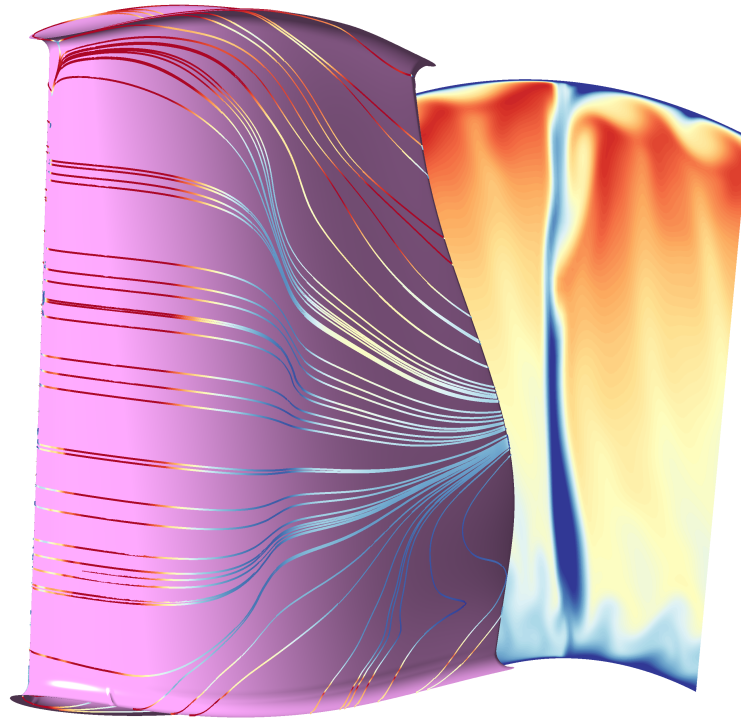




CHALMERS
UNIVERSITY OF TECHNOLOGY



CFD Evaluation of a Turbine Rear Structure with an Integrated Heat Exchanger

A comparative study between measured data and CFD simulation results

Master's thesis in Engineering Mathematics and Computational Science & Sustainable Energy Systems

Elias Chalhoub & August Skoglund

DEPARTMENT OF Mechanics and Maritime Sciences

CHALMERS UNIVERSITY OF TECHNOLOGY

Gothenburg, Sweden 2023

www.chalmers.se

MASTER'S THESIS 2023

CFD Evaluation of a Turbine Rear Structure with an Integrated Heat Exchanger

A comparative study between measured data and CFD simulation
results

Elias Chalhoub & August Skoglund



Department of Mechanics and Maritime Sciences
Division of Fluid Dynamics
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2023

CFD Evaluation of a Turbine Rear Structure With an Integrated Heat Exchanger
A comparative study between measured data and CFD simulation results
Elias Chalhoub & August Skoglund

© Elias Chalhoub & August Skoglund, 2023.

Supervisor: Sonny Andersson, GKN Aerospace
Supervisor: Isak Jonsson, Chalmers
Examiner: Valery Chernoray, Chalmers

Master's Thesis 2023
Department of Mechanics and Maritime Sciences
Division of Fluid Dynamics
Chalmers University of Technology
SE-412 96 Gothenburg
Telephone +46 31 772 1000

Cover: The OGV is presented with streamlines depicting the airflow around the blade, along with a contour plot showing the total pressure coefficient downstream of the TRS.

Typeset in L^AT_EX
Printed by Chalmers Reproservice
Gothenburg, Sweden 2023

CFD Evaluation of a Turbine Rear Structure With an Integrated Heat Exchange
A comparative study between measured data and CFD simulation results
Elias Chalhoub, August Skoglund
Department of Mechanics and Maritime Sciences
Chalmers University of Technology

Abstract

The expanding aviation industry is currently experiencing a substantial transition in order to become more sustainable. A more environmentally friendly alternative to traditional jet fuel is liquid hydrogen, an option that has been investigated by GKN Aerospace among others. Employing liquid hydrogen imposes many engineering challenges, one of which is the design on the turbine rear structure part (TRS) of the engine. This section of an aeroplane engine is of special interest to GKN as it is one of their main products. As liquid hydrogen has to be kept at low temperatures in order to remain in its liquid state, a state most suited for use in aircraft due to the low energy density of gases, and as the air exiting an engine is typically very hot this opens the possibility, if not necessity of heat exchange between the fluids. A model of a heat exchanger that can be used for this purpose was constructed by GKN and tested at Chalmers university of technology where the hydrogen was replaced by heated water. A patent for the heat exchanger model had been filed but not obtained during the thesis work which meant that the exact geometry of it was omitted from the thesis.

In this thesis the tests at Chalmers were used in order to perform CFD simulations of the turbine rear structure with an integrated heat exchanger. The results obtained from the CFD were then validated by comparing turbulence models and boundary condition specification methods after which the results were compared to the test data. The validity of the used CFD model was confirmed using tests and reasoning although the CFD model was shown to over predict the magnitude of separation zones. The measured outlet temperature of the water was shown to be accurately predicted by the CFD simulations. Additional simulations of a TRS without the heat exchanger were also carried out to highlight the difference between using and not using a heat exchanger. Finally the performance of the heat exchanger was assessed by calculating its effectiveness for the different simulations which gave indications for the optimal air flow through it.

Keywords: Heat Exchanger, CFD, TRS, Liquid Hydrogen, Aeroplane Engine, GKN Aerospace, OGV, high BPR turbofan, Turbulence modelling

Acknowledgements

We would like to express our sincere gratitude to all who have helped and guided us throughout our master thesis both at GKN Aerospace and at Chalmers. We would firstly like to thank our supervisor at GKN, Sonny Andersson for his invaluable guidance and support throughout this project. His expertise, insightful feedback, and encouragement were instrumental in helping us navigate through the challenges we encountered and achieve our goals. We appreciate his dedication, patience, and availability whenever we needed assistance. We express our gratitude to Michal Nitulescu at GKN for his valuable contribution and guidance throughout our work on this project, and for providing us with helpful suggestions for this report. We express our gratitude to Richard Avellan for his commendable efforts in ensuring our comfort during our stay at GKN Aerospace in Trollhättan, and for providing us with all the necessary resources required to accomplish our assigned tasks. Finally, we express our gratitude to GKN for affording us the opportunity to write the master thesis for their organization. This experience has been highly informative, captivating, and enjoyable.

We would also like to thank our supervisor at Chalmers, Isak Jonsson for his guidance throughout the project. His insightful feedback and constructive criticism have been crucial in refining our research and improving its quality.

We would also like to express our gratitude to our desk buddies, David Hanås and Frowin Winkes, who are fellow master's students at Chalmers and who have been working on their own thesis projects at GKN. We extend our appreciation to them for their insightful and entertaining discussions, both related and unrelated to our master's thesis.

Finally, we express our gratitude to our families and friends for their unwavering support and encouragement throughout our thesis.

Elias Chalhoub, August Skoglund, Gothenburg, 06 2023

List of Acronyms

Below is the list of acronyms that have been used throughout this thesis listed in alphabetical order:

1D	One dimensional
2D	Two dimensional
BL	Boundary layer
BPR	Bypass Ratio
CC	Combustion chamber
CFD	Computational Fluid Dynamics
DS	Downstream
EES	Engine Exit Structure
FC	Flow Coefficient
HPC	High pressure compressor
HPT	High pressure turbine
HX	Heat Exchanger
LH_2	Liquid hydrogen
LPT	Low Pressure Turbine
MHP	Multi holed probe
NGV	Nozzle Guide Vane
N-S	Navier-Stokes
NTU	Number of transfer units
OGV	Outlet Guide Vane
PS	Pressure Side
RANS	Reynold averaged Navier-Stokes
Re	Reynolds Number
RPM	Rotations Per Minute
SS	Suction Side
SST	Shear stress transport
TE	Trailing Edge
TRS	Turbine Rear Structure
US	Upstream

Nomenclature

Below is the nomenclature of indices, sets, parameters, and variables that have been used throughout this thesis.

Symbols

μ	Dynamic viscosity [Pa · s]
ν	Kinematic viscosity [m^2/s]
ρ	Density [kg/m^3]
C_P	Static pressure coefficient
C_{P0}	Total pressure coefficient
c_p	Heat Capacity [kJ/kgK]
L	Characteristic Length [m]
U	Blade Speed [m/s]
V	Velocity [m/s]
$\dot{m}$	Mass flow [kg/s]
α	Swirl angle [°]
β	Yaw angle [°]
P	Pressure [Pa]
C	Specific heat capacity Rate [kJ/Ks]
ϵ	Effectiveness
q	Heat transfer rate [W]
Q	Heat [J]
t	Time [s]
T	Temperature [K]
x	Spatial dimension [m]
τ	Shear stress [Pa]
g	Gravitational acceleration [m/s^2]

h	Enthalpy [J]
λ	Mean free path [m]
S	Source term
∂	Partial derivative
ξ	Generic variable
δ_{ij}	Kroenecker delta
k	Turbulent kinetic energy [m^2/s^3]
ε	turbulent dissipation rate [m^2/s^4]
y^+	y plus
u_*	Characteristic velocity scale
C_f	Friction coefficient
e	Surface roughness
D	Diameter [m]
y	Distance [m]
ω	Specific Dissipation [1/s]
γ	intermittency factor
Re_θ	Momentum-thickness Reynolds number

Subscripts

θ	Tangential
r	Radial
s,ref	Static Reference
t,ref	Total Reference
x	Axial
c	Cold
h	Hot
in	Inlet
o	Outlet
max	Maximum
min	Minimum
i	Dimension
j	Dimension (other than i)
tot	Total

M	Momentum
E	Energy
w	Wall

Superscripts

$\overline{x}$	Mean
'	Fluctuation

Contents

List of Acronyms	ix
Nomenclature	ix
List of Figures	xix
List of Tables	xxiii
1 Introduction	1
1.1 Background	1
1.2 Aim	2
1.3 Limitations	2
2 Theory	5
2.1 The High BPR Turbofan Engine	5
2.1.1 TRS	6
2.1.2 Operating Points	8
2.2 Heat Exchange	8
2.2.1 Hydrogen Propelled Aircraft	9
2.2.2 Heat Exchange Using Hydrogen	9
2.2.3 HX Effectiveness	10
2.3 Equations of State	11
2.4 Governing Equations	11
2.4.1 Navier-Stokes	11
2.5 Turbulence Modeling	12
2.5.1 RANS	13
2.5.2 The Boussinesq Assumption	15
2.5.3 Two-equation models	16
2.5.4 The $k - \varepsilon$ Model	17
2.5.5 Shear Stress Transport	18
2.5.6 Transition SST Model	20
2.5.7 Turbulence Constants	21
2.6 Near Wall Modelling	22
3 Methods	25
3.1 Experimental Test Facility	25
3.1.1 Implementation of the HX	26

3.1.2	Instruments	27
3.2	Studied Cases	27
3.3	Meshing	28
3.3.1	Surface Mesh	28
3.3.2	Volume Mesh	30
3.4	Numerical Setup	32
3.4.1	Boundary Conditions	32
3.5	Solver Settings	35
3.6	Generic Procedure for Setting up Case in <i>CFX</i>	36
3.7	Other Tests	37
3.8	Gathering of Results	39
4	Results	41
4.1	Inlet Specification	41
4.1.1	Inlet Dimension	41
4.1.2	Inlet Extrapolation	43
4.2	Impact of Turbulence Model	44
4.3	Near wall Modeling	48
4.4	Mesh Independence Study	49
4.5	Cases 6 & 7	50
4.6	Cases 1-3	55
4.7	Split Variation	58
4.8	Comparison of Simulations for Different Geometries	60
4.9	Effectiveness of the HX	62
5	Discussion	65
5.1	Inlet Specification	65
5.2	Turbulence Modelling	66
5.3	Near Wall Modelling	68
5.4	Mesh Independence Study	68
5.5	Cases 6 & 7	69
5.6	Cases 1-3	70
5.7	Split Variation	71
5.8	Comparison of Simulations for Different Geometries	72
5.9	Effectiveness of HX	73
5.10	Possible Improvements and Further Research	74
5.10.1	Further Turbulence Modelling Details	74
5.11	Compilation of Results	74
5.11.1	Data for Reynolds Number = 350000	74
5.11.2	Further Investigations	75
6	Conclusion	77
	Bibliography	79
A	Appendix 1	I
A.1	Experimental data	I

A.2	Case 1 Re235FC0.69	III
A.3	Case 2 Re235FC0.73	V
A.4	Case 3 Re235FC0.61	VII
A.5	Case 6 Re235FC0.69, reduced split	IX
A.6	Case 7 Re235FC0.73, reduced split	XII
A.7	Different Geometries	XIV
A.8	1D inlet, Re235FC0.69	XVI
A.9	y^+	XVII

List of Figures

2.1	Schematic of a conceptual BPR engine with a cut-out of the turbine section	5
2.2	Schematic visualizing the general frame of reference	6
2.3	CAD geometry of the entire TRS as well as a single OGV	7
2.4	Front, side and back view of a thick vane	8
2.5	Schematic of engine with components, notably a recuperator. HPC: High-pressure compressor, CC: Combustion chamber, HPT: High-pressure turbine, LPT: Low-pressure turbine, OGV: Outlet guide vane, LH_2 : Liquid hydrogen	9
2.6	Visualisation of the decomposition of velocity into a mean and a fluctuating part	13
2.7	Friction coefficients for different empiric expressions	23
3.1	Schematic overview of the rig at Chalmers with an integrated heat-exchanger, Provided by Isak Jonsson	25
3.2	Description of the rigg downstream of the contraction, showing the integrated HX with its water circuit	26
3.3	Surface mesh for the hub and the shroud	29
3.4	Blade surface mesh, both at PS and SS	29
3.5	Structured volume mesh for the TRS with a total of 3097654 hexas	30
3.6	This figure represents a zoomed in picture of the inflation surrounding the shroud	31
3.7	Zoomed in picture of the inflation used around the shroud	31
3.8	Comparison of generic contours for 2D and 1D inlets	32
3.9	Plot showing the actual size of the inlet and the area measured by the probe	33
3.10	Comparison of the different modes of extrapolating the boundary layers shown together with experimental values	34
3.11	Domain setup for the TRS showing some of the BCs	35
3.12	General procedure for <i>CFX</i>	37
3.13	Domain for the extended OGV	38
3.14	Figure showing the location where wake results will be gathered	39
3.15	Figure showing an entire DS profile obtained in <i>CFX</i> and the area measured by the DS MHP	40
4.1	The figures are showing contour plots for Cp_0 both upstreams and downstreams, different inlets (2D and 1D) used	42

4.2	Figures showing a comparison of tangentially averaged DS values between test data, CFX data using a 1D inlet and CFX data using a 2D inlet	42
4.3	Figures showing a comparison of wake data values between test data, CFX data using a 1D inlet and CFX data using a 2D inlet	43
4.4	Figures showing a comparison of wake data values between test data, CFX data using a 1D inlet and CFX data using a 2D inlet	43
4.5	The figures are showing contour plots for Cp_0 both upstream and downstream of the OGV. Three different extrapolation schemes has been used and is highlighted in the figure	43
4.6	The differences between the extrapolation schemes is shown in these figures. Both figures are illustrating circumferential average axial velocities at both US and DS. The left figure is showing the axial velocity upstreams and the right figure the axial velocity downstreams. The different extrapolation schemes are highlighted with a specific color.	44
4.7	The contour plot to the left is illustration Cp_0 downstream where the Transition SST turbulence model has been used. The right contour plot is illustrating the same variable while the turbulence model is instead replaced with the $k - \varepsilon$ model	44
4.8	This figure is showing C_{P_0} at different blade spans for two different turbulence models, namely the SST Transition model and the $k - \varepsilon$ model	45
4.9	Streamlines of the airflow at the OGV for the Transition SST $k - \omega$ model	45
4.10	Streamlines of the airflow at the OGV for for the $k - \varepsilon$ model	45
4.11	These contour plots display Cp_0 for two distinct scenarios using the standard SST $k - \omega$ model, where the a_1 coefficient has been assigned two different values	46
4.12	Contour plot of the blending function F_1 for a SST transition simulation	46
4.13	Contour plot of the blending function F_2 for SST $k - \omega$ simulations with different values of a_1	47
4.14	Figure illustrating contour plots of C_{P_0} , case 6 for both test and simulations	47
4.15		48
4.16	y+ values at the TRS walls	48
4.17	The figure is showing contour plots over the total pressure coefficient, C_{P_0} downstream for three different refinements on the mesh.	49
4.18	The figure above is showing C_{P_0} for three different refinements of the mesh at a blade span of 50%. The graph below is instead showing the swirl.	49
4.19	Zoomed in figure around 0 for figure 4.18	50
4.20	tangentially averaged data for US, case 6	51
4.21	tangentially averaged data for US, case 7	51
4.22	Contour plots for Cp_0 at the outlet of the TRS. The figure is presenting results for both measured data and data from simulation results, cases 6 and 7	52
4.23	tangentially averaged data for DS, case 6	52

