Compressor CFD simulation method development

*A CFD study*

Johan Björk

Engineering Physics and Electrical Engineering, master's level

2018

Luleå University of Technology
Department of Computer Science, Electrical and Space Engineering
Compressor CFD simulation method development: A CFD study

Johan Björk
johbjr-2@student.ltu.se
Department of Engineering Sciences and Mathematics

Scania supervisor: Olle Bodin
Examiner LTU: Gunnar Hellström

June 26, 2018
Abstract

This master thesis project consisted of three parts that all were performed through CFD simulations with the purpose to develop Scania’s methods in the subject of CFD. All parts included simulations on Scania’s SC92T70 centrifugal compressor.

Part one consisted of performing a mesh study for the purpose of reliability, to investigate the convergence of different parameters by refining the boundary layer. The method used is an inflation option called First layer thickness. Five different meshes were generated where the Richardson extrapolation method was used to examine the parameters between the mesh refinements.

From the result from the examined parameters, an approximate relative error could be calculated to be less than 0.52 %, and a numerical uncertainty of less than 0.35 %, between Mesh3 and Mesh4. In addition to that, Mesh3 had a simulation time of one hour less than for Mesh4. These results motivated the use of mesh3 to be refined enough for further work in this thesis project. This mesh ended at 37,915,257 number of elements.

The second part consisted of performing steady state CFD simulations, to examine different parameters in order to find indications of the phenomena surge. Here, experimental data was used as reliance to perform CFD simulations on the compressor. Design points from experimental data was used, that ranged from low mass flow rates where surge arises, to high mass flow rates where another phenomena called choke occur. Except for the design points taken from experimental data, a few extra design points where included at low mass flow rates (in the region of surge). The goal was that the analysis of the different parameters would generate fluctuations on the result for the design points in surge region. Four different rotational speeds on the compressor were examined, 56k, 69k, 87k and 110k revolutions per minute. A total of 140 different parameters were examined, where 10 of these indicated on surge.

All of these parameters that indicated on surge where found in regions of vicinity to the compressor wheel, which are the regions subjected to the phenomena. The parameters indicating on surge where mass flow, pressure coefficient, static pressure and temperature. Indications where found at the wheel inlet, ported shroud, and wheel outlet interfaces. The indications were only found for the two lower rotational speeds of the compressor wheel. To capture the behaviour on higher rotational speeds, more design points in the region of surge are needed, or transient simulations.

Part three of the thesis project consisted of investigating the methodology of performing a Conjugate Heat Transfer model (CHT) with the CFD code CFX. This part has not been performed by Scania before, so a big part of the problem was to investigate if it actually was achievable. The goal was to use this model to calculate the heat transfer between fluid and solid parts, as well as between
the solid parts and the ambient. One question Scania wanted to answer was if the CHT model could generate aerodynamic performance that corresponds to Scania’s traditional adiabatic model, as well as to experimental data of the compressor.

In this part, both solid and fluid domains were included in the geometry to calculate heat transport, in contrast to the traditional adiabatic model that only uses the fluid domains. Because of that, a big part of the work consisted of defining all interfaces connecting together surfaces between all domains. This is needed to model heat transport between the domains. In the set up part in CFX, the CHT model differed a lot from the traditional adiabatic model in that way that the outer walls was not set up as adiabatic anymore. In the CHT model, instead heat transfer is allowed between the outer walls of the fluids and the solids.

From the result simulations, one could see that the CHT model was able to compute the heat transfer between fluids and solids. It also managed to export thermal data such as heat flux and wall heat transfer coefficient to be used for mechanical analysis, which is an important part in Scania’s work. From the analysis of aerodynamic performance, a conclusion was drawn that the CHT model was able to compute efficiency and pressure ratio that followed the behaviour of the traditional adiabatic model as well as experimental data. However, for lower mass flows, the CHT model started to underpredict which could be explained by the geometrical differences between the CHT and adiabatic model.

By analysis of temperature, one could see quantitative differences compared to the traditional adiabatic model. For other parameters (static and total pressure), there were no experimental data to be used for comparison. Because of that, an important part in future work of this CHT method development is to perform more experimental test for CFD data to be compared against. Another important part to compare the models is to have an identical geometry. Without an identical geometry, deviations in result will occur that depends on geometry.
Sammanfattning

Detta examensarbete har bestått i tre moment som alla har utförts genom CFD-bärökningar med syfte att utveckla Scaniás metoder inom CFD. Alla moment har omfattat beräkningar på Scaniás SC92T70 centrifugalkompressor.

Del ett bestod i att utföra en meshstudie för att i reliabilitetssyfte undersöka olika parametrars konvergens genom att förfinna meshens gränsskikt. Metoden som användes är en inflationstyp som heter First layer thickness. Fem olika meshförfiningar togs fram där olika parametrar undersöktes med hjälp av Richardssons extrapolation.

Från resultatet för de undersökta parametrarna kunde ett approximativt relativt fel beräknas till att vara mindre än 0.52 %, och en numerisk osäkerhet på mindre än 0.35 %, mellan Mesh3 och Mesh4. Mesh3 var dessutom en timme snabbare att simulera på. Med dessa resultat motiverades Mesh3 till att vara tillräckligt förfinad kring gränsskiktet för resterande arbeten inom detta examensarbete. Mesh3 slutade på 37,915,257 antal element.


Samtliga av dessa parametrar som indikerade på surge fanns i områden i närhet till kompressorrhjulet, vilket är den del av kompressorn som utsätts för detta fenomen. Mer specifikt kunde parametrar som massflöde, tryckkoefficient, statiskt tryck och temperatur ge utslag. Indikationerna återfanns på hjulutlopps-, ported shroud- och hjulinloppsgränssnittet. För de indikationer som fanns för fenomenet surge, så kunde det bara spåras för de två lägsta undersökta rotationshastigheterna. För att finna fenomenet för högre rotationshastigheter på kompressorrhjulet så krävs flera designpunkter i området kring surge, eller transenta simuleringar.

Del tre i detta examensarbete gick ut på att undersöka metodologin för att ställa upp en Conjugate Heat Transfer-modell (CHT) med hjälp av Ansys CFX. Detta har inte gjorts tidigare på Scania så stor del av problemet var att undersöka om det faktiskt gick att utföra en CHT-modell. Tanken var att använda denna modell för att beräkna värmetransporten mellan fluid- och soliddelar, men också mellan soliddelar och omgivning. En frågeställning som
Scania ville ha besvarat var om modellen kunde generera aerodynamisk data som motsvarar deras traditionella adiabatiska CFD-metod.

I detta moment användes både solider och fluider i geometrin för att beräkna värmetransport, till skillnad mot den traditionella adiabatiska modellen som endast använder fluiddelarna. På grund av det bestod en stor del av detta arbete i att definiera upp alla gränssnitt som kopplar samman olika ytor. Detta för att kunna modellera värmetransport mellan delarna. I själva modelleringen skiljde sig denna modell från Scanias traditionella modell på så sätt att ytterväggarna inte modelleras som adiabatiska längre, utan nu tillåter värmetransport.


Vid analys av temperatur gick det att urskilja stora kvalitativa skillnader jämfört med den traditionella metoden. För övriga parametrar (statiskt tryck och totaltryck) fanns ingen experimentell data att jämföra med. Därför krävs det för framtida metodutveckling av denna modell en mer omfattande experimentell analys, så CFD-data kan evalueras. En annan viktig del för att kunna jämföra resultat mellan modellerna är en identisk geometri. Utan det kan avvikelser förekomma som kan bero på de geometriska skillnaderna.
Acknowledgement

I would like to thank my examiner and supervisor Gunnar Hellström at the division of fluid and experimental mechanics at LTU for providing me with help during this thesis. His door has always been open both for this project but also during many of the courses I have taken during my years at the university. My supervisor Olle Bodin and co-worker Thomas Svensson at the department NMGD at Scania for taking up a big part of their time to assist me in my work. I also want to thank my family for supporting me and for always being there when I have needed it.
## Contents

1 Introduction 12  
1.1 Background .................................................. 12  
1.2 Problem formulation ........................................ 12  
1.3 Software ....................................................... 13  

2 Theory 15  
2.1 General Theory on centrifugal compressors ............... 15  
2.1.1 Limitations .................................................. 16  
2.1.2 Ported shroud ............................................. 17  
2.1.3 Governing equations ...................................... 18  
2.1.4 Reynolds-Average Navier-Stokes ....................... 19  
2.1.5 Eddy-Viscosity modeling ................................ 20  
2.1.6 SST Turbulence model .................................... 21  
2.1.7 Boundary layer ........................................... 22  
2.1.8 Performance ............................................... 23  
2.1.9 Aerodynamics ............................................. 26  
2.2 Mesh dependency study ...................................... 28  
2.3 Surge line prediction ....................................... 29  
2.4 Conjugate Heat Transfer .................................... 32  

3 Method 34  
3.1 Mesh Independency study ................................... 34  
3.1.1 Geometry ................................................... 34  
3.1.2 Meshing ..................................................... 34  
3.2 Surge line prediction ....................................... 42  
3.2.1 Boundary Conditions .................................... 48  
3.3 Conjugate Heat Transfer .................................... 50  

4 Results 54  
4.1 Mesh independency study ................................... 54  
4.2 Surge line prediction ....................................... 57  
4.2.1 Mass flow .................................................. 58  
4.2.2 Pressure recovery coefficient ......................... 60  
4.2.3 Static pressure ......................................... 61  
4.2.4 Temperature ............................................. 61  
4.3 Conjugate Heat Transfer .................................... 64  
4.3.1 Temperature fields ...................................... 64  
4.3.2 Efficiency .................................................. 66  
4.3.3 Pressure Ratio ........................................... 67  
4.3.4 Temperature ............................................. 68  
4.3.5 Static pressure ......................................... 69  
4.3.6 Total pressure ........................................... 70
## Analysis & Discussion

5.1 Mesh dependency study ............................ 72
5.2 Surge line prediction ............................... 72
  5.2.1 Future work .................................. 73
5.3 Conjugate Heat Transfer ......................... 73
  5.3.1 Future work .................................. 76
## List of Figures

1. CFX module and workflow in Ansys. ........................................ 13
2. The anatomy of a turbo. ...................................................... 16
3. Illustrative schematic of the centrifugal compressor. Adapted from Hill & Peterson [10]. ........................................ 17
4. Illustration of the ported shroud. .......................................... 18
5. Laminar and turbulent boundary layer ................................. 19
6. h-s schematic of the ideal and real compression process. ....... 23
7. Velocity triangles on compressor wheel. Adapted from Sundström [13] ................................................................. 27
8. Illustrative schematic of an idealized compressor wheel. ....... 28
9. An illustrative image of how a compressor map can look. ....... 31
10. Default tab of mesh. ........................................................... 34
11. Sizing tab of mesh. ............................................................ 35
12. Inflation tab of mesh. ........................................................ 35
13. RSM residuals for the standard mesh simulation. ............... 39
14. Illustration of the 3 design points included together with the design point taken from experimental data. ............... 43
15. Input and output parameters defined for use by the parametric pack license. ............................................................... 44
16. Location of inlet and outlet interface. ................................ 45
17. Inlet-to-wheel domain interface. .......................................... 46
18. Interfaces connecting wheel domain with adjacent domains. . 46
19. Diffuser-to-wheel interface. ................................................ 47
20. Adiabatic geometry consisting of 3 domains, inlet, wheel and diffuser /volute domain. .................................................... 50
21. CHT geometry consisting of 8 domains. ............................... 50
22. Geometrical differences between CHT and adiabatic model. ... 51
23. Separately meshes of fluid and solid domains. ....................... 52
24. Final mesh of the solids and fluids of the CHT geometry. ....... 52
25. Pressure ratio and efficiency plotted as a function of inflation layers. ................................................................. 54
26. Mass flow at outlet as a function of inflation layers. ............ 54
27. Mesh 3 in cross section. ..................................................... 56
28. The average and standard deviation of critical parameters ....... 56
29. Efficiency compressor maps. .............................................. 57
30. Pressure ratio compressor maps. ........................................ 58
31. Mass flow at wheel inlet interface ....................................... 59
32. Mass flow ratio, wheel inlet over compressor inlet. ............. 59
33. Mass flow ratio, ported shroud over compressor inlet. ......... 60
34. Pressure recovery coefficient. ............................................ 60
35. Static pressure at ported shroud and 1 cm upstream in ported shroud. ................................................................. 61
36. Mean temperature ratio on measure lines on wheel inlet interface. ................................................................. 62
37. Static temperature at wheel inlet interface. ......................... 62
38. Total temperature at wheel inlet interface. ........................... 63
<table>
<thead>
<tr>
<th>Page</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>39</td>
<td>Cross section plane of temperature distribution in solids.</td>
</tr>
<tr>
<td>40</td>
<td>Two views to illustrate the temperature distribution on the compressor wheel.</td>
</tr>
<tr>
<td>41</td>
<td>Comparison of temperature distribution, CHT model to the left, adiabatic model to the right.</td>
</tr>
<tr>
<td>42</td>
<td>Comparison of temperature distribution, CHT model to the left, adiabatic model to the right.</td>
</tr>
<tr>
<td>43</td>
<td>Efficiency comparison between CHT model, traditional adiabatic model and experimental data.</td>
</tr>
<tr>
<td>44</td>
<td>Pressure ratio comparison between CHT model, traditional adiabatic model and experimental data.</td>
</tr>
<tr>
<td>45</td>
<td>Difference in power between the CHT and adiabatic model.</td>
</tr>
<tr>
<td>46</td>
<td>Total temperature comparison.</td>
</tr>
<tr>
<td>47</td>
<td>Total temperature comparison.</td>
</tr>
<tr>
<td>48</td>
<td>Static pressure comparison.</td>
</tr>
<tr>
<td>49</td>
<td>Static pressure comparison.</td>
</tr>
<tr>
<td>50</td>
<td>Total pressure comparison.</td>
</tr>
<tr>
<td>51</td>
<td>Total pressure comparison.</td>
</tr>
</tbody>
</table>
Nomenclature

