{2} = h_2 + \frac{V_{n2}^2}{2} \quad (8) \]

Relative to the upstream flow, the shock wave inclination angle is denoted as \( \beta \), and the downstream flow has an inclination angle of \( \theta \). These are expressed as:

\[ V_{n1} = V_1 \sin \beta \]
\[ V_{t1} = V_1 \cos \beta \]
\[ V_{n2} = V_2 \sin (\beta - \theta) \]
\[ V_{t2} = V_2 \cos (\beta - \theta) \quad (10) \]

In order to find the shock angles, a relatively simple expression is given which relates the two inclination angles to the normal velocities.

\[ \frac{\tan(\beta - \theta)}{\tan \beta} = \frac{V_{n2}}{V_{n1}} \quad (11) \]

However for flows involving real gases, this closed form expression cannot be used. The Rankine-Hugoniot relations are invalid and relations involving shock angles must be calculated for using an alternative way. An iterative procedure that utilises trigonometric identities to solve for \( \tan(\beta) \) can be used, where the solution is given as [7]:

\[ \tan \beta = \frac{(1 - v) \pm [(1 - v)^2 - 4v \tan^2 \theta]^{1/2}}{2v \tan \theta} \quad (12) \]

Calculating for \( \beta \) will be a good indication of how valid the high-fidelity computational results obtained throughout this project will be. In general, for ideal gases, the ratio of quantities across a normal shock such as the stagnation pressure loss ratio are given using the Rankine-Hugoniot relations where expressions are given as a function of upstream Mach number. For real gases,
expressions are dependent not only of upstream Mach number, but also upstream velocity and two thermodynamic variables such as pressure and density.

### 3.2.2 Supersonic Blockage

To allow the establishment of supersonic flow with a particular model and support system, a phenomenon known as supersonic blockage must be investigated. This is generally described by the one dimensional theory assuming an entropy increase across a normal shock wave [8] [9] given by:

\[
\frac{\Delta A}{A_{core}} = 1 - \frac{P_{02}}{P_{01}} \frac{A^*}{A_{core}}
\]

where:

- \(\Delta A\) = Model frontal area;
- \(A_{core}\) = Aerodynamic test section area;
- \(A^*\) = Nozzle throat area;
- \(P_{02}\) = Stagnation pressure downstream of normal shock wave, and
- \(P_{01}\) = Facility stagnation pressure upstream of normal shock wave.

This ratio between the model frontal area, \(\Delta A\) to the test section area at the tip of the model, \(A_{core}\) represents the theoretical limit which will permit establishment of supersonic flow in the nozzle. Figure 10 labels the variables used to determine this ratio.

![Figure 10 - Labeling of variables used to calculate supersonic blockage](image)

Tests conducted by Scheuler [8] shows the blockage results obtained under ideal air conditions for various cone and wedge models, shown below in Figure 11. The blockage ratio is plotted as a function of Mach number at the tip of the model before the normal shock. The theoretical result based on the described one-dimensional flow theory is shown as the dotted line. Scheuler conducted tests below and above the limit at various Mach numbers. The solid symbols above the limit indicate that the nozzle becomes blocked while an open symbol indicate that the tunnel has started. Figure 12 shows similar tests conducted by Czysz [9] for differing nose drag coefficients.
From the result shown, it is apparent that the theory of calculating for the blockage ratio based on one-dimensional normal shock theory does not completely correlate with the experimental results.
Scheuler obtained although the trends are similar. However, results obtained by Czysz shows good agreement for drag coefficients less than 0.7, greater than this, there is a considerable change in the trendline for calculating the limiting size of the model frontal area.

Both Scheuler and Czysz state that reasons that the experimental data does not exactly correlate with the theoretical results may be because of other factors outside the blockage ratio. Factors such as the nature of the flow-starting process, tunnel boundary layer, operating conditions and the layout of the support system can influence the correlation of the permissible model size results to theoretical results. Due to these reasons, Dayman [10] states that a good rule-of-thumb when designing for the model sizing is to calculate the maximum model size area to be 60% of the theoretical values obtained.

Overall Czysz concludes that model blockage results for various model types can be reasonably well correlated for drag coefficients up to about 0.7 and for most shapes up to angles of attack of 40 degrees. Scheuler concludes his findings by stating that the proposed theoretical method of correlating model blockage results was found to be applicable over a limited range of Mach numbers only. An investigation of supersonic blockage in the context of this project is given in Section 6.3.

### 3.3 The Fluid-Structure Interaction Problem Definition

#### 3.3.1 FSI Procedure Overview

One of the research questions posed dealt with how to couple the CFD and FEA such that the results of one domain can influence the other and vice versa. This can be achieved by utilising a field of study known as Fluid-Structure Interaction (FSI) which is an example of a multi-physics problem where interaction between two independent domains take place. The basic steps for the theory can be summarised as such:

1. Begin with a fluid mesh domain;
2. Run and simulate the flow equations;
3. Extract values of importance along the interfacing surface;
4. Transfer the results onto the structural mesh domain as loadings for the FEA setup;
5. Solve the structural equations to obtain structural results such as deformation;
6. Transfer the deformation data onto the fluid mesh domain;
7. Update fluid mesh nodes corresponding to deformation data.

These steps represent one timestep and is repeated until the solution converges to a suitable threshold or the number of timesteps are met. This is termed the two-way fluid-structure problem.
definition as the results of one domain influence the other and vice-versa [11]. de Boer et al. [12] provides a basic visual of these steps to achieve the coupled behaviour shown below.

![Figure 13 - Steps for conducting the Two-Way FSI Procedure [12]](image)

Assuming the state of the fluid mesh is known at time $t_n$ and $t_{n+1}$, the fluid solution $W_{n+1}$ is computed. The pressure data obtained from the simulation is transferred to the structural mesh model $U_n$ where the structural solver will update the displacement and structural mesh for $U_{n+1}$ and $u_{n+1}$ respectively. Displacement is transferred to the fluid model where deformation can be captured in this domain and solutions for the next fluid CFD simulation can be found at $W_{n+2}$ corresponding to time $t_{n+2}$.

The interaction steps described take place at the common boundaries that the mechanical model shares with the fluid domain. An important aspect is the method of transferral of pressure loads from the fluid nodes at the interface to its corresponding structural nodes. It is usually not desirable to generate matching meshes, as the fluid flow requires a much finer mesh than the structural mesh to obtain accurate results [13]. This is because the spatial discretisation of the flow field is usually based on a finite volume formulation using Eulerian coordinates, and the structural model consisting of finite elements in a Lagrangian description [13] [14]. This means that generally, the two meshes do not share the same amount of nodes and nodal co-ordinates at their common boundary, as shown in Figure 14.
As a result, a coupling scheme interpolating values of interest between the meshes will need to be devised. This coupling approach should allow independent discretisation of each mesh, allowing the structural and fluid domains to remain independent of each other. Various interpolation methods will be discussed in the next section with regards to their efficiency and accuracy and conclusions will result in a chosen method for use in this project.

### 3.3.2 Interpolation Methods

To transfer values of importance between non-matching meshes, there are typically two types of interpolation approaches within many methodologies observed in literature; a consistent and conservative approach. The conservative method approaches the methodology through global conservation of virtual work over the interface. This approach uses one transformation matrix to perform the transfer of pressure loads between two discrete interfaces. This can however, lead to unphysical oscillations in the pressure forces received by the structure [14].

Hence, instead of using the same transformation matrix, two different matrices are used for each interpolation process; the consistent approach. However, the virtual work over the interface is generally not conserved due to the errors caused in time lag between the flow and structural meshes. But when this error introduced is smaller than the spatial and temporal discretisation error, it generally will not affect the stability and accuracy of the computation [12].

It can be generally said that whichever interpolation method is chosen, the outcome and use of the transformation matrices are formulated as such [12]:

\[
P_s = H_{fs}P_f
\]

\[
T_s = H_{fs}T_f
\]
Where $H$ represents the transformation matrix to transfer values of one mesh to the other. To investigate and compare between different methods of interpolation, Beckert [13] provides an analysis for a test case. The paper sets up a quasi one-dimensional channel with a flexible curved wall shown in Figure 15.

![Figure 15 - Test case for interpolation testing of different methodologies [13]](image)

To couple the structural and flow domains, the iterative approach described in section 3.3.1 was implemented and compared to with a numerically 'exact' solution. Resulting interpolated pressures for the structural domain and interpolated displacements for the fluid domain were computed. The exact steady state solution was solved on a much finer mesh and to set up the interpolation, the number of cells used were $n_f = 21 \times 2^k$ flow cells and $n_s = 6 \times 2^k$ structure cells plotted over the same boundary in the form of the initial shape of the channel $z_0(x) = a_0 - a_1 e^{-a_2 x^2}$ ($-0.5 < x < 0.5$).

Many various methods of interpolation methods were conducted and are listed below:

<table>
<thead>
<tr>
<th>Notation</th>
<th>Interpolation Method Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>NN</td>
<td>Nearest-neighbour</td>
</tr>
<tr>
<td>TPS</td>
<td>Thin-plate Splines</td>
</tr>
<tr>
<td>MQ</td>
<td>Multi-quadric Biharmonic splines</td>
</tr>
<tr>
<td>GI</td>
<td>Gauss Integration</td>
</tr>
<tr>
<td>RBF 0.25</td>
<td>Radial Basis Function with radius $r = 0.25$</td>
</tr>
<tr>
<td>RBF 5</td>
<td>Radial Basis Function with radius $r = 5$</td>
</tr>
</tbody>
</table>

![Table 4 - Interpolation schemes used by Beckert [13]](image)

Figure 16 shows the log-log error for the interpolated displacement and pressure values compared to the number of structure cells for both conservative and consistent approaches of each method.
The solid line is obtained with the conservative approach and dotted lines being the consistent approach.

![Figure 16 - Interpolation error vs no. of structural cells; Left: displacement, Right: pressure [13]](image)

It can be noted that for both the error in displacement and pressure, the NN interpolation method is quite inaccurate relative to the other methods with errors never reaching a magnitude of below 1e-2. The RBF methods are the most accurate with the consistent approach providing lower errors. Other methods such as the TPS and MQ methods are comparable to RBF but RBF interpolation methods can be more accurate by increasing the value of its radius as observed in the figures. Figure 17 shows the error compared to CPU time and results indicate that the consistent RBF method with a large radius is again the most efficient way and the most inefficient being the NN method.

![Figure 17 - Interpolation error vs CPU time; Left: displacement, Right: pressure [13]](image)

If a weighted residual formation is to be used the highest accuracy is obtained with a conservative approach. All other methods obtain more accurate results with the consistent approach. The paper concludes that considering the number of cells and CPU time, the methods based on radial basis function interpolation are preferred. Other methods such as nearest-neighbour are only first order
accurate and while Gauss interpolation or the Thin Spline method also show accurate interpolations, the search and projection algorithms needed result in expensive computational times relative to RBF methods [12].