4.24	tangentially averaged data for DS, case 7	53
4.25	C_{p0} data, case 6 and 7	53
4.26	Swirl data, case 6 and 7	54
4.27	Tangentially averaged DS plots for radial velocity and yaw, case 6 and 7	54
4.28	Circumstantially averaged data for DS, case 1	55
4.29	Contour plots for C_{p0} at the inlet of the TRS. The figure is presenting results for both measured data and data from simulation results, cases 1-3	56
4.30	Contour plots for C_{p0} at the outlet of the TRS. The figure is present- ing results for both measured data and data from simulation results, cases 1-3	56
4.31	Figure showing streamlines and velocity contours around the blade for cases 1, 2 and 3	56
4.32	Wake data for swirl and C_{p0} downstreams for case 1 where $Re=235k$ and $F_c=0.69$	57
4.33	Streamlines of the airflow at the OGV, suction-side	57
4.34	Streamlines of the airflow at the blade, pressure-side	58
4.35	C_{min} as a function of split	58
4.36	Split as a function of the HX back pressure	59
4.37	DS profiles for different geometries	60
4.38	DS profiles for experiments and CFD, both only using the TRS ge- ometry and the entire domain (TRS + HX) for cases 1-3	60
4.39	Streamlines of the airflow at the OGV for the geometry containing only the OGV	61
4.40	Streamlines of the airflow at the OGV for the geometry containing OGV and HX, Water enabled	61
4.41	Water outlet temperature comparison	63
4.42	Effectiveness plotted against split	64
A.1	cp_0 for the test data, US, (set to scale of -0.4 to 0.5)	I
A.2	Swirl for the test data, US, (set to scale of -30° to 30°)	II
A.3	cp_0 for the test data, DS, (set to scale of -0.4 to 0.5)	II
A.4	Swirl for the test data, DS, (set to scale of -30° to 30°)	III
A.5	cp_0 at DS	III
A.6	tangentially averaged data for US and DS, case 1	IV
A.7	Streamlines of the airflow at the OGV for case 1	IV
A.8	Wake data, case 1	V
A.9	cp_0 at DS	V
A.10	tangentially averaged data for US and DS, case 2	VI
A.11	Streamlines of the airflow at the OGV for case 2	VI
A.12	Wake data, case 2	VII
A.13	cp_0 at DS	VII
A.14	tangentially averaged data for US and DS, case 3	VIII
A.15	Streamlines of the airflow at the OGV for case 3	VIII
A.16	Wake data, case 3	IX
A.17	cp_0 at DS	IX

A.18 tangentially averaged data for US and DS, case 6	X
A.19 Wake data, case 6	XI
A.20 Streamlines of the airflow at the OGV for case 6	XI
A.21 cp_0 at DS	XII
A.22 tangentially averaged data for US and DS, case 7	XII
A.23 Wake data, case 7	XIII
A.24 Streamlines of the airflow at the OGV for case 7	XIII
A.25 tangentially averaged data for US and DS, only OGV case	XIV
A.26 tangentially averaged data for US and DS, case with TRS + HX and water enabled	XV
A.27 Streamlines of the airflow at the OGV for radial inlet	XV
A.28 cp_0 at DS	XVI
A.29 tangentially averaged data for US and DS, radial inlet	XVI
A.30 Wake data, radial inlet	XVII
A.31 Streamlines of the airflow at the OGV for radial inlet	XVII
A.32 y^+ values at the hub	XVII
A.33 y^+ values at the shroud	XVIII
A.34 y^+ values at the OGV, PS and SS	XVIII

List of Tables

3.1	Characteristics for the studied cases	27
3.2	Availability and status of the US and DS profiles for the different cases	28
3.3	Statistics for the mesh parts	32
3.4	Various constants used for turbulence modelling in <i>CFXpre</i>	36
4.1	Pressure outlet setting and resulting splits	50
4.2	The presented table shows the back pressure applied in the simulation to achieve inlet profiles that correspond to the experimental data for specific cases.	55
4.3	Data for the cases where the split was varied by changing back pressure.	59
4.4	Solution time for various simulations	61
4.5	Air temperatures from CFX	62
4.6	Both the inlet and outlet temperatures for the water is tabled here. The table presents values both from measurments and from simulations	62
4.7	Mass flow data from CFX	63
4.8	The HX effectiveness for the different cases can be seen from this table together with a averaged values for case 1-3 and 6-7 separately	63

1

Introduction

1.1 Background

The aviation industry is experiencing continuous growth with a significant increase in the number of flights each year. As a result, sustainability has become a critical issue, a part of which is the green new deal proposal by the EU. Currently, the primary cause of airplane pollution is the use of fossil fuel-driven gas turbines. To address this issue, several ideas have been proposed to transform the aviation industry and make it more environmentally friendly. These include concepts such as electrifying airplanes and replacing fossil fuels with eco-friendly alternatives [1]. GKN Aerospace recognizes liquid hydrogen as a suitable pathway towards achieving future carbon neutrality. Engines fueled by liquid hydrogen are regarded as a long-term solution for transforming the commercial aviation industry into a sustainable and environmentally responsible sector.

The revision from the classical use of fossil fuels to green fuels such as hydrogen in airplane engines entails further research in new technology's which is a necessity to evaluate the impact and changes that the hydrogen-fuel imposes. For instance, hydrogen has a comparatively lower volumetric energy density. As a consequence, the storage of hydrogen fuel requires different methods compared to traditional fuels. Moreover, there is a desire to store hydrogen fuel in a liquid state due to its high specific energy content, which presents a significant challenge in itself. Storing hydrogen as a liquid in airplanes implies that the hydrogen is cooled, compressed hydrogen requires thick and heavy walls of the fuel tank which is less desired [2]. GKN's research includes exploring the possibility of storing hydrogen in a cooled liquid state, which opens up a vast area for research. Due to the thermal properties of hydrogen it is of interest to GKN to investigate the possibilities of heat exchange between hydrogen and the combustion product, more specifically using the hot combustion product at the outlet of the engine to pre-heat the fuel before entering the combustion chamber. This pre-heating would also reduce unwanted material stresses caused by thermal gradients in the combustion chamber

The performance and environmental requirements for aeroplane engines increases as seen in "Flightpath 2050" set out by the EU which, among other things, includes a reduction in CO_2 of 75%, a reduction in NO_x of 90% and a reduction of 65% in noise emissions by 2050 [3]. These higher standards for aircraft give engineers opportunities to continue to develop engines which can manage the demands. One

of the emerging trends is turbofan engines with a high bypass ratio (BPR) which can be highly efficient [4]. These high BPR turbofans imposes high demands on the low pressure turbine (LPT) that drive the fans. Turbines with higher demand tend to produce more swirl at the exit rotor and since axial flow is desired at the exit of the aeroplane in order to maximize thrust outlet guide vanes (OGV) are imposed in order to make the flow more axial. The characteristics of a LPT and OGV system has been studied at Chalmers by building a test rig which produces relevant Reynolds number for air at flight condition and then letting it pass a LPT and OGV [5; 6; 7; 8; 9; 10]. A more detailed explanation of the test rig is presented under the "Methodology" section below. To further improve the efficiency of the engine proposals to use heat exchangers in order to capture some of the heat in the exhaust gas which otherwise would be released to the atmosphere and using it to preheat fuel has been made [11]. There are several possible heat exchanger configurations for possible integration into a hydrogen engine [11] but further studies are needed to evaluate their performance.

1.2 Aim

The gas turbine engines are intended to use stored cooled hydrogen in airplanes as fuel. The hydrogen is desired to be preheated before combustion in order to achieve a higher thermodynamic performance of the engine. The aim of the thesis is to investigate the capability's of an-already designed heat-exchanger using simulation tools. Further on CFD simulation will also be performed using different simulation setups in order to compare the simulation results with results given from tests executed on the heat-exchanger at Chalmers. A simulated environment has been set up at Chalmers so that the heat-exchanger could be developed and tested, the numerous tests that have been conducted at Chalmers will also be included in the thesis were evaluation of the test results will be assessed. The end goal is to validate numerical models and also evaluate the performance of the heat exchanger, especially with focus on the external side (the air side) using both results from the previously done experiments as well as the CFD simulations.

1.3 Limitations

The collected experimental data that will serve as the basis of the CFD simulations is from a test rig which uses air and water rather than flue gases and hydrogen which would be more correct to how it is supposed to look in an aeroplane. This means that only the hot side of the real heat exchanger is studied in this project, leaving the behavior of the cold side, hydrogen, unstudied. The limiting factor for heat transfer will however be air as both water and hydrogen have higher heat transfer coefficients than air laving the air as the fluid of highest interest in this project

Moreover the heat exchanger model has not been constructed to accommodate flow of water in the entire model, only a portion will actually have water running and thus phenomena that might occur when the whole capacity of the heat exchanger

is used is not investigated, although periodicity has been taken into account when designing the experiments. Due to the expensive nature of the tests (both monetary and timewise) there will only be data for a certain number of running conditions specified by Reynolds number and flow coefficient which represent a few operating points, more operating points would naturally give a more stable foundation for the CFD analysis. The study does however includes operation conditions typical for a TRS at take-off and cruise. The tests will also have concluded before the CFD simulations meaning that observed phenomena in CFD will have to be strictly derivative of the test.

The specific design of the heat exchanger (HX) will not be divulged in the report as a patent from GKN is pending and will not be secured before the due date of the master thesis, making the detail of the HX geometry presented in the thesis limited.

2

Theory

2.1 The High BPR Turbofan Engine

There are several different types of engine designs for aircraft, although the most common for long distance passenger aircraft is the turbofan engine where the air entering the engine is split into two different streams, one "hot stream" entering the core of the engine and one "cold stream" flowing around the core only traversing the fan before exiting the engine. The ratio of these streams is the bypass ratio (BPR);

$$BPR = \frac{\dot{m}_{air,cold}}{\dot{m}_{air,hot}} \quad (2.1)$$

Design of turbofan engines strives to increase the BPR as the desired thrust is produced from the cold stream which has led to the design of "high BPR turbofan engines" of which a schematic is presented below;

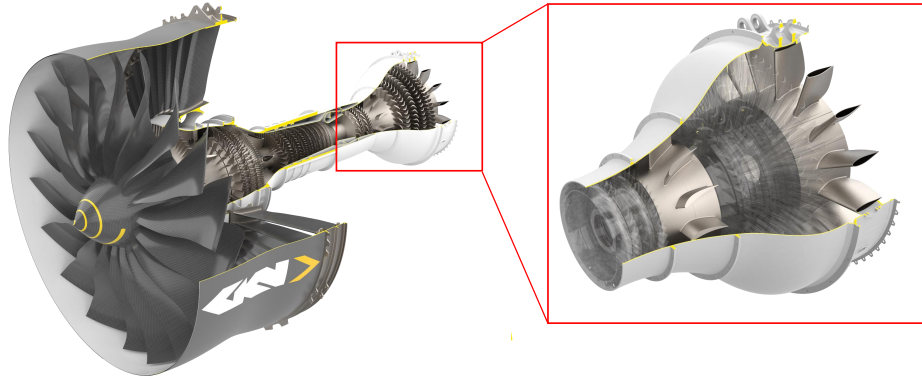


Figure 2.1: Schematic of a conceptual BPR engine with a cut-out of the turbine section

In figure 2.1 the entire engine is shown as well as a cut-out of the turbine section where the turbine rear structure is the rightmost metallic part. Having a high BPR implies increased size of the fan blades which changes the nature of the LPT as well as the fan is driven by the LPT. There are two different approaches to the high

BPR turbofan engine, having a geared fan or an un-geared fan [10]. Using a geared turbofan entails decoupling the rotation speed of the fan and the LPT enabling the LPT to rotate substantially faster than the fan which means that fewer LPT stages are required for the same engine performance. The integration of such a gear does however pose difficulties that arise due to the complexity of such an apparatus, which means that both geared and un-geared engines are present as of today. This variation implies a high variance of LPT operation and design which in turn means that the flow exiting the core of the engine can be vastly different in terms of several qualities such as the velocity. Since the velocity at the outlet of the high BPR engine may vary, the swirl, which is the ratio of tangential velocity to axial velocity ($\alpha = \tan^{-1}\left(\frac{V_\theta}{V_x}\right)$), also varies. Another quantity of note is the yaw angle which is similar to the swirl but using the radial rather than the tangential velocity, $\beta = \tan^{-1}\left(\frac{V_r}{V_x}\right)$. The coordinates that will be used in this thesis are shown in figure 2.2, where $\mathbf{X}$ is denotes the axial direction, θ the tangential and $\mathbf{r}$ the radial.

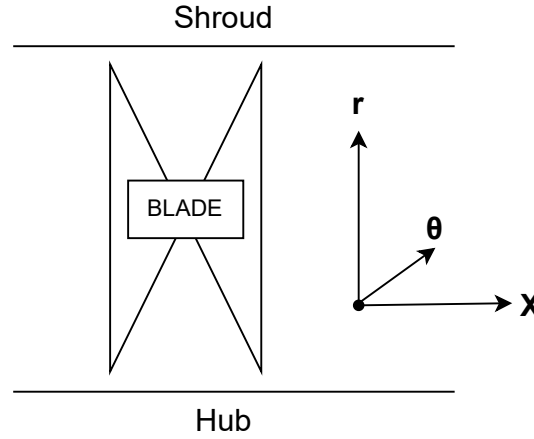


Figure 2.2: Schematic visualizing the general frame of reference

2.1.1 TRS

A Turbine rear structure (TRS) is an engine component placed downstream of the LPT at the outlet of an engine. An overview of the OGVs situated in the TRS is presented in figure 2.3, note that this only shows the blade portion of the TRS and not the surrounding casing which holds significant structural components.

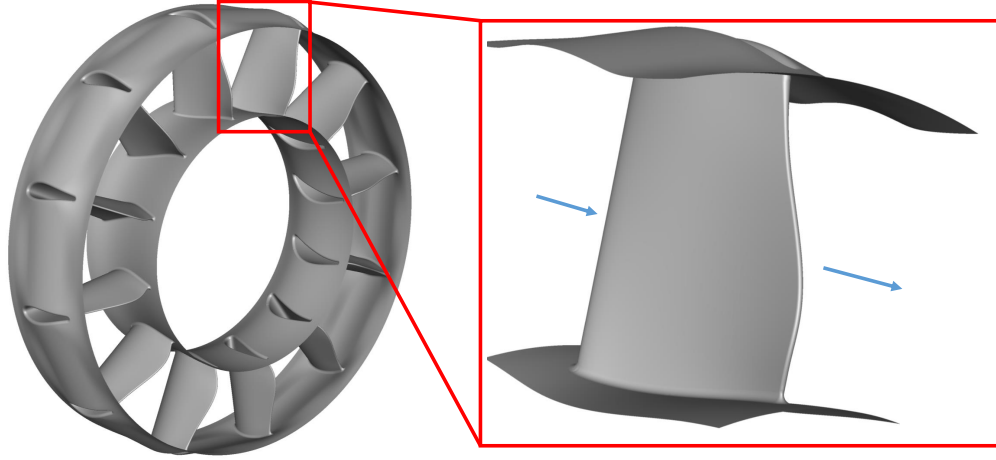


Figure 2.3: CAD geometry of the entire TRS as well as a single OGV

This component is of structural importance as it holds bearings for the LPT [8] but the central role of the TRS is to guide the flow of air exiting the engine. The TRS is designed to reduce the amount of tangential flow out of the engine and maximize the axial flow to increase the thrust delivered by the core air stream. To accomplish this redirection of the flow OGVs are employed which reduce the tangential components of the flow forcing the flow in the axial direction. As the amount of swirl that the TRS has to handle differs depending on the engine operating conditions, which gives different operating points for the OGV, yielding engineering challenges for this component. Different geometries for these OGVs have been tested [8] including thicker vanes able to accommodate oil pipes within them. These thicker vanes are the vanes used in the experiments utilized in this thesis. In order to measure the pressure distribution at different points in the TRS the non-dimensional static and total pressure coefficients C_p and C_{p0} defined below are used;

$$C_p = \frac{P_s - P_{s,ref}}{P_{t,ref} - P_{s,ref}} \quad (2.2)$$

$$C_{p0} = \frac{P_t - P_{t,ref}}{P_{t,ref} - P_{s,ref}} \quad (2.3)$$

The TRS is an annular structure but in order to save computational time only a twelfth, corresponding to a singular OGV, will be studied. A schematic of a singular thick vane along with the hub and shroud outlines is shown in 2.4 below and the blade profiles at the hub, the midspan and the shroud is displayed in figure ??.

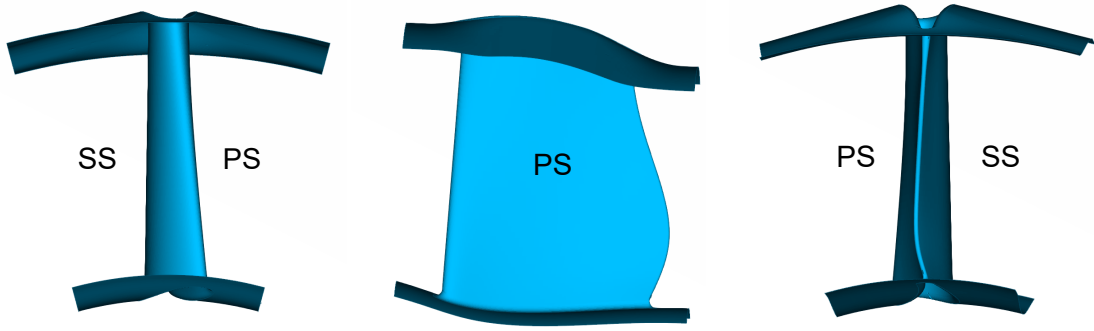


Figure 2.4: Front, side and back view of a thick vane

Figure 2.4 also shows the suction side (SS) and the pressure side (PS) of the OGV blade, the pressure side is the side of the blade experiencing the highest pressure as the flow impinges on it directly where as the suction side experiences lower pressure and higher velocity as the air flow is suctioned over it.

2.1.2 Operating Points

As aircraft are expected to operate at a variety of operating conditions it is of interest to study the TRS with integrated heat exchanger at several operating points. These different operating points can be represented using two dimensionless numbers, the Reynolds number (Re) and the flow coefficient (FC). The Reynolds number is the relation between inertial and viscous forces, defined as

$$Re = \frac{\rho V L}{\mu} = \frac{v L}{\nu} \quad (2.4)$$

As the geometry of the test rig will be constant and the variation in dynamic viscosity of air is small Re will essentially indicate the speed of the aircraft to which the engine would be mounted. The Re of a TRS in a typical aircraft ranges between 100.000 and 600.000 [12]. The definition of the flow coefficient is

$$FC = \frac{V_x}{U} \quad (2.5)$$

which means that for a constant axial velocity, and constant Re, the FC indicates different blade speeds, or swirl as the fan performance dictates the swirl of the flow exiting the LPT, which is caused by a faster rotation of the fan usually seen in off performance points such as take off.