Table 1: Parameters used throughout the report.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Parameter</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time</td>
<td>$t$</td>
<td>$s$</td>
</tr>
<tr>
<td>Cartesian coordinates in $i$ direction</td>
<td>$x_i$</td>
<td>$m$</td>
</tr>
<tr>
<td>Velocity component in $x_i$ direction</td>
<td>$u_i$</td>
<td>$m/s$</td>
</tr>
<tr>
<td>Density</td>
<td>$\rho$</td>
<td>$kg/m^3$</td>
</tr>
<tr>
<td>Stress tensor</td>
<td>$\sigma$</td>
<td>$N/m^2$</td>
</tr>
<tr>
<td>External / body force</td>
<td>$F$</td>
<td>$N$</td>
</tr>
<tr>
<td>Heat flux</td>
<td>$q$</td>
<td>$W/m^2$</td>
</tr>
<tr>
<td>Total energy</td>
<td>$e_0$</td>
<td>$J/kg$</td>
</tr>
<tr>
<td>Inner energy</td>
<td>$e$</td>
<td>$J/kg$</td>
</tr>
<tr>
<td>Specific heat at constant pressure</td>
<td>$C_p$</td>
<td>$J/kg$</td>
</tr>
<tr>
<td>Specific heat at constant volume</td>
<td>$C_v$</td>
<td>$J/kg$</td>
</tr>
<tr>
<td>Enthalpy</td>
<td>$h$</td>
<td>$J$</td>
</tr>
<tr>
<td>Entropy</td>
<td>$s$</td>
<td>$J/kg/K$</td>
</tr>
<tr>
<td>Temperature</td>
<td>$T$</td>
<td>$K$</td>
</tr>
<tr>
<td>Dynamic viscosity</td>
<td>$\mu$</td>
<td>$Pa \cdot s$</td>
</tr>
<tr>
<td>Kinematic viscosity</td>
<td>$\nu$</td>
<td>$m^2/s$</td>
</tr>
<tr>
<td>Angular velocity</td>
<td>$\omega$</td>
<td>$rev/min$</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>$Re$</td>
<td>Unit less</td>
</tr>
<tr>
<td>Mass flow</td>
<td>$\dot{m}$</td>
<td>$kg/s$</td>
</tr>
<tr>
<td>Tangential blade velocity</td>
<td>$\vec{U}$</td>
<td>$m/s$</td>
</tr>
<tr>
<td>Work</td>
<td>$W$</td>
<td>$J$</td>
</tr>
<tr>
<td>Efficiency</td>
<td>$\eta$</td>
<td>[%]</td>
</tr>
<tr>
<td>Pressure</td>
<td>$p$</td>
<td>$Pa$</td>
</tr>
</tbody>
</table>

Table 2: Sub and superscripts.

<table>
<thead>
<tr>
<th>Subscript</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet of compressor</td>
<td>1</td>
</tr>
<tr>
<td>Outlet of compressor</td>
<td>2</td>
</tr>
<tr>
<td>Reference value</td>
<td>ref</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Superscript</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Total condition</td>
<td>0</td>
</tr>
<tr>
<td>Average</td>
<td>$\overline{\cdot}$</td>
</tr>
<tr>
<td>Favre average</td>
<td>$\tilde{\cdot}$</td>
</tr>
<tr>
<td>Fluctuation</td>
<td>$\cdot'$</td>
</tr>
</tbody>
</table>
1 Introduction

1.1 Background

Scania works continuously to develop internal combustion engines which can achieve low pollutant emissions and high efficiency. This requires designing components with high performance and low weight while maintaining design constraints and cost. Of high importance is also driver ergonomics, for example noise, and driving “feel”.

The capability to model compressor performance and characteristics is important in order to have a virtual gas exchange design loop. Having this loop in place enable Scania to save time and/or cut development cost by excluding some hardware acquisition from the design loop. Presently Scania does not have in place a fully robust simulation method for the compressor.

The background to this thesis is based on Scania’s demands to perform performance simulations that generates results with high accuracy and precision. Firstly to perform performance predictions that the department GT Power can use for their engine performance simulations. Secondly to supply the department TMF with data and boundary conditions for their calculations of thermomechanical fatigue.

To meet this demands, Scania needs to improve their understanding of how the performance of the turbo affects the engine. In the end, it is all about being an "educated customer", so Scania as the intermediate link can set high and righteous demands on their supplier of turbos. Therefore Scania needs to continuously improve their CFD methods.

1.2 Problem formulation

The problem formulations in general is to:

- Perform a mesh study by refining the boundary layer
- Compressor map surge line prediction using steady state simulations
- Conjugate heat transfer modeling

The first point is performed with the goal to refine the boundary layers in the mesh to investigate parameters, and find a converging behaviour of these parameters. This point is also performed to in terms of trade-off determine how a refined mesh will affect the computational time. Earlier work in this subject has been performed at Scania, but regarding the refinement of the volume. This mesh study is focusing on resolving the boundary layers.

The surge line prediction is performed to get a better understanding of the phenomena surge. To investigate if there are any parameters that can indicate
on the phenomena surge, by means of steady state simulations. Surge line prediction has been performed before at Scania using transient simulations. This investigation is performed purely with steady state simulations.

Conjugate heat transfer modeling is a modeling technique where the solids are included in the geometry. The aim of this type of modeling is to investigate the methodology of computing the heat transfer between the fluid and solid, but also between the solid and ambient using Ansys CFX. This has never been performed by Scania before, so the goal is both to investigate if it is achievable, and if so, will the aerodynamic performance of this method generate data close enough to Scania’s traditional adiabatic method.

This thesis project is performed at Scania NMGD in Södertälje. Because of the confidentiality in the work, all numerical plots will be normalized. This also means that all graphical vector field is presented without legends.

1.3 Software

Ansys Workbench 18.1 is the software used throughout this thesis project, with the CFD code CFX. In Figs. 1a and 1b below, the module used for CFD simulations is shown together with the work flow.

![CFX module in Ansys Workbench 18.1](image-a)

![Workflow in Ansys](image-b)

**Fig. 1:** CFX module and workflow in Ansys.

A geometry can be created by design modeler or spaceclaim. It can also be imported from an external CAD software used by i.e a designer. When the geometry is done, it is discretized using the mesh cell. In this step, a grid is put on the geometry to divide it into smaller cells, where every cell later will be solved for. After that, boundary conditions like pressure, temperature, rotational speeds, turbulence models etc, is included in the setup cell. When this is done, the problem can be solved for in the CFX-Solver. The final step is
to post process the result file from the solver. This is done in CFD-Post where either graphical vector fields for parameters can be used, or the data can be exported for numerical analysis in i.e Matlab.
2 Theory

2.1 General Theory on centrifugal compressors

Because of ever increasing demands to increase engine efficiency and decrease vehicle emissions, the heavy-duty truck engine manufacturers are continuously investigating ways to satisfy both customers and legislation using old and new technology. Turbocharging is certainly not a new technology but is essential in order to achieve high engine efficiency and the engine/turbo interaction is far from fully understood making turbocharging a highly active research topic for the heavy-duty manufacturers.

Turbochargers works by letting waste gas from the engine be lead into the turbine housing. The gas will enter radially and start a rotation of the turbine wheel. The gas then exit the turbine axially and is lead away to the exhaust system. The turbine wheel is connected to the compressor wheel via a shaft, letting the energy from the rotating turbine wheel be transferred to the compressor wheel. The rotation of the compressor wheel will help to suck ambient air into the compressor axially. The air entering the compressor will flow down to the compressor wheel, where the rotation of the wheel will compress the air, and radially send it towards the volute (\textit{outlet}). The now high pressurized gas has a increased density when passing through the air cooler and back into the engine. Modern direct-injection diesel engine combustion concepts are reliant on high air/fuel ratios in order to achieve high combustion efficiency. By using the otherwise wasted exhaust energy to achieve this, higher total engine efficiency can be realized for the turbocharging.

The anatomy of the turbocharger is presented in Fig. 2 below.
2.1.1 Limitations

The domains of the turbocharger that this study will cover is the adjacent parts of the centrifugal compressor, that includes the inlet, impeller wheel, diffuser and volute. See Fig. 3 below for a more detailed view of the considered parts of the centrifugal compressor. This means that the turbine and bearing housing is not considered in this study.

A centrifugal compressor consists of several parts. The inlet, where the air is passing through to the impeller (compressor wheel). The impeller consist of several blades that will rotate around the axis and convert- the axial velocities of the fluid from the inlet into radial velocity. The inlet of the impeller is called inducer, and the outlet of the impeller is the exducer. The air will be compressed in the wheel and with a radial velocity flow out of the impeller wheel through the diffuser. From the diffuser, the flow is then finally passing through the volute and outlet of the compressor.
There are different types of blades in the impeller wheel with different advantages and usages. The type of impeller wheel studied in this master thesis consists of 12 inclined blades. Six full blades and six splitter blades, aligned alternated around the circumference of the wheel. The full blades spans from the top of the wheel down to the diffuser, whilst the splitter is shorter in height and starts further down the shaft. The combination of full and splitter blades is a design that allows for higher airflow when operating at higher rotational speeds of the wheel.

The centrifugal compressor is limited in operating range by the surge and choke lines respectively. The surge appears in the lower mass flow-rates of the operating range whilst the choke appears at high.

2.1.2 Ported shroud

There are several methods to widen the operational range of a compressor. Two main approaches are active and passive flow control. Active flow control includes variable varying vanes installed in the diffuser and inlet guided vanes [8]. Both of these methods widens the operational range of the compressor but has the disadvantages of not being easy to design and produce. Passive flow control on the other hand consists of methods such as different kind of casing treatments like the ported shroud.

The ported shroud is a cavity in the housing that allows the gas to recirculate back upstream. It is used to widen the operational range of the compressor, by letting gas pass through it at low mass flow rates. The reason for needing a ported shroud is because of the phenomena surge that occurs in the vicinity of the compressor wheel. Surge or stall has the affect of letting the gas disrupt from its natural flow, hindering the gas from passing through down to the compressor wheel. The cavity of the ported shroud can then be used to redirect the gas back upstream into the main flow and later flow down through the compressor wheel when the mass flow rate has increased. The widening of the compressor
map will allow for the compressor to work at a greater span of mass flow rates. In Fig. 4 below, the ported shroud can be seen.

Fig. 4: Illustration of the ported shroud.

2.1.3 Governing equations
Three essential equations are solved for in Ansys CFX in order to capture the flow characteristics and its evolution through the compressor. They are called the governing equations in the subject of fluid mechanics and are the unsteady Navier-Stokes equations. The Navier-Stokes equations are the conservation of mass (continuity), the conservation of momentum, and the conservation of energy, and are defined below in cartesian tensor notation.

The continuity equation

\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0 \]  

(1)

The momentum equation

\[ \frac{\partial \rho u_i}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_j} + \frac{\partial \sigma_{ij}}{\partial x_j} + \rho F_i \]  

(2)

The conservation of energy

\[ \frac{\partial (\rho e_0)}{\partial t} + \frac{\partial (\rho u_i e_0)}{\partial x_i} = -\frac{\partial (\rho u_i)}{\partial x_i} - \frac{\partial q_i}{\partial x_i} + \frac{\partial (u_i \sigma_{ij})}{\partial x_i} \]  

(3)

Here, \( t = \) time, \( \rho = \) density, \( x_i = \) cartesian coordinate, where \( i = 1, 2, 3 \), \( u_i = \) fluid velocity component in \( x_i \) direction, \( p = \) pressure, \( \sigma = \) stress tensor.
\[ F_i = \text{body and external force in the system and } q_i = \text{heat flux.} \]  

In eq. (3), \( e_0 = e + \frac{1}{2} u_i u_j \) and denotes the total energy per unit mass, where \( e \) is the inner energy. An assumption is made that there is no external forces acting on the system leading to \( F_i = 0 \).