As a result, the radial basis function method will be chosen to implement into the FSI problem definition, with further review and tests conducted to compare between the consistent and conservative approach.

### 3.3.3 Radial Basis Functions for Interpolation between Non-Matching Meshes

The chosen method of RBF interpolation is based on the use of spline functions [13]. Suppose a quantity is to be transferred from mesh A to mesh B. For the consistent approach, this is approximated by a sum of basis functions both at the interface of mesh A and B:

\[
s_i(x_i) = \sum_{j=1}^{n_A} \gamma_j \phi(||x_i - x_A||) + q(x)
\]

where:

- \( s_i(x_i) \) = the discrete interpolated value at domain \( i \)
- \( n_A \) = number of node points at mesh A
- \( q(x) = \alpha_0 + x\alpha_1 + y\alpha_2 \) a linear polynomial
- \( \gamma \) = coefficients for the radial basis function \( \phi \)
- and \( \phi \) the given radial basis function with respect to Euclidean distance \( ||x|| \).

The coefficients \( \gamma \) and the polynomial \( q \) can be determined by the interpolation of the discrete values of the known mesh A, along with the additional requirements that the summation of these values equal to 0.

\[
D_A = (Known \ values \ at \ mesh \ A) \tag{16}
\]

\[
D_A = s_A(x_A) = \sum_{j=1}^{n_A} \gamma_j \phi(||x_A - x_A||) + q(x)
\]

\[
\sum_{j=1}^{n_A} \gamma_j q(x) = 0 \tag{17}
\]

Expressions (17) and (16) are coupled together in matrix form to give the following:

\[
[D_A^T]_0 = \begin{bmatrix} \phi_{AA} & Q_A \\ Q_A^T & 0 \end{bmatrix} \gamma' \tag{18}
\]
where:

- $\phi_{AA}$ is an $n_A \times n_A$ matrix containing the evaluation of $\phi(\|x_A - x_A\|)$
- $Q_A$ is a $n_A \times 4$ matrix with row $j$ of the node points $[1 \ X_{Aj} \ Y_{Aj}]$
- $\gamma$ containing the coefficients of $\gamma$
- and $\beta$ containing the coefficients of $q(x)$.

To obtain the interpolated value on mesh B, equation (15) is evaluated again but for points at mesh B. The matrix form of the equation is also expressed:

$$D_B = (\text{Unknown values at mesh B})$$

$$D_B = s_B(x_B) = \sum_{j=1}^{n_A} \gamma \phi(\|x_B - x_A\|) + q(x)$$

$$[D_B] = [\phi_{BA} \ Q_B] [\gamma \beta]$$

where:

- $\phi_{BA}$ is an $n_B \times n_A$ matrix containing the evaluation of $\phi(\|x_B - x_A\|)$
- $Q_B$ is a $n_B \times 4$ matrix with row $j$ of the node points $[1 \ X_{Bj} \ Y_{Bj}]$

Combining expressions (20) and (18) gives a relationship for the transformation derived as follows:

$$[\gamma \beta] = \begin{bmatrix} \phi_{AA} & Q_A \
Q_A^T & 0 \end{bmatrix}^{-1} [D_A]$$

$$[D_B] = [\phi_{BA} \ Q_B] \begin{bmatrix} \phi_{AA} & Q_A \
Q_A^T & 0 \end{bmatrix}^{-1} [D_A]$$

$$[D_B] = H_{AB} [D_A]$$

The variable $D$ denotes the value of interest that is to be interpolated. In the context of this project, $D$ is the pressure and temperature values along the interfacing boundary. Therefore the transformation matrix $H$ for converting values of a known quantity in mesh A to mesh B is calculated as:

$$H_{AB} = [\phi_{BA} \ Q_B] \begin{bmatrix} \phi_{AA} & Q_A \
Q_A^T & 0 \end{bmatrix}^{-1}$$

The evaluation of this transformation matrix is computationally inexpensive compared to other methods explored by de Boer et al [12] and Beckert and Wendland [13]. No orthogonal projection or search algorithms are needed and the computation of the inverse matrix is relatively small compared to other methods.
With regards, to the choice of the radial basis function \( \phi \), many options are available, divided into two groups; functions with compact support and global support. Beckert and Wendland \([13]\) uses a compact supported radial basis function defined as:

\[
\phi(||X||) = (1 - \frac{||X||}{r})^2 + \left(\frac{4||X||}{r} + 1\right)
\]  

(25)

The choice of the radius \( r \) will need to be considered. A large number yields a good approximation order, but too large leads to a singular matrix, while a small number leads to a solution not accurate enough.

For the conservative approach, energy is globally conserved over the interface, when \([15]\)

\[
\int u_f \cdot p_f n_f ds = \int u_s \cdot p_s n_s ds
\]  

(26)

where:

- \( u_{f,s} \) = displacement;
- \( p_{f,s} \) = pressure and;
- \( n_{f,s} \) = the outward normal of the flow and structure interface.

The pressure values can be defined by the approximations:

\[
p(x)n(x) = \sum_{j=1}^{n_p} D_j(x) P_j
\]  

(27)

Writing out equation (18) in terms of (17) shows that energy is globally conserved when:

\[
[M_{ff} U_f]^T P_f = [M_{ss} U_s]^T P_s
\]  

(28)

\[
U_f^T M_{ff}^T P_f = U_s^T M_{ss}^T P_s
\]

\[
U_f^T H_{sf}^T M_{ff}^T P_f = U_s^T M_{ss}^T P_s
\]  

(29)

Thus, the method of converting pressure values to the structural domain is expressed in a simple manner. Simplification of expression (20) leads to:

\[
P_s = [M_{ff} H_{sf} M_{ss}^{-1}]^T P_f
\]  

(30)

where the matrices \( M_f \) and \( M_s \) are defined as:

\[
M_{ff}^{ij} = \int D_f^i N_f^j ds
\]
\[ M_{ss}^{ij} = \int D_s^i N_s^j ds \] (31)

To test both the conservative and consistent approach, these equations will be applied to a simple test case of our own to observe their accuracy, which will be discussed in the section 5.1, concluding with which choice will be chosen for implementation into the FSI problem definition.
4. Software Overview

Table 5 provides an overview of the programs that are used throughout the project. A number of commercial tools are utilised and additionally, two in-house tools such as SU2 and UMG2. This section will provide a brief overview and description of these in-house programs and how they relate to the project.

Table 5 - Overview of chosen programs

<table>
<thead>
<tr>
<th>Program</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solidworks</td>
<td>Draw the geometry of the model and its support system</td>
</tr>
<tr>
<td>UMG2</td>
<td>Generate the initial unstructured fluid mesh of the nozzle with the geometry created in Solidworks</td>
</tr>
<tr>
<td>SU2</td>
<td>Chosen program to conduct the CFD simulations with the mesh obtained from UMG2</td>
</tr>
<tr>
<td>Nastran</td>
<td>Chosen program to conduct the FEA with the results obtained from CFD</td>
</tr>
<tr>
<td>Matlab</td>
<td>Will act as the interfacing between the fluid and structural domain for the FSI problem definition</td>
</tr>
</tbody>
</table>

4.1 The Mesh Generator

The in-house mesh generator UMG2 developed by Ghidoni [16] emphasises the necessity of a fast and automatic tool for mesh generation on geometries of an arbitrary, complex shape for unstructured forms. The methodology to generate the unstructured mesh is described in Table 6 with the exact syntax for carrying out these steps provided in Appendix A.

Table 6 - Steps for the generation of unstructured meshes in UMG2

<table>
<thead>
<tr>
<th>Step</th>
<th>Process</th>
<th>Description of Method</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Definition of the domain boundaries</td>
<td>Points along the CAD model of interest defined and modeled</td>
</tr>
<tr>
<td>2</td>
<td>Specification of the sizing of elements</td>
<td>A function H will be used to specify the size and directional features of the mesh elements pertaining to each wall.</td>
</tr>
<tr>
<td>3</td>
<td>Generation of the mesh</td>
<td>Meshing technique based on the Delaunay criteria (out of scope) is employed</td>
</tr>
</tbody>
</table>
| 4 | Optimisation of the element shapes | Three automatic processes are used to improve quality:  
  - Diagonal Sweeping - a topological operation leading to swap edges only if there is an improvement of element quality  
  - Edge Collapsing - allows to remove an edge if elements are below a threshold  
  - Node Movement - Fictitious springs are implemented and mesh elements are spaced such that these springs are in equilibrium |
The ability to automate the mesh generation process provided by this software, especially for unstructured meshes will be beneficial. Unstructured meshes are recommended as they are popular for complex geometries because of their flexibility to discretise arbitrary complex domains [17]. As a result, the types of meshes to be generated for the fluid domain will be a 2D, unstructured, triangular mesh. Boundary layers will be added if Navier stokes fluid models are used. Ghidoni’s presentation on his code [16] provides more in-depth detail about the generation process along with a user guide.

4.2 CFD Simulation Software
The CFD program to be used will be SU2 which is an open source tool. This program is currently being developed at TU Delft alongside a number of other institutions. One of the extensions that is planned for future use with this program is the inclusion of an in-built FSI procedure (out of scope) [18] [19]. Therefore, usage of this CFD program for this project will benefit the developers to ensure its validation for FSI use. For post-processing analyses, solution files are readable in ASCII format to allow easy extraction of information using 3rd party tools such as Matlab. Refer to Appendix B for a breakdown of the syntax, pre-processing and post-processing steps to conduct CFD simulations.
5. Fluid-Structure Interaction Setup

This section gives an overview of the fluid-structure methodology utilised to set up the methodology of the project. With the radial basis interpolation method chosen for use, further tests to justify its accuracy and chosen approach is conducted in Section 5.1. A detailed explanation is then provided in Section 5.2 that describes how the written coupling tool was conceived to conduct the interpolation.

5.1 Justification of Chosen Interpolation Method

The chosen method to interpolate values of interest from the fluid mesh to the structural mesh was the radial basis function with the selected function \( \phi \) being the \( C^2 \) function shown in Equation (32) below:

\[
\phi(||x||) = \begin{cases} 
(1 - \frac{||x||}{r})^4 & ||x|| \leq 0 \\
\frac{4||x||}{r} + 1 & ||x|| > 0 
\end{cases}
\]

where the support radius \( r = 1 \) will be chosen to begin the initial analysis. A test case to apply the accuracy and efficiency of each approach (consistent or conservative) was written in Matlab. The purposes of the test case is to decide which approach to use and also to determine firsthand how accurate this interpolation method is. For each discrete point \( i \), each mesh has the discrete co-ordinate \((X_{D,i}, Y_{D,i})\) where the y-value is graphed in the form of a cosine wave in two dimensions:

\[
D(x_i) = \cos(2\pi x_i); 0 \leq x \leq 1
\]

\[
D = \text{(Fluid mesh F, Structural Mesh \textit{S})}
\]

To simulate the pressure points, the x-values of the cosine wave are then graphed over a sine wave.

\[
p(x_{s,i}) = \sin(2\pi x_{s,i}) \text{ for structural mesh points}
\]

These curves were chosen as it is simple enough to detect the accuracy and to compare it with the exact results. After interpolation of the pressure values in the fluid domain (interpreted as a sine wave) to the structural domain, the points obtained in the structural domain will be graphed against the exact sine wave to determine its accuracy.