2.2 Heat Exchange

This section discusses heat exchange in general terms and heat exchange between air and water or hydrogen in particular. The theory presented in this section includes the basis for evaluating the HX used in this thesis.

2.2.1 Hydrogen Propelled Aircraft

The prospect of utilizing hydrogen as fuel for aircraft has been investigated by engineers for several years, the soviets having tested a hydrogen fueled engine in 1988 [13]. Replacing traditional aircraft fuel with hydrogen introduces several engineering problems as the properties of hydrogen is different than that of aircraft fuel such as JetA as the evaporation temperature of hydrogen is $-260\text{ }^{\circ}\text{C}$ at atmospheric pressure. Using liquid hydrogen as fuel is generally desired as the energy content per volumetric unit is much higher for liquids than for gas (the volumetric energy content for liquid hydrogen is already lower than that of JetA) which is significant for aircraft design [13]. Maintaining the hydrogen in a liquid state imposes high demands on the aircraft equipment but also opens the possibility of heat exchange with the hot air flow typically present in aircraft engines.

2.2.2 Heat Exchange Using Hydrogen

Other parts of the H2JET project indicate that hydrogen should be injected into the combustion chamber at around 500 K . This requires a significant amount of heating for the hydrogen. To achieve this, a heat exchanger (HX) can be incorporated in various ways, such as utilizing the exhaust air from the engine, which would otherwise be released into the colder environment and waste the heat energy. This mode of heat exchange between the air flowing through the core and the fuel is denoted as recuperation and has the greatest potential for this type of heat exchange [14]. A schematic showing an aircraft engine with typical components as well as a recuperator for pre-heating liquid hydrogen is presented in figure 2.5, note that this is only a schematic showing how LH_2 can be pre-heated and not the exact location of the recuperator. The location of this component is of importance but outside of the scope of this report due to the pending patent.

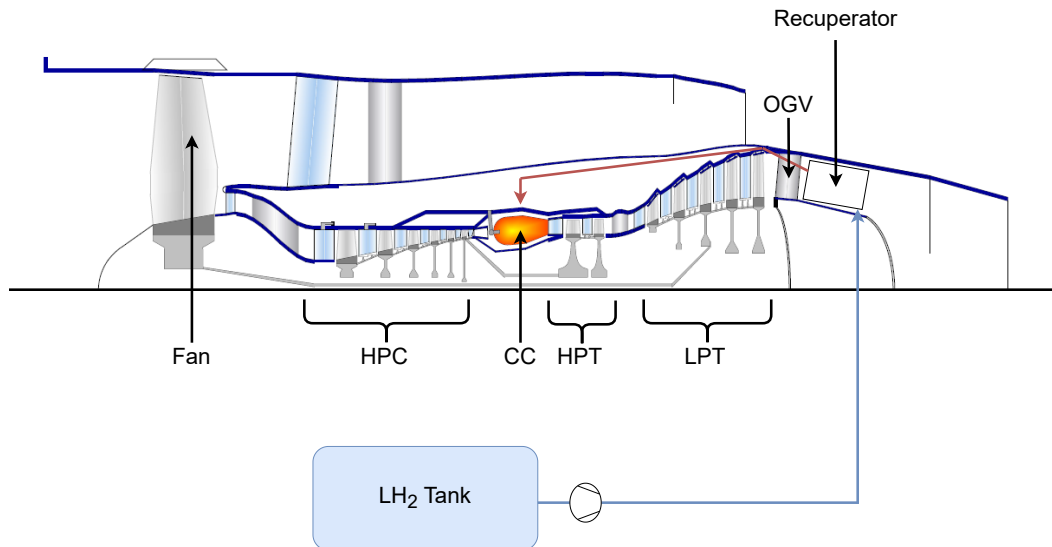


Figure 2.5: Schematic of engine with components, notably a recuperator. HPC: High-pressure compressor, CC: Combustion chamber, HPT: High-pressure turbine, LPT: Low-pressure turbine, OGV: Outlet guide vane, LH_2 : Liquid hydrogen

The HX studied in this thesis will be a recuperator as defined above.

2.2.3 HX Effectiveness

The *Effectiveness*-NTU (number of transfer units) or the NTU method is a method for analyzing and designing heat exchangers. The effectiveness, ϵ , of a heat exchanger is a dimensionless parameter that represents the ratio of heat transfer to maximum possible heat transfer in a heat exchanger. The maximum possible heat transfer rate, q_{max} , is the theoretical maximum amount of heat that can be transferred from one fluid to another under ideal conditions and is limited by the physical properties of the fluids and the temperature difference between them [15]. q_{max} can be expressed as the following;

$$\begin{cases} \text{If } C_c < C_h : & q_{max} = C_c(T_{h,in} - T_{c,in}) \\ \text{If } C_h < C_c : & q_{max} = C_h(T_{h,in} - T_{c,in}) \end{cases} \quad (2.6)$$

Where $T_{h,in}$ is the inlet temperature for the hot fluid, $T_{h,o}$ is the outlet temperature for the hot fluid and $T_{c,in}$ is the inlet temperature for the cold fluid. From equation 2.6 the parameters C_c and C_h represent the heat capacity rate for the cold and hot fluid respectively. In the first case where $C_c < C_h$ the cold fluid would experience the larger temperature change and would eventually be heated into the inlet temperature for hot fluid, $T_{h,in}$ [15]. In the second case where $C_h < C_c$ the hot fluid would experience the larger temperature change and would eventually be cooled into the inlet temperature for the cold fluid [15]. Hence q_{max} can be written into the generalized expression,

$$\begin{aligned} q_{max} &= C_{min}(T_{h,in} - T_{c,in}) \\ C_{min} &= \min[C_h, C_c] \end{aligned} \quad (2.7)$$

C_{min} is thus equal to C_c or C_h whichever is smaller. The heat capacity rate is defined as the specific heat capacity, c_p , for the fluid property multiplied with the mass flow-rate and can be expressed as,

$$C_h = \dot{m}_h c_{p,h} \quad C_c = \dot{m}_c c_{p,c} \quad (2.8)$$

The heat exchanger effectiveness can now be defined as the ratio between the actual heat transfer rate q , and the maximum possible heat transfer rate q_{max} ,

$$\epsilon = \frac{q}{q_{max}} \quad (2.9)$$

Expanding equation 2.9 gives the following expressions for the heat exchanger effectiveness;

$$\epsilon = \frac{C_h(T_{h,in} - T_{h,o})}{C_{min}(T_{h,in} - T_{c,in})} \quad \text{or} \quad \epsilon = \frac{C_c(T_{c,o} - T_{c,in})}{C_{min}(T_{h,in} - T_{c,in})} \quad (2.10)$$

The test data from the experiments at Chalmers measured in and out temperature of the water on the hot side, meaning that the first way of calculating effectiveness using equation 2.10 is employed.

It is assumed that no heat is lost during the heat transfer process, all heat lost from the water is absorbed by the air;

$$q_c = q_h \quad \leftrightarrow \quad C_c(T_{c,o} - T_{c,in}) = C_h(T_{h,in} - T_{h,o}) \quad (2.11)$$

Using this equation the outgoing air temperature can be calculated;

$$T_{c,o} = T_{c,in} + \frac{C_h(T_{h,in} - T_{h,o})}{C_c} \quad (2.12)$$

Using the effectiveness obtained from equation 2.10 several metrics of the heat exchanger such as the amount of energy exchanged between the fluids and the overall heat transfer coefficient using the "NTU-method". In order to use this method the configuration of heat exchanger has to be specified and since this is confidential the effectiveness value will be the final detail of heat exchange presented in this thesis.

2.3 Equations of State

The equations of state used to determine fluid properties such as the viscosity (ν_i) and the heat capacity ($c_{p,i}$) is the GERG-2008 equations of state [16] which are available through the "pyfluids" package in *Python*. These equations give various parameters for certain fluids such as water and air (which is assumed to be a combination of oxygen and nitrogen) as functions of temperature and pressure. The GERG-2008 equation have shown to give similar results to the Peng-Robinson equation and for temperature ranges of 90-450 K and pressures up to 35 MPa the equations give accurate values when compared to experimental results [16].

2.4 Governing Equations

This section includes the definition of the fluid equations that are solved and modelled in the CFD program.

2.4.1 Navier-Stokes

The definition of the Navier-Stokes equations are the momentum equations presented below but in this project the central equations are the conservation of mass and momentum and hence these are collectively denoted as the Navier-Stokes equations. The equations presented below are defined using a tensor notation according to [17].

The Continuity Equation

The equation for conservation of mass commonly denoted as the continuity equation

is derived using a control volume and formulation the mass balance "accumulation of mass = mass transport in - mass transport out" [17]. This yields the continuity equation;

$$\frac{D\rho}{Dt} = -\rho \frac{\partial V_i}{\partial x_i} \quad (2.13)$$

Here $\frac{D}{Dt}$ denotes the material derivative $\frac{D}{Dt} = V_i \frac{\partial}{\partial x_i} + \frac{\partial}{\partial t}$. If the flow is assumed to be incompressible, the density is assumed to be constant, 2.13 reduces to

$$\frac{\partial V_i}{\partial x_i} = 0 \quad (2.14)$$

The assumption of incompressibility is applied in this project since the speeds are far below the speed of sound, a Mach number substantially below 0.3 is expected and therefore 2.14 will be used for the conservation of mass.

The Momentum Equation

The momentum equation is derived in a similar manner to the continuity equation, a control volume is defined and then the following momentum balance is written "Rate of accumulation of momentum= Rate of momentum in -rate of momentum out + sum of forces acting of system" [17]. After some mathematical manipulation the resulting momentum equation is the following;

$$\frac{\partial V_i}{\partial t} + V_j \frac{\partial V_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{1}{\rho} \frac{\partial \tau_{ji}}{\partial x_j} + g_i \quad (2.15)$$

In both the equation for continuity and momentum the subscript i denotes the 3 spatial dimensions i=1,2,3 so for example $x_i = \{x_1, x_2, x_3\} = \{x, y, z\}$. In equation 2.15 j denotes a vector element different from the i subscript. If the fluid in question is Newtonian equation 2.15 can be rewritten using the more common form [17];

$$\frac{\partial V_i}{\partial t} + V_j \frac{\partial V_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \nu \frac{\partial^2 V_i}{\partial x_j \partial x_j} + g_i \quad (2.16)$$

The Energy Equation

The final equation solved by the CFD solver is the energy equation, more accurately denoted as the transport equation for total energy. The derivation for this equation will not be presented as it is lengthy and available in other places [18]. The energy equation is thus

$$\frac{\partial(\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \frac{\partial}{\partial x_i}(\rho V_i h_{tot}) = \frac{\partial}{\partial x_i}(\lambda \frac{\partial T}{\partial x_i}) + \frac{\partial}{\partial x_i}(V_i \tau) + V_i S_M + S_E \quad (2.17)$$

where h_{tot} is the total enthalpy defined as $h_{tot} = h + \frac{1}{2}V_i^2$. In the main CFD solver used in this thesis, *ANSYS CFX*, the term $V_i S_M$ related to external momentum sources is neglected.

2.5 Turbulence Modeling

The majority of flows, including the ones dealt with in this thesis are turbulent, meaning that they exhibit properties such as irregularity and diffusivity that are

strenuous to model with accuracy [19]. The Navier-Stokes equations are still possible to solve for turbulent flow but in order to do this the computational mesh must be small as to capture the smallest scales of the turbulent eddies and the time step must also be small to capture the inherent invariance of turbulent flows. To avoid such costly simulations turbulent variables are decomposed into a mean part and a fluctuating part as the following equation [17], which is also graphically illustrated in figure 2.6;

$$V_i = \bar{V}_i + V'_i \quad (2.18)$$

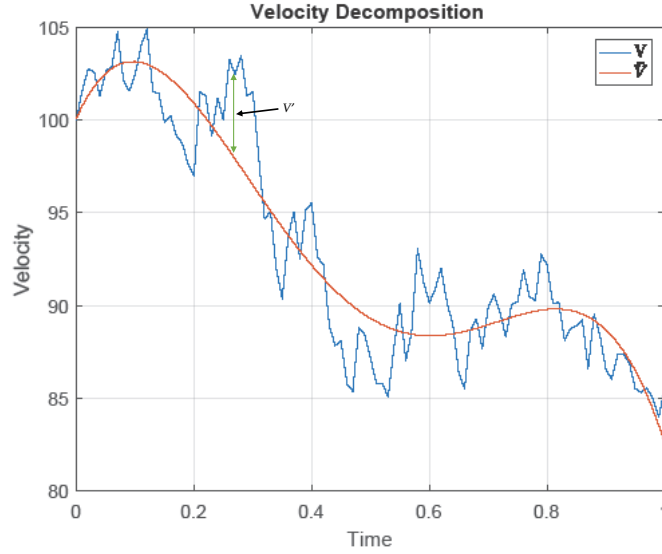


Figure 2.6: Visualisation of the decomposition of velocity into a mean and a fluctuating part

The above is an example showing the decomposition of the (turbulent) velocity into a mean velocity ($\bar{V}_i$) and a fluctuating velocity (V'_i). If this is done to the Navier - Stokes equations, equations 2.14 and 2.16, the resulting equations (omitting the constant gravity term in the momentum equation for brevity) are;

$$\begin{cases} \frac{(\bar{V}_i + V'_i)}{\partial x_i} = 0 \\ \frac{\partial(\bar{V}_i + V'_i)}{\partial t} + (\bar{V}_i + V'_i) \frac{\partial(\bar{V}_i + V'_i)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial(\bar{V}_i + V'_i)}{\partial x_i} + \nu \frac{\partial^2(\bar{V}_i + V'_i)}{\partial x_j \partial x_j} \end{cases} \quad (2.19)$$

2.5.1 RANS

When simulating flows the equations above need to be averaged as to separate the computationally draining turbulent fluctuations from the mean quantities. This averaging can be done in a number of ways, one of which is time averaging. Equation 2.20 below shows the time averaging for a general variable ξ , a function of space and time over the time scale τ .

$$\bar{\xi} = \frac{1}{\tau} \int_t^{t+\tau} \xi(x, \tilde{t}) d\tilde{t} \quad (2.20)$$

After applying this time averaging to the decomposed Navier-Stokes equations they read

$$\begin{cases} \overline{\frac{(\bar{V}_i + V'_i)}{\partial x_i}} = 0 \\ \overline{\frac{\partial(\bar{V}_i + V'_i)}{\partial t}} + \overline{(\bar{V}_i + V'_i) \frac{\partial(\bar{V}_i + V'_i)}{\partial x_j}} = -\frac{1}{\rho} \overline{\frac{\partial(\bar{V}_i + V'_i)}{\partial x_i}} + \nu \overline{\frac{\partial^2(\bar{V}_i + V'_i)}{\partial x_j \partial x_j}} \end{cases} \quad (2.21)$$

The time average of a linear turbulent term is zero by definition, $\overline{\xi'} = 0$ and time averaging a mean term does nothing, $\overline{\bar{\xi}} = \bar{\xi}$. Averaging non linear turbulent terms does not equal to zero as shown in the following example

$$\begin{aligned} \overline{V_i V_j} &= \overline{(\bar{V}_i + V'_i)(\bar{V}_j + V'_j)} = \overline{\bar{V}_i \bar{V}_j + V'_j \bar{V}_i + V'_i \bar{V}_j + V'_i V'_j} = \\ &= \bar{V}_i \bar{V}_j + \bar{V}'_i \bar{V}_j + \bar{V}_j \bar{V}'_i + \overline{V'_i V'_j} = \bar{V}_i \bar{V}_j + \overline{V'_i V'_j} \end{aligned} \quad (2.22)$$

This means that when the terms in equation 2.21 are time averaged the continuity equation becomes

$$\frac{\partial \bar{V}_i}{\partial x_i} = 0 \quad (2.23)$$

and the momentum equation becomes

$$\frac{\partial \bar{V}_i}{\partial t} + \bar{V}_j \frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \overline{V'_i V'_j}}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{P}}{\partial x_i} + \nu \frac{\partial^2 \bar{V}_i}{\partial x_j \partial x_j} \quad (2.24)$$

which can be rewritten as

$$\frac{\partial \bar{V}_i}{\partial t} + \bar{V}_j \frac{\partial \bar{V}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial}{\partial x_j} \left[\bar{P} \delta_{ij} + \mu \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) - \rho \overline{V'_i V'_j} \right] \quad (2.25)$$

The averaging procedure produced above was introduced by Reynolds Which explains the collective name of equations 2.23 and 2.25 as "Reynolds averaged Navier-Stokes" or RANS for short. The RANS equations introduces the term $-\rho \overline{V'_i V'_j}$, alternatively written as $\overline{V'_i V'_j}$ referred to as the Reynolds stress and must be closed in order to solve the equations.

The Reynold stress tensor, $\overline{V'_i V'_j}$, is accurately given by solving a transport equation for it assuming steady, incompressible flow with constant viscosity and density. The procedure for deriving the transport equation for $\overline{V'_i V'_j}$ is given by the following steps:

1. Set up the momentum equation for the instantaneous velocity $V_I = \bar{V}_i + V'_i$ (equation 2.19)
2. Now time-average equation 2.19 to get the time averaged momentum equation for $\bar{V}_i$ (equation 2.21)
3. Subtract equation 2.21 from equation 2.19 to get an equation for V'_i , **Equation. (A)**
4. Do the same for V_j to get the equation for V'_j , **Equation. (B)**

5. Multiply **Equation. (A)** with V_j' and **Equation. (B)** with V_i' . Time average the products and add them together which together with some simplification and tricks yields an equation for $\overline{V_i'V_j'}$.