The gas investigated throughout the thesis is air where the gas is assumed to be calorically perfect, so the specific heat at constant volume \((C_v)\) and at constant pressure \((C_p)\) is constant. This gives the two relations:

\[ e = C_v T \quad (4) \]

and

\[ h = C_p T. \quad (5) \]

Air is a newtonian fluid meaning that the viscous stresses emerging from the fluid is linearly. This is true at every point in the fluid, so a constitutive relation for the newtonian fluid can be described as:

\[ \sigma_{ij} = 2\mu S_{ij} - \frac{2}{3}\mu S_{kk}\delta_{ij}. \quad (6) \]

In eq. (6) above, \( \mu \) denotes the dynamic viscosity of the fluid, \( \delta_{ij} \) is the Kronecker delta. \( \delta_{ij} \) is always in unity for \( i = j \) and zero for \( i \neq j \). The rate of deformation, \( S_{ij} \) is described as:

\[ S_{ij} = \frac{1}{2}\left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right). \quad (7) \]

The gas will because of the compression in the system get a rise in temperature, which leads to the assumption that the dynamic viscosity is temperature dependent. This is described by Sutherland’s law:

\[ \frac{\mu}{\mu_0} = \left( \frac{T}{T_0} \right)^{3/2}\left( \frac{T_0 + T_S}{T + T_S} \right), \quad (8) \]

where \( \mu_0 = 1.7231 e^{-5 \left( \frac{k_B T_0}{m \cdot \text{s}} \right)} \) is the reference viscosity at \( T_0 = 0^\circ C \), and \( T_S \) is the Sutherland’s constant.

### 2.1.4 Reynolds-Average Navier-Stokes

The above equations are used for simpler cases with steady laminar flow. However, for the turbo compressor the flow is highly turbulent and chaotic. The general flow type throughout the compressor is in fact turbulent, with some small scale regions with laminar flow. In a turbulent flow, the flow parameters varies with time in form of unsteady fluctuations. To capture the characteristics of a turbulent flow, a method called Reynold’s decomposition is used, where each quantity of the flow (i.e velocity or pressure) is split up into its time-averaged and time-dependent fluctuation part. For instance, the velocity:

\[ u = \bar{u} + u', \quad (9) \]
where \( \bar{u} \) denotes the time-averaged and \( u' \), the fluctuation part. In addition to Reynold’s decomposition, when dealing with a compressible flow as in a compressor, density variations due to turbulent fluctuations has to be considered.

This introduces the Favre averaging, which is a density weighted average that splits the previously mentioned quantity into a once again time-averaged and time-dependent fluctuating part:

\[ u = \bar{u} + u'', \]  

where the difference between Reynold’s decompositon and Favre averaging is the density weighted averaging applied in the latter, so the velocity \( u \) is related to the density via \( \bar{u} = \frac{\rho \bar{u}}{\rho} \).

Applying Favre averaging to the Navier-Stokes equations (eqs. 1-3) yields the Reynold-Average Navier-Stokes equations (RANS equations):

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho \bar{u}_i)}{\partial x_i} = 0, \]

\[ \frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j}, \]

\[ \frac{\partial}{\partial t}(\bar{e} + \frac{\bar{u}_i \bar{u}_i}{2}) + \frac{\partial \rho \bar{u}_i \bar{u}_j}{\partial x_j} + \frac{\partial}{\partial x_j}(\bar{p} \bar{u}_j (\bar{e} + \frac{\bar{u}_i \bar{u}_i}{2} + \bar{u}_j \frac{\rho \bar{u}_i \bar{u}_i}{2})) \\
= \frac{\partial}{\partial x_j}(\bar{u}_i (\sigma_{ij} - \frac{\rho \bar{u}_i \bar{u}_j}{2})) + \frac{\partial}{\partial x_j}(-\bar{q}_j - \rho \bar{u}_j h'' + \sigma_{ij} u'' \bar{u}_j - \rho \bar{u}_j \bar{u}_j)) \]

where \( \tau_{ij} = -\rho \bar{u}_i \bar{u}_j \) is the Reynolds stress term, which is introduced in order to capture the velocity fluctuations in all directions at macro scale. The Reynold stress tensor is in fact six different equations leading to a closure problem since the system of equations to solve consists of more variables then equations.

### 2.1.5 Eddy-Viscosity modeling

To close this system of equations, the six components of the Reynolds stress tensor should be expressed in the averaging way described above. This type of modeling, known as Eddy-viscosity modeling, will not introduce any new variables and is a concept introduced by Joseph Valentin Boussinesq where the Reynolds stress tensor and time-averaged velocity gradients are related to each other as seen in eq. (14) below:

\[ -\rho \bar{u}_i \bar{u}_j \approx 2 \nu_T S_{ij} - \frac{2}{3} \rho \nu_T S_{kk} \delta_{ij} \]  

Here, \( \nu_T \) is the turbulent eddy viscosity and represents the local property of the fluid. The Eddy-viscosity modeling is based on assumptions that limits the usage of it for turbulent flows. Firstly, the Reynolds stress tensor can be
acquired from single point quantities and secondly, the strain rate tensor, $S_{ij}$ assumes that the behaviour of the time-averaged velocity gradients will have a linear dependency. This implies that the strain rate tensor will not be affected by rotational velocities. Therefore it will be limited in centrifugal compressors with swirling flows or boundary layer separations and is most suited for wall-bounded flows.

### 2.1.6 SST Turbulence model

Because of the limitations of the eddy viscosity modeling, flow models have been developed to better suit the turbulent flows found in turbo compressors. One of those are the Shear Stress Transport model that from now on will be addressed as the SST model. The SST is based on the $K – \omega$ model and is the standard turbulence used for simulations on centrifugal compressors because of its ability to predict the flow separation under unfavourable pressure gradient that are found in compressors. The model takes into account for the transport of the turbulent shear stress and take advantage of both the Wilcox and $K – \epsilon$ model. It solves both transport equation for kinetic energy as well as specific dissipation rate. This makes the SST model more sophisticated and accurate than its predecessors and is suitable for a larger range of different flows, like compressors and aerodynamics.

The SST model is identical to the $K – \epsilon$ model in the free shear flows. At near walls, the prediction of separation from a smooth surface with high accuracy is difficult, especially under unfavourable pressure gradients like in the compressor.

It is recommended in the user guide to properly resolve the boundary layer with at least 10 layers for the SST model to capture the flow correctly and with high accuracy at boundaries. This is especially of importance when solving for heat transfer [3]. Throughout this study, the SST model is used for simulations together with an automatic wall function.

This automatic wall function will switch from wall function to a low-Re near wall treatment when the mesh is refined. As already mentioned, SST is recommended for accurate predictions near wall, but is mathematically identical to the $K – \epsilon$ model in the free stream. By using the automatic near-wall treatment, the SST model will shift from a low-Re number form to a wall function formulation when the mesh is refined. This shift will be carried out to smoothly shift between the two formulations, so the low-Re formulation is used for $y^+ < 2$. This is why at least 10 layers of inflation should be used, so the wall can be resolved enough. Since the condition of $y^+ \leq 2$ is difficult to adapt in all parts of the geometry, the SST model with automatic near-wall treatment will shift between low-Re and wall function for $y^+ < 11.06$.

By resolving the mesh by the use of inflation layer, so the mesh is below this criteria, the simulation does not have to switch between different wall functions, which may lead to an increased accuracy for the solver.
2.1.7 Boundary layer

The Reynolds number in a centrifugal compressor is usually determined as:

\[ Re = \frac{\omega L^2}{\nu}, \]  

(15)

where \( \omega \) denotes the rotational velocity and \( L \) is the height of the compressor wheel outlet diameter. For this compressor, the diameter is \( L = 99.5 \text{ mm} \). If measured at a rotational speed of \( \omega = 109400 \text{ revolutions per minute (RPM)} \), and a temperature around \( 25^\circ C – 200^\circ C \), the Reynolds number will be in the regions of \( 10^6 \leq Re \leq 10^7 \).

The high Reynolds number is mainly due to a low dynamic viscosity of the fluid \( (13.4 \cdot 10^{-6} \leq \mu \leq 35 \cdot 10^{-6} \text{ m}^2/\text{s}) \). One might think that the low viscosity should result in a negligible shear stress of the fluid. Because of the no-slip boundary condition at the walls, the viscosity is still of great importance.

Depending on the Reynolds number the flow in the boundary layer can be in three stages. Laminar, transitional or turbulent. In the laminar case, the velocity profile is increasing linearly from the wall out to the boundary layer edge and will have a more parabolic form than for a turbulent flow where the velocity profile is flatter. The laminar velocity profile most often have its maximum velocity in the middle of the channel with a magnitude around twice of the average. For a turbulent flow, the velocity profile will instead have a logarithmic behavior, see Fig. 5 below. The slope will be more steep for the turbulent velocity profile which will lead to an increase in magnitude of the shear stress. The only exchange of mass and momentum of laminar boundary layers will take place between adjacent layers. This exchange is not possible to witness since it occur on a microscopic scale. This exchange of mass and momentum will for a turbulent boundary layer also occur between multiple layers and not only adjacent ones. Because of that, the exchange or mixing is possible to witness since it occurs on a greater (macroscopic) scale. In the boundary layer region closest to the wall (viscous sub-layer) the flow will however remain laminar even for a turbulent flow.
2.1.8 Performance

In order to quantify how well or efficient a centrifugal compressor is performing a set of parameters are introduced. The parameters that often is considered in the subject is related to the air flow, pressure ratio and efficiency. Concerning the air flow, parameters such as pressure \( p \), temperature \( T \) and mass flow \( \dot{m} \) changes are considered. Since the geometry between two compressors can vary a lot, the best way to express the performance of a compressor would be to use a dimensionless parameter. This is where the pressure ratio is introduced. It is often described in two terms namely total-to-total and total-to-static pressure ratio, see eqs. (16) and (17) below. The reason to use two ways to express the pressure ratio is mainly that the static way will use the stagnation pressure at the outlet of the domain measured over, whilst total uses total pressure at the outlet instead. Total-to-total is the most common way to express the ratio but when performing test on the pressure, it can often be difficult to measure the total pressure. Therefore both terms are used to express the ratio.

\[
PR_{tt} = \frac{p_2^0}{p_1^0}, \quad (16)
\]
\[
PR_{ts} = \frac{p_2}{p_1^0}. \quad (17)
\]

The compressor wheel is applying energy to the fluid by the rotation of it. Sucking air into the inlet and compressing the air into the diffuser. The compressor is considered as a open system since air is transported into and out of the boundaries of the inlet and the volute (outlet). This allows for matter
in motion to be carried in and out of the system. The matter carries internal energy with it as well as small scales of potential energy.

The law of conservation of energy states that the energy for an open system must be in balance:

\[ \Delta U_{\text{system}} + \Delta U_{\text{surrounding}} = 0, \quad (18) \]

thus with the first law of thermodynamics, the energy balance can be written as:

\[ \dot{q} + W = \dot{m} \left( e + pv + \frac{C^2}{2} \right), \quad (19) \]

where \( \dot{q} \) and \( W \) on the left hand side is the heat transfer into and work done on the system. The work done on the system is due to the compressor wheel rotating and is expressed in angular velocity multiplied with the torque from the shaft, \( \vec{\omega} \cdot \vec{r} \). The heat transfer \( \dot{q} \) is due to the energy from heat transferred through the systems. Since the compressor wheel will produce much larger temperatures at the outlet than the solid domains, the heat transfer will work as heating on the solid domains. The ambient temperature of the outside walls of the solids are set to \( T_0 = 45^\circ \text{C} \), together with an wall heat transfer coefficient of \( htc = 35 \left[ \frac{\text{w}}{\text{m}^2 \text{K}} \right] \). This will contribute with a cooling effect by means of heat transfer energy on the solid.

On the right hand side of eq. (19) the changes in internal energy \( e \), work required to move the fluid \( pv \) and the kinetic energy \( \frac{C^2}{2} \) is multiplied with the mass flow \( \dot{m} \) to balance the equation.

\( e + pv \) will together form the specific enthalpy, \( h \) that with the kinetic energy added expresses the total enthalpy, \( h^0 \).

A common way to express the energy of the compressor is hence the equation below:

\[ W = \dot{m} (h^0_2 - h^0_1) = \dot{m} \Delta h_0, \quad (20) \]

The compressor is assumed to be in a reversible process under specific conditions, so the total amount of heat added to the system can be expressed as:

\[ ds = \frac{d\dot{q}}{T} \quad (21) \]

where \( s \) is the specific entropy. This lead up to:

\[ dh = T ds - pdV, \quad (22) \]

This is true also for a non-reversible flow since both the energy, entropy and volume are thermodynamic functions of state. For a compressor that both
are adiabatic and reversible, the specific entropy cannot change, and are called isentropic. The efficiency for an isentropic compressor is expressed as:

\[ \eta_{is} = \frac{W_s}{W}, \]

(23)

where \( W_s \) denotes the work done for a constant entropy, and \( W \) is the actual work into the compressor, \( W = h_2 - h_1 \). This is illustrated in Fig. 6 below in a schematic of the compression process.

\[ ds = \frac{dh}{T} - \frac{R dp}{p}. \]

(24)

In an isobaric process, the pressure remains constant, meaning \( dp = 0 \), so by integrating eq. (24) yields the total enthalpy as:

\[ h^0 = h_{ref}^0 \cdot e^{\Delta s/C_p}, \]

(25)

Which defines the enthalpy as a function of entropy change. Since the efficiency as stated in eq. (23) is the ideal work as a function of actual work that defines the efficiency of the centrifugal compressor, the equation can be rewritten as:

Fig. 6: \( h-s \) schematic of the ideal and real compression process.