In addition, the root mean square error of both the consistent and conservative approach will be computed. The transformation matrix \( H_{fs} \) (subscript \( f \) to \( s \) denoting fluid to structural domain) was computed using the RBF method in a Matlab script so that the following calculations can be made:

\[
p_s = H_{fs} p_f
\]
Figure 13 shows the comparison of the interpolated values against the exact sine curve. Table 7 shows the RMS errors obtained with both approaches.

Table 7 - RMS Error obtained with the RBF interpolation test case

<table>
<thead>
<tr>
<th></th>
<th>Pressure Interpolation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Consistent RMS Error</td>
<td>0.001</td>
</tr>
<tr>
<td>Conservative RMS Error</td>
<td>0.0038</td>
</tr>
<tr>
<td>% difference</td>
<td>73.6%</td>
</tr>
</tbody>
</table>

From the resulting graph, both approaches matched the sine curve to a close degree. From observation of the RMS errors, the consistent approach showed a lower error of 0.001 compared to the RMS error obtained from the conservative approach of 0.0038. This showed that the interpolated values from the consistent approach were overall closer to the exact curve than the conservative approach. From this simple test, it is apparent that the consistent method is the superior choice.

Further tests were also conducted to observe the effect of the RMS error with increasing numbers of node points in the meshes, shown in Figure 19. To decide on a number of nodes for the meshes a similar approach to Beckert [13] was used where the number of structural nodes and fluid nodes was determined with the expression:

\[ N_s = 3 \times 2^{i-1} + 1 \]
\[ N_f = 5 \times 2^{i-1} + 1 \]
\[ 1 < i < 8 \]

It is clearly seen that the consistent RBF method has a lower RMS error than the conservative error. As number of elements increase, the error decreases as more elements mean a higher degree of accuracy when transferring values between domains. For the consistent approach, RMS errors decrease linearly, but the conservative approach seems to plateau after 100 structure elements. It is likely the error will converge as elements increase, meaning the conservative approach has a limit on its accuracy.

A final test was conducted to test the accuracy of the interpolation method with varying \( r \) values, shown in Figure 20.
Radius values between 1 and 10 were tested, and the graph provided no change in the error result as radius increases. One reason for this may be that a large enough radius value was not tested for the matrix to reach a singular level due to the simplicity of the chosen test case. However, this particular test reinforces the result that the consistent approach is more accurate than the conservative approach. It can also be concluded that this particular test justifies the use of using a radius of $r = 1$ being large enough to be accurate while also remaining low to save computational time.

From the test case described, the tests conducted included:

- a comparison between the conservative and consistent approach against a simple curve;
- an observation of the error as mesh nodes increase; and
- an observation of the error for increasing radius value.

All tests resulted that using the consistent method provided more accurate results, and thus will be the approach used to conduct the one-way FSI problem definition.

### 5.2 Modeling Procedure

A two-way FSI procedure was explained in detail where the results of one domain would influence the other and vice versa. Due to time constraints and the complexity of implementing the loop that deforms the initial fluid mesh in an efficient and automated manner, this project will utilise instead a one-way FSI procedure [20].
Shown in Figure 21 is the one-way procedure within the full two-way procedure. In a one-way FSI procedure, the results of the CFD analysis at the matching boundaries are transferred and applied as loads to the mechanical model. The subsequent calculated displacements from the FEA solver at the interface are not transferred back. Limitations arise from only utilising a one-way method and these include:

- Only being able to conduct one timestep in the FSI procedure;
- Flow field changes due to the structural deflection of the model can’t be analysed; and
- Only initial deflections and stresses are analysed. Fatigue or excitations can’t be properly analysed as the structural results cannot be looped back to the fluid domain to simulate the two-way procedure. As a result, only the stressing that occurs during a steady state analysis and deflection of the model tip for one timestep can be analysed.

![Figure 21 - 1-Way FSI Procedure within the full 2-Way Procedure](image)

This analysis is usually appropriate if the displacements calculated are not large enough to have a significant impact on the fluid analysis. After results are obtained, conclusions regarding whether a full two-way FSI procedure should be conducted for the future will be discussed.

To conduct the 1-way FSI procedure, Figure 22 provides a visual layout of how the structural results starting from the CFD fluid analysis were obtained in a procedural manner. A number of Matlab functions were written to carry out specific tasks needed for the RBF interpolation method. These included:

- Extracted pressure values from the CFD simulation along the matching boundary of interest;
- Nodal co-ordinates and number of nodes for the fluid domain at the matching boundary;
- Nodal co-ordinates and number of nodes for the structural domain at the matching boundary.

The written Matlab code, \textit{F2S.m} reads the acquired data and performs the RBF interpolation method. The results are then extracted to (.csv) files and applied as loadings for the FEA setup. Appendix C provides a full documentation on the written Matlab functions.

![Diagram](image-url)

\textbf{Figure 22 - Conceived tool to carry out One-Way FSI Procedure}
6. Fluid Model Setup

This section describes the methodology and the design choices made pertaining to the fluid domain. Section 6.1 describes the results used from conducting the Method of Characteristics to obtain the diverging nozzle section. Section 6.2 conducts a mesh sensitivity analysis where initial CFD results are obtained for several test meshes with different densities. A convergence study is discussed with conclusions leading to a chosen mesh density to be used for the one-way FSI simulations. The CFD configuration settings, along with the initial and boundary conditions are also given in this section. Section 6.3 then investigates the supersonic blockage phenomenon where results are obtained through Matlab of the blockage ratios for Mach numbers along the midline of the nozzle. Using these plots, a suitable sizing of the model is designed followed by CFD checks to ensure blockage is avoided.

6.1 Nozzle Profile Generation

To generate the diverging section of the de Laval nozzle, a Fortran executable which implements the method of characteristics was used. Refer to Appendix G for Matlab file documentation on the generation of the nozzle profile. Figure 23 shows the Mach number along the generated diverging nozzle profile and Figure 24 the pressure distribution. Thirty characteristic lines were specified in the configuration file to allow enough starting points for an accurate profile.

The line starting at 27mm from the throat to the end of the nozzle profile represents the last expansion wave before flow parameters become constant at the midline of the nozzle, seen in the figures below. This results in the diverging section to start at co-ordinates of (0, 5)mm and ends at (79, 24.53)mm. The converging section which was designed prior, will connect to this profile and will be used to accelerate the flow to obtain sonic conditions at the throat. Following this profile will be the testing channel where the wall will remain at a constant length from the midline and will be where the model support will be implemented to hold the model statically in place.
6.2 Mesh Sensitivity Analysis

6.2.1 Test Case

In this section, an analysis of the accuracy of the flow solution with different mesh densities of the same unstructured test case are performed. With the aim of choosing a suitable density for the FSI simulations, the objective is to determine this density such that convergence rates of the solution will not be affected if the mesh was any denser. This mesh will output an accurate solution while also being the least computationally expensive.

The test case for the simulations is of the de Laval nozzle in 2D form with a diamond and sting represented as wall1 in the figure below. Euler simulations will take place as it will save computational time to ignore viscous boundaries for now. A symmetric geometry is used with the boundaries labeled in Figure 25. This geometry was drawn in Solidworks, then exported to UMG2 which generated the unstructured mesh shown in Figure 26.
Seven different meshes were tested with their densities determined in UMG2 by specifying the face spacing of the individual elements. The smaller the number indicating a smaller element face. Table 8 displays the chosen meshes and their resulting number of nodes.

### Table 8 - Summary of meshes used with test densities

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Face Spacing</th>
<th>No. of Nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>M_1.0</td>
<td>1.0</td>
<td>5,986</td>
</tr>
<tr>
<td>M_0.8</td>
<td>0.8</td>
<td>10,600</td>
</tr>
<tr>
<td>M_0.6</td>
<td>0.6</td>
<td>23,707</td>
</tr>
<tr>
<td>M_0.4</td>
<td>0.4</td>
<td>36,990</td>
</tr>
<tr>
<td>M_0.35</td>
<td>0.35</td>
<td>42,057</td>
</tr>
<tr>
<td>M_0.3</td>
<td>0.3</td>
<td>44,990</td>
</tr>
<tr>
<td>M_0.25</td>
<td>0.25</td>
<td>48,224</td>
</tr>
</tbody>
</table>

The plan to determine a suitable mesh density is to conduct a mesh sensitivity analysis. This will include graphs of the following for each density proposed:

- Residuals vs iterations;
- Computational time vs mesh density;
- Computational time per iteration vs mesh density;
• Pressure distribution over the model and sting (wall1) vs surface along model and sting

The pressure distribution over the model and sting is chosen as these will be the values that will be transferred over to the structural domain during the FSI simulations. Therefore, testing for the convergence rate with this parameter is appropriate. The residuals plot can give an indication of how many iterations the simulation needs for that particular mesh to converge to a solution.

For comparative reasons, the computational time per iteration will be used to test for computational time. This is done because simulations do not contain the same number of nodes, and as a result the times should be normalised [17]. The normalised time is determined with:

\[ t_{\text{iteration}} = \frac{t_{10} - t_2}{8 \times n_{\text{nodes}}} \]

where:

• \( t_{\text{iteration}} \) = computational time per iteration;
• \( t_{10}, t_2 \) = the starting time for iterations 10 and 2 respectively;
• \( 8 \) = the number of iterations between iteration 10 and 2;
• \( n_{\text{nodes}} \) = the no. of nodes for that particular mesh.

These plots can give an overall impression of what to expect when conducting the CFD simulations while also concluding with a chosen mesh density to be used for the FSI procedure.

6.2.2 Initial and Boundary Conditions

The flow parameters and boundary conditions for the test case simulations are presented here. An Euler, steady state, RANS simulation with symmetry along the mid-plane will be conducted for these cases. The simulations will be performed on a machine with the following specifications:

• i7 - 4710H @ 2.50 GHz
• 16 GB RAM

The parameters defining the flow in the domain along with the global initial conditions are listed in Table 9. The gas constants pertaining to MM provided by Head et al. [1] are specified with a residual target of \( 1 \times 10^{-8} \) to be deemed an accurate solution. The shear stress transport model (SST) is chosen as the turbulence model. This turbulence model takes on the \( k-\omega \) model in the near wall regions for adverse pressure gradients where this formulation yields accurate results [17]. When moving away from the boundary walls the model is gradually transformed to the \( k-\varepsilon \) model which is
less sensitive to the free-stream conditions. As a result the SST model takes on the advantages of both the $k$-$\omega$ and $k$-$\varepsilon$ models.