The final Reynolds Stress Equation reads,

$$\begin{aligned}
 \frac{\partial \overline{V_i'V_j'}}{\partial t} + \underbrace{\bar{V}_m \frac{\partial \overline{V_i'V_j'}}{\partial x_m}}_{C_{ij}} = & \underbrace{-\overline{V_i'V_m'} \frac{\partial \bar{V}_j}{\partial x_m} - \overline{V_j'V_m'} \frac{\partial \bar{V}_i}{\partial x_m}}_{P_{ij}} + \underbrace{\frac{p'}{\rho} \left(\frac{\partial V_i'}{\partial x_j} + \frac{\partial V_j'}{\partial x_i} \right)}_{\Pi_{ij}} \\
 & - \underbrace{\frac{\partial}{\partial x_m} \left[\overline{V_i'V_j'V_m'} + \frac{\overline{p'V_j'}}{\rho} \delta_{im} + \frac{\overline{p'V_i'}}{\rho} \delta_{jm} \right]}_{D_{ij,t}} + \underbrace{\nu \frac{\partial^2 \overline{V_i'V_j'}}{\partial x_m \partial x_m}}_{D_{ij,\nu}} \\
 & \underbrace{-g_i \beta \overline{V_j'\theta'} - g_j \beta \overline{V_i'\theta'}}_{G_{ij}} - \underbrace{2\nu \frac{\partial \bar{V}_i}{\partial x_m} \frac{\partial \bar{V}_j}{\partial x_m}}_{\varepsilon_{ij}}
 \end{aligned} \tag{2.26}$$

Equation 2.26 can be expressed symbolically and is then written as,

$$C_{ij} = P_{ij} + \Pi_{ij} + \underbrace{(D_{ij,t} + D_{ij,\nu})}_{D_{ij}} + G_{ij} - \varepsilon_{ij} \tag{2.27}$$

Where C_{ij} present the convection, P_{ij} production, Π_{ij} pressure-strain, G_{ij} Buoyancy production and ε_{ij} dissipation. $D_{ij,t}$ denotes the turbulent diffusion and $D_{ij,\nu}$ the viscous diffusion, adding $D_{ij,t}$ and $D_{ij,\nu}$ gives the total diffusion $D_{ij} = D_{ij,t} + D_{ij,\nu}$. The RANS procedure for the energy equation is omitted due to its complicated nature.

2.5.2 The Boussinesq Assumption

The Boussinesq assumption is utilized in order to simplify the Reynolds stresses in the momentum equation. This is done by introducing a model to the unknown Reynolds stresses, $\overline{V_i'V_j'}$. The Boussinesq can be derived by considering the right hand side of the momentum equation, equation 2.25 and replacing the Reynolds stresses with a turbulent viscosity ν_t ,

$$-\overline{V_i'V_j'} = \nu_t \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) \tag{2.28}$$

The right hand side of the momentum equation 2.25 can be simplyfied in the following maner,

$$-\frac{1}{\rho} \frac{\partial}{\partial x_j} \left[\bar{P} \delta_{ij} + \mu \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) - \rho \overline{V_i'V_j'} \right] = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left\{ \nu \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) - \overline{V_i'V_j'} \right\} \tag{2.29}$$

Replacing the Reynolds stresses in equation 2.29 with the modeled Reynold stresses expressed in 2.28 results in the following,

$$\frac{\partial}{\partial x_j} \left\{ (\nu + \nu_t) \left(\frac{\partial \bar{V}_i}{\partial x_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) \right\} \quad (2.30)$$

Adding the trace back to make the equation valid under contraction (when taking the trace). Otherwise it would be 0 due to continuity, yields the final expression for the modeled Reynold stresses,

$$\overline{v'_i v'_j} = -\nu_t \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) + \frac{1}{3} \delta_{ij} \overline{v'_k v'_k} = -2\nu_t \bar{s}_{ij} + \frac{2}{3} \delta_{ij} k \quad (2.31)$$

2.5.3 Two-equation models

The ***k*-equation** can be derived by introducing a turbulent kinetic energy, k which is defined as the sum off all normal Reynolds stresses, i.e.

$$k = \frac{1}{2} \left(\overline{V_1'^2} + \overline{V_2'^2} + \overline{V_3'^2} \right) \equiv \frac{1}{2} \overline{V'_i V'_i} \quad (2.32)$$

With the definition above the equation for the turbulent kinetic energy, k is given by taking the trace (setting the indices $i = j$) of the Reynold stress equation, equation 2.26 and dividing by two. The equation for k is hence written as,

$$\begin{aligned} \frac{\partial k}{\partial t} + \underbrace{\bar{V}_j \frac{\partial k}{\partial x_j}}_{C^k} = & \underbrace{-\bar{V'_i V'_j} \frac{\partial \bar{V}_i}{\partial x_j}}_{P^k} - \underbrace{\nu \frac{\partial \bar{v'_i}}{\partial x_j} \frac{\partial \bar{v'_i}}{\partial x_j}}_{\varepsilon} - \underbrace{\frac{\partial}{\partial x_j} \left[\bar{V'_j} \left(\frac{p'}{\rho} + \frac{1}{2} \bar{v'_i v'_i} \right) \right]}_{D_t^k} \\ & + \underbrace{\nu \frac{\partial^2 k}{\partial x_j \partial x_j}}_{D_\nu^k} - \underbrace{g_i \beta \overline{v'_i \theta'}}_{G^k} \end{aligned} \quad (2.33)$$

In symbolic form the ***k* - equation** can be written as

$$C^k = P^k + D^k + G^k - \varepsilon \quad (2.34)$$

where again C^k denotes the convection, P^k the production, D^k the total diffusion, G^k Buoyancy production and ε the dissipation.

A similar equation to the k -equation can be set up for the turbulent dissipation rate, ε - equation, this can be done by using the terms in the k -equation to formulate corresponding terms in the ε -equation. The ε -equation should also include a convection term, C^ε , a production term, P^ε , a diffusion term. D^ε , a buoyancy production term, G^ε and lastly a destruction term Ψ^ε . Hence a symbolic expression for the ε -equation should be in the following form,

$$C^\varepsilon = P^\varepsilon + D^\varepsilon + G^\varepsilon - \Psi^\varepsilon \quad (2.35)$$

The terms in the ε -equation have the units $[\frac{m^2}{s^4}]$ whereas the k -equation has the units $[\frac{m^2}{s^3}]$, it is therefore necessary to multiply P^k and ε in the k -equation by a

quantity with the unit $\frac{1}{s}$. A good choice is to multiply the k -equation with $\frac{\varepsilon}{k} = [\frac{1}{s}]$. This results in the following symbolic ε -equation,

$$P^\varepsilon + G^\varepsilon - \Psi^\varepsilon = \frac{\varepsilon}{k} \left(c_{\varepsilon_1} P^k + c_{\varepsilon_1} G^k - c_{\varepsilon_2} \varepsilon \right) \quad \text{and where} \quad D^\varepsilon = D^k \quad (2.36)$$

$$D^k = \frac{\partial}{\partial x_j} = \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial \varepsilon}{\partial x_j} \right], \quad D^\varepsilon = \frac{\partial}{\partial x_j} = \left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \quad (2.37)$$

It is also noticed from the equation above, equation 2.36 that two new coefficients are introduced in front of each term but where the diffusion term is formulated and expressed similarly as in the k -equation but with it's own Prandtl number, σ_ε . The final form of the ε transport equation is hence a modeled equation since some of the terms are modeled and is given by the following equation,

$$\frac{\partial \varepsilon}{\partial t} + \bar{V}_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\varepsilon}{k} \left(c_{\varepsilon_1} P^k + c_{\varepsilon_1} G^k - c_{\varepsilon_2} \varepsilon \right) + \frac{\partial}{\partial x_j} = \left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \quad (2.38)$$

2.5.4 The $k - \varepsilon$ Model

The $k - \varepsilon$ model can now be derived by modeling the production, buoyancy and the diffusion term. The diffusion term has already been dealt with and the modeled diffusion terms for both the k -equation and the ε -equation can be seen in equation 2.37. The diffusion terms have been modeled by using the standard gradient hypothesis which assumes that turbulent diffusion is transporting turbulent kinetic energy, k from regions where k is large to regions where k is small. D^k is hence playing a role with evening out inhomogeneities in the regions.

The production term in the $k - \varepsilon$ model is modeled by using the Boussinesq assumption for the Reynold stresses, $\bar{V}_i' v_j'$. With some simplification the modeled production term can be expressed as,

$$P^k = 2\nu_t \bar{s}_{ij} \bar{s}_{ij} \quad \text{where} \quad \bar{s}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) \quad (2.39)$$

Now the buoyancy production term needs to be expressed. The heat flux vector, $\bar{v}_i' \theta_i'$ is modeled using the Boussinesq assumption which results in the following expression,

$$\bar{v}_i' \theta_i' = -\alpha_t \frac{\partial \theta}{\partial x_i} \quad (2.40)$$

Where α_t denotes the turbulent thermal diffusivity which is commonly given from the turbulent viscosity as

$$\alpha_t = \frac{\nu_t}{\sigma_\theta} \quad (2.41)$$

Where σ_θ is the turbulent Prandtl number which defines the efficiency between the transportation of momentum through turbulent diffusion compared to the transportation of thermal energy. The turbulent Prandtl number is usually assigned a value between $0.7 \leq \sigma_\theta \leq 0.9$

The modeled equation for k can now be derived and is expressed as,

$$\frac{\partial k}{\partial t} + \bar{V}_j \frac{\partial k}{\partial x_j} = \nu_t \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) \frac{\partial \bar{v}_i}{\partial x_j} + g_i \beta \frac{\nu_t}{\sigma_\theta} \frac{\partial \bar{\theta}}{\partial x_i} - \varepsilon + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad (2.42)$$

Where the turbulent viscosity is given by the following expression,

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \quad (2.43)$$

The same can now be done for the modeled equation for ε and is expressed as

$$\begin{aligned} \frac{\partial \varepsilon}{\partial t} + \bar{V}_j \frac{\partial \varepsilon}{\partial x_j} = & \frac{\varepsilon}{k} C_{\varepsilon_1} \nu_t \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) \frac{\partial \bar{v}_i}{\partial x_j} + C_{\varepsilon_1} g_i \frac{\varepsilon}{k} \frac{\nu_t}{\sigma_\theta} \frac{\partial \bar{\theta}}{\partial x_i} \\ & - C_{\varepsilon_2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \end{aligned} \quad (2.44)$$

The commonly used values for the coefficients in the modeled k and ε equations above reads

$$(C_\mu, C_{\varepsilon_1}, C_{\varepsilon_2}, \sigma_k, \sigma_\varepsilon) = (0.09, 1.44, 1.92, 1, 1.3)$$

2.5.5 Shear Stress Transport

For turbulent flows with high Re two equation turbulence models are the most commonly used turbulence models. These models are able to independently characterize both the turbulent length and velocity scales and are therefore suitable for engineering applications [17]. The specific two equation model used in this thesis is the $k-\omega$ Shear stress transport (SST) model which solves equations for the specific dissipation (ω) as well as the turbulent kinetic energy (k). The difference between the regular $k-\omega$ model and the $k-\omega$ SST model is that the SST variant is a blend of two models, the $k-\omega$ and the $k-\varepsilon$ models [20]. This is employed because the $k-\varepsilon$ model is accurate in free streaming region and the $k-\omega$ model is accurate in the flow in the boundary layer, close to the wall. Another advantage of employing the $k-\omega$ model is that this model does not use wall functions to model the viscous sub-layer as it has been shown to reliably predict this region [17]. Due to this flexibility in the $k-\omega$ SST model it has become widely used for several applications, but especially for simulations of aircraft [21].

The governing equations of the $k-\omega$ SST closer model is described by four equations including the continuity equations, the Reynolds momentum equation, the turbulent kinetic energy equation and the transport equation for the specific dissipation rate, ω . The standard $k-\omega$ SST model reads

$$\frac{\partial k}{\partial t} + \frac{\partial}{\partial x_j}(\bar{V}_j k) = \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P^k - \beta^* k \omega \quad (2.45)$$

$$\begin{aligned} \frac{\partial \omega}{\partial t} + \frac{\partial}{\partial x_j}(\bar{V}_j \omega) &= \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \alpha \frac{P^k}{\nu_t} - \beta^* \omega^2 \\ &+ 2(1 - F_1) \sigma_{\omega_2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_j} \end{aligned} \quad (2.46)$$

Where the constant $\beta^* = 0.09$ according to [19]. A blending function F_1 is introduced which smoothly switches the desirable futures of either the $k - \omega$ or $k - \varepsilon$. The blending function should be equal to unity near the boundary layer to ensure and preserve features of the $k - \omega$ model, but go to zero at the boundary layer edge to preserve desirable features of the $k - \varepsilon$ in the free stream. The blending function F_1 is expressed as,

$$F_1 = \tanh(\xi^4), \quad \xi = \min \left[\max \left\{ \frac{\sqrt{k}}{\beta^* \omega d}, \frac{500\nu}{d^2 \omega} \right\} \frac{4\sigma_{\omega_{k-\varepsilon}} k}{CD_\omega d^2} \right] \quad (2.47)$$

Where CD_ω is given by

$$CD_\omega = \max \left\{ 2\sigma_{\omega_{k-\varepsilon}} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right\} \quad (2.48)$$

The distance to the closest wall node is denoted d in the equation 2.47. As previously mentioned the SST model adopts features for the $k - \omega$ model near walls where $F_1 = 1$ and features for the $k - \varepsilon$ model in free stream regions, i.e. far from walls, hence $F_1 = 0$. For regions where the blending function is either 0 nor 1, $0 < F_1 < 1$ all coefficients is blended between the $k - \omega$ and $k - \varepsilon$ model using the F_1 function in the following manner,

$$\alpha = F_1 \alpha_{k-\omega} + (1 - F_1) \alpha_{k-\varepsilon} \quad (2.49)$$

The constants used in the $k - \varepsilon$ SST model takes the following values,

$$\begin{cases} \alpha_{k-\omega} = 0.553167 \\ \alpha_{k-\varepsilon} = 0.44 \end{cases} \quad \begin{cases} \beta_{k-\omega} = 0.075 \\ \beta_{k-\varepsilon} = 0.0828 \end{cases} \quad \begin{cases} \sigma_{k,k-\omega} = 1.17647 \\ \sigma_{k,k-\varepsilon} = 1 \end{cases} \quad \begin{cases} \sigma_{\omega,k-\omega} = 2 \\ \sigma_{\omega,k-\varepsilon} = 1.16822 \end{cases}$$

The turbulent viscosity, ν_t in the $k - \omega$ SST is expressed as,

$$\nu_t = \frac{a_1 k}{\max(a_1 \omega, SF_2)} \quad (2.50)$$

which includes a second blending function, F_2 given by,

$$F_2 = \tanh(\eta^2), \quad \eta = \max \left\{ \frac{2k^{\frac{1}{2}}}{\beta^* \omega d}, \frac{500\nu}{d^2 \omega} \right\} \quad (2.51)$$

Where S is the strain-rate tensor magnitude. The second blending function, F_2 is preventing the SST limiter, to be activated near walls. The purpose of the SST limiter is to control and limit the production to dissipation ratio inside turbulent boundary layers. The SST limiter is dependent on the constant a_1 and has therefore been calibrated to the value of $a_1 = 0.31$ which is the default value used in both ANSYS FLUENT and CFX. Previous studies has however indicated that the a_1 coefficient has a great effect on the prediction of separation regions. The studies concluded that an increase of the constant can completely remove the influence of the SST limiter but yields better results in various flow configurations. Increasing a_1 to 0.355 is according to [22] enough to completely remove the influence of the SST limiter which should result in improved results.

2.5.6 Transition SST Model

The Transition SST model also know as the $\gamma - Re_\theta$ model extends the traditional $k - \omega$ SST model by incorporating two other transport equations, one for the intermittency factor, γ and one for the transition onset criteria in terms of the momentum-thickness Reynolds number Re_θ . The purpose for momentum-thickness Reynolds number is to capture the non-local influences of the turbulent intensity which changes because of two reasons. The first reason is due to the decay of the turbulent kinetic energy in the free stream and the second reason is due to changes in the free stream velocity outside the boundary layer [23].

The intermittency, γ provides a fraction of the time as the flow transition into turbulent flow and is implemented to alter the turbulent eddy-viscosity μ_t . A value of $\gamma = 0$ signifies a laminar state and a value, $\gamma = 1$ implies a turbulent state. The model is described in a manner that the intermittency should rise from zero to unity in order to approach and transition from a laminar-to-turbulent state [23; 24].

The transport equation for the intermittency γ reads as follows:

$$\frac{\partial(\rho\gamma)}{\partial t} + \frac{\partial(\rho V_j \gamma)}{\partial x_j} = \underbrace{P_{\gamma 1} - E_{\gamma 1}}_{\text{Transitions Sources}} + \underbrace{P_{\gamma 2} - E_{\gamma 2}}_{\text{Relaminarization Sources}} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\gamma} \right) \frac{\partial \gamma}{\partial x_j} \right] \quad (2.52)$$

Where the transition sources are defined as follows [25],

$$\begin{cases} P_{\gamma 1} = 2F_{\text{length}}\rho S \left(\gamma F_{\text{onset}} \right)^{c_{\gamma 3}} \\ E_{\gamma 1} = P_{\gamma 1}\gamma \end{cases} \quad (2.53)$$

Where the length of the transition region is controlled by the empirical correlation F_{length} and S is the strain rate magnitude.. The second pair of source terms from equation 2.52 called the destruction/relaminarization sources are defined as follows [25],

$$\begin{cases} P_{\gamma 2} = (2c_{\gamma 1})\rho\Omega\gamma F_{\text{turb}} \\ E_{\gamma 2} = c_{\gamma 2}P_{\gamma 2}\gamma \end{cases} \quad F_{\text{turb}} = e^{-(\frac{R_T}{4})^4} \quad (2.54)$$

Where Ω is the vorticity magnitude, the F_{onset} is described and controlled by the following functions:

$$Re_V = \frac{\rho y^2 S}{\mu} \quad \text{and} \quad R_T = \frac{\rho k}{\mu \omega} \quad (2.55)$$

$$F_{\text{onset}_1} = \frac{Re_V}{2193 Re_{\theta_c}} \quad (2.56)$$

$$F_{\text{onset}_2} = \min\left(\max\left(F_{\text{onset}_1}, F_{\text{onset}_1}^4\right), 2.0\right) \quad (2.57)$$

$$F_{\text{onset}_3} = \max\left(1 - \left(\frac{R_T}{2.5}\right)^3, 0\right) \quad (2.58)$$

$$F_{\text{onset}} = \max\left(F_{\text{onset}_2} - F_{\text{onset}_3}, 0\right) \quad (2.59)$$

Where Re_{θ_c} is the critical Reynolds number where the intermittency first starts to increase in the boundary layer and y is the wall distance. In the $\gamma - Re_{\theta}$ model the onset function. F_{onset} is used to trigger the intermittency production, hence the onset function should activate equation 2.53 [22].