The schematic shows both the ideal and the isobaric (real) process. The ideal process is for a constant entropy and the isobaric process is determined from the change in entropy of the fluid:
\[ \eta_{is} = \frac{W_s}{W} = \frac{h_{2,s}^0 - h_1}{h_2 - h_1}. \]  
(26)

The parameter \( \gamma \) is introduced as the fraction of the specific heat at constant temperature over specific heat at constant volume:

\[ \gamma = \frac{c_p}{c_v}. \]  
(27)

With the use of \( \gamma \), the isentropic temperature ratio can be expressed as:

\[ \frac{T_{2,s}}{T_1} = \left( \frac{p_2}{p_1} \right)^{\frac{\gamma-1}{\gamma}} , \]  
(28)

which yields another expression for isentropic efficiency as:

\[ \eta_{is} = \frac{\left( \frac{p_2}{p_1} \right)^{\frac{\gamma-1}{\gamma}} - 1}{\left( \frac{T_2}{T_1} \right)^{\frac{\gamma-1}{\gamma}} - 1}. \]  
(29)

As well as for the pressure ratio, also the efficiency is measured in term of total-to-total and total-to-static:

\[ \eta_{tt} = \frac{PR_{tt}^{\frac{\gamma-1}{\gamma}}}{\left( \frac{T_2}{T_1} \right)^{\frac{\gamma-1}{\gamma}} - 1} , \]  
(30)

\[ \eta_{ts} = \frac{PR_{ts}^{\frac{\gamma-1}{\gamma}}}{\left( \frac{T_2^0}{T_1^0} \right)^{\frac{\gamma-1}{\gamma}} - 1} . \]  
(31)

### 2.1.9 Aerodynamics

The flow in the compressor is highly unsteady because of the presence of both stationary and rotational parts, which makes it hard to investigate. A way around this problem is to use a reference system for the rotational part (compressor wheel), and by that have the ability to define the velocity triangles of the flow around the compressor wheel. The use of velocity triangles is a common method for turbo machinery to represent velocity components of the fluid and can express both the absolute and relative velocity in the vicinity of the compressor wheel.

Therefore a stationary reference frame is applied with its z-axis that coincide with the axis of rotation of the compressor wheel. In addition to that, a rotating reference frame is also introduced with the same axis, meaning it will rotate around the z-axis with the angular velocity of the compressor wheel. The velocity triangle are expressed in cylindrical coordinates, so the power \( (W) \) from the wheel generated onto the fluid can be formulated. It also has the benefit of expressing the radial and tangential velocity vectors.

In Fig. 7 below, the velocity triangles are illustrated both on the exducer and the inducer.
Fig. 7: **Velocity triangles on compressor wheel. Adapted from Sundström [13]**

Here $\vec{U}$ denotes the tangential blade velocity and represents the local velocity of the compressor wheel at a specific position $\vec{r}$. Having the angular velocity $\vec{\omega}$ of the wheel, $\vec{U}$ can be calculated as:

$$\vec{U} = \vec{\omega} \times \vec{r}. \quad (32)$$

The absolute velocity $\vec{c}$ expresses the velocity in the stationary frame of reference and $\vec{\omega}$ is the relative wheel velocity of the rotating frame of reference. The work input per unit mass flow generated on the fluid from the compressor wheel can be determined by Euler’s equation for turbomachinery:

$$W = \omega (r_2 U_{\theta 2} - r_1 U_{\theta 1}), \quad (33)$$

where $r_1$ and $r_2$ represents the radius of the inducer and exducer, as seen in Fig. 8 below.
The flow enters axially at \( r_1 \) and leaves radially at \( r_2 \) for a centrifugal compressor. Eq. (33) for turbomachinery can also be written in terms of the local blade speed:

\[
W_{1,2} = U_{2c_{\theta,2}} - U_{1c_{\theta,1}}.
\]  

By combining eq. (20) with eq. (34), the change in enthalpy can be expressed by the equation for conservation of rothalpy:

\[
\Delta h_0 = \int_{p_1}^{p_2} \frac{dp}{\rho} + \int_{T_1}^{T_2} Tds = \frac{1}{2}(U_2^2 - U_1^2) - \frac{1}{2}(\omega_2^2 - \omega_1^2).
\]

As seen in the above equation, the pressure rise is dependent for a change in enthalpy of the system.

### 2.2 Mesh dependency study

Besides turbo knowledge, a big part of CFD simulation demands high precision in meshing to avoid errors. For that reason, a mesh study is great importance. The use of a mesh study is a way to prove the reliability of the mesh. If one wants to publish a work at for example the organization ASME (American Society of Mechanical Engineering), the reliability has to be proven. A common standard to do this, is with the Richardson extrapolation. Therefore, as a part of the mesh study, literature regarding Richardson extrapolation has been studied, where focus is on grid convergence criteria (GCI).

The process of the mesh study is to generate different meshes, where every mesh will be more refined than the latter. Then by examining parameters of interest, one can study how much the function value of that parameter has changed for the refined mesh. By using Richardson extrapolation, the goal is to examine the approximate relative error and the numerical uncertainty for that parameter to numerically see how much the parameter has changed in
function value between the meshes. By examining the parameter graphically, it is possible to also illustrate the converging behaviour that the mesh refinement will have on the parameter.

2.3 Surge line prediction

Surge is a phenomenon that has been around for as long as the turbo compressors. To this day, it is still a challenge to predict when it occurs with good accuracy. It is a widely studied problem, and regarding centrifugal compressors in particular, work by Cumpsty [5], Semlithsch et al [8] and Nyström [13] to mention a few, could be considered to better understand the phenomena.

Surge occurs at low mass flows, where the flow is connected to large system oscillations or flow instabilities governed by high pressure gradients in the compressor. Fatigue from alternating stresses, both on the compressor blades and upstream components can occur during surge and this will limit the operational range of the compressor. In order to increase the operational range without risk of going into surge, the compressor as well as upstream and downstream components must be designed with surge in mind. To achieve this, one has to understand why and when surge occurs. By increasing the knowledge of the phenomena, it is easier to avoid it to increase the lifetime of the compressor.

Surge is appearing at lower mass flow rates of the compressor, and is often detected by studying the mass flow. The best evidence of surge is an audible turbulence [5]. When lowering the mass flow rate in the compressor, it is common to get an increase in pressure rise. At some point, the pressure rise will reach its maximum value. For mass flows below this point, the compressor will go into either surge or stall.

At this phenomena, the time dependent averaged mass flow will vary, compared to stall. This means that the compressor is changing between unstalled to stalled flow over time. This kind of behaviour is known as surge. The latter described process can impact the compressor in such a powerful way that the mass flow will indeed partially flow backwards leading already compressed gas to flow back out of the inlet. This process is known as deep surge. Deep surge may however be acting in a more mild way where the operating point is near the top of the pressure rise. The time frame of surge (surge cycle) when the compressor is acting in the regions of lower flow rates the, flow may temporarily be stalling. At deep surge the oscillations are characterized by frequencies lower than the natural frequencies of the system.

The other type of surge is called mild surge, and is characterized by mass and pressure oscillations governed by the Helmholtz resonance frequency. At mild surge, reversed flow does not occur.

In general, surge is close to, or axisymmetric when fully developed. Initially the surge is not axisymmetric and this may have a huge impact on axial compressors where large transverse loads acts on the rotor, leading to wear on the blades.
A centrifugal compressor can still perform in presence of rotating stall. Both axisymmetric stall in the regions of the inducer tip and stationary non axisymmetric stall produced by asymmetry in the volute are operable conditions. The reason for that is the fact that the compressor will still perform a centrifugal effect on the gas leading to pressure rise even though it operates in stall.

When it comes to predicting surge, there are several uncertainties to consider. Uncertainties in form of geometry of the system and also the operating conditions of the system. Regarding the geometry, studies have found that a system with larger volume downstream from the compressor often more easily can register surge behaviour, in contrast to a smaller downstream volume [5]. The design of the impeller blades such as tip and axial clearance, and also damage on the blades in form of e.g. blade erosion has to be considered. For the operating conditions, throttle changes in form of acceleration of the gas in the compressor is a contribution to uncertainty. Throttle change is partly due to the fact that a compressor seldom operates at a constant rotational speed, since the waste gas entering the turbine is varying in mass flow rate. Throttling is also affected by gear shifting, due to an abrupt stop of mass flow entering the turbine during gear shift. The uncertainties is the reason to introduce a concept called surge line. This line is the margin that displays at what regions of the compressor map that surge occurs. Below the surge line, the compressor cannot operate in a stable regime. There are several definitions of where the surge line is situated and fabricators of compressor often have their own definition. Because of the uncertainty in different surge line definitions, many producers use a save margin in the compressor map, often called surge control line. This is a great reason for getting a better understanding of surge and when it occurs, so the range of the compressor can be widened.

When setting up a compressor map, a parameter of interest (typically pressure ratio or efficiency) is plotted as a function of the mass flow rate that creates a speed line for a specific rotational sped of the compressor. This is done for different rotational speeds. The common denominator of the different speed lines of the compressor map is that they all have a surge margin, as well as a choke margin on the other side of the mass flow range. The design point for every speed line, where the maximum pressure ratio of that speed line is situated is often linked together with a design line to display where the compressor is acting most efficiently for different rotational speeds. The surge line can be lowered to widen the range of the compressor. This is achievable by changing the shape of the blades, reducing the tip clearance or perform changes of the casing [5](s.367).

To predict the behaviour of surge, one needs to be reminded that a compressor can work satisfactory even though a vast part of the flow is separated. Because of the fact that a centrifugal compressor wheel generally have several zones of stall or separated flow even though the compressor isn’t in an operation point of surge, there is a difference in opinion whether or not rotating stall is an origination of surge. Also the fact that the components should be investigated as a system and not treated as several separated components. According to Cumpsty [5], the components of the compressor should be treated as a coupled
system, so the combined effect of the parts is to be considered in the aim of predicting surge.

Manufacturers of turbos often illustrate where the phenomena surge is by presenting it in a compressor map. A compressor map is often shown as the pressure ratio or efficiency of the compressor as a function of mass flow at the outlet. In Fig. 9 below, an example of how a compressor map can look, is presented.

![Compressor Map](image)

**Fig. 9:** *An illustrative image of how a compressor map can look.*

This compressor map is presented as pressure ratio as a function of mass flow rate for different speed lines. Every speed line corresponds to a certain rotational velocity that the compressor is operating at for mass flows ranging from surge (low mass flow) to choke (high mass flow).

Five speed lines means that the compressor is tested for five different rotational speeds. The bold line on the left connecting the speed lines is the surge limit line. It tells at which mass flow rate the compressor is possible to operate at the different speed lines. In the other end of the speed lines is the choke limit line. This phenomena will not be managed in this thesis project. The black dashed line correspond to where the compressor works most efficient for all the
tested speed lines. Finally, the red surge control line, is a safety margin that some manufacturers includes in their compressor map to illustrate where the compressor should be operated above, to avoid damage. The surge control line is not used by all manufacturers, and there is no general equation of where it lies, so it rather is the manufacturers own choice of safety margin.

2.4 Conjugate Heat Transfer

The aim of this part is to investigate the methodology of setting up a Conjugate Heat Transfer (CHT) model for CFD simulation, to solve heat transport between fluids and solids, but also between solids and ambient. Also to investigate how the CHT model aerodynamic performance will do in comparison against the traditional adiabatic CFD method, as well as experimental data.

When performing CFD simulations on centrifugal compressors, Scania usually use the traditional method of only considering the fluid components of the geometry and apply adiabatic walls as boundary condition. This method is fairly straightforward when it comes to solving.

The CHT model introduces conducting solids to the model that surrounds the fluid domains. This setup visually gives a better understanding on how the different domains is interacting with each other. The method is however more complex to implement because of the increased number of domains to consider as well as the number of interfaces and boundary conditions that has to be included in the setup. Between every domain (solid to solid, solid to fluid or fluid to fluid) an interface is needed (on both sides) to connect the domains properly. The amount of interfaces subjected to frame change is increased by using CHT, so to effectively set up the case, it is essential to pre-define which faces that should be considered as a interface. This will speed up the process in CFX Pre since a named selection is easier to use than the auto generated surface names created by Ansys. The autogenerated names are typically called $FXXXX.XXXX$, where X is some arbitrary number between 0 and 9. It is a vital part of the modelling to define all this interfaces. If not done, Ansys CFX will automatically apply a wall where an interface is not defined. Not only that, but also per default set it as adiabatic, which is not preferable when solving for heat transfer.

When performing a CHT case, CFX will include terms for heat transport, conduction and volumetric heat sources into the energy equation. This is done automatically when domains are specified as solids in the geometry.

The benefit och using a CHT method is the ability to determine heat transfer characteristics of the simulation such as heat flux through between fluid and solid parts, or the heat transfer coefficients of the material. Information of heat transfer characteristic is of great importance because of reduced engine sizes and increased exhaust gas temperatures. By performing a CHT analysis, heat flow distribution can be analyzed in both the fluid and the solid. Knowledge
of this parameters can be used by engineers when testing compressors in laboratories in order to assess more trustworthy data. The parameters can also be used in order to calculate stress and fatigue of the compressor in Ansys’s other simulation environments, such as Ansys Mechanics.