Table 9 - Fluid Domain and Global Conditions

<table>
<thead>
<tr>
<th>General Options</th>
<th>Fluid Models</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Basic Settings</strong></td>
<td><strong>Heat Transfer Model</strong></td>
</tr>
<tr>
<td>- Location</td>
<td>Assembly</td>
</tr>
<tr>
<td>- Domain type</td>
<td>Fluid</td>
</tr>
<tr>
<td>- Fluid Model</td>
<td>PR Gas</td>
</tr>
<tr>
<td>- Physical Problem</td>
<td>Euler</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Domain Models</strong></td>
<td><strong>Domain Motion</strong></td>
</tr>
<tr>
<td>- Ref. Pressure</td>
<td>16 Bar</td>
</tr>
<tr>
<td>- Ref. Temp</td>
<td>530 K</td>
</tr>
<tr>
<td>- Ref. Density</td>
<td>633.77 kg/m$^3$</td>
</tr>
</tbody>
</table>

**Global Initial Conditions**

<table>
<thead>
<tr>
<th><strong>Freestream Conditions</strong></th>
<th><strong>Turb. Convective</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>- Mach</td>
<td>0.1</td>
</tr>
<tr>
<td>- Freestream P</td>
<td>15 Bar</td>
</tr>
<tr>
<td>- Freestream T</td>
<td>525.15 K</td>
</tr>
<tr>
<td>- Freestream density</td>
<td>1.2886 kg/m$^3$</td>
</tr>
<tr>
<td>- Reynolds No.</td>
<td>6e6</td>
</tr>
<tr>
<td><strong>Gas Constants</strong></td>
<td><strong>Convergence Criteria</strong></td>
</tr>
<tr>
<td>- Ratio of spec. heat</td>
<td>1.0186</td>
</tr>
<tr>
<td>- Specific gas constant</td>
<td>51.2045</td>
</tr>
<tr>
<td>- Acentric factor</td>
<td>0.419</td>
</tr>
</tbody>
</table>

**Solver Control**

<table>
<thead>
<tr>
<th><strong>Geometric Initial Conditions</strong></th>
<th><strong>Solver Control</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>- Ref. Temperature</td>
<td>528.86 K</td>
</tr>
</tbody>
</table>

The boundary conditions used for the test simulations are listed in Table 10. The MM inlet and outlet conditions reviewed in Section 3.1.3 are specified here.

Table 10 - Boundary Conditions

<table>
<thead>
<tr>
<th>Basic Settings</th>
<th>Boundary Details</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inlet</strong></td>
<td><strong>Boundary Details</strong></td>
</tr>
<tr>
<td>- Boundary Name</td>
<td>inflow</td>
</tr>
<tr>
<td>- Location on Mesh</td>
<td>INLET</td>
</tr>
<tr>
<td><strong>Symmetry</strong></td>
<td><strong>Boundary Details</strong></td>
</tr>
<tr>
<td>- Boundary Name</td>
<td>symmetry</td>
</tr>
<tr>
<td>- Location on Mesh</td>
<td>SYM</td>
</tr>
<tr>
<td><strong>Diamond Wall</strong></td>
<td><strong>Boundary Details</strong></td>
</tr>
<tr>
<td>- Boundary Name</td>
<td>wall1</td>
</tr>
<tr>
<td>- Location on Mesh</td>
<td>DIAMOND FRONT</td>
</tr>
<tr>
<td></td>
<td>DIAMOND BACK</td>
</tr>
<tr>
<td></td>
<td>DIAMOND STING</td>
</tr>
<tr>
<td><strong>Outlet</strong></td>
<td><strong>Boundary Details</strong></td>
</tr>
<tr>
<td>- Boundary Name</td>
<td>outflow</td>
</tr>
<tr>
<td>- Location on Mesh</td>
<td>OUT</td>
</tr>
<tr>
<td><strong>Top Nozzle Wall</strong></td>
<td><strong>Boundary Details</strong></td>
</tr>
<tr>
<td>- Boundary Name</td>
<td>wall2</td>
</tr>
<tr>
<td>- Location on Mesh</td>
<td>TOP</td>
</tr>
</tbody>
</table>
6.2.3 CFD Simulations

This section will present the results of the CFD simulations. Firstly, to determine the quality of the different meshes, the aspect ratio and skewness were visually plotted. These are included in Appendix D. It can be seen that as the mesh gets finer, the quality of both these criteria gets progressively better. An aspect ratio of one is mostly observed throughout the meshes with high value regions along the boundary walls. These are improved dramatically as density increases, as shown in Figure 27 which shows the aspect ratios of the coarsest and finest mesh for the leading edge of the diamond. Similarly, the skewness for the meshes remain close to zero throughout the nozzle with improvements along the wall boundaries as density increases.

![Aspect Ratio of mesh, Left: Finest mesh; Right: Coarsest mesh](image)

Figure 27 - Aspect Ratio of mesh, Left: Finest mesh; Right: Coarsest mesh

Figure 28 and Figure 29 shows the Mach number and pressure distribution of the coarsest and densest test meshes respectively. As the pressure ratio across the nozzle indicates, a high pressure enters the nozzle and a low pressure exits, creating the pressure differential that drives the flow through the nozzle. The CFD results of all test meshes are included in Appendix E. Initial subsonic flow enters the converging area with the onset of supersonic flow reached at the throat. Mach number increases as the area diverges until an oblique shock is induced at the onset of the tip of the model. A Mach number of 2.1 is induced at the tip of the model just before the oblique shock. It is very apparent that as mesh density increases, the visual contrast of the resulting shocks are more clearly defined. Resulting reflected shocks are then observed downstream of the model. To avoid bow shocks and normal shocks occurring at the onset of the model tip, the design of the model accounting for supersonic blockage will be discussed in the next section.
1.0 Spacing

0.25 Spacing

Figure 28 - Mach number between coarsest and finest mesh densities

1.0 Spacing

0.25 Spacing

Figure 29 - Pressure distribution between coarsest and finest mesh densities

6.2.4 Discussion of Simulations

As mentioned at the start of this section, a number of plots will be observed to test the accuracy, convergence rates and computational times of the resulting simulation results. The resulting plots have been written and conducted in Matlab and the documentation of the Matlab scripts included in Appendix F. The written code extracts history and pressure data from the solution file outputted
from the CFD solution and plots them in a simple manner. The first being the residuals plot over the number of iterations, shown in Figure 30.

![Residuals vs Iterations](image)

**Figure 30 - Residuals vs Iterations**

With a residual criteria of $1 \times 10^{-8}$, it is shown that the finer the mesh, the longer it takes for the solution to reach this residual limit, which is expected. A steady rate progression of the residuals are observed until a dramatic drop is experienced to reach the residual criteria, occurring in all the meshes. The next set of graphs shows that the total computational time and the computational time per iteration.

![Total Computational Time and Avg Computational Time per Iteration](image)

**Figure 31 - Left: Total Computational Time, Right: Avg Computational Time per Iteration**

The total computational time increases as density increases, which was also expected. However, the computational time per iteration shows a small plateau in the time it takes for one iteration from 0.35 spacing onwards. A prediction can be made that denser meshes will have a similar time per
iteration of around 0.12 seconds. The last plot to be observed is the pressure distribution along the model and sting shown in Figure 32 of the seven different test meshes.

![Pressure Distribution along surface of model and sting](Image)

This graph shows that as mesh density increases, the pressure distribution will converge to a curve as displayed. The behaviour of the curve reflects the pressure observed in the CFD results correctly along the model and sting. For the leading edge of the model (0mm < x < 13mm) the pressure undergoes a drastic decrease as the oblique shock is induced. Along the back edge of the model (13mm < x < 20mm) the pressure increases and stays at this rate until it further increases leading up to the location of the reflected shock. Downstream of this reflected shock (60mm < x < 113mm) the pressure gradually decreases to the output pressure. A close up of the reflected shock location, shown in Figure 33 is a good indication of the convergence behaviour.
An observation to note is the curve corresponding to 0.35 spacing lying in between 0.3 and 0.25 spacing. This is possibly an indication of the exact solution lying in this region with the minimum limit being the 0.4 spacing. It can be predicted that denser meshes will output a curve within this region. Therefore, it can be confidently stated that convergence of the pressure distribution has been reached at a minimum density of 0.4 spacing.

To sum up on the observations, the plots obtained concerning computational times and residual accuracy were expected. As the mesh gets finer (density increase) it takes more iterations for the residual threshold to be met and computational time increases. The inclusion of the average computational time per iteration was useful as it was observed that for meshes of 0.35 spacing and finer the average time remained somewhat constant and can be predicted to remain this way for even finer meshes. This tells us that on an average basis the computational time to solve one timestep won’t differ, possibly leading to the time convergence of the solution.

From the discussion of the pressure distribution plot in Figure 32 and Figure 33, the chosen mesh density to use for the subsequent FSI simulations will be the 0.35 spacing. This solution lies within the region where it is believed the exact solution lies based on the convergence behaviour observed while also being the least computationally expensive. The results from the average computational time support this as a time convergence also suggests this face spacing to be accurate. Even finer meshes will not be observed due to time constraints and limited hardware to effectively analyse these meshes.
6.3 Supersonic Blockage

6.3.1 Verification Case

A verification case investigated by Head [1] identifies the validity of the supersonic blockage theory. Figure 34 compares two cases over a Mach number range from 1 to 2.5. The dotted line represents an analytical approach where the stagnation pressure loss ratio over a normal shock, referred to in Equation (6), is used to calculate Equation (13), shown below.

\[
\frac{\Delta A}{A_{\text{core}}} = 1 - \frac{P_{02}}{P_{01}} \frac{A^*}{A_{\text{core}}}
\]  

Equation 13

The solid line is an iterative numerical approach explained by Grossman [7] to find the downstream stagnation pressure condition of the normal shock. The pressure, density and velocity is calculated after the shock, by employing the iterative approach and then finds the corresponding stagnation pressure. This is subsequently used for the calculation of the blockage ratio. Figure 34 calculates the ratio under ideal air conditions under the ideal gas law. The same cannot be done for the MM working fluid because the analytical expression would be different and in fact, does not exist.

![Figure 34 - Blockage ratio vs Mach for air conditions](image)

It can be seen that the curves plotted matched closely to each other, with deviations occurring for increasing Mach number. However, given the Mach number domain of interest, the results indicate that usage of the iterative procedure presented by Grossman [7] for calculation of thermodynamic properties after discontinuities for non-ideal vapours is valid.
6.3.2 Investigation and Results

Appendix G describes the Matlab file documentation to acquire the following results relating to supersonic blockage for the nozzle. To theoretically calculate for the ratio between the model frontal area and test section area, results from obtaining the nozzle profile with the Method of Characteristics were extracted. These included the x and y co-ordinates of the diverging nozzle profile and in addition, the velocity, Mach number, density and pressures along the midline of the nozzle. These were used as the inputs to calculate the stagnation pressure loss across the normal shock and area ratios needed for that particular Mach along the midline. This allowed calculation for the ratio that defines supersonic blockage.