Therefore, this report will use the Transition SST $k - \omega$ model along with the $\gamma - Re_{\theta}$ model for the transitional turbulence.

2.5.7 Turbulence Constants

The above described turbulence modelling approach utilizes several coefficients which are used by CFD software when solving the numerous equations. As mentioned the transition SST model introduces two new equations for the intermittency and the momentum-thickness Re number, the latter of which is defined in equation 2.60 as a transported scalar quantity $\overline{Re_{\theta t}}$ [26].

$$\frac{\partial(\rho \overline{Re_{\theta t}})}{\partial t} + \frac{\partial(\rho V_j \overline{Re_{\theta t}})}{\partial x_j} = P_{\theta t} + \frac{\partial}{\partial x_j} \left[\sigma_{\theta t} (\mu + \mu_t) \frac{\partial \overline{Re_{\theta t}}}{\partial x_j} \right] \quad (2.60)$$

The production term $P_{\theta t}$ can be written, once again using a blending function $F_{\theta t}$ as

$$P_{\theta t} = c_{\theta t} \frac{\rho}{t} (Re_{\theta t} - \overline{Re_{\theta t}}) (1.0 - F_{\theta t}) \quad (2.61)$$

In this equation for the production term the constant $c_{\theta t}$ determines the magnitude of the production and is commonly set to 0.03 [26] which it is by default in *ANSYS CFX* [25]. The constants $C_{\gamma 1}$, $C_{\gamma 2}$ and $C_{\gamma 3}$ in equation 2.53 and 2.54 can also be chosen in *CFX* where they are denoted as *CI1*, *CI2* and *CI3* and have the default values of 0.03, 50 and 0.5 respectively [25].

CFX also uses a plethora of constants outside of the intermittency model. The shear stress transport model uses two constants, β^* and a_1 , as described in the SST

theory section (2.5.5). Since the SST model is a mix of the $k - \varepsilon$ and $k - \omega$ models a few constants for these models also need specification in *CFX*. These constants are also described in the SST theory section (2.5.5).

2.6 Near Wall Modelling

As mentioned above the $k - \omega$ SST model resolves the viscous sub-layer without needing wall functions, but some concessions still has to be made when meshing close to walls in order to determine the shear stress at the wall. In order to measure the thickness of the mesh layers at a particular wall a dimensionless distance, $y+$, is imposed and defined as

$$y+ = \frac{yu_*}{\nu} \quad (2.62)$$

where y is the wall distance, ν is the kinematic viscosity and u_* is a characteristic velocity scale in turn defined using the density of the fluid ρ and the wall shear stress τ_w as $u_* = \sqrt{\tau_w/\rho}$. The wall shear stress can be defined as a function of an empirically derived friction coefficient C_f according to the following [27]:

$$\tau_w = \frac{1}{2}C_f\rho V^2 \quad (2.63)$$

This implies that the calculation of $y+$ is dependent on the choice of friction coefficient, of which many correlations are available. Some of these are correlations for turbulent flow over a flat plate, turbulent flow in a smooth pipe, flow in a completely turbulent pipe with specified roughness, the Colebrook equation [15]. These three correlations are presented below;

$$\text{Flat plate: } C_f = 0.0592Re^{-1/5} \quad (2.64)$$

$$\text{Smooth turbulent pipe: } C_f = 0.25(0.790\ln(Re) - 1.64)^{-2} \quad (2.65)$$

$$\text{Colebrook equation: } \frac{1}{\sqrt{4C_f}} = -2.0\log\left[\frac{e/D}{3.7} + \frac{2.51}{Re\sqrt{C_f}}\right] \quad (2.66)$$

In the Colebrook e is the roughness of the material that the pipe is made out of (the material was set as aluminum) and D is the diameter of the pipe. The *fluids* package in *Python* also has a function used to calculate the friction coefficient although the methodology is unknown as the source is unavailable. These different friction coefficients calculated for the TRS geometry using the highest Re case (Re=350.000) are presented below.

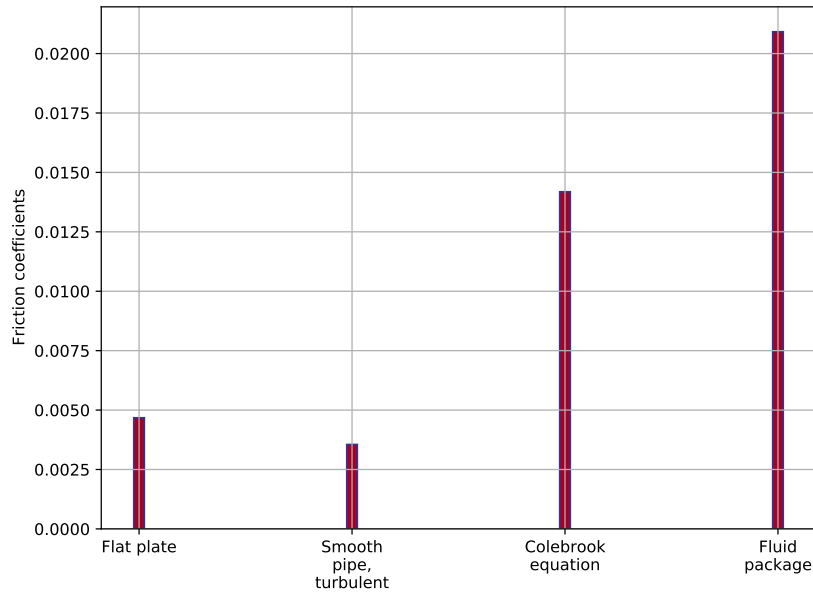


Figure 2.7: Friction coefficients for different empiric expressions

As can be seen in figure 2.7 there is some variance to the friction coefficient depending on which expression is used, which in turn will yield different wall distances when equation 2.62 is solved for y . The Fluid package in *Python* yield the highest friction coefficient, but seeing as the the equation used is unknown it is hard to argue for the employment of this correlation. The correlation for flat plates and the correlation for smooth pipes give similar results, although the Colebrook equation was chosen as the expression for the friction coefficient due to its more encompassing nature. In order to solve for y however a value of y^+ must be set. A common rule of thumb for simulations employing the SST model is to use $y^+ = 1$ [28; 29], although there are issues with using this rule of thumb for all applications. It has been shown that the SST model can predict nonphysical behaviour for drag and friction coefficients for flow around airflow if a y^+ value of one is used [28], but it has also been shown that the SST model can reliably predict gas turbine applications [30; 29]. As the model has different result there is no obvious way to determine the y^+ value required for this application. In order to reduce the computational cost a y^+ value of unity was decided as a preliminary target, which is used to determine the thickness of the inflation layers when meshing.

3

Methods

3.1 Experimental Test Facility

The test facility built at Chalmers is built in order to simulate flow-characteristics that is similar to what can be found in real aero-engines. The purpose of the facility is to simulate an environment where it is possible to investigate the flow around outlet guided vanes (OGV) located down streams of a low pressure turbine (LPT). The rig at Chalmers is set to provide realistic inlet boundary conditions to the engine exit structure (EES) which later can be used in CFD-simulations. The facility consists of different components that have different functions. Down below is a schematic figure of the test-facility together with some components highlighted. Figure 3.1 [6]

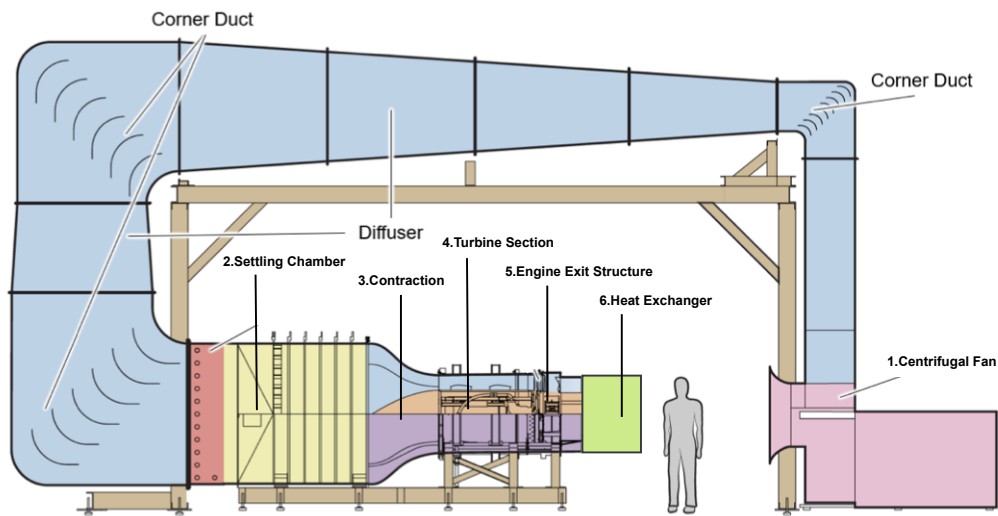


Figure 3.1: Schematic overview of the rig at Chalmers with an integrated heat-exchanger, Provided by Isak Jonsson

1. **Centrifugal Fan:** The main task for the centrifugal fan is too provide flow-rate and pressure to the system. The centrifugal fan provides appropriate pressures drops which later is needed to run the turbine. The centrifugal fan is controlled with a motor that allows the user to alter the mass flow rate in the system upon request [6].
2. **Settling Chamber:** The purpose of the settling chamber is to reduce non-

uniformities in the axial direction for the flow, further on the settling chamber should also reduce the turbulent intensity for the flow. The flow from the duct is therefore firstly forced through *honeycombs* which are hexagonal guiding devices and secondly the airflow passes through five turbulence *screens* [6].

3. **Contraction:** In between the turbine and the settling chamber a body is placed to connect these two components. Another use for the contraction is to calculate the mass-flow in the system by measuring the pressure at both the inlet and the outlet of the contraction-body [6].
4. **Turbine Section:** The purpose of the turbine section is to produced realistic boundary conditions at the inlet of the Engine Exit Structure (EES). The test section is also placed at the EES where heat and aero measurements is supposed to be taken. It is therby important to create realistic conditions [6].
5. **Engine Exit Structure:** The EES consist of 12 OGVs which purpose is to decrees the swirl coming from the LPT, hence obtaining an axial flow at the outlet of the EES. The design of the EES that is studied was made at GKN Aerospace Sweden [6].
6. The **heat exchanger** is mounted right after the EES.

The TRS both holds structural integrity of the engine as it holds bearings and is also an aerodynamic components as it contains outlet guide vanes OGVs whose purpose is to lessen the swirl of the flow out of the engine structure [8]. Several papers investigating the performance of different TRS configurations on this particular rig have been published confirming the ability of the rig to provide realistic conditions[7],[8]. These conditions include Reynolds numbers as high as 350 000 which is comparable to engines such as geared turbofan engines [5] and the flow coefficient (FC), the ratio between the rotational speed of the LPT and the axial velocity of the flow.

3.1.1 Implementation of the HX

The heat-exchanger that is studied is mounted right after the EES in the test facility. After the TRS the flow continuous into the heat-exchanger where it is first split up where a portion flows towards and through the heat exchanger. A figure representing this events is illustrated down below in figure 3.2.

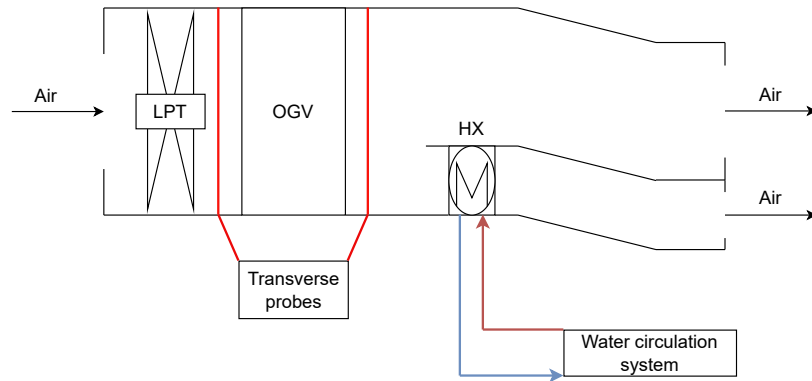


Figure 3.2: Description of the rigg downstream of the contraction, showing the integrated HX with its water circuit

Measurements of the flow in and out of the TRS are done by an L-shaped probe which measures several quantities over a section of the annular TRS, a 30 degree span, which yields planes of the examined variables. The location of these probes can be seen from figure 3.2.

3.1.2 Instruments

For measurement's of the flow a multi-hole probe (MHP) measurement technique has been used. A multi-hole probe is a probe with multiple holes at the tip. For the measurements of the flow in and out of the TRS a L-shaped multi-hole probe has been used. A five-hole MHP was used US and a seven-hole MHP DS. [12]. The multi-hole probe measurements technique is a accurate measurement technique for determining fluid properties such as velocities and densities and are usually designed to either be five- or seven-hole probes. The multi-hole probe predicts the velocity vector by recording the port pressure and comparing them with a calibration database, it is therefore important to calibrate the probes in known flow-fields for accurate results when measuring unknown flow fields. The calibration is carried out in a flow-field of known magnitude and direction where the probe is also rotated and pitched through a range of determined angles. The rotation and pitching of the probe is done in order to simulate every possible velocity path [31]. More information about the MHP that has been used at Chalmers can be found in the paper "*Experimental Aerothermal Study Of Internal Jet Engine Structures*" written by Isak Jonsson [12].

3.2 Studied Cases

As described in the "Operating points" part of the theory section the two variables that characterize different flow regimes are the Reynolds number and the flow coefficient. In order to test the performance of the TRS integrated with the heat exchanger a number of operating points has been tested in the test rig. These different cases are presented in table 3.1

Table 3.1: Characteristics for the studied cases

Test number	RE	FC	Characteristic
1	235k	0.69	At design
2	235k	0.73	high swirl
3	235k	0.61	low swirl
4	350k	0.69	High RE
5	235k	0.69	Variation in HX geometry
6	235k	0.69	Lower bleed
7	235k	0.73	Lower bleed

As described in section 3.1 the main measurement of the flow is done by two different multi-hole probes upstream (US) and downstream (DS) of the OGV. For some cases this MHP was partially blocked leaving the data compromised, which is seen in table 3.2 as cases 1-3 have this in common. This table also highlights the fact that no US data is available for two cases, 4 and 5. As the US data is the basis for the CFD calculations, US is used for boundary conditions which is explained later, the lack of US data effectively makes these cases unable to simulate. Hence cases 4 and 5 will not be investigated in this thesis.

Table 3.2: Availability and status of the US and DS profiles for the different cases

Test number	US	DS	Probe status
1	Yes	Yes	One hole blocked for DS
2	Yes	Yes	One hole blocked for DS
3	Yes	Yes	One hole blocked for DS
4	No	Yes	
5	No	No	
6	Yes	Yes	
7	Yes	Yes	

In the table above the five first tests were done with the nearly the same back pressure, approximately one atmosphere, at the outlet of the HX as well as the outlet of the air that does not enter the HX. For the last two tests the back pressure at the outlet downstream of the HX was set in order to obtain a specific mass flow over the HX (this flow is denoted as bleed) of about ten per cent of the inlet flow. For the first three and the last two cases the data obtained are planes both upstream (US) and downstream (DS) of the OGV where as the collected data for the high Re case only is the DS profile. The collected test data was obtained at Chalmers by Isak Jonsson and Valentin Vikhorev prior to the start of the thesis and was provided at the beginning of the thesis.

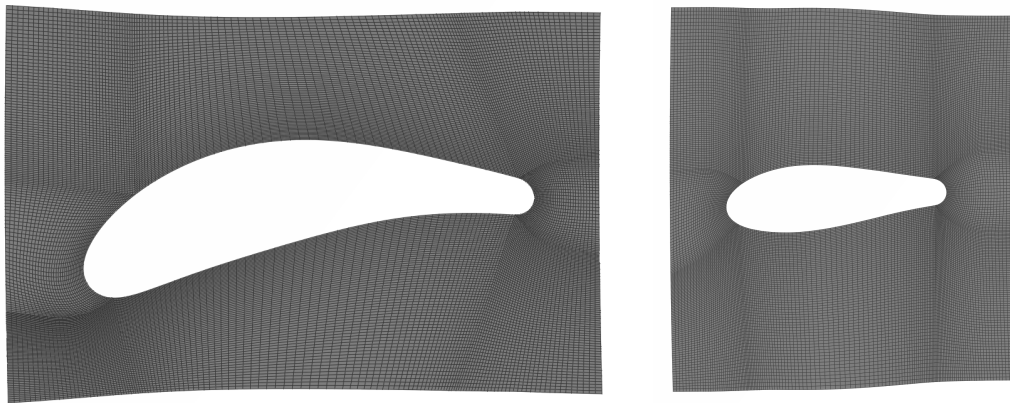
3.3 Meshing

As a CAD-model had been made previous to the master thesis the needed preparation in order to start simulating was to mesh the domain, including the TRS as well as the HX. This section of the thesis will however only cover the meshing of the TRS as the geometry of the heat exchanger is confidential. The meshing was done using the meshing software *ANSA* in which the provided geometry was imported.