There are also drawbacks with CHT methods. For instance, the geometry needs to be created both for the fluid and solid parts, which is a time demanding process. The mesh will also gain an increase in generation time since both solid and fluid parts need a mesh.
3 Method

3.1 Mesh Independency study

3.1.1 Geometry

The first problem considered, is to import the geometry and check for problems with it. The geometry consisted of three parts, namely the inlet, wheel and diffuser. In Ansys, the usual tool for editing a model is by using Design modeler. For geometry checks, there is however more convenient to use the second tool for geometries which is Spaceclaim. In Spaceclaim, the different parts of a model can be highlighted and checked for problems. For the compressor investigated there was a lot of problems concerning the boundaries. By checking which faces of the parts that had problems and cut out and paste back the faces, a built in stitch feature was used to sew the geometry back. This method would take away all boundary problems that Spaceclaim flagged about. The same procedure were performed for all three parts of the geometry.

3.1.2 Meshing

When the geometry was checked and problem free, the mesh could be set up. At Scania, there are some standard settings when setting up the mesh, which is presented below.

<table>
<thead>
<tr>
<th>Default</th>
<th>Setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Physics Preference</td>
<td>CFX</td>
</tr>
<tr>
<td>Solver Preference</td>
<td>CFX</td>
</tr>
<tr>
<td>Relevance</td>
<td>66</td>
</tr>
<tr>
<td>Element Order</td>
<td>Program Controlled</td>
</tr>
</tbody>
</table>

Fig. 10: *Default tab of mesh.*

As seen in Figure 10, the solver preference is set to CFX with relevance at 66. The relevance is an general option to control the fineness of the mesh, and by setting it to a value, it will have that relevance through the entire model. Scania is using 66 as a default value in Ansys.
The next tab is sizing, see Figure 11. Also for sizing, there are default settings that Scania uses, like size function, that is set to proximity and curvature. This option will allow the mesher to minimum number of element layers in regions that constitute gaps. It also examines the curvature on edges and faces and computes element sizes on these entities such that the size will not violate the maximum size or curvature normal angle. There is also a Scania standard to use \( \text{Min size} = 5e^{-4} \), proximity min size \( = 2.5e^{-4} \), max face size \( = 2e^{-3} \) and max tet size \( = 3e^{-3} \).

![Table](image)

**Fig. 11: Sizing tab of mesh.**

In Figure 12 above, the inflation tab is presented. The inflation layer is used to be able to have a great refinement near walls. This is a vital point in order to capture the viscous effects that rise and has a significant role of the flow, close to no-slip walls. First layer thickness works by defining the first layer height, meaning what height the first inflation layer will have from the wall. Then by varying the growth rate and number of layers, a total inflation layer height will be produced.

A key feature of inflation is which method used for collision avoidance. This is once again a Scania default to use layer compression, which task is to compress inflation layers in areas of collision. The defined heights and ratios in these areas are reduced to guarantee the same number of layers throughout the entire
inflation region.

Having the basic mesh tabs set up for wanted options, the next task is to assign all included parts in the mesh with suitable settings to assure great mesh design. There are three settings used with different refinement for all three parts. *Patch conforming method, inflation* and *face sizing*. Patch control method is a method control that provides support for 3D inflation with a built-in growth and smoothness control. This method cannot be interpreted for the whole geometry, so it is inserted for all three parts of the model.

The inflation is used to assign which faces of a part that should have inflation layers. Often a whole part is considered, except the interfaces that connects different parts. It is especially of importance to use inflation at faces with a lot of curvature or where a higher refinement is required to capture the flow characteristics, i.e the blades of the wheel.

The face sizing is used to control the adjacent faces to the faces within the scope, which means that meshes on the adjacent faces will transition smoothly to the size on the scoped face.

A key part of the meshing process is to meet the design criteria that Scania have to not have a higher skewness of the mesh than 0.95. The criteria of 0.95 is based on experience at Scania as a best practice guideline and is not an official demand. Skewness is a quality measure that determines how close to ideal a face or cell is. It ranges from 0.0 to 1.0, where values above 0.9 is considered to be bad. One has to take into account that the geometry of the turbo has some extremely complex parts, such as the blades. For that reason the criteria of values below 0.95 is a sacrifice that has to be made. A lower skewness is achievable but with the result of a high cost in computational time. For that reason, a sacrifice has to be made. As long as the elements of high skewness is not located at important locations of the domain that will consist of flow characteristics that needs to be captured, it is acceptable to keep some elements of higher skewness.

One way of dealing with high skewness is by using a structured mesh instead of the unstructured tetrahedron mesh. This method uses a block-structured grid that easier controls the mesh cell distribution and is highly space efficient. Since the geometry of the turbo is very complex and this mesh study focuses on the boundary layer and the outcome of refining it, an unstructured mesh is considered. Furthermore, Scania has through experience found out that CFD simulations with unstructured mesh will produce just as good result as for a structured mesh.

To determine how refined your mesh needs to be and still be able to generate expected result, a mesh study is performed. Similar work has been done earlier [14] regarding a mesh study on the volume mesh. From that study,
the conclusion drawn was that a mesh of 13,887,787 number of cells was the best choice in order to capture the behaviour of the fluid correctly and with as low computational effort as possible. The aim of the mesh study is to find a grid convergence index (GCI) small enough to ensure that a more refined mesh would not change the solution anymore. The GCI tells in percentage the numerical uncertainty a certain parameter has, compared between two meshes [12].

As mentioned earlier, for this mesh study, the focus is on refining the boundary layer. This is done purely with the inflation option First layer thickness.

The results from a simulation will partly depend on the discretization of the domain but also on the approximation of the mathematical model. Both truncation errors and round-off errors will contribute to the solution leading to a difference between numerical and exact solution. Truncation errors is the difference between the discretized equation and the exact one and is often estimated by replacing the nodal values in the discrete approximation by a Taylor series expansion about a single point. The discretized equation will consist of the original differential equation and a remainder which corresponds to the truncation error. For the discretization to be consistent, one has to avoid truncation errors, which is done by letting the mesh spacing approach zero.

The round-off error is a form of a quantization error that occurs due to the representation of floating point numbers on the computer and limits the accuracy at which the computer stores numbers.

The generated result from simulations needs to be validated to ensure good quality with high precision when solving. This is where the mesh study is introduced, to validate if the numerical and physical approximations from the solver is good enough to capture the flow characteristics of the domain.

The first step in the process before analyzing the discretization error is to show that the iterative convergence of the solver has preferably four orders of magnitude decrease in the normalized residuals. This needs to be fulfilled for every equation solved for. ASME Journal of Fluids Engineering has a criteria [11] of at least an order of three, but to be able to capture the flow characteristics correctly, a better convergence criteria (smaller) can be justified if the time cost is within reasonable frames. Usually, the convergence criteria is therefore set to $1 \cdot 10^{-4}$. A common method for discretization error estimation is Richardson extrapolation (RE), which is also the method that ASME recommends. The method has been around since 1910 [7] and is a commonly used standard. It has its limitations but is the most reliable method available for numerical uncertainty prediction to this date.

For this study, mainly three different meshes will be investigated, but to easier find the convergence to an asymptotic numerical value, two additional meshes are introduced. As mentioned earlier the volume mesh will not be actively treated, since the goal of the study is to investigate how the boundary
layer refinement will affect the result. That means that different approaches of inflation layer usage will be the method in focus. For all five meshes, the previously mentioned inflation layer method First layer thickness is used. The approach was to keep the first layer height constant to $3 \cdot 10^{-6}$ m, and by varying the growth rate and number of layers to try to achieve a fairly constant total inflation layer height between the five meshes.

Throughout this study, the subscript 1, 2 3 4 & 5 is used declaring the refinement of the mesh, where a lower subscript indicates a more refined mesh. The used parameters are seen in Table 3. Mesh3 will because of its great usage in the thesis also be refered to as the standard mesh.

**Table 3: Parameters used for the three meshes.**

<table>
<thead>
<tr>
<th>Mesh</th>
<th>$h_1$ [m]</th>
<th>Number of layers</th>
<th>$GR$ [%]</th>
<th>$h_{tot}$ [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh5</td>
<td>$3 \cdot 10^{-6}$</td>
<td>5</td>
<td>2.9445</td>
<td>$3.9994 \cdot 10^{-4}$</td>
</tr>
<tr>
<td>Mesh4</td>
<td>$3 \cdot 10^{-6}$</td>
<td>9</td>
<td>1.743</td>
<td>$3.3993 \cdot 10^{-4}$</td>
</tr>
<tr>
<td>Mesh3</td>
<td>$3 \cdot 10^{-6}$</td>
<td>10</td>
<td>1.5</td>
<td>$3.3999 \cdot 10^{-4}$</td>
</tr>
<tr>
<td>Mesh2</td>
<td>$3 \cdot 10^{-6}$</td>
<td>16</td>
<td>1.2284</td>
<td>$3.3994 \cdot 10^{-4}$</td>
</tr>
<tr>
<td>Mesh1</td>
<td>$3 \cdot 10^{-6}$</td>
<td>18</td>
<td>1.188378</td>
<td>$3.3993 \cdot 10^{-4}$</td>
</tr>
</tbody>
</table>

where $h_1$ denotes the first layer height, $GR$ is the growth rate and $h_{tot}$ is the total inflation layer height. In Ansys CFX Manual, a recommendation when using inflation methods is to use at least 10 layers or more to properly resolve the boundary layer, and capture the flow characteristics close to the wall. For that reason, it will be of greater interest to compare Mesh3-Mesh5.

Below, the residuals of the simulation is presented for the standard mesh of 37,915,257 elements. The chosen design point for the simulation is a mass flow of $\dot{m} = 0.2$ [kg/s], $\omega = 87000$ [RPM], that is run at the inlet temperature $T_{ref} = 298.15$ [K] with relative pressure of $p_{ref} = 100$ kPa.
As seen in Fig. 13, the U-, V- and W-momentum residuals are from approximately 450 time steps oscillating around the iterative convergence criteria of $1 \cdot 10^{-4}$. Besides the iterative convergence criteria, a solution control is set to terminate the solution after 800 accumulated time steps. The more time steps used, the longer the simulation will take, so this limit of 800 is also a value based on experience at Scania from earlier works. By setting the value to a higher number of maximum time steps, one might achieve residuals that converges to values below $1 \cdot 10^{-4}$. However, for this mesh study, the solution is considered to be converged enough. The fourth function in the figure corresponds to the P-mass residual and it is noticeable that it converges faster than the three others without any problems.

The RE method is consisting of several steps. First of all one has do define a representative cell size. For a 3-D problem, the approach is done using eq (36) below,

$$h = \frac{1}{N} \sum_{i=1}^{N} (\Delta V_i),$$  \hspace{1cm} (36)

where $\Delta V_i$ is the volume of the $i$th cell and $N$ corresponds to the number of cells in your mesh. In Ansys Meshing, there is an inbuilt quality tool that lets you investigate the cell characteristic length for the mesh. This length is the one that is used as the representative cell size $h$, and it is presented for all the meshes in Table 4 below.
Table 4: Mesh statistics

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Number of elements</th>
<th>h [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh 5</td>
<td>29,579,983</td>
<td>3.8392 · 10^{-4}</td>
</tr>
<tr>
<td>Mesh 5</td>
<td>34,445,237</td>
<td>3.2874 · 10^{-4}</td>
</tr>
<tr>
<td>Mesh 4</td>
<td>37,915,257</td>
<td>2.9869 · 10^{-4}</td>
</tr>
<tr>
<td>Mesh 2</td>
<td>48,571,877</td>
<td>2.326 · 10^{-4}</td>
</tr>
<tr>
<td>Mesh 1</td>
<td>52,111,560</td>
<td>2.1747 · 10^{-4}</td>
</tr>
</tbody>
</table>

The second step is to verify that the grid refinement factor $r$ is greater than 1.3 which is a recommended value generated from experience [11]. The grid refinement factor is the ratio between the coarse and refined characteristic cell length. Therefore, the grid refinement factor is calculated using $Mesh_5$ and $Mesh_1$ as the coarse and refined mesh respectively. The calculated refinement factor is presented in eq (37) below.

$$
r = \frac{h_{\text{coarse}}}{h_{\text{refined}}} = \frac{h_5}{h_1} = \frac{3.8392 \cdot 10^{-4}}{2.1747 \cdot 10^{-4}} = 1.7654
$$

The grid refinement factor is greater than 1.3, hence the RE method proceeds to step 3 where you first define the variables $r_{21}$ and $r_{32}$.

$$
r_{21} = \frac{h_2}{h_1} = 1.3735
$$

and

$$
r_{32} = \frac{h_3}{h_2} = 1.2853.
$$

The apparent order $p$ of the method is calculated using eq (40a-40c) below,

$$
p = \frac{1}{\ln(r_{21})} \left[ \ln \frac{\epsilon_{32}}{\epsilon_{21}} \right] + q(p)
$$

$$
q(p) = \ln \left( \frac{r_{21}^p - s}{r_{32}^p - s} \right)
$$

$$
s = 1 \cdot sgn \left( \frac{\epsilon_{32}}{\epsilon_{21}} \right)
$$

where $\epsilon_{32} = \phi_3 - \phi_2$, $\epsilon_{21} = \phi_2 - \phi_1$ and $\phi_k$ denotes the solution of the $k^{th}$ grid. $\phi$ is the variable of interest, which of this study consists of mass flow, pressure ratio and efficiency.