Figure 35 shows the resulting graph plotting the ratio against midline Mach number. The top curve represents the theoretical limit for the sizing of the model frontal area while the bottom curve represents the 60% curve as recommended by Dayman [10].

The trendline shows a similar behaviour to what Scheuler and Czysz obtained however, a maximum was calculated at around a Mach number of 1.78. After this point, the ratio for maximum model area decreased in value. To explain this behaviour, Figure 36 shows the x-position plotted against the area ratio and stagnation pressure loss ratio respectively which form part of the expression to calculate for the blockage ratio. For points after 6mm the pressure loss ratio becomes constant, indicating that the flow field at the midline becomes uniform first then the entire flow field becomes uniform at the exit. This is the location where the nozzle profile becomes constant as seen from Figure 23, being the Mach plot from the Method of Characteristics. For points after 6mm, the area ratio however, varies and shows a less steeper curveline, also indicating the nozzle profile is
diverging less as it connects to the test section. Due to there being less influence in these ratios, this could explain the parabolic nature when compared to the results found in literature. Furthermore, it may also be important to note that the data that Scheuler and Czysz obtained were taken from the exit of the nozzle under ideal air conditions. No such literature has been found that theoretically or numerically evaluates the blocking effects in supersonic de Laval nozzles for real gases.

![Figure 36 - Plots of theoretical ratios against x position. Left: nozzle area ratio; Right: pressure loss ratio](image)

A diamond model will be used to configure for the model frontal area. Also, the sizing will be based off the theoretically obtained curve and won’t account for the 60% factor as stated by Dayman [10]. This is because the CFD simulations won’t be able to account for the experimental factors stated by [8] and [9]. The experimental results obtained by Schueler or Czysz won’t be reflected in the CFD simulation, hence if a computational validation check for the chosen design is conducted accounting for the 60% correction factor, results above the limit would still show supersonic flow established.

A Mach number of 2.1 at a specified angle of 20 degrees and chosen length of 25mm from the throat was induced during the sensitivity analysis, hence will be used as the basis for designing the model sizing. By observing the graph in Figure 35, this correlates to a maximum model sizing ratio of 0.3027. Under symmetric conditions, using this ratio along with a size of 16.73mm for the model test sectional area, the theoretical limit for the frontal area ΔA can be calculated.

\[
\frac{\Delta A/2}{16.73} = 0.3027 \\
\Delta A/2 = 5.064 \text{ mm} \\
\Delta A = 10.12 \text{ mm}
\]

Therefore, the theoretical maximum size for the full model frontal area that will permit establishment of supersonic flow in the nozzle is 10.12mm. This value is based on one-dimensional
normal shock theory assuming an entropy increase across the normal shock. To avoid blocking effects, a chosen design height of 9mm will be implemented as the model size of the diamond.

\[
\text{Maximum } \Delta A = 10.12\text{mm} \\
\text{Chosen } \Delta A = 9\text{mm}
\]

This is just one of many possible configurations that make it possible to start supersonic flow in the nozzle. Various conditions that allow the nozzle to induce oblique shocks from the model can be concluded by interpreting Equation (13). To start the supersonic flow process within the nozzle, by using as large a model as possible, the model may be first moved back to have a larger Mach at the tip, the pressure ratio could be increased by increasing the inlet pressure or the test section area may also be increased to have a larger nozzle area ratio [10].

### 6.3.3 CFD Blockage Verification

This section presents a validation check for the chosen model sizing of 9mm. To achieve this, a CFD analysis under viscous conditions using a Peng Robinson equation of state for the thermodynamic model was used. The concluded mesh density found in section 6.2 along with the same initial and boundary conditions was applied.

Figure 37 shows the result of using the designed 9mm model size where an oblique shock is successfully induced. However if a size above the calculated limit of 10.12mm is chosen, as seen in Figure 38, the nozzle is effectively blocked. A chosen length of 14mm is applied to the diamond model, and it seems that a normal shock occurs at the tip of the model. Hence, the limit that was calculated and imposed was verified and using the chosen model frontal size of 9mm is justified.

![Figure 37 - Blockage validation check - at 9mm](image-url)
Figure 38 - Blockage Validation check - at 14mm
7. Structural Model Setup

The methodology for the structural domain is now given in this section. Details on the chosen material and stress criteria model is outlined in Section 7.1. A description of the chosen design for the support system with justifications is given in Section 7.2. Finally, the test cases which will take the implemented design choices from both the fluid and structural domain is described in Section 7.3 to conduct the one-way FSI simulations.

7.1 Structural Analysis Criteria

7.1.1 Chosen Material

The material to be used for the model and support system will be of the AISI low carbon 1010 steel. This particular steel exhibits a sufficient yield strength and a low enough thermal coefficient expansion such that it may withstand high temperature and pressure loadings. Table 11 presents the relevant material properties of the chosen material [21]. The model will also be assumed to be solid and homogenous to simplify the FEA analysis. However, in reality, small tubes exist within the model for mounting of pressure gauges along the surface.

Table 11 - Material properties of 1010 steel

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/m³)</td>
<td>7700</td>
</tr>
<tr>
<td>Poisson's Ratio</td>
<td>0.3</td>
</tr>
<tr>
<td>Elastic Modulus (GPa)</td>
<td>210</td>
</tr>
<tr>
<td>Tensile Strength (MPa)</td>
<td>365</td>
</tr>
<tr>
<td>Yield Strength (MPa)</td>
<td>305</td>
</tr>
<tr>
<td>Thermal Expansion Coefficient (10⁻⁶/°C)</td>
<td>15</td>
</tr>
</tbody>
</table>

7.1.2 Stress Analysis

Several key notes regarding the stress analysis should be taken into account for the design of the model support system:

- To show that allowable stresses do not exceed the worst case loadings;
- To identify a set of stress criteria based on a set of justified assumptions.

The following procedure is a set of guidelines set forth by Langley Research Centre at NASA [22] and provides a set of criteria for the design of wind-tunnel model systems, of which its stress criteria can be applied to this project. A von Mises stress criteria will be used for the FEA solver in Nastran which
employs a conservative method that involves calculation of a single equivalent stress. This equivalent stress is then compared to the yield or ultimate tensile strength of the material. Principal stress directions will need to be identified for this method. The maximum allowable stresses are given in the table below.

Table 12 - Von Mises Stress Criteria

<table>
<thead>
<tr>
<th>Calculated Value</th>
<th>Yield Criteria</th>
<th>Ultimate Tensile Strength Criteria</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma_e$</td>
<td>$S_y$</td>
<td>$\frac{2}{3} S_u$</td>
</tr>
</tbody>
</table>

For a tri-axial stress state involving normal stresses in the principal 1, 2 and 3 directions the equivalent stress is calculated with:

$$\sigma_e = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_1 - \sigma_3)^2 + (\sigma_2 - \sigma_3)^2}{2}}$$

(36)

7.2 Model and Support System Layout

To simulate the one-way FSI simulations, a diamond model will be chosen. Out of various model types between the diamond, wedge or a needle, the diamond would be the model to exhibit the most stress onto the support system. This is due to the diamond having the largest weight, thus having the largest moment arm for the stress and deflection to take place. Hence choosing this model would be the most conservative to test for the structural integrity of the support system.

For the design of the support system, three possible configurations were conceived. Out of these three, a simple sting a support layout fixed to the top of the testing section shown in Figure 39 is employed. The main advantages of this design include little hindrance on the flow field to avoid flow detachment. To withstand the flow from a structural standpoint while also maintaining the feature of low hindrance, a laterally thick support system is used. As this system is constrained at the base of the support, it is expected that stresses occurring from the flow will translate onto the base of this support, hence this thick support will be able to distribute the stress more evenly across its length. Refer to Appendix H for dimensions of the nozzle and the model support system.

The two other conceived systems can be seen in Appendix I. Circular tubing and curved lengths have been used for the design of the support for these iterations. The main disadvantages with these designs are that it may hinder the downstream flow field results. While these supports are designed
with the mindset to structurally withstand the oncoming flow, turbulent eddy effects and separation may occur behind these supports which is undesirable for the flow.

7.3 FSI Simulation Cases

To test and analyse the structural stability of the proposed model support system, two individual test cases will be simulated using the conceived coupling tool written in Matlab along with the configuration chosen for the model itself. The first test case is an arrangement where the model is completely aligned with the flow. In this situation, the oncoming flow is applied head-on to the diamond model and support system. The second case is to investigate how the support behaves under the pre-conceived notion of it being deflected due to a physical mounting error or initial perturbation. These are explained in more detail in Section 7.3.1 and Section 7.3.2 respectively.

7.3.1 Aligned Model

CFD simulations of the de Laval nozzle under the aligned test case were conducted with the stated initial and boundary conditions from section 3.1.3 and with the CFD configuration presented in section 6.2.2. Viscous conditions are accounted for using the Navier stokes model in SU2. The flow domain used is shown in Figure 40.

Pressure results obtained from the simulation along the front surface of the diamond model was then extracted and interpolated to represent the pressure values in the structural domain. These are
applied as the loadings along the same boundary interfacing for the structural domain shown in Figure 41.

Figure 40 - Fluid Domain (Nozzle) of aligned case

The results obtained from the FEA solver represent the stresses and deflections the support system undergoes when it is perfectly aligned to the oncoming flow during steady state operation of the nozzle. These results allow us to analyse how well the designed support system structurally reacts to the flow field and decide whether further iterations to the design need to be done.

Figure 41 - Structural Domain (Model and support system) of aligned case

7.3.2 Deflected Model

The second test case where the one-way FSI simulation is conducted is for a model and sting that is already deflected. This case is chosen to investigate the magnitude of the resulting deflections if the model was off-balanced or already pitched. To model the initial deflection, the tip of the model has been rotated clockwise by 2 degrees and the model translated down by 3mm.

As the nozzle is small in size and problems due to supersonic blockage dictate that the model and support also be kept small, deflected cases may arise due to the loading generated from the flow during a transient start-up process.
For this instance, as the nozzle will no longer be symmetric under a pitched and translated model, a CFD simulation over the whole nozzle will be conducted with the domain boundaries shown in Figure 42. The results along the surfaces of the front, back and sting of the top and bottom surface as labeled in Figure 43 are extracted and interpolated, and applied at the same boundary interfacing on the structural domain of Figure 43. In addition to investigating the influence of pressure loading, temperature results from the CFD simulation will also be applied to the structural domain. This provides a more thorough understanding of how the model responds to the specified inlet conditions of non-ideal vapours under steady state flow for a deflected anomaly.
8. FSI Simulation Results

8.1 Validity of Test Cases

To allow a 2D one-way FSI simulation to take place, certain assumptions needed to be stated and investigated to ensure the validity of the test cases. Simulating flow downstream of the model support itself wasn’t achievable due to the nature of the geometry. Therefore, the outlet of the fluid domain represented the edge of the model support system, meaning that pressure and temperature results along this edge could not be extracted.