3.3.1 Surface Mesh

Since the probe measures 25 values per radius in the circumferential direction and 20 different radii the total amount of measurements on the inlet is 500. This means

that the mesh has to be sufficiently fine so that the data values can be used as inlet conditions for the simulations. A similar reasoning applies to the outlet of the TRS, and the number of cells on these surfaces will dictate the behaviour of the other surface meshes. The main concern with the TRS surface mesh is the blade and the impact of its curved geometry which makes surface meshing using quadrilaterals difficult. The resulting surface mesh at the hub and shroud is shown in figure 3.3, note that these are not to scale.



(a)

(b)

Figure 3.3: Surface mesh for the hub and the shroud

The four sides perpendicular to the hub and shroud are however easily meshed using quadrilaterals. The surface meshing of the blade itself also poses problems when trying to use quadrilateral elements. The resulting mesh is presented in figure 3.4, where the mesh for both PS and SS is shown.

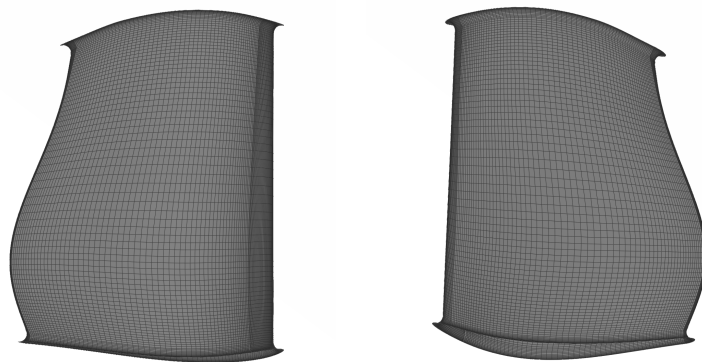


Figure 3.4: Blade surface mesh, both at PS and SS

3.3.2 Volume Mesh

A structured mesh type was chosen for this simulation. This was executed with the *map* function in ANSA which construct a structured grid of rectangular blocks or cubes also called hexas. The illustration of the TRS down below is visualizing this grid of rectangular blocks. The volume mesh for the TRS including the OGV contained a total of 3140214 hexas. A inflation was also built near the boundary layers for the hub, shroud and blade. This was accomplished by using ANSAs *Layers* function where the first height of the first cell closest to the wall was set to a value of 0.01. The inflation is thereafter build according to a growth-factor and a maximum number of layers. The growth factor was set to 1.2 and as the SST transition model has an target $y+$ value of 1 the number of layers had to be adjusted to try to replicate this $y+$ value in the simulations. To acquire a $y+$ value of unity or less the most demanding case in terms of $y+$ was chosen as the target to try and model towards. This most demanding case is the case where the Re value is the highest, as per equations 2.62 - 2.66, meaning the Re=350 case. Using the Re number the friction coefficient was calculated using equation 2.66 with a roughness value of aluminum. With the friciton factor the wall shear stress could be obtained using equation 2.63, which in turn yields a characteristic velocity and the desired distance of the inflation layer could be calculated using equation 2.62 and solving for y given a $y+$ of unity. Since an initial height and growth-factor were specified the number of layers was set the number of layers could be determined.

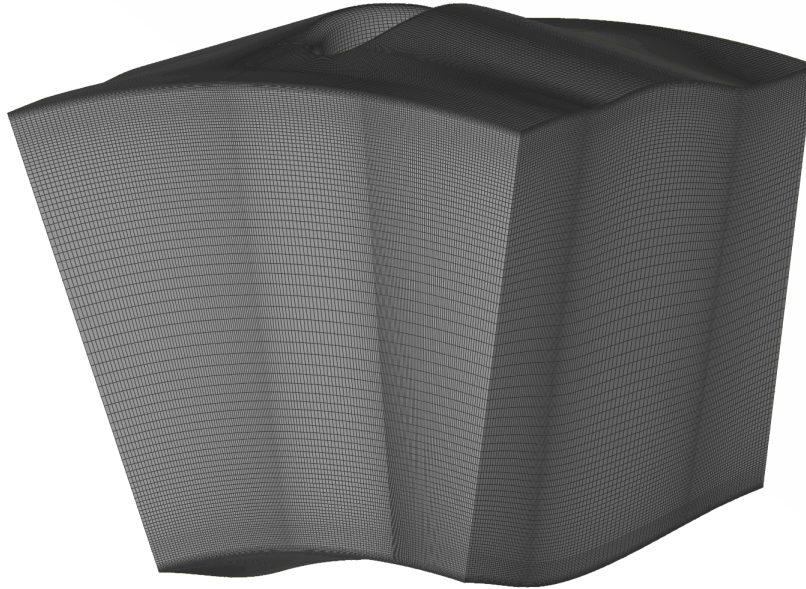


Figure 3.5: Structured volume mesh for the TRS with a total of 3097654 hexas

The volume mesh around the shroud can be observed in Figure 3.6 and 3.7, which provides a visualization of the inflation around the shroud. It is worth noting that the same inflation technique was also applied around the hub.

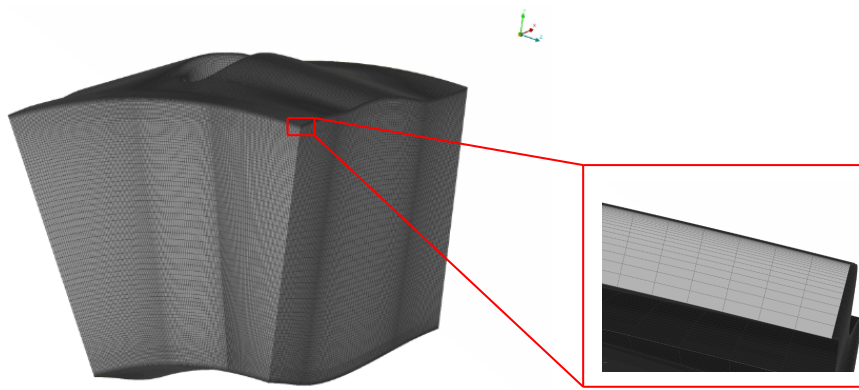


Figure 3.6: This figure represents a zoomed in picture of the inflation surrounding the shroud

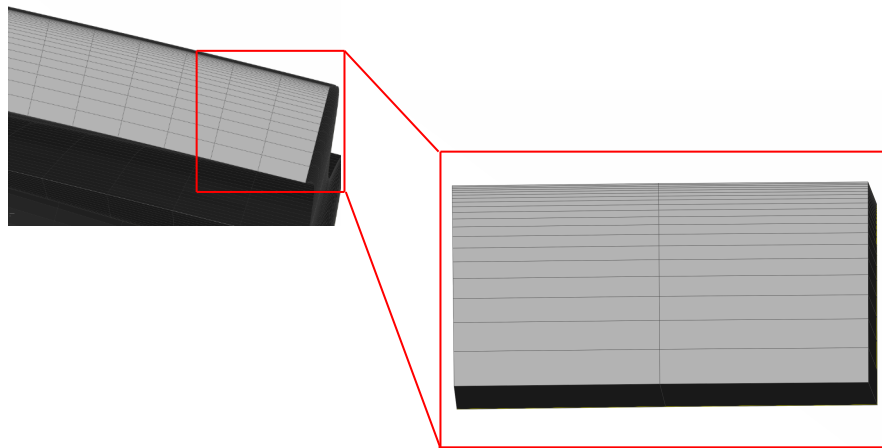


Figure 3.7: Zoomed in picture of the inflation used around the shroud

In essence three different meshes was done and then merged, the above described and presented TRS mesh, the mesh of the air side of the HX and the mesh at the water side of the HX. The meshes for the air side of the HX and the TRS mesh had to be matched as both domains will contain the same fluid with no physical border. The mesh at the water side could however be done independently of the other meshes as the water and air are separated by a heat conducting solid, and this mesh had been done previous to the thesis work. Some statistics for the meshes are presented in table 3.3, note that wedges are 5 sided shapes where as hexas have 6 sides.

Table 3.3: Statistics for the mesh parts

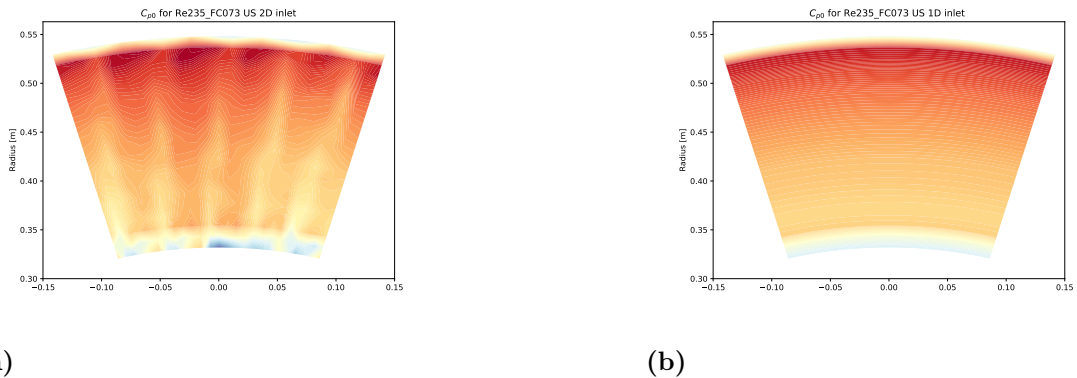
Mesh part	Number of elements	Hexas [%]	Wedges [%]
TRS	3,140,214	100	0
HX, air side	28,521,173	46.32	53.68
HX, water side	18,183,396	60.32	39.68
Total	49,844,783	54.81	45.19

3.4 Numerical Setup

The simulations were carried out in the 2022r1 edition of the turbo machinery focused CFD program *ANSYS CFX* using a *Linux* interface. In order to set up a simulation the program *CFXpre* was used where the mesh was imported from *ANSA* and initial and boundary conditions was set up as well as the settings that the solver *CFXsolver* uses when solving the equations.

3.4.1 Boundary Conditions

From the experiments done at Chalmers using the traversing probe profiles both upstream and downstream of the OGV had been measured. The profiles that were set as inlet profiles in CFX were both the exact 2D profile that the traversing probe produces and 1D radial inlet profiles in order to asses the impact of tangentially averaging the inlet data. An example of a 2D and a 1D inlet can be seen in figure 3.8.

**Figure 3.8:** Comparison of generic contours for 2D and 1D inlets

The US 2D and 1D profiles could be used as boundary conditions at the inlet of the domain, with the caveat that the profiles had to be extrapolated as the probe only measures a certain portion of the inlet as seen in figure 3.9 below.

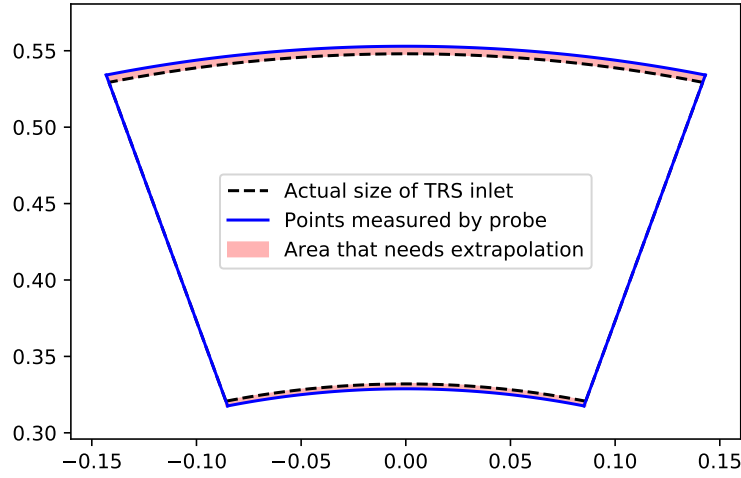


Figure 3.9: Plot showing the actual size of the inlet and the area measured by the probe

This implies that values in the inlet file has to be extrapolated in order to give the entire inlet appropriate values. There are different ways to extrapolate the data however [9]. The three ways presented here are different ways of handling the boundary layers (BL) and are therefore called BL0, BL1 and BL2 using previously established notation [9]. For the BL0 option the end-wall boundary layers are removed completely; BL1 simply extrapolates the measured end values to the hub and shroud coordinates and BL2 utilizes linear extrapolation according to equation 3.1 to predict the hub and shroud values.

$$y(x^*) = y_{k-1} + \frac{x^* - x_{k-1}}{x_k - x_{k-1}}(y_k - y_{k-1}) \quad (3.1)$$

An example of the different modes of extrapolation is shown in figure 3.10 where the extrapolation of the C_{p0} value for a radial inlet profile is shown.

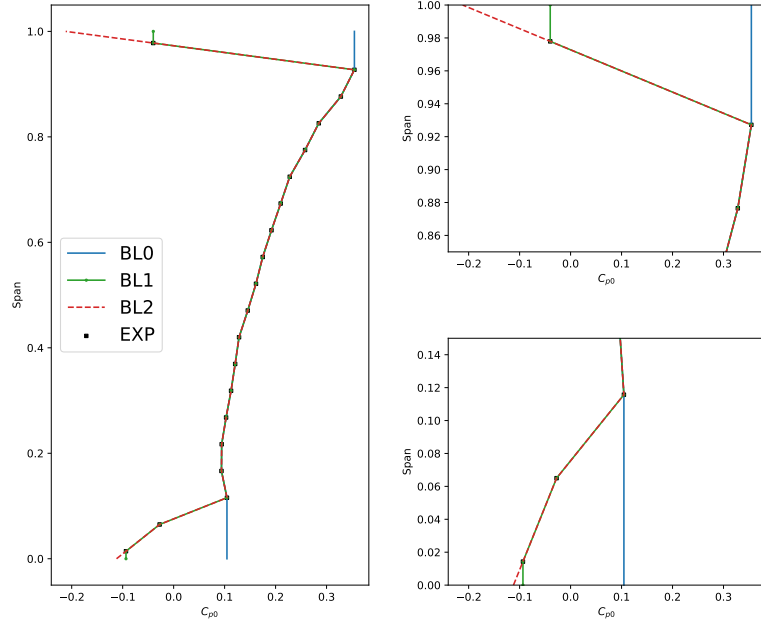


Figure 3.10: Comparison of the different modes of extrapolating the boundary layers shown together with experimental values

There are a few different ways to set the inlet as a boundary condition, the inlet can be set as a velocity inlet where the velocity components in the three directions (since cylindrical coordinates is used these are the axial, the circumferential and the radial components) is set in each point from the data. An alternative to the velocity inlet is a pressure inlet where the total pressure is set as well as the directions of the flow given as follows:

$$d_i = \frac{V_i}{\sqrt{\sum V_i^2}} \quad i = \{x, \theta, r\} \quad (3.2)$$

In accordance with previous studies the pressure inlet was used for all simulations unless otherwise stated which means that the data from the tests that was extrapolated and used at the inlet was the total pressure, the flow direction and the temperature.

The outlets were set as static pressure outlets where the relative pressure, the pressure set compared to the pressure in the surrounding, was set in order to match key parameters US such as the static pressure and velocity magnitudes to the test data. To measure whether this was accomplished tangentially averaged plots were produced at the US location and compared with test data. It is important to note that the domain with the heat exchanger has two outlets, one for the undisturbed air and one for the air interacting with the heat exchanger, but that the settings are identical for both. To connect the TRS mesh with the extended mesh which

includes the HX a "General Connection" interface is used at the location of DS to merge the meshes. The hub, shroud and the blade are set as no slip, smooth and adiabatic walls. As one twelfth of the domain is modelled the two sides normal to the blade in the tangential direction is set as periodic interfaces. A schematic of the TRS is shown in figure 3.11 where the inlet is blue, the general connection interface is green and The walls are grey. The periodic interfaces are not displayed as this would hide the other surfaces.

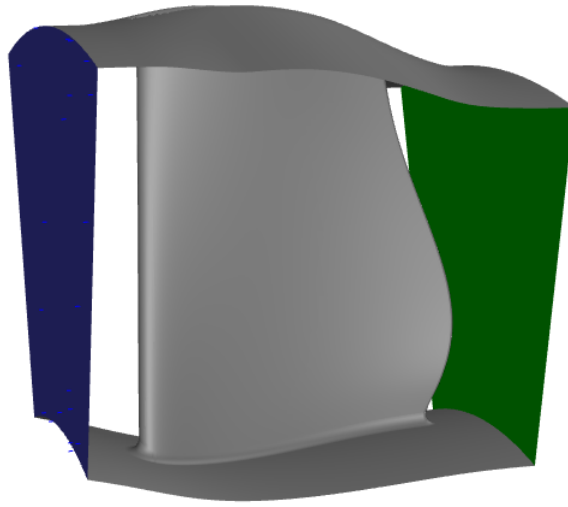


Figure 3.11: Domain setup for the TRS showing some of the BCs

Similar BCs are set for the extended HX geometry although they are not visualized, the inlet temperature of water is set using test data and type of inlet is set to a mass flow inlet as the mass flow of water is also measured. The water outlets are set as pressure outlets with the pressure being the same as the ambient pressure (0 gauge pressure).

3.5 Solver Settings

The model was set as steady state and the fluid in the domain was set as air, ideal gas. As heat transfer model the "total energy" option was used which allowed for the temperature to be set at the inlet. As discussed in the theory section the turbulence model can be varied in numerous ways but the default turbulence setting that was used unless other models are specified is the SST $k - \omega$ model utilizing the $\gamma - Re_\theta$ approach of describing transitional turbulence. The various empirically derived coefficients that are applied are presented in table 3.4.