Initially, one has to guess the apparent order $p$ by setting it to be the first term of eq (40a) and neglecting the second. Eq (40b-40a) is computed iterative with the initial guess of $p$ for the first iteration and the updating it for every step until $p$ gets stable.
Step 4 of the process consists of calculating the extrapolated values from

\[ \phi_{\text{ext}}^{21} = \frac{r_{21}^p \phi_1 - \phi_2}{r_{21}^p - 1} \]  
(41a)

\[ \phi_{\text{ext}}^{32} = \frac{r_{32}^p \phi_2 - \phi_3}{r_{32}^p - 1} \]  
(41b)

The fifth and final step is to calculate error estimates, along with the apparent order \( p \). This can be done both for approximate relative error,

\[ e_{a}^{21} = \left| \frac{\phi_1 - \phi_2}{\phi_1} \right|, \]  
(42)

and extrapolated relative error,

\[ e_{\text{ext}}^{21} = \left| \frac{\phi_{\text{ext}}^{21} - \phi_1}{\phi_{\text{ext}}^{21}} \right|. \]  
(43)

The extrapolated relative error is a discretization uncertainty between two meshes (coarse and standard, or refined and standard mesh).

Finally, the fine-grid convergence index \( GCI_{fine}^{21} \)

\[ GCI_{fine}^{21} = \left| \frac{1.25 e_{a}^{21}}{r_{21}^p - 1} \right| \]  
(44)

The latter corresponds to a numerical uncertainty of the fine grid, for the different variables of interest \( \phi \), between two meshes.
3.2 Surge line prediction

Experimental study has been performed on the SC92T70 compressor in Pankl Turbosystems laboratory located in Mannheim Germany. The laboratory has performed test on several operation points. The Different rotational speeds used varies from approximately 41000 to 116000 RPM. Where different mass flows have been used ranging from surge to choke line. For this thesis, 4 different operational speeds has been picked from the experimental dato to produce a compressor map with CFD simulations. The laboratory performing the experimental data had some deviation for the different examined rotational velocities, so the velocities chosen for CFD simulations are takes as the average. These rotational velocities are 56250, 68000, 87000 and 109400.

The experimental data is ranging from surge to choke, but not in the region of surge. This is due to the laboratory not having the ability to perform test in this region. Therefore three design points per speed line is included that is beneath the region of surge in mass flow rate for this CFD analysis. By including these design points with the ones from experimental data, it should from post processing of CFD data be able to find parameters that can indicate on the phenomena. These three design points are shown as an illustration in Fig.14.
Fig. 14: *Illustration of the 3 design points included together with the design point taken from experimental data.*

The model used for the map was the standard mesh of the mesh independence study (Mesh 3). When performing a compressor map, an initial run is performed to retain data from a case known to converge. In Ansys post processor (CFD Post), expressions is set up to monitor different flow characteristics through the compressor system. These expressions are based on surfaces, lines and points of calculations.

An expression in CFD-Post can be checked as an input or output parameter. When performing the initial run, the exit-corrected mass flow was used as an input parameter. The input parameter is then constraining the solver with that mass flow as a boundary condition. For the compressor map, expressions of interest are declared and set as output parameters instead. The compressor map will then calculate all declared output parameters for every design point.

A compressor map is solved using *Ansys HPC Parametric Pack license*, which is a license option offered to be used for Ansys workbench jobs. The benefit of the license is to compute multiple design points in parallel for a single design study. In the license, a total of five HPC parametric pack can be utilized. The first parametric pack allows for four simultaneous jobs to be calculated in parallel, and for every addition of parametric pack license, the previous amount
of simultaneous jobs to be calculated is doubled. A total of four parametric pack licenses has been used for this simulation, resulting in 32 simultaneous design points being calculated in parallel [4]. This means that for every design point calculated, all declared output parameters for that particular case will be calculated.

For solving the compressor map, the parametric pack is initialized by solving for the current design point, meaning the previously solved design point used to set up the compressor map. This is due to the fact that from the results of it, it can be ensured of convergence. Knowing that your simulations of multiple design points is initialized with a design point known to converge is of great importance, since the solver uses parts of the already solved information when solving for the later ones. Therefore, it is more efficient for the solver to solve all design points at a certain rotational velocity before solving for the next [1]. In addition to that, for a certain rotational velocity, the order of the design points to solve should be set to start from a point known to be near maximum efficiency, since it will speed up the convergence.

By performing a compressor map, the settings used for the different design points will be the same except for the mass flow and rotational speed. This means that the setup used only needs to be defined once, and then configure the parametric pack to calculate for the varying mass flow and rotational velocity. The following setup for the simulation is therefore the setup used for all cases, except for the mass flow and rotational velocity. In Fig. 15 below, a view of the input, and output parameters can be seen.

![Fig. 15: Input and output parameters defined for use by the parametric pack license.](image)

The two first columns that are circled, \textit{ecmflow} and \textit{wheelspeed} corresponds to the input parameters used for the parametric pack. To the right of \textit{wheelspeed} the output parameters are seen. These are defined in CFD-Post. By varying the input parameters \textit{ecmflow} which is the exit corrected mass flow, a
whole speed line can be solved for. Exit corrected mass flow will be explained later in section 3.2.1. By varying wheelspeed which is the rotational speed, multiple speed lines can be solved for. For example, ten different mass flows, and four different rotational speeds will generate a total of 40 different inputs to the solver. A total of 140 different output parameters are defined, so when all input parameters are solved for, a big table will be generated with data of the output parameters. These output parameters are then later examined in order to find indications of the surge phenomena.

As mentioned in section 5.1, the mesh chosen for later simulations are the standard mesh (Mesh3) of 37,915,257 elements. The inlet and outlet is extruded 8 diameters to ensure fully developed ans stable flow in the system. Boundary condition for inlet and outlet are visualized in Fig. 16 below.

Fig. 16: Location of inlet and outlet interface.

An interface is connecting the inlet and wheel domains.
For the wheel domain, three interfaces are used to link the fluid from wheel to inlet, ported shroud and diffuser respectively as illustrated in Fig. 18 below.

The top red interface is wheel inlet interface, middle red is the ported shroud interface, and the bottom red is the wheel outlet interface (also known as wheel-to-diffuser interface). These interfaces will later be of interest when investigating the surge phenomena.

When performing a steady state simulation including a moving reference frame (MRF) that is connected to a stationary with an interface, must be set up using the proper frame changing model. In Ansys CFX-Pre, three different options are available as well as the option to disable frame change. Earlier work on compressors have proven that the best suitable frame change option for this kind of problem is Frozen rotor [14]. This model works by letting the frame of
reference change, but let the relative orientation of all the components across the interface be fixed. This is done by letting both the frames of reference connect so they keep a fixed relative position throughout the solving.

Disregarding the outlet interface of the volute domain, one interface is used to connect the diffuser with the wheel domain. It is shown in Fig. 19 below.

![Diffuser-to-wheel interface](image)

**Fig. 19:** Diffuser-to-wheel interface.

The analysis type is steady state, meaning it will use Reynolds Averaged Navier Stokes equation, mentioned earlier in section 2.1.4. Air (compressible) is used as fluid with the thermodynamic state of gas. The gas will reach speeds over Mach number $Ma = 0.3$, and is therefore modelled with total energy. This energy model differs from the usual Thermal energy just by taking into account that the velocities will reach subsonic and transonic velocities. Defined data for air is presented in Table 5 below, where $T$ is temperature, $\rho$ is density, $\kappa$ is thermal conductivity and $C_p$ is specific heat at constant pressure and $\mu$ is dynamic viscosity.
Table 5: Air data for different temperatures.

<table>
<thead>
<tr>
<th>T, [°C]</th>
<th>ρ, [kg/m³]</th>
<th>κ, [W/mK]</th>
<th>Cₚ, [J/kgK]</th>
<th>µ, [Ns/m²·10⁻⁶]</th>
</tr>
</thead>
<tbody>
<tr>
<td>-150</td>
<td>2.866</td>
<td>0.01171</td>
<td>983</td>
<td>8.636e-06</td>
</tr>
<tr>
<td>0</td>
<td>1.292</td>
<td>0.02364</td>
<td>1006</td>
<td>1.729e-5</td>
</tr>
<tr>
<td>5</td>
<td>1.268</td>
<td>0.02401</td>
<td>1006</td>
<td>1.754e-5</td>
</tr>
<tr>
<td>10</td>
<td>1.246</td>
<td>0.02439</td>
<td>1006</td>
<td>1.778e-5</td>
</tr>
<tr>
<td>15</td>
<td>1.225</td>
<td>0.02476</td>
<td>1007</td>
<td>1.802e-5</td>
</tr>
<tr>
<td>20</td>
<td>1.204</td>
<td>0.02514</td>
<td>1007</td>
<td>1.825e-5</td>
</tr>
<tr>
<td>25</td>
<td>1.184</td>
<td>0.02551</td>
<td>1007</td>
<td>1.849e-5</td>
</tr>
<tr>
<td>30</td>
<td>1.164</td>
<td>0.02588</td>
<td>1007</td>
<td>1.872e-5</td>
</tr>
<tr>
<td>35</td>
<td>1.146</td>
<td>0.02625</td>
<td>1007</td>
<td>1.895e-5</td>
</tr>
<tr>
<td>40</td>
<td>1.127</td>
<td>0.02662</td>
<td>1007</td>
<td>1.918e-5</td>
</tr>
<tr>
<td>45</td>
<td>1.110</td>
<td>0.02699</td>
<td>1007</td>
<td>1.941e-5</td>
</tr>
<tr>
<td>50</td>
<td>1.093</td>
<td>0.02735</td>
<td>1007</td>
<td>1.963e-5</td>
</tr>
<tr>
<td>60</td>
<td>1.060</td>
<td>0.02808</td>
<td>1007</td>
<td>2.008e-5</td>
</tr>
<tr>
<td>70</td>
<td>1.029</td>
<td>0.02881</td>
<td>1007</td>
<td>2.052e-5</td>
</tr>
<tr>
<td>80</td>
<td>1.000</td>
<td>0.02953</td>
<td>1008</td>
<td>2.096e-5</td>
</tr>
<tr>
<td>90</td>
<td>0.9720</td>
<td>0.03024</td>
<td>1008</td>
<td>2.139e-5</td>
</tr>
<tr>
<td>100</td>
<td>0.947</td>
<td>0.03095</td>
<td>1009</td>
<td>2.181e-5</td>
</tr>
<tr>
<td>120</td>
<td>0.898</td>
<td>0.03235</td>
<td>1011</td>
<td>2.264e-5</td>
</tr>
<tr>
<td>140</td>
<td>0.8544</td>
<td>0.03374</td>
<td>1013</td>
<td>2.345e-5</td>
</tr>
<tr>
<td>160</td>
<td>0.815</td>
<td>0.03511</td>
<td>1016</td>
<td>2.420e-5</td>
</tr>
<tr>
<td>180</td>
<td>0.779</td>
<td>0.03646</td>
<td>1019</td>
<td>2.504e-5</td>
</tr>
<tr>
<td>200</td>
<td>0.745</td>
<td>0.03779</td>
<td>1023</td>
<td>2.577e-5</td>
</tr>
<tr>
<td>250</td>
<td>0.675</td>
<td>0.04104</td>
<td>1033</td>
<td>2.760e-5</td>
</tr>
<tr>
<td>300</td>
<td>0.617</td>
<td>0.04418</td>
<td>1044</td>
<td>2.934e-5</td>
</tr>
<tr>
<td>600</td>
<td>0.404</td>
<td>0.06093</td>
<td>1115</td>
<td>3.846e-5</td>
</tr>
<tr>
<td>2000</td>
<td>0.155</td>
<td>0.11113</td>
<td>1264</td>
<td>6.630e-5</td>
</tr>
</tbody>
</table>

3.2.1 Boundary Conditions

By using a boundary condition for the pressure or for the temperature at the outlet to control the mass flow, it is hard cover the whole mass flow rate of a speed line without entering surge or choke. Therefore it is common to use an exit corrected mass flow, that corrects the mass flow rate to total conditions at the outlet. With this method, the exit corrected mass flow can be maintained constant. It relates the mass flow with pressure and temperature at the outlet as well as the reference pressure and temperature at the inlet. Besides the benefit of not having to change the total conditions, it also makes it possible to be used for different compressors, since it is adjusted with reference pressure and reference temperature. Therefore two compressors operating at different mass flows can use the same exit corrected mass flow. The equation for exit corrected mass flow is:
\[
\dot{m}_{ce} = \dot{m} \sqrt{\frac{T_2}{T_{ref}}} \times \frac{p_f}{p_{ref}}.
\] (45)

The outer walls of the fluid domains are modelled as adiabatic walls. This is a standard boundary condition that Scania uses for their CFD simulations of centrifugal compressors. By using adiabatic walls, no energy will pass through the walls, they are fully isolated. This means that the only energy contribution to the system is the energy that the rotating compressor wheel will transfer to the fluid in form of heat and kinetic energy.