An assumption can be made that the results along the frontal edge of the support system can be considered negligible such that it won’t significantly impact the structural results of the conceived support system. To investigate this assumption, a 3D simulation was conducted involving the aligned nozzle test case. As can be seen in Figure 44, the results revealed a low static pressure along the frontal edge surface. As such, ignoring this surface and assuming negligibility towards the structural integrity of the whole support system was valid.

8.2 Aligned Model

8.2.1 CFD Simulations

Figure 45 shows the CFD results of Mach number and pressure distribution. As the Mach distribution shows, initial subsonic flow enters the nozzle with the flow transitioning to sonic at the throat. At the leading edge of the diamond there is an oblique shock on each of the sides. Expansion waves
result behind the model and then the flow leaves the trailing edge through another shock. Interactions between the initial oblique shock and expansion take place in the far field flow where the oblique shock is reflected off the nozzle profile which subsequently cuts into the expansion fan. This does not generally have a significant effect on the surface pressure as seen in the pressure distribution result. Therefore, by observing this pressure distribution, the effect of the surface pressure along the back and sting seem to be small relative to the effect of the oblique shock on the front edge of the diamond. Reflected shocks are seen downstream of the sting and it is when this shock hits the sting as seen in the pressure distribution of Figure 45 that there is an increase in pressure.

A total of 33,714 nodes was used to obtain these solutions. The chosen model frontal sizing of 9mm was used and the Navier Stokes model was employed which took into account viscous effects to capture the boundary layers in the near field flow. Other initial and boundary conditions were kept the same from the mesh sensitivity analysis. A Mach number of 2.12 is at the tip is observed from this solution.

![Mach Number](image1)

![Static Pressure](image2)

Figure 45 - Mach number and static pressure results of aligned model
The shock wave resulting from the computation is $\beta = 45.0^\circ$. To compare the theoretical result, the iterative procedure of Equation (12) can be used. As it is well known that conditions across surfaces of discontinuities are governed by integral conservation equations, this expression accounts for the non-ideal case as the Rankine-Hugoniot relations are rendered obsolete. Consequently, the iteration procedure uses more complicated equations of state, given in [7] to obtain the thermodynamic change across a shock wave. Providing for inputs for upstream conditions such as $\theta$, $V_1$, $P_1$ and $\rho_1$, properties after the shock may be calculated. The resulting $\beta$ calculated is $45.7^\circ$, which means there is a 1.55% deviation in the solution between integral based numerical results to the CFD result. These are summarised in Table 13.

Table 13 - Summary of oblique shock wave angle results

<table>
<thead>
<tr>
<th>Theoretically calculated $\beta$ ($^\circ$)</th>
<th>Computational $\beta$ ($^\circ$)</th>
<th>Percentage difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>45.7</td>
<td>45.0</td>
<td>1.55</td>
</tr>
</tbody>
</table>

### 8.2.2 Interpolation of CFD Results

The pressure results along the front of the diamond model from the CFD were then extracted and interpolated with the RBF method using the conceived tool written in Matlab. Figure 47 shows the interpolated pressure compared to the CFD pressures obtained. The blue line represents the CFD result and the red represents the results after undergoing the interpolation. Notice there are less red dots than blue dots and this simply shows the number of nodes of each respective domain, meaning less nodes at the structural domain.
The increase in pressure at the start is a result of oblique shock with the max pressure occurring just after the tip at 300kPa. Further downstream of the pressure gradually decreases until it hits the expansion region where a large drop-off represents the back of the model. By observing the graph, the RBF method seemed to visually follow the pressure distribution with less nodes. Quantitatively, the RMS error is calculated to show how close the interpolated structural values were to the original fluid values. A value of 0.107 was obtained for this particular case.

Table 14 - RMS Errors calculated for aligned case

<table>
<thead>
<tr>
<th>Location</th>
<th>RMS Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diamond front</td>
<td>0.107</td>
</tr>
</tbody>
</table>

8.2.3 FEA Structural Results

The interpolated pressure values were applied along the same boundary surface of the structural domain in Nastran for the FEA to take place. Refer to Appendix J for the setup of the FEA. Table 15 summarises the results obtained and Figure 48 shows the visual results obtained.

Table 15 - Structural results for aligned case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max Von Mises Stress (Mpa)</td>
<td>0.564</td>
</tr>
<tr>
<td>Max displacement magnitude (mm)</td>
<td>$6.074 \times 10^{-4}$</td>
</tr>
</tbody>
</table>
A max stress of 0.564 MPa using the von Mises criteria was computed at the base of the support system. This is as expected as it is only at this point where the system is constrained, hence the design for a laterally wide support with a thin thickness was chosen. This max stress is insignificant, meaning that during a steady state operation using this support system, the stressing is safe from yielding with a safety factor of 540.

To determine if the support system would alter the fluid flow from its intended path the displacements of the system can be observed. A max value of $6.074 \times 10^{-4}$ mm at the tip of the model results and is also insignificant that it would not affect the flow field. From these results obtained, it can be stated that the influence of pressure from the flow field won't have any significant impact towards the structural integrity of the chosen support system during steady state operation of the nozzle.

Figure 48 - Structural Results of aligned model
8.3 Deflected Model

8.3.1 CFD Simulations

For the case of the deflected model, Figure 49 shows the results of the CFD simulations. As the model is no longer symmetric, a simulation involving the full nozzle had to be used. Similar flow characteristics are observed, although due to the model being deflected, this essentially applies an angle of attack for the model. As a result, the oblique shocks at the leading edge of the diamond are not the same strength, with Figure 50 showing a close-up of this location. The results along the bottom surface have a higher temperature and pressure distribution due to the flow hitting the bottom surface at a higher intensity caused by the deflection upwards. This inequality along the top and bottom surface will lead to an imbalance in the forces imposed onto the structural domain, hence why values will be extracted from all surfaces of the model and sting.

A total of 45,655 nodes was used with the same initial and boundary conditions.
Figure 49 - Mach number, pressure and temperature of deflected case

Figure 50 - Temperature and pressure at tip of diamond
### 8.3.2 Interpolation of CFD Results

The interpolation process is applied across all surfaces. Table 16 shows the visual results where interpolated values are graphed against the obtained CFD values. Temperature plots shown in Table 17 were also interpolated. The first three sets of the tables show the front, back and sting respectively while the last set of graphs combine these results to show the full pressure distribution. The left column shows the results along the top surface while the results of the right column show the bottom surface.

Table 16 - Pressure comparison between top and bottom surface of model and support system

- **Pressure of Top Surface**
- **Pressure of Bottom Surface**

![Graphs showing pressure comparison](image-url)
Comparing the full pressure distributions between the top and bottom surface, it is clearly evident that there is a higher intensity of values along the bottom surface. A high region of pressure is noted at the start which is the front of the model. A high drop-off occurs where expansion takes place along the back of the model. A constant pressure remains along the sting until an increase occurs, which is where the reflected shock hits the wall of the sting. This behaviour is also similar when observing the temperature results.
Table 17 - Temperature comparison between top and bottom surface of model and support system
The RMS errors are summarised in Table 18 for the pressure interpolation and Table 19 for temperature. Calculated values for the front and sting of pressure and temperature remained below 0.5, indicating that the interpolated values followed the trend close to the CFD results. However, slightly higher values of the back were calculated with a high error of 0.8 occurring for the back along the bottom surface of the pressure. This is also visually evident as can be seen in Table 16, the interpolated results seem to almost oscillate about the CFD result. Reasons for this has not been investigated, but general reasons could be the complexity of the curve and a low number of nodes along the back surface.

Table 18 - RMS Error for pressure interpolation

<table>
<thead>
<tr>
<th>Top Surface</th>
<th>RMS Error</th>
<th>Bottom Surface</th>
<th>RMS Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Front</td>
<td>0.10204</td>
<td>Front</td>
<td>0.1553</td>
</tr>
<tr>
<td>Back</td>
<td>0.73088</td>
<td>Back</td>
<td>0.82739</td>
</tr>
<tr>
<td>Sting</td>
<td>0.0669</td>
<td>Sting</td>
<td>0.10746</td>
</tr>
</tbody>
</table>

Table 19 - RMS Error for temperature interpolation

<table>
<thead>
<tr>
<th>Top Surface</th>
<th>RMS Error</th>
<th>Bottom Surface</th>
<th>RMS Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>Front</td>
<td>0.5427</td>
<td>Front</td>
<td>0.4232</td>
</tr>
<tr>
<td>Back</td>
<td>0.6012</td>
<td>Back</td>
<td>0.6101</td>
</tr>
<tr>
<td>Sting</td>
<td>0.1073</td>
<td>Sting</td>
<td>0.1321</td>
</tr>
</tbody>
</table>
8.3.3 FEA Structural Results

Two sets of results were obtained for the structural analysis concerning the deflected model. The first set shown in Figure 51 looked at how the pressure affected the model support system. A maximum stress of 31.1MPa was observed, located at the end of the support. Realistically, the maximum stress would be located at the base of the support. The maximum displacement recorded was 0.23 deflected further upwards at the tip of the model, implying that a further 0.23mm deflection to be added onto the already deflected model.

To further investigate the effect of the deflection, temperature loadings were applied to the structural boundary. These results are shown in Figure 52. The max stress recorded for this case was 187MPa, resulting in a 501% increase in stress. The structure is still safe from yielding however, with a calculated safety factor of 1.63.
The max displacement however, showed a small change of a 7% increase from the result of just the pressure influence. This can be due to the chosen material to have a low thermal expansion coefficient among the low carbon metals.

Table 20 below summarises the FEA structural results and compares the values obtained between the influence of pressure with the addition of temperature.
Table 20 - Structural results for deflected case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Result (Pressure only)</th>
<th>Result (Pressure and Temperature)</th>
<th>Percentage Increase</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max Von Mises Stress (MPa)</td>
<td>31.1</td>
<td>187</td>
<td>501.28%</td>
</tr>
<tr>
<td>Max Displacement Magnitude (mm)</td>
<td>0.23</td>
<td>0.247</td>
<td>7.127%</td>
</tr>
<tr>
<td>No. of nodes</td>
<td>1,423</td>
<td>1,423</td>
<td></td>
</tr>
</tbody>
</table>

The reason behind conducting a case without temperature was to identify how significant of a parameter temperature is when simulating for real gases. The usage of real gases dictate high temperature flow, which must be accounted for to get a full picture of the structural results. In ideal ambient conditions, temperature would be otherwise neglected but it was found in this case, that including the parameter of temperature was indeed significant.

The stress results obtained for the deflected case are safe from yielding, however the deflection of 0.247mm of the tip of the model observed may change the results of the flow field. As steady state operation continues, this deflection, relative to the sizing of the model can be considered significant enough to start unwanted oscillations or instability in the model. This can only be investigated further by conducting the full 2-way FSI procedure.