Table 3.4: Various constants used for turbulence modelling in *CFXpre*

Intermittency coefficients	
CI1	0.03
CI2	50.0
CI3	0.03
Re_θ coefficient	
CTHETA	0.03
Advanced Turbulence Control	
β^*	0.09
a_1	0.35
k-ω regime	
k-coefficients	Eddy Diffusivity Closure
Turbulent Schmidt Number	1.176471
ω -coefficients	Eddy Diffusivity Closure
Turbulent Schmidt Number	2.0
α	0.553167
β	0.075
k-ε regime	
k-coefficients	Eddy Diffusivity Closure
Turbulent Schmidt Number	1.0
ω -coefficients	Eddy Diffusivity Closure
Turbulent Schmidt Number	1.168224
α	0.44
β	0.0828

For the inlet turbulent properties the turbulent intensity was set to 5% and the turbulent length scale was set to 0.7 percent of the TRS midspan axial chord, which is $0.007 \times 0.2566 \approx 2e - 3$. The target residuals were set as 10^{-5} .

3.6 Generic Procedure for Setting up Case in *CFX*

The general CFD procedure for a case is presented in figure 3.12 below, such a procedure was followed for all cases where an US profile had been obtained by tests (all tests except 4 and 5).

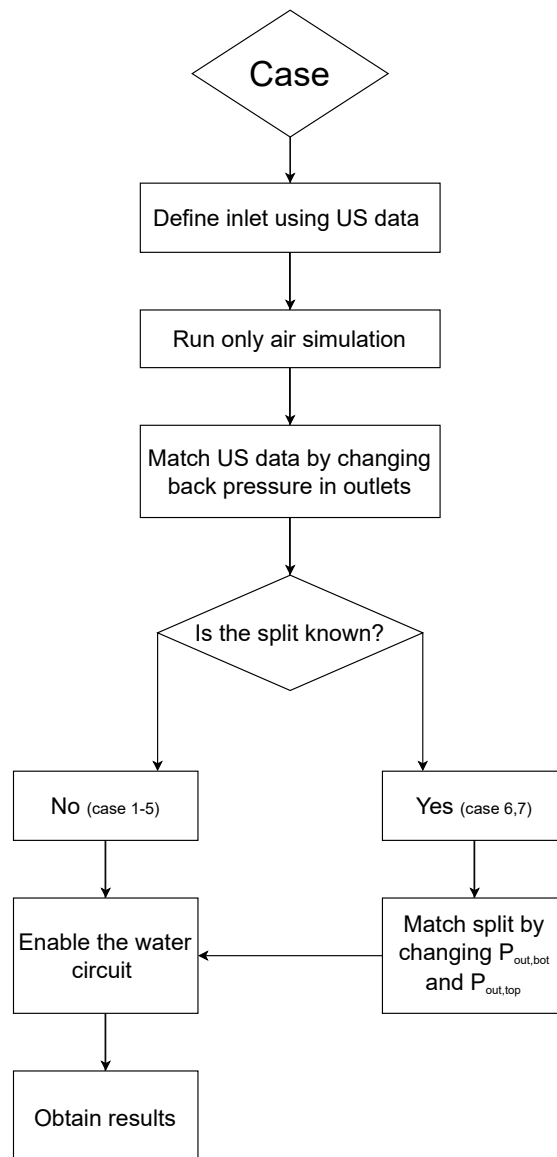


Figure 3.12: General procedure for *CFX*

”Run only air simulation” means that the simulations are done with the entire geometry of the TRS and HX but that the the hot side of the heat exchanger is left empty, leaving only the geometry of the HX to interact with the air. The water circuit is enabled after a solution to the model only including air is obtained after which final results can be gathered.

3.7 Other Tests

To asses the impact of the HX presence tests with an extended version of the OGV geometry without the HX geometry were carried out as well. The geometry of the

extended OGV domain is seen in 3.13.

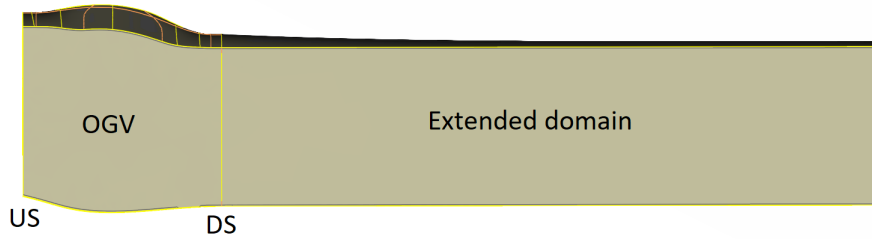


Figure 3.13: Domain for the extended OGV

The reason behind the extension of the domain is to minimize the effect of having an outlet at the same location as the DS probe, which could yield non-physical results as the outlet could give back flows. Using this geometry and comparing with the results from the HX integrated simulations the impact of the HX geometry on the flow characteristics can be evaluated. The meshing of the extended domain was done in accordance with the previous meshing described above.

A mesh independence study was conducted in order to determine the appropriate level of mesh refinement required to obtain accurate and reliable results. It involved analyzing the sensitivity of the solution to changes in the mesh density or size. When doing the study the geometry that was used was the OGV together with the extended domain shown in figure 3.13. For each mesh, an inflation was applied to the hub, shroud, and blade. The inflation consisted of tetrahedral elements, similar to previous cases. The volume mesh, on the other hand, was constructed using tetrahedral elements. The mesh refinements resulted in varying numbers of cells: the coarsest mesh had 2.7 million cells, the finer mesh had 8 million cells, and the finest mesh had 20 million cells. The mesh refinement process involved increasing the number of elements, thereby leading to smaller cell sizes.

As described in theory section 2.5.5 the SST limiter is dependent on the constant a_1 where studies have shown that the calibrated value of $a_1 = 0.31$ can overpredict separation regions. Due to this, simulations where a_1 is set to its default value of 0.31 and $a_1 = 0.35$ has been run in order to verify the impact of a_1 . Another variation in turbulence modelling that was studied was investigating the $k-\varepsilon$ model. Tests with this model were made for case 1 with all other settings kept the same.

Additional simulations were also done in order to compare how the effectiveness and pressure drop change with the split fraction. As changing the geometry is time consuming the split percentage was simply done by changing the back pressure at

the outlet of the HX which in turn effects the split.

3.8 Gathering of Results

The results presented in the section below is mainly gathered at three points in the domain, at the locations corresponding to the US and DS probes as well as at the OGV. At the location of the probes entire contour plots will be presented as well as plots showing tangentially averaged variables chosen by the authors. At the DS profile results will also be gathered for certain radii corresponding to 25, 50 and 75 percent of the DS total span, as seen in figure 3.14.

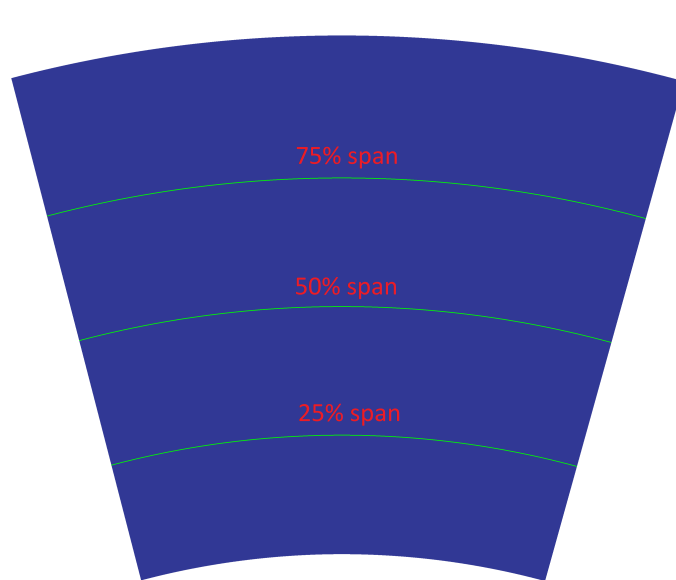


Figure 3.14: Figure showing the location where wake results will be gathered

At the OGV streamlines will be compiled to illustrate the air flow characteristics over the surface. These figures will be gathered for the studied cases and are available in their entirety in the Appendix.

Just as the inlet cannot be measured in its entirety due to the nature of MHP the same applies DS of the OGV. Figure 3.15 shows a typical downstream contour plot where the black arrows mark the full extent of the DS profile obtained from CFD and the green arrows mark the limitations to the test data. This has to be taken into account when comparing experiments and simulations at the hub and shroud regions of the domain.

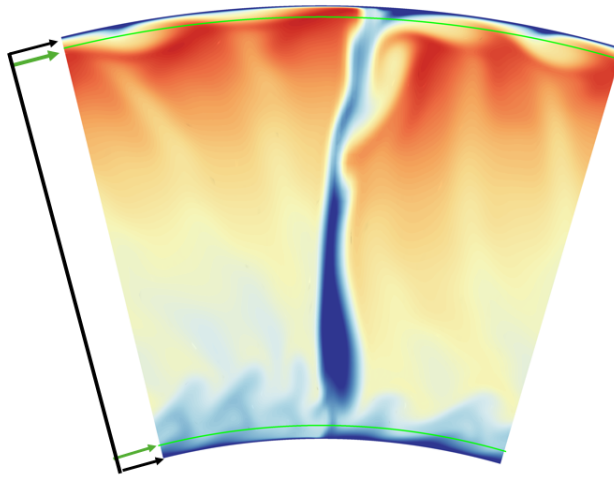


Figure 3.15: Figure showing an entire DS profile obtained in *CFX* and the area measured by the DS MHP

4

Results

In order to compare the results from the CFD simulations with the test data in a clear and expansive manner the results have been divided into a number of subsections. The order of these subsections is to firstly present the used settings for the CFD solver and then to show the results of the different cases presented in table 3.1. The specification of the inlet will be the starting point after which results for the employed turbulence models will be shown. Cases 6 and 7 will be the first cases shown as these cases match the test data to the largest extent, cases 1-3 having partial blockage of the MHP and cases 4 and 5 missing US data. Following the results for cases 6 and 7 the results for cases 1-3 will be presented. After the results for the test cases has been presented some CFD results beyond the test cases were studied. The first of these investigations was the change in split percentage due to the back pressure at the HX outlet, after which simulations with a geometry not including the heat exchanger will be presented. Following this the near wall modelling results and finally the heat exchanger performance will be presented.

4.1 Inlet Specification

Two different modes of inlet investigations were carried out, specifying the dimension of the inlet using either one or two dimensions as well as the hub and shroud extrapolation method.

4.1.1 Inlet Dimension

In this section results for simulations done using a 1D inlet, as described in section 3.4.1, for case 1 is compared with the results obtained when employing a 2D inlet. Figure 4.1 shows US as well as DS profiles for C_{p0} for experiments, CFD using 1D inlet and CFD using 2D inlet.

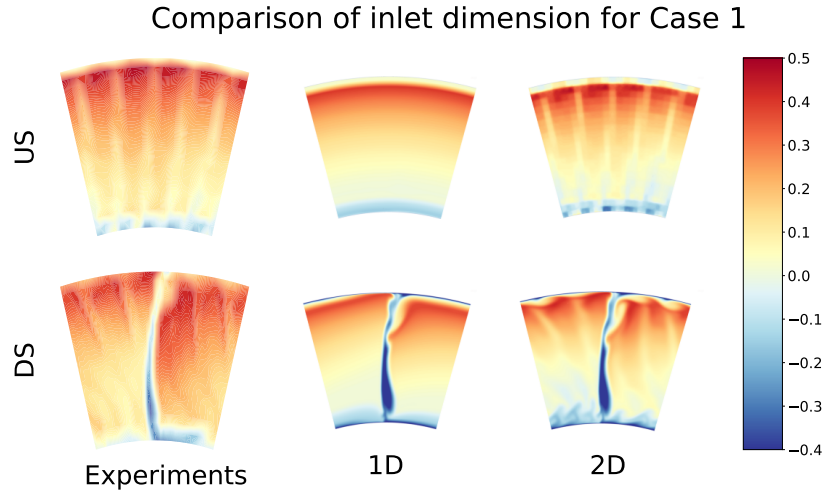


Figure 4.1: The figures are showing contour plots for C_{p0} both upstreams and downstreams, different inlets (2D and 1D) used

Figure 4.2 shows the comparison of tangentially averaged variables between the usage of 1D and 2D inlet, and figures 4.3 and 4.4 shows values for C_{p0} and swirl at different blade spans for the DS profile.

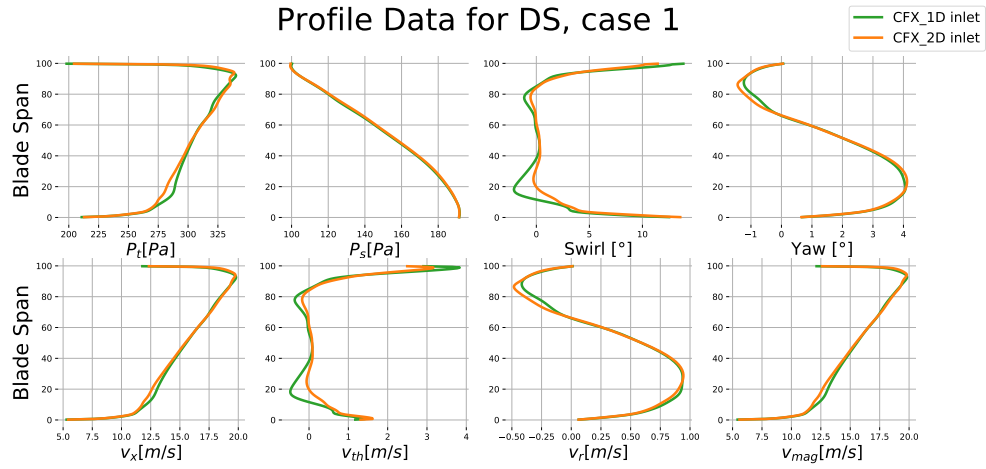


Figure 4.2: Figures showing a comparison of tangentially averaged DS values between test data, CFX data using a 1D inlet and CFX data using a 2D inlet

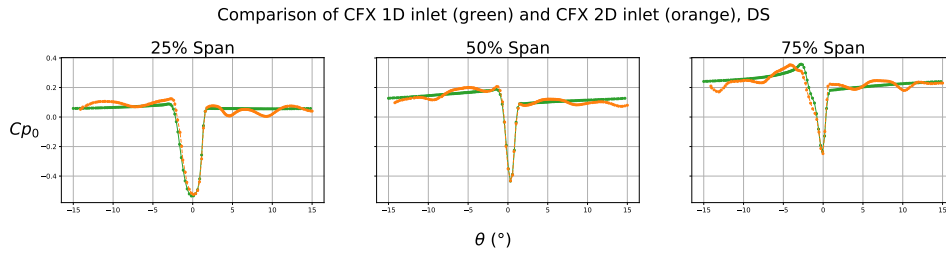


Figure 4.3: Figures showing a comparison of wake data values between test data, CFX data using a 1D inlet and CFX data using a 2D inlet

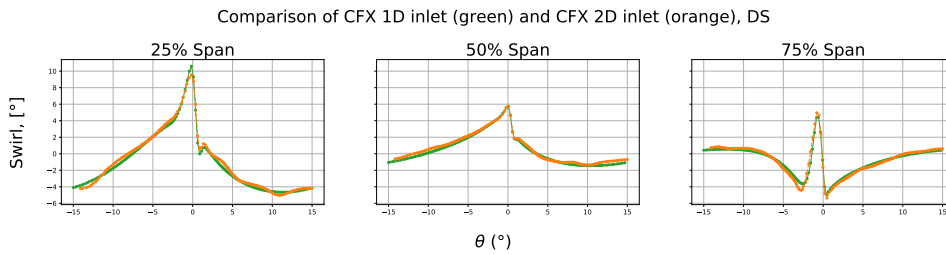


Figure 4.4: Figures showing a comparison of wake data values between test data, CFX data using a 1D inlet and CFX data using a 2D inlet

4.1.2 Inlet Extrapolation

Due to the physical nature of the MHP the entire US and DS profiles could not be covered by the probe and extrapolation measures were required. As discussed in section 3.4.1 the boundary layer could however be handled through extrapolation schemes. Three different extrapolation schemes was used in this report and compared. In the figure below, figure 4.5 contour plots for Cp_0 has been illustrated both US and DS for the data and for three different cases where three different extrapolation schemes has been used, BL0, BL1 and BL2 respectively.