As mentioned above, a total of 140 output parameters were defined to examine the phenomena to find indications of surge. The ones found will be presented in section 4.2.
3.3 Conjugate Heat Transfer

For the mesh study and the surge line prediction, a geometry of the fluid parts were imported from the designers at Scania. This geometry will also be called as the adiabatic or traditional geometry later on in this report. For the CHT case, a geometry was once again imported from the designers, but this time for the solids. Therefore the fluid parts had to be created using Spaceclaim. The fluid geometry can be seen in Figs. 20a and 20b below.

(a) Adiabatic geometry viewed from the side.  
(b) Cross section of the adiabatic geometry.

Fig. 20: Adiabatic geometry consisting of 3 domains, inlet, wheel and diffuser/volute domain.

In Fig. 21a and 21b below, the CHT geometry is presented.

(a) CHT geometry viewed from the side.  
(b) Cross section of the CHT geometry.

Fig. 21: CHT geometry consisting of 8 domains.

As seen in the Fig. 21b to the right, the CHT geometry consists of 8 domains. The fluids in pink, compressor house solids in green, compressor wheel (solid) in blue, and finally shaft and shaft nut in gray.
Even though the geometry imported from the designers used for the two first problems in this thesis should nominally be the same as the geometry for the CHT problem, there was geometrical differences on several locations around the compressor wheel. In Table 6 below, these differences are presented.

**Table 6:** Geometrical differences between CHT and adiabatic geometry

<table>
<thead>
<tr>
<th></th>
<th>CHT geometry</th>
<th>Adiabatic geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Trailing edge, [mm]</td>
<td>0.4002</td>
<td>0.3999</td>
</tr>
<tr>
<td>( d_{r1r2} ), [mm]</td>
<td>29.7874</td>
<td>30.1061</td>
</tr>
<tr>
<td>( d_{ps} ), [mm]</td>
<td>1.9997</td>
<td>1.4344</td>
</tr>
<tr>
<td>( h_{ps} ), [mm]</td>
<td>3.0113</td>
<td>5.8780</td>
</tr>
<tr>
<td>( h_{W2D} ), [mm]</td>
<td>4.3561</td>
<td>4.3267</td>
</tr>
</tbody>
</table>

Here, *Trailing edge* corresponds to the distance from the end of the blade (closest to the wheel outlet) to the shroud. \( d_{r1r2} \) is the distance from the shaft to the shroud. \( d_{ps} \) is the radial distance from the shroud to the ported shroud interface. \( h_{ps} \) is the height of the ported shroud interface. \( h_{W2D} \) corresponds to the height of the wheel-to-diffuser interface.

When it comes to meshing, the method was to divide the parts into two modules. One for fluids and one for solids. The benefit with this approach is to have global mesh settings that will apply for all solids, and vice versa for the fluids. Ansys is good at writing big files of data for every CFD project, so by meshing solids and fluids separately, the memory usage in the computer can be reduced. In Fig. 23 below, the two mesh modules are illustrated. The image also shows how the two mesh modules later links together into the setup cell.
One drawback of the separate mesh modules is the interface generation. This has to be set up for all interfaces, connecting two faces, and needs to be done on both sides to link them together for heat transfer. This is a time demanding process that in the end lead up to 74 interfaces. By having only one mesh module, it is easier to keep track of all interfaces and ensure that both sides of the interface are defined.

For mesh settings, the fluid parts are meshed with the exact same settings as for the mesh study and surge line case. The solids will however not have the same refinement due to the linearity of heat transfer in solids. The solids are therefore meshed with default settings, and without any refinement of the boundary layer. The final mesh of the solids and fluids is presented in Figs. below.

![Mesh of solid domains.](image1)

![Mesh of fluid domains.](image2)

**Fig. 23:** Separately meshes of fluid and solid domains.

In the surge line case a frame change option was introduced to connect MRF faces with stationary faces on the wheelfluid. In the CHT case, in addition to the wheel fluid, also the wheel solid, shaft and shaft nut is rotating, which means that a lot more frame change interfaces must be defined. This is another benefit of having pre defined all faces with named selection, to easier connect
All solids must also have the right thermodynamic data, such as thermal conductivity, specific heat and density in order for the solver to properly compute the heat transfer in the solids. The outer walls of the traditional case were modeled with adiabatic walls. In this model the outer walls are modelled with an ambient temperature of $T_{\text{out}} = 45^\circ C$ to correspond to the temperature in the engine room. The outer walls is also set up with a heat transfer coefficient of $htc = 35 [\text{W/m}^2 \cdot \text{K}]$, to allow heat transport out of the solid walls.

Since the CHT model also includes the rotating shaft, that will rotate in close vicinity to the compressor house, friction and thermal energy will arise in this region. This is modelled by introducing a thin layer of air (0.5 mm) that will lie between the shaft and compressor house as a thermal resistance.

Another difference in the setup compared to the traditional adiabatic case is the time step used for the solver. Normally when performing CFD simulations on a compressor, a physical time step is used of $1/\omega$, but for this CHT case, an automatic time scale is introduced, with a timescale factor of 10 for the fluid parts, and a timescale factor 100 for the solid parts, [6].

Other than that, the CHT model is identical in setup compared to the traditional adiabatic model. The simulations were performed on one speed line of 109400 RPM. The solid mesh ended at 5,754,398 elements, and the fluid on 38,844,114, leading to a total of 44,598,512 elements of the final mesh. The simulation time for one design point on this speed line took 7h 27min running on 168 CPUs.
4 Results

4.1 Mesh independency study

In Fig. 25a, 25b and 26 below, a graphical view of the examined parameters are shown. Here, the parameters are plotted as a function of refined mesh, to illustrate the converging behaviour resulting of the mesh refinement.

(a) Total-to-total & total-to-static efficiency. (b) Total-to-total & total-to-static pressure ratio.

Fig. 25: Pressure ratio and efficiency plotted as a function of inflation layers.

Fig. 26: Mass flow at outlet as a function of inflation layers.

With the Richardson extrapolation method, the approximate relative error and numerical uncertainty ($GCI_{fine}$) is calculated for the above parameters comparing $Mesh_3$ and $Mesh_4$, with the steps presented in Section 3.1.2. The
Table 7: Approximate relative error and numerical uncertainty for examined parameters between Mesh$_3$ & Mesh$_4$.

<table>
<thead>
<tr>
<th>$\dot{m}$</th>
<th>$\eta_{tt}$</th>
<th>$\eta_{ts}$</th>
<th>$PR_{tt}$</th>
<th>$PR_{ts}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\varepsilon_{approx}$, [%]</td>
<td>0.110</td>
<td>0.509</td>
<td>0.519</td>
<td>0.0348</td>
</tr>
<tr>
<td>$GCI_{fine}$, [%]</td>
<td>0.0374</td>
<td>0.3339</td>
<td>0.3421</td>
<td>0.0033</td>
</tr>
<tr>
<td>CPUs</td>
<td>240</td>
<td>240</td>
<td>240</td>
<td>240</td>
</tr>
<tr>
<td>Simulations time</td>
<td>2h</td>
<td>2h 12min</td>
<td>2h 27min</td>
<td>3h 27min</td>
</tr>
</tbody>
</table>
The standard mesh is illustrated in Figs. 27-28d below.

**Fig. 27:** Mesh 3 in cross section.

(a) Close up of wheel domain.  
(b) Inlet-to-wheel fluid

(c) Wheel-to-diffuser.  
(d) Ported shroud.

**Fig. 28:** The average and standard deviation of critical parameters: Region R4
4.2 Surge line prediction

In order to know that the result from the simulations are trustworthy, an analysis of the compressor map against experimental data is first performed. In Figs. 29a-29b below, the compressor map of efficiency as a function of mass flow is plotted. CFD data as circles, and experimental data as squares.

As seen in the plots, the CFD data follows the trend of the experimental data quite good. The same investigation where made on the compressor map for the pressure ratio, as seen in Figs. 30a-30b below.

Fig. 29: Efficiency compressor maps.
Here, the experimental values are plotted as stars. Also in these compressor maps, the CFD data follows the same characteristics as the experimental data. With that knowledge, investigation of surge indicators could be performed. As mentioned, over 140 parameters was investigated, but only parameters with indicators will be presented below. These indicators are found on the interfaces of the wheel fluid, and can be seen in Fig. 18, in section 3.2, where the examined interfaces are illustrated.

4.2.1 Mass flow

First out was investigation of the mass flow. By examining the wheel inlet interface (inducer), it was noticeable for the two lowest speed lines a oscillating behaviour at low mass flow rates, as seen in Fig. 31.
The oscillating behaviour is very small and hard to catch. Therefore a new plot was examined where the mass flow at the wheel inlet interface was plotted as a ratio over the mass flow over the compressor inlet. This is seen in Fig. 32 below.

By examining this plot, the behaviour is more noticeable. An indication of surge is spotted at the two lowest speed lines, for low mass flow rates. The
same mass flow ratio was also examined for the ported shroud interface over the compressor inlet, as seen in Fig. 33.

![Mass flow ratio, ported shroud over inlet](image)

**Fig. 33:** Mass flow ratio, ported shroud over compressor inlet.

Once again is an indication found for the two lowest speed lines at low mass flow rates.

### 4.2.2 Pressure recovery coefficient

In Fig. 34a and 34b below, the pressure recovery coefficient is examined at the ported shroud and wheel outlet interface.

![Pressure coefficient at ported shroud](image)

(a) Ported shroud interface.

![Pressure coefficient at W2D](image)

(b) Wheel outlet interface.

**Fig. 34:** Pressure recovery coefficient.
In both cases, indication of surge can be spotted for the two lowest speed lines for low mass flow rates.

### 4.2.3 Static pressure

From investigation of static pressure, one interface picked up a behaviour that differed at the two lower speed lines at low mass flow rates. It was found by examining the ported shroud interface. The indication was however small oscillations hard to see, if not knowing what to look for. Therefore, a measure plane was introduced 1 cm upstream of the ported shroud cavity, where examined. In Figs. 35a-35b, the result of these two figures are plotted.

As seen in Fig. 35a the behaviour can be spotted for the two lowest speed lines for low mass flow rates as small oscillations. The indication is more apparent by investigating the upstream measure plane in Fig. 35b. Still only for the two lowest speed lines.

### 4.2.4 Temperature

By placing two circumferential lines, one at a 50% radial distance from the axis, and one at a 90% radial distance from the axis, and take the mean temperature on this lines. The ratio of these temperatures are plotted in Fig. 36 below.
This examination shows the most apparent indication of surge for the two lowest speed lines, for low mass flow rates.

In Fig. 37 below, the static temperature on the wheel inlet was taken as an area average.

Fig. 36: Mean temperature ratio on measure lines on wheel inlet interface.

Fig. 37: Static temperature at wheel inlet interface.
This indication, besides for the two lowest speed lines also picks up the behaviour for the highest rotational speed of 110k RPM.

The last indication of surge was found by examining the area average of the total temperature, once again on the wheel inlet. This is seen in Fig. 38 below.

![Total temperature at wheel inlet interface](image)

**Fig. 38:** Total temperature at wheel inlet interface.

Once again, the indication of surge is only spotted at the lowest two speed lines, for low mass flows.
4.3 Conjugate Heat Transfer

4.3.1 Temperature fields

In Fig. 39 the temperature distribution in the compressor house (solid) is plotted. The temperature distribution for the compressor wheel can be seen in Figs. 40a-40b.

Fig. 39: *Cross section plane of temperature distribution in solids.*

(a) View 1 of compressor wheel.    (b) View 2 of compressor wheel.

Fig. 40: *Two views to illustrate the temperature distribution on the compressor wheel.*
A comparison of the temperature on the compressor wheel was made between the two cases and is presented in Table 8 below.

**Table 8:** Temperature distribution on compressor wheel, between CHT and adiabatic models.

<table>
<thead>
<tr>
<th>Model</th>
<th>Min temp, °C</th>
<th>Max temp, °C</th>
</tr>
</thead>
<tbody>
<tr>
<td>CHT model</td>
<td>55</td>
<td>140</td>
</tr>
<tr>
<td>Adiabatic model</td>
<td>25</td>
<td>180</td>
</tr>
</tbody>
</table>

In Figs. 41a and 41b below, the temperature distribution of the fluids are shown. CHT model to the left and adiabatic model to the right.

(a) Cross section view of temperature distribution on CHT model.
(b) Cross section view of temperature distribution on adiabatic model.

**Fig. 41:** Comparison of temperature distribution, CHT model to the left, adiabatic model to the right.

Another comparison of temperature distribution in the fluid is made once again, but here for the diffuser. CHT model to the left and adiabatic to the right, as seen in Figs. 42a and 42b below.
4.3.2 Efficiency

Next up is numerical plots for comparison of the different models. In Figs. 43a and 43b below, the efficiency is plotted together with adiabatic traditional model as well as experimental data for comparison.

**Fig. 42:** Comparison of temperature distribution, CHT model to the left, adiabatic model to the right.

**Fig. 43:** Efficiency comparison between CHT model, traditional adiabatic model and experimental data.
4.3.3 Pressure Ratio

The pressure ratio of the CHT model is also compared against the traditional adiabatic model and the experimental data, as seen in Figs. 44a and 44b below.

Fig. 44: Pressure ratio comparison between CHT model, traditional adiabatic model and experimental data.