9. Conclusions

The aim of this project was to assist with the development of the supersonic nozzle test section of the ORCHID plant facility currently under development at TU Delft. An investigation into simulating computational test cases of the nozzle and using these results to see how it influences the structural model support system was outlined. A number of research questions was posed that asked how this could be done. As a result, four objectives were identified in an effort to answer these questions as the motivation behind this project. These were to:

- Generate 2D high fidelity, steady state, non-ideal compressible flow simulations;
- Design a suitable model and support system that remains minimally intrusive to the results of the flow field;
- Write a tool that couples results from the CFD analysis to an FEA solver;
- Use this tool to investigate structurally how the designed model support system will react to the oncoming flow field during steady state analysis.
After a thorough insight to the literature review, the first task was to conceive the tool that performed the one-way FSI simulation. The full FSI procedure which utilised a two-way procedure where the structural results would influence the fluid domain was not conducted due to its complex nature and time restraints. As a result of this, interpolation methods which transferred values from the fluid mesh to the structural mesh only was chosen. This was the radial basis interpolation method and further tests conducted showed the consistent approach to be more accurate than the conservative approach. This is the approach chosen to write in Matlab for the coupling tool that was able to transfer values between non-matching domains between the fluid and structural domains.

An investigation of the fluid domain was conducted next, where a sensitivity analysis was performed to find the ideal mesh density. This would be the density chosen to conduct the subsequent CFD simulations to investigate blockage and for the FSI analysis. This mesh density was chosen with the idea to balance computational time and accuracy and was concluded that a density of 0.35 spacing as defined in UMG2 would be chosen. An explanation into how the diverging section of the de Laval nozzle was conducted and was justified with results showing the Mach and pressure distribution. The third part included configuring the sizing of the diamond model such that it would avoid supersonic blockage. The blockage ratio was calculated specific to the de Laval nozzle and a frontal size of 9mm was chosen. Further CFD validation checks were conducted to justify this design decision.

The design of the structural domain was then taken into consideration. A detailed layout of the chosen design of the support system was described. AISI 1010 low carbon steel was chosen as the material and a diamond model employed as it would exhibit the highest stresses onto the support system. Using these aspects along with the design configuration of the fluid domain, the one-way FSI simulations were conducted using the Matlab tool that conducts the RBF interpolation method.

Two cases were described, one being an aligned case; this would investigate the stresses that occur if the model and support system was perfectly aligned and to see how the pressure loadings from the fluid flow influenced the structural integrity of this support system. The second case was the deflected model where an initial deflection of the model and sting was modeled. This investigates how the support system would react under an imbalance of the mounting or if the model underwent deflections due to unsteady loadings generated from a transient start-up. For the fluid domain simulations, a 2D unstructured, triangular mesh type was used with UMG2 being the chosen software tool to create these meshes. The in-house tool, SU2 was used to conduct the CFD simulations under a RANS code, fully turbulent and high fidelity simulations for a steady state analysis of the nozzle. A Peng Robinson equation of state was also used to model the
Concerning the structural domain, an assumption was made that the model and support system would be solid and homogenous with a linear elastic analysis for the FEA solver which used Nastran.

For the one-way FSI simulations, CFD results were generated, extracted and transferred to the structural domain where FEA results were then obtained. The CFD results obtained for real gases characterised the flow with an oblique shock at the leading edge, expansion waves occurring at the back of the model followed by another shock at the trailing edge. It was also investigated during supersonic blockage that many factors can play into the configuration of the model and the chosen design was just one of many that would allow for induced oblique shocks at the working conditions of the nozzle. If the chosen model was inserted further than the evaluated position or if the nozzle size was increased, than the nozzle could fail to establish oblique shocks in steady state.

For the aligned case, the maximum stress and deflection computed was insignificant and concludes that for the chosen design of the model and support system, the influence of flow field would have no considerable impact on the structural integrity nor with the results of the flow field.

For the deflected case, the influence of temperature in addition to the pressure was investigated. This is due to the nature of real gases that dictate high temperature inlet flows. Therefore, the affect of temperature must be included whereas for ideal gases, it would typically be neglected. The effect of temperature showed a 500% increase in stress and this is due to the material expansion at the location of the constraint. However, only a 7% increase in maximum magnitude of deflection resulted due to the chosen materials low thermal coefficient of expansion. It can be concluded that temperature effects are a serious consideration when working with dense organic vapours and should not be neglected for future analyses involving the nozzle test section.

Even though the maximum stresses observed was safe from yielding, the resulting deflection for the deflected case should be further considered. Given the small scale of the nozzle and support, if instability results, the deflections could lead to unwanted and unstable excitations. Also important to note is how significant the change in deflection may have on the fluid flow, which cannot be fully investigated without conducting the full 2-way FSI procedure.

Overall, the objectives posed were achieved, with resulting simulations providing an initial investigation into the insight of how the structural support system behaves due to the oncoming flow of non-ideal compressible flow within the de Laval nozzle. These were:

- CFD Simulations involving non-ideal compressible flows generated
- Model configuration designed
Matlab tool to couple fluid and structural domain for a one-way FSI simulation conceived

Structural analysis of model support under steady state conditions analysed

From the deliverables obtained, the main conclusions are that the support system would not yield under the stress of the nozzle under steady state conditions. Also important to note is the level of accuracy when transferring values between domains. Low RMS error values were calculated, thus validating the choice of using the RBF method. Certain assumptions to come to this statement were that the pressure and temperature data along the frontal edge of the support system be deemed negligible. However, due to the deflections observed for a deflected case, which may occur due to a transient startup or if the model is mounted incorrectly, this may hinder the results of the flow field, and thus a 2-way FSI simulation would be necessary.

10. Recommendations and Future Advancements

To further the development of the nozzle test section of the ORCHID, specifically regarding the support system, some recommendations are proposed that can advance the work begun with this project. These are summarised in Table 21

Table 21 - Summary of proposed future recommendations

<table>
<thead>
<tr>
<th>Recommendation</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Conceive an automation tool to conduct the two-way FSI Procedure</td>
<td>Note that the development team at SU2 is currently developing a fully integrated fluid-structure interaction branch. Further assistance regarding this field for implementation into SU2 could be done.</td>
</tr>
<tr>
<td>Transient Analysis (to analyse excitation of model due to unstable pressure loading during start-up)</td>
<td>This aspect of the model support system could not be analysed due to the one-way FSI procedure set in place. To conduct a transient analysis, the two-way procedure is used.</td>
</tr>
<tr>
<td>3D RANS Simulations</td>
<td>Turbulence effects such as vortices can be captured more accurately by conducting a 3D simulation. Data along all surfaces could also be used to investigate structural behaviour of the support system.</td>
</tr>
<tr>
<td>Experimental procedures regarding real gas behaviour</td>
<td>To validate theoretical calculations and CFD results, experimental procedures involving real gases in nozzles could be conducted as a future project topic.</td>
</tr>
<tr>
<td>Additional mechanical design</td>
<td>As the basis of the results obtained in this project assume a static support system, if the system needs to be set up such that a mechanical cart and mounting system is preferred, additional mechanical design work can be completed.</td>
</tr>
</tbody>
</table>
11. References


12. Appendix

12.1 Appendix A - UMG2 Syntax

For the generation of 2D unstructured triangular meshes, Table 22 labels the configuration files needed and their function. Four commands are carried out in subsequent order in a windows command console that performs the steps defined by Table 6 of section 4.1. This is summarised in Table 23.

Table 22 - Configuration setup for UMG2 to generate unstructured meshes

<table>
<thead>
<tr>
<th>Configuration File</th>
<th>Description of Contents</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometry</td>
<td>- sets the number of surfaces or edges</td>
</tr>
<tr>
<td></td>
<td>- geometrically defines and labels the domain boundaries</td>
</tr>
<tr>
<td>Options</td>
<td>- an options summary for overall settings</td>
</tr>
<tr>
<td>Topology</td>
<td>- defines the zones within the domain</td>
</tr>
<tr>
<td></td>
<td>- labels the fluid domain of the geometry (inlet, outlet, symmetry, etc)</td>
</tr>
<tr>
<td>SpacingControl</td>
<td>- sets the sizing of each boundary condition</td>
</tr>
<tr>
<td></td>
<td>- sets the initial boundary layer size</td>
</tr>
<tr>
<td></td>
<td>- controls the settings and inclusion of boundary layers</td>
</tr>
</tbody>
</table>

Table 23 - Commands needed to generate unstructured meshes in UMG2

<table>
<thead>
<tr>
<th>Step</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>..\MCURVE.exe</td>
<td>The definition of the domain boundaries and sizing of the elements are created with this command.</td>
</tr>
<tr>
<td>2</td>
<td>..\BGRID.exe</td>
<td>This command optimises the element shapes as described in Table 6</td>
</tr>
<tr>
<td>3</td>
<td>..\UMG2D.exe</td>
<td>This command generates the mesh in .su2 format.</td>
</tr>
<tr>
<td>4</td>
<td>..\HYB2D.exe</td>
<td>If boundary layers need to be added to the mesh to account for viscous effects, this command applies them to the generated mesh.</td>
</tr>
</tbody>
</table>

12.2 Appendix B - SU2 Syntax

SU2 requires several input files as defined by the user and produces several output files as a result. The configuration file is a text file that contains user defined global and domain parameters carrying the file extension .cfg. A native mesh format (.su2) is also needed for the input which contains the discretisation of a physical domain into specified elements. This file is also readable in ASCII format to extract information such as number and location of elements and nodes. The mesh generator UMG2 has been written to output the mesh in this native .su2 format.
Regarding the execution, installation of a command window such as Cygwin can be used to carry out the syntax. To utilise the SU2 parallel suite where simulations can be executed on multiple CPU cores the following command is entered after locating the relevant directory:

\[ \texttt{mpirun} -n 8 \texttt{SU2.CFD.exe 'configfile'.cfg} \]

Where 'configfile' is the name of the configuration file. After simulation to obtain the flow solution the following command is then entered:

\[ \texttt{SU2_SOL.exe 'configfile'.cfg} \]

These two commands will be the main executables for carrying out the CFD computations described by Palacios et al [23] which also contains a user manual regarding compilation instructions, template configuration examples and other advanced features not used for this project. For the post processing of results, SU2 outputs a number of solution files and are summarised below in Table 24.