In figure 45 below, the power of the compressor wheel is compared between the two models.
4.3.4 Temperature

In the next following plots, there is no experimental data to compare against, so these plots are only to compare the CHT model against the traditional adiabatic model. First is the total temperature ratio and total temperature at wheel outlet, as seen in Figs. 46a and 46b below.

Fig. 45: Difference in power between the CHT and adiabatic model.

Fig. 46: Total temperature comparison.
In Figs. 47a and 47b below, the total temperature at the ported shroud and the wheel inlet is plotted.

![Total Temperature at ported shroud](image1)

![Total Temperature at wheel inlet](image2)

(a) **Total temperature at ported shroud.**  
(b) **Total temperature at wheel inlet.**

**Fig. 47:** **Total temperature comparison.**

### 4.3.5 Static pressure

The static pressure at the outlet and ported shroud is seen in Figs. 48a and 35a below,

![Static Pressure at outlet](image3)

![Static Pressure at ported shroud](image4)

(a) **Static pressure at outlet.**  
(b) **Static pressure at ported shroud.**

**Fig. 48:** **Static pressure comparison.**

and the same parameter measured at the wheel inlet and wheel outlet is
plotted in Figs. 35a and 49a below.

(a) Static pressure at wheel inlet.  
(b) Static pressure at wheel outlet.

**Fig. 49:** Static pressure comparison.

### 4.3.6 Total pressure

The last results is for the total pressure, where the parameter is measured at the outlet and wheel outlet can be seen in Figs. 50a and 50b below.

(a) Total pressure at outlet.  
(b) Total pressure at wheel outlet.

**Fig. 50:** Total pressure comparison.

The same parameter measured over the ported shroud and wheel inlet is plotted in Figs. 51a and 51b below.
Fig. 51: Total pressure comparison.

(a) Total pressure at ported shroud.

(b) Total pressure at wheel inlet.
5 Analysis & Discussion

5.1 Mesh dependency study

From Fig. 25a, 25b and 26 the examined parameters is plotted as a function of mesh refinement. In all three figures, the converging behaviour of the function values is found. This converging behaviour is sought, and can graphically tell how the function value eventually will not change anymore.

The approximate relative error for the examined parameters between \( \text{Mesh}_3 \) and \( \text{Mesh}_4 \) are less than 0.52\% and the numerical uncertainty for the same parameters are less than 0.35\%.

The added simulation time to perform computations on \( \text{Mesh}_4 \) is approximately one hour longer than for \( \text{Mesh}_3 \). This is only the time for computing the case, and the time for discretization is not included. That process increased in man-hours for every refinement.

As mentioned earlier, the CFX manual in Ansys recommends to use at least \( \geq 10 \) layers or more when using inflation methods, to properly resolve the boundary layer.

All above points has been used to motivate the use of \( \text{Mesh}_3 \) for future work in this thesis project, since it is refined enough. As seen in Table 4, \( \text{Mesh}_3 \) consisted of 37,915,257 elements.

5.2 Surge line prediction

From the performed steady state simulations, indicators of the surge phenomena was found. These indicators where often found by examining wheel inlet and ported shroud. This did not come as a surprise since surge arises because of large system oscillations or flow instabilities due to the large pressure gradients in the region around the compressor wheel. With other words, the compressor wheel is the region in the compressor that should be affected by the phenomena, and for all indicators found, they were found in the vicinity of the compressor wheel.

From the plots that could indicate of surge, it could be traced specifically to a certain region in the mass flow for the two lower speed lines. Because of the confidentiality of this thesis, no numerical value of this mass flow regions will be presented.

The higher rotational speeds simulated for, the harder was it to capture this phenomena. In fact only one indication of surge was found. Static temperature at wheel inlet at 110k \( \text{RPM} \). Because only one indication was found, no conclusion can be drawn that surge will appear at this mass flow. Therefore more indications are needed to ensure the mass flow region of surge of higher speed lines. The same indicator was not found for the speed line of 87k \( \text{RPM} \). By examining Fig. 37, it almost looks like the design points simulated over has not reach the surge region yet and needs more design points.
It can also be answered by the fact that surge is both a stationary and transient behaviour, and might need transient simulations to indicate on surge for higher speed lines.

5.2.1 Future work

Since it in many plots looks like the design points for higher speed lines are not in the region of surge, it might mean that more design points are needed in the region of surge to capture the behaviour. The design points was taken from experimental data, so it should be checked that the experimental part was performed correct for all speed lines.

Because the simulations could not capture behaviour of surge at higher speed lines using steady state simulations, the next step would to perform this study with transient simulations.

Surge is a phenomena that not only depends on the compressor but the the system as a whole. The inlet was extruded 8 diameters to ensure a stable flow computed over. It would have been interesting to compare this extruded inlet with an original inlet, but also against other inlet types of compressors, such as radial inlets. To see if this geometrical change makes it easier to predict surge. For that reason, it would also be of interest to compare two different compressors with different size and volume, to see if volumetric or geometric differences can effect the ability to indicate surge.

5.3 Conjugate Heat Transfer

The geometry for the CHT model as stated in section 3.3 should nominally be the same as for the adiabatic model. When all simulations were performed and the results where analyzed the geometry was compared to justify the result. From Table 6, it is noticeable that there are several distances that differ between the two geometries. Especially $h_{ps}$, the height of the ported shroud interface. The height of the CHT model is almost half the height of the adiabatic model, which makes it harder for the gas to flow through the ported shroud cavity. This also means that when comparing results from the CHT model against the adiabatic model and experimental data, the geometry is not the same. It has been shown from earlier work at Scania that geometrical differences in the regions in vincinity of the compressor wheel can have great affect on the results. Therefore one has to be reminded of this when analysing the results comparing the different cases.

As seen in Fig. 39, the temperature distribution in the solids, (compressor house) can be solved for. The same goes for the heat transfer on the compressor wheel and shaft, as seen in Figs. 40a and 40b. This is one of the benefits of performing a CHT model, compared to using adiabatic outer walls of the fluid,
as usually is done at Scania. These temperature fields, but also wall heat flux, is vector fields that can be exported from CFD-Post to be used for mechanical analysis.

The lowest temperature of the CHT model is more than double the temperature of the adiabatic model as seen in Table 8. It also differ at the maximum temperature, where the CHT model is 40°C lower. This might depend on the fact that the CHT model allows heat transport between fluids and solids. Therefore the gas will pick up heat from the solids on its way down to the compressor wheel, leading to a higher minimum temperature on the wheel. With the same reason, the compressor wheel will emit heat to the fluid and surrounding solids, leading to a decrease in maximum temperature.

In Figs. 41a and 41b the temperature distribution in the fluid is presented. It its clearly noticeable the quantitative difference in how the CHT model (left) handles the heat transport compared to the adiabatic model (right). This gets more obvious by also examining the temperature distribution in the diffuser fluid domain, Figs. 42a and 42b. There is a quantitative difference in the distribution of temperature once again. Worth to mention is that the temperature scales for these plots are the same when comparing the CHT against the adiabatic model.

When comparing the efficiency in Figs. 43a and 43b, it is noticeable that the CHT model follows the same trend as the traditional adiabatic model, but also as experimental data. However when approaching lower mass flows, the CHT model starts to underpredict in function value. This underprediction of efficiency could have to do with the fact that the CHT model allows for heat transfer out of the fluid walls, whilst the adiabatic model does not, and is considered a isolated system that does not allow heat or matter transport through walls. Therefore the CHT model was also tested without the solid domains, and with adiabatic outer walls of the fluid domains. This CHT model with adiabatic walls were performed at one design point of 0.1671 kg/s in mass flow, and is plotted as the red star in Figs. 43a and 43b. Corresponding design point with CHT model is plotted as a blue star. Unfortunately the efficiency dropped even more as seen in the figures, which lead to the conclusion that the allowance of heat transfer in the CHT model is not the reason for the efficiency to underpredict.

Instead an analysis of the pressure ratio was performed, as seen in Figs. 44a and 44b. Once again the CHT model follows the same characteristics as the adiabatic model and experimental data, but starts to underpredict when approaching lower mass flow rates. Because of that, the power in the system was compared between the CHT and adiabatic model. The allowance of heat transfer in the CHT model might lead to a loss of energy in the system that could be traced back to the underprediction of efficiency and pressure ratio. As seen in Fig. 45, the power for the CHT model and the adiabatic model has the same curve and is only shifted in function value. If the energy loss would have
been the reason for the under prediction, the power for the CHT model should then have a deviating behaviour for lower mass flows compared to the adiabatic model. This was not the case, which lead to the conclusion that the underprediction of efficiency and pressure ratio of the CHT model rather depends on the geometrical differences shown earlier in section 3.3, in Fig. 22, and discussed above.

The total temperature ratio between outlet and inlet as seen in Fig. 46a shows a quantitative difference in function value, but both models have a similar behaviour.

From Fig. 46b, showing the total temperature at the wheel outlet, the two models behave relatively the same with little deviation.

When analyzing the total temperature at the ported shroud and wheel inlet, seen in Figs. 47a and 47b the CHT model have a whole other behaviour than the adiabatic model for lower mass flows. For increasing mass flow the CHT model starts to follow the characteristics of the adiabatic model. The reason for the strange behaviour at low mass flows may depend on the usage of the ported shroud. The ported shroud is used to recirculate the flow when the compressor is performing under low mass flows. In the CHT model the gas will from the heat transfer in the system pick up heat energy leading to this behaviour. When the mass flow is increased, the ported shroud is not needed since the gas naturally can flow downstream of the compressor without stalling. This is why the behaviour of the CHT model starts to follow the trend of the adiabatic model for increasing mass flow rates.

The static pressure in Fig. 48a and 35a both shows a similar behaviour. The pressure on the ported shroud is however much higher in the CHT model. This can be explained by the interface height of the ported shroud in the CHT geometry that is almost the half of the height of the adiabatic geometry. The smaller opening of the interface will produce an increase in pressure over the interface.

By comparing the static pressure at the wheel inlet and wheel outlet seen in Figs. 49a and 49b, the pressure is similar at the wheel inlet. In the CHT model, the static pressure at the outlet is however increased, meaning that the CHT model computes a pressure ratio over the compressor wheel that is higher compared to the adiabatic model.

The total pressure at the outlet is following the same characteristic between the models, as seen in Fig. 50a. In Fig. 50b, the total pressure at the wheel outlet shows a qualitative difference, but follows the same trend.

The behavior at the ported shroud is once again deviating for lower mass flows, but straightens out when the flow increases. This is seen in Fig. 51a and
can also be justified by the geometrical differences at the ported shroud interface.

At the wheel inlet, the total pressure looks to differ a lot. This is not true and can be explained by the normalized plot that gives a bad scale. The function values of the total pressure at the wheel inlet is in fact fairly close between the two models. This means, by looking at Figs. 50b and 51b that the total pressure ratio over the compressor wheel is lower for the CHT model.

Overall based on efficiency and pressure ratio, the CHT model shows a characteristic fairly close to the adiabatic model and experimental data but underpredicts when approaching lower mass flows. These underpredictions may depend on the geometrical differences.

5.3.1 Future work

When conducting CHT simulations including MRF, there are recommendations of letting every MRF solid domain be encased by a fluid domain that should be rotating along with the solid at the same rotational velocity. From earlier work at Scania, it has been found out that the positioning of the wheel domain interfaces have a significant effect on parameters such as efficiency. For that reason, the wheel fluid domain is starting at the same axial distance as the traditional model is, and the same goes for the wheel outlet to diffuser interface. A problem with this approach is that for the CHT model, a shaft and shaftnut domain is also included in the system. Both of these domains are starting inside the inlet fluid domain, which is set as a stationary domain. Since the shaft and shaft nut is rotating with the same rotational velocity as the compressor wheel, those domains should also be encased by a rotating fluid, which is not the case. If this has a negative influence on parameters such as efficiency would be interesting to investigate.

The time scale used in the solver is based on recommendations from an Ansys webinar in the subject of CHT, that states that an automatic time scale should be used [6]. In the finishing parts of this thesis, I found an article about CHT modeling, stating that the solid time scale should be set up using the equation:

\[ T_s = \frac{L_s \rho_c C_{p,s}}{\lambda_s} \]  

where \( s \) denotes the solid. Here \( L \) is a characteristic length of the solid, and \( \lambda_s \) the thermal conductivity of the material. Since the CHT simulations is performed with the automatic time scale factors for the fluid and solid of 10 and 100 respectively, it would be of interest to see the difference in the result by using the traditional physical time scale for the fluid, together with eq. (46) for the solid.

It would also be of interest to perform a mesh study, both on fluid and solid domains in the geometry. Especially for the solids, so the assumption of using
a coarse mesh because of the linearity of the heat transfer in solids, can be tested.

The CHT model was also performed with adiabatic walls, for one design point. Because of the time limit, I had no time to perform this CHT model with adiabatic walls for the complete speed line, which would have been interesting to do. Firstly because it should behave more like the adiabatic traditional model. Secondly because the CHT model had geometrical differences and could therefore not fully be compared to the adiabatic model, since the differences in the result is dependent on the geometry.

To perform a transient simulation to compare against adiabatic model and experimental data would also be of interest, to see if a time dependent solutions have better prediction.

Finally, to perform a more comprehensive experimental study with more data for evaluation of the CHT model to better understand if it is a good model.
References