### Table 24 - Collection of outputted solution files from SU2 for post processing

<table>
<thead>
<tr>
<th>File Name</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>flow.dat</td>
<td>Full volume flow solution (can be viewed with tecplot or ParaView depending on preset parameters)</td>
</tr>
<tr>
<td>surface_flow.dat</td>
<td>Flow solution along a surface of interest (specified in the .cfg file)</td>
</tr>
<tr>
<td>surface_flow.csv</td>
<td>Comma separated flow solution along a surface of interest (specified in the .cfg file)</td>
</tr>
<tr>
<td>history.dat</td>
<td>File containing the convergence and time history information</td>
</tr>
<tr>
<td>restart_flow.dat</td>
<td>A restart file for restarting simulations if specified in the .cfg file</td>
</tr>
</tbody>
</table>

#### 12.3 Appendix C - One-Way FSI Matlab Script Documentation

**Input_file.f.m**

- \[ function[f\_dia\_top, f\_dia\_bot, f\_dia\_top\_no, f\_dia\_bot\_no ] = Input\_file\_f(fluid\_P) \]
- Reads nodal co-ordinates and counts the number of nodes along the model and support system of the fluid mesh.

<table>
<thead>
<tr>
<th>Input</th>
<th>Variable Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>In</td>
<td>fluid_P</td>
<td>fluid flow solution (.csv) file</td>
</tr>
<tr>
<td>Out</td>
<td>f_dia_top</td>
<td>Nodal co-ordinates along top surface</td>
</tr>
<tr>
<td>Out</td>
<td>f_dia_bot</td>
<td>Nodal co-ordinates along bottom surface</td>
</tr>
<tr>
<td>Out</td>
<td>f_dia_top_no</td>
<td>Number of nodes along the top surface</td>
</tr>
<tr>
<td>Out</td>
<td>f_dia_bot_no</td>
<td>Number of nodes along the bottom surface</td>
</tr>
</tbody>
</table>
Input_file_s.m

- `function [s_dia_top, s_dia_bot, s_dia_top_no, s_dia_bot_no ] = Input_file_s(struct)`
- Reads nodal co-ordinates and counts the number of nodes along the model and support system of the structural mesh

<table>
<thead>
<tr>
<th>Input</th>
<th>Variable Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>In</td>
<td><code>struc</code></td>
<td>Structural mesh in (.dat) file format</td>
</tr>
<tr>
<td>Out</td>
<td><code>s_dia_top</code></td>
<td>Nodal co-ordinates along top surface</td>
</tr>
<tr>
<td>Out</td>
<td><code>s_dia_bot</code></td>
<td>Nodal co-ordinates along bottom surface</td>
</tr>
<tr>
<td>Out</td>
<td><code>s_dia_top_no</code></td>
<td>Number of nodes along the top surface</td>
</tr>
<tr>
<td>Out</td>
<td><code>s_dia_bot_no</code></td>
<td>Number of nodes along the bottom surface</td>
</tr>
</tbody>
</table>

Extract_Pressure.m

- `function [dia_top_Fp, dia_bot_Fp, dia_top_no_Fp, dia_bot_no_Fp ] = Extract_pressure(fluid_P)`
- Reads nodal pressure values from CFD solver (from surface of interest) and compiles into matrix form

<table>
<thead>
<tr>
<th>Input</th>
<th>Variable Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>In</td>
<td><code>fluid_P</code></td>
<td>Fluid flow solution (.csv) file</td>
</tr>
<tr>
<td>Out</td>
<td><code>dia_top_Fp</code></td>
<td>Static pressure results along top surface</td>
</tr>
<tr>
<td>Out</td>
<td><code>dia_bot_Fp</code></td>
<td>Static pressure results along bottom surface</td>
</tr>
<tr>
<td>Out</td>
<td><code>dia_top_no_Fp</code></td>
<td>Number of discrete static pressure values along top surface</td>
</tr>
<tr>
<td>Out</td>
<td><code>dia_bot_no_Fp</code></td>
<td>Number of discrete static pressure values along bottom surface</td>
</tr>
</tbody>
</table>

Extract_Temp.m

- `function [t_dia_top, t_dia_bot, t_dia_top_no, t_dia_bot_no ] = Extract_temp(fluid_P)`
- Reads nodal temperature values from CFD solver (from surface of interest) and compiles into matrix form

<table>
<thead>
<tr>
<th>Input</th>
<th>Variable Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>In</td>
<td><code>fluid_P</code></td>
<td>Fluid flow solution (.csv) file</td>
</tr>
<tr>
<td>Out</td>
<td><code>t_dia_top</code></td>
<td>Static temperature results along top surface</td>
</tr>
<tr>
<td>Out</td>
<td><code>t_dia_bot</code></td>
<td>Static temperature results along bottom surface</td>
</tr>
<tr>
<td>Out</td>
<td><code>t_dia_top_no</code></td>
<td>Number of discrete static temperature values along top surface</td>
</tr>
</tbody>
</table>
### Out

| t_dia_top_no | Number of discrete static temperature values along bottom surface |

---

**F2S.m**

- \( \text{function} \ [ \text{Struc}_P, \text{err}_Ps \ ] = \text{F2S}( \text{F}_\text{no_nodes}, \text{S}_\text{no_nodes}, \text{Fp}_\text{nodes}, \text{S}_\text{nodes}, \text{Fp}_\text{values} ) \)
- Runs the consistent radial basis interpolation method to transfer values from fluid domain to structural domain

<table>
<thead>
<tr>
<th>Input</th>
<th>Variable Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>In</td>
<td>F_no_nodes</td>
<td>Number of nodes for the fluid mesh</td>
</tr>
<tr>
<td>In</td>
<td>S_no_nodes</td>
<td>Number of nodes for the structural mesh</td>
</tr>
<tr>
<td>In</td>
<td>Fp_nodes</td>
<td>Nodal co-ordinates of the fluid mesh</td>
</tr>
<tr>
<td>In</td>
<td>S_nodes</td>
<td>Nodal co-ordinates of the structural mesh</td>
</tr>
<tr>
<td>In</td>
<td>Fp_values</td>
<td>Nodal fluid values obtained from CFD simulation</td>
</tr>
<tr>
<td>Out</td>
<td>Struc_P</td>
<td>Nodal structural values from performing RBF method</td>
</tr>
<tr>
<td>Out</td>
<td>err_Ps</td>
<td>RMS Error of structural values to obtained fluid values</td>
</tr>
</tbody>
</table>

**Master.m**

Couples the fluid results to the structural results by utilising the functions written.

- Interpolation plots
- RMS Errors for RBF procedure
- Input (.csv) files for FEA solver
- CFD Solution file
- Fluid Mesh
- Structural Mesh
- \( \text{Input\_file\_f.m} \)
- \( \text{Input\_file\_s.m} \)
- \( \text{Extract\_Pressure.m} \)
- \( \text{Extract\_Temp.m} \)
- \( \text{F2S.m} \)
12.4 Appendix D - Mesh Quality results for Sensitivity Analysis

1.0 Spacing

0.8 Spacing

0.6 Spacing
0.4 Spacing

0.35 Spacing

0.3 Spacing
Figure 53 - Aspect Ratio of test meshes for Sensitivity Analysis

0.25 Spacing

1.0 Spacing

0.8 Spacing
12.5 Appendix E - CFD Simulations for Sensitivity Analysis
Figure 55 - Mach Number of CFD simulations conducted for sensitivity analysis
Figure 56 - Static pressures of CFD Simulations conducted for sensitivity analysis
12.6 Appendix F - Mesh Sensitivity Analysis Matlab Script Documentation

res_it.m

- \textit{function} \[ \text{Res} = \text{res}_\text{it}( \text{history} ) \]
- Returns the residuals, iteration and length of time and compiles in matrix form

<table>
<thead>
<tr>
<th>Input</th>
<th>Variable Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>In</td>
<td>History</td>
<td>Fluid history solution (.dat) file</td>
</tr>
<tr>
<td>Out</td>
<td>Res</td>
<td>Matrix values of residual values, iteration number and time of each iteration</td>
</tr>
</tbody>
</table>

extract_surface.m

- \textit{function} \[ \text{surf} = \text{res}_\text{it}( \text{surface} ) \]
- Returns the x-points along model and corresponding pressure values

<table>
<thead>
<tr>
<th>Input</th>
<th>Variable Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>In</td>
<td>History</td>
<td>Fluid flow solution (.csv) file</td>
</tr>
<tr>
<td>Out</td>
<td>Surf</td>
<td>Matrix values of x-points along model and corresponding pressure values</td>
</tr>
</tbody>
</table>

mesh_analysis.m

- Performs the mesh sensitivity analysis

<table>
<thead>
<tr>
<th>Plots of residuals</th>
<th>mesh_analysis.m</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plots of CFD Times</td>
<td></td>
</tr>
<tr>
<td>Plot of pressure distribution</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>res_it.m</th>
<th>CFD Solution file</th>
</tr>
</thead>
<tbody>
<tr>
<td>extract_surface.m</td>
<td></td>
</tr>
</tbody>
</table>
12.7 Appendix G - Supersonic Blockage and Profile Generation

Matlab Script Documentation

centrelinecord.m

- `function centrelinecord`
- Reads the co-ordinates of the centreline of the nozzle and corresponding flow parameters

<table>
<thead>
<tr>
<th>Input</th>
<th>Variable Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>In</td>
<td>RGND_Solution.dat</td>
<td>Solution file created from running the method of characteristics generated</td>
</tr>
<tr>
<td></td>
<td></td>
<td>from the fortran executable</td>
</tr>
<tr>
<td>Out</td>
<td>centrelinecord.txt</td>
<td>Text file containing flow parameters (velocity, Mach, density, pressure)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>along the midline of the nozzle</td>
</tr>
</tbody>
</table>

nozzlecord.m

- `function nozzlecord`
- Reads the co-ordinates of the diverging nozzle profile

<table>
<thead>
<tr>
<th>Input</th>
<th>Variable Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>In</td>
<td>RGND_Solution.dat</td>
<td>Solution file created from running the method of characteristics generated</td>
</tr>
<tr>
<td></td>
<td></td>
<td>from the fortran executable</td>
</tr>
<tr>
<td>Out</td>
<td>nozzlecord.txt</td>
<td>Text file containing the x and y co-ordinates of the diverging nozzle profile</td>
</tr>
</tbody>
</table>

blockage.m

- `function blockage`
- Performs the one-dimensional shock theory to obtain model frontal area to test section ratio

```
Graphs for supersonic blockage
```

```
centreline.txt
centreline.m
blockage.m
nozzlecord.txt
nozzlecord.m
```
**Console1.exe**

- Conducts the Method of Characteristics and outputs the diverging profile in Tecplot.
12.8 Appendix H - Model and Support System Dimensions
12.9 Appendix I - Layout of other conceived Model Support Systems

Figure 57 - Design using circular tubes for the support

Figure 58 - Design using curved support to reduce stress concentration
12.10 Appendix J - FEA Model Setup

12.10.1 Aligned Model

Figure 59 - Visual representation of applying interpolated structural pressures onto model domain

Figure 60 - Setup of aligned model, showing the mesh, loadings and constraints

12.10.2 Deflected Model

Figure 61 - Setup of deflected model, showing the mesh, loadings with temperature included and constraints
Figure 62 - Pressure distribution shown in Nastran, confirming pressure results applied from CFD simulation