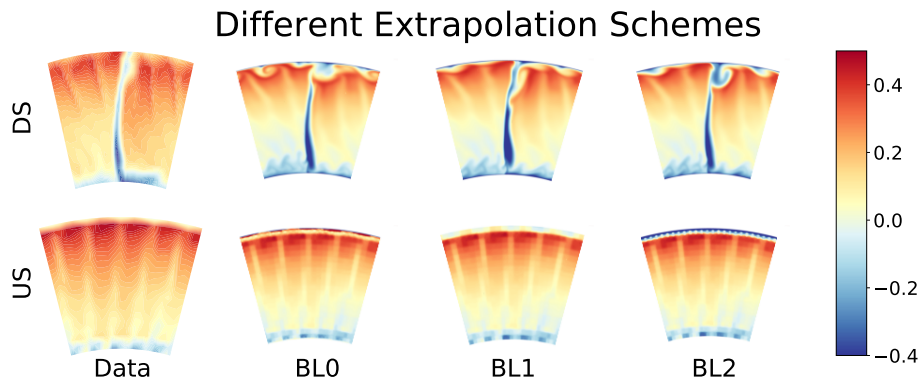


Figure 4.5: The figures are showing contour plots for Cp_0 both upstream and downstream of the OGV. Three different extrapolation schemes has been used and is highlighted in the figure

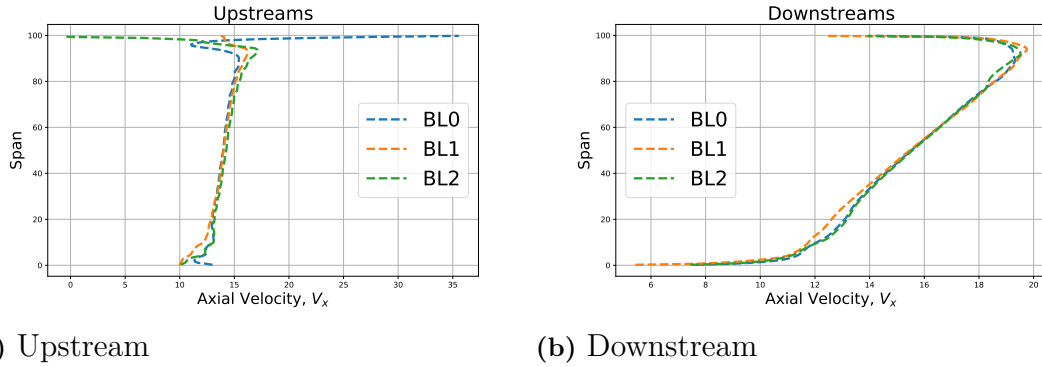


Figure 4.6: The differences between the extrapolation schemes is shown in these figures. Both figures are illustrating circumferential average axial velocities at both US and DS. The left figure is showing the axial velocity upstreams and the right figure the axial velocity downstreams. The different extrapolation schemes are highlighted with a specific color.

4.2 Impact of Turbulence Model

This section presents simulation outcomes for different turbulence models and settings. The section compares contour plots of C_{P_0} and streamlines at the blade between simulations executed with the $k - \varepsilon$ model and the Transition SST $k - \omega$ model where the transitional turbulence is modeled with the $\gamma - Re_\theta$ model. Additionally, results obtained using the conventional SST model are presented, and outcomes for various values of the a_1 coefficient tested for the standard SST-model are showcased in this section. The first and second blending function from the SST model will also be illustrated around the OGV.

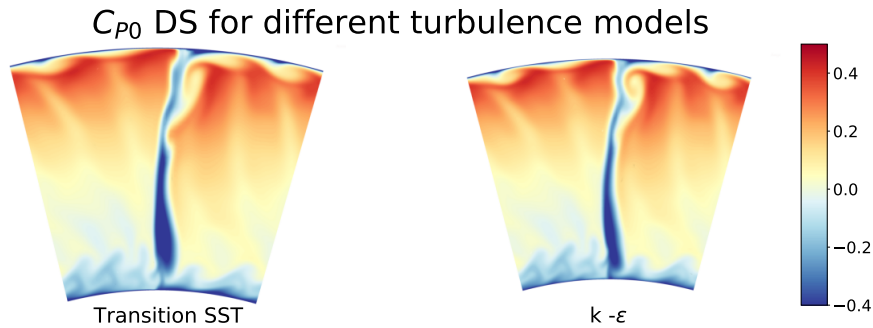


Figure 4.7: The contour plot to the left is illustration C_{p_0} downstream where the Transition SST turbulence model has been used. The right contour plot is illustrating the same variable while the turbulence model is instead replaced with the $k - \varepsilon$ model

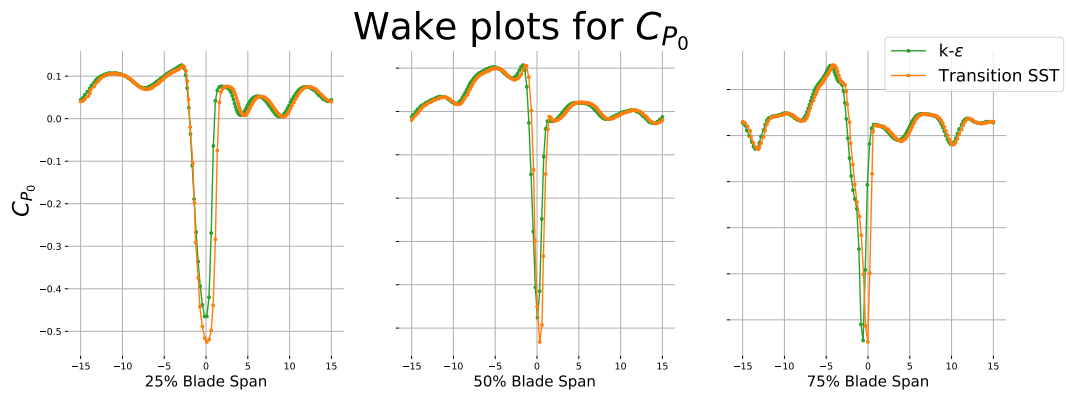


Figure 4.8: This figure is showing C_{P0} at different blade spans for two different turbulence models, namely the SST Transition model and the $k - \varepsilon$ model

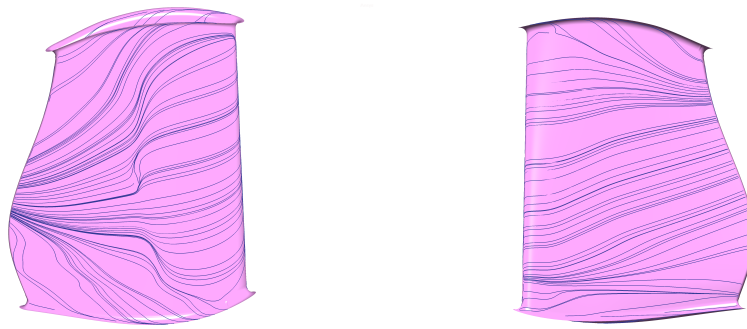


Figure 4.9: Streamlines of the airflow at the OGV for the Transition SST $k-\omega$ model

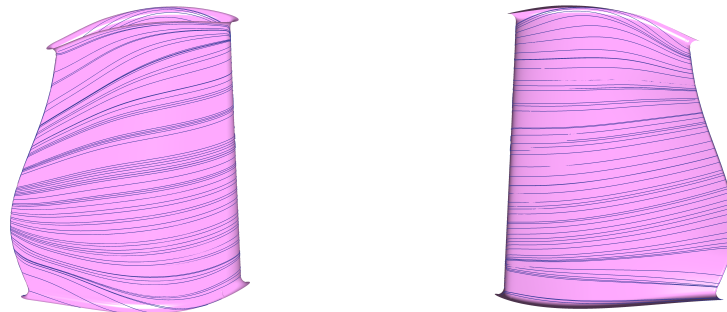


Figure 4.10: Streamlines of the airflow at the OGV for for the $k-\varepsilon$ model

Figures 4.9 and 4.10 is illustrating airflow streamlines over the OGV for two different turbulence models. The two figures on the top row is showing the streamlines at the blade for the Transition SST model and the figures at the bottom row is showing the results for the $k - \varepsilon$ model.

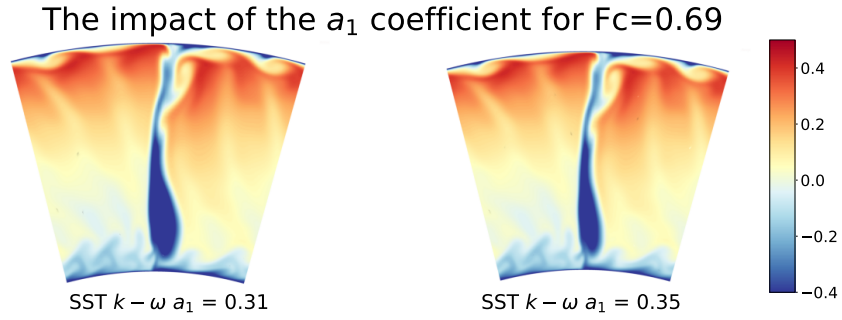


Figure 4.11: These contour plots display Cp_0 for two distinct scenarios using the standard SST $k - \omega$ model, where the a_1 coefficient has been assigned two different values

Figure 4.11 is showcasing results from simulation where the regular SST $k - \omega$ model was used with different values for the a_1 coefficient. The contour plots are showing Cp_0 downstreams where the left plot visualise the result for when $a_1 = 0.31$ and the right plot for when $a_1 = 0.35$.

Figure 4.12 shows a contour plot of the blending function F_1 (described in section 2.5.5) at 50 percent of the blade chord. The red color indicates a value of 1, signifying where the $k - \omega$ model is active and blue a value of zero, indicating the use of the $k - \epsilon$ model

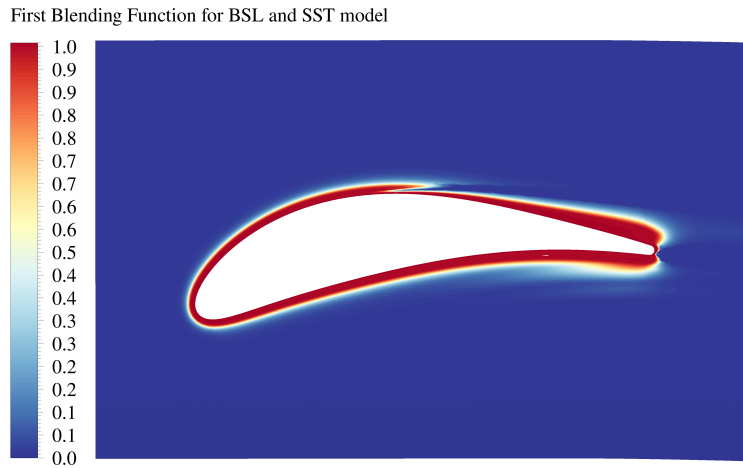


Figure 4.12: Contour plot of the blending function F_1 for a SST transition simulation

Figure 4.13 illustrates the second blending function around the OGV at the radius $r = 0.335m$ for two different cases where the a_1 coefficient has been tested for two different values.

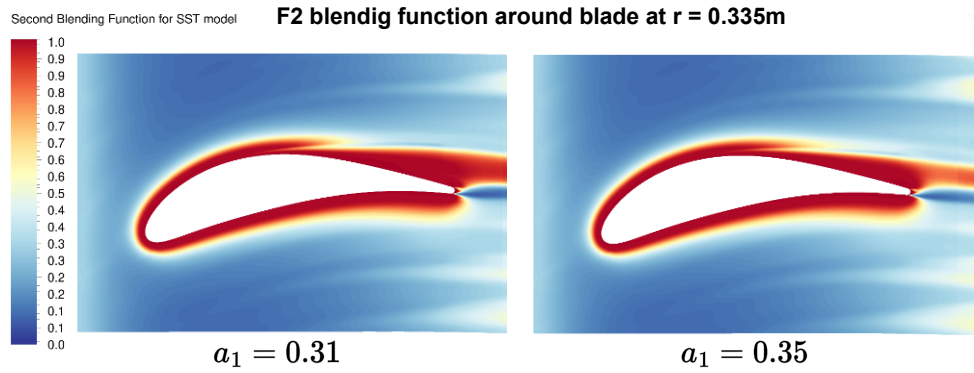


Figure 4.13: Contour plot of the blending function F_2 for SST $k - \omega$ simulations with different values of a_1

Figure 4.14 demonstrates the influence of the predefined value for the eddy length scale on the DS contour plots and figure 4.15 show the comparison of C_{P0} at 25 and 75 percent span.

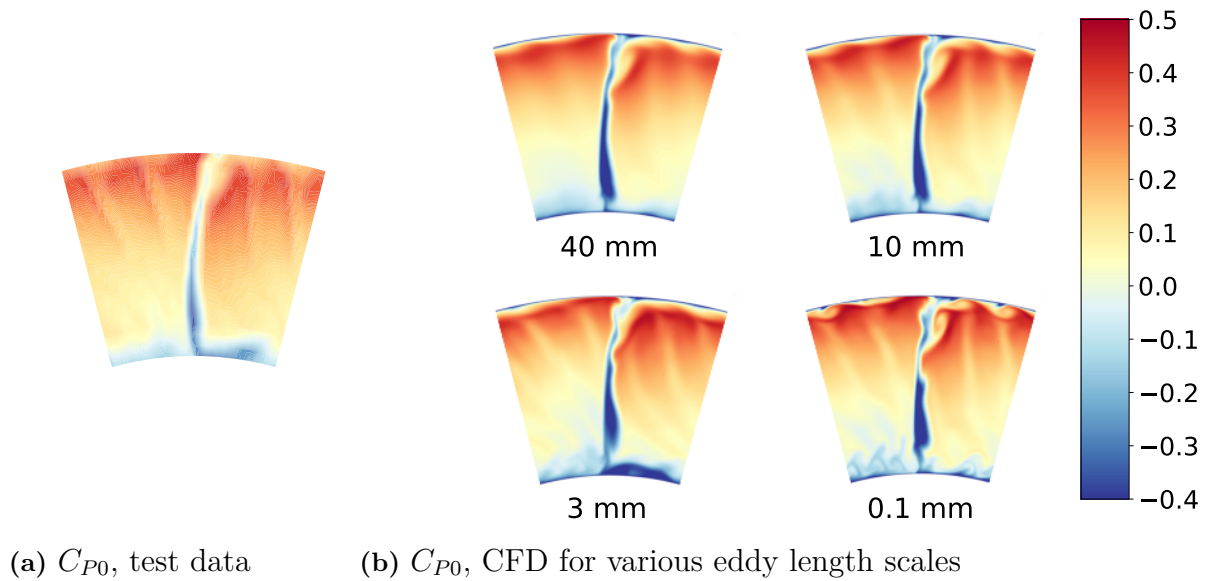


Figure 4.14: Figure illustrating contour plots of C_{P0} , case 6 for both test and simulations

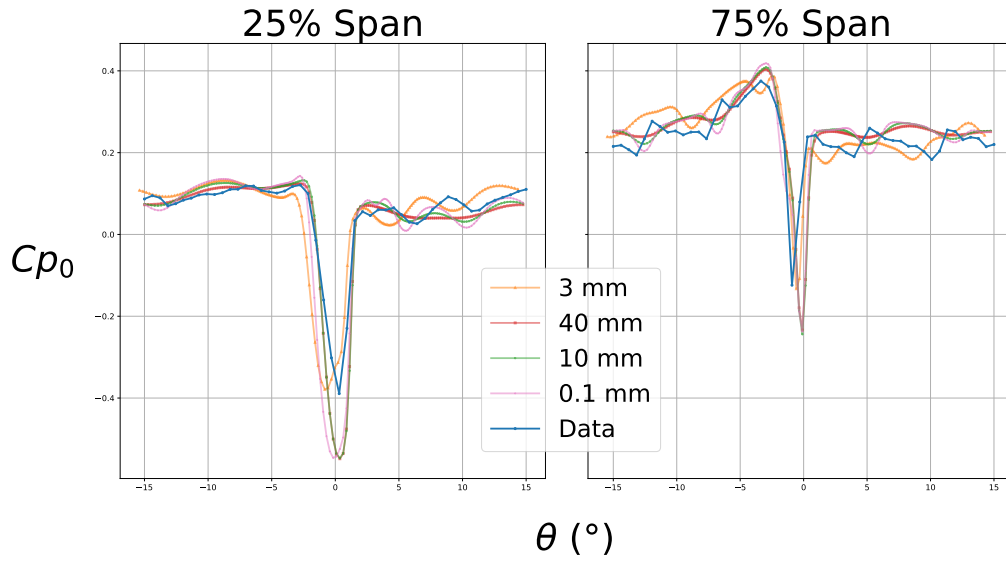


Figure 4.15

4.3 Near wall Modeling

The resulting y^+ values at the walls of the TRS domain (hub, shroud and the blade) from the simulations, only the results from case 1 is presented, are shown in figure 4.16 below.

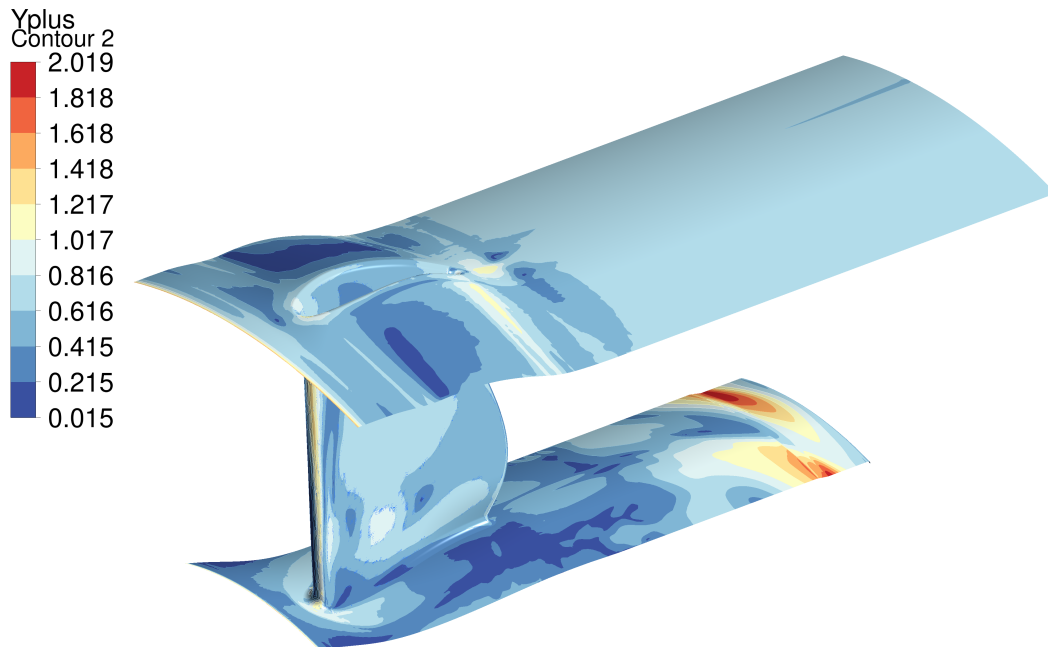


Figure 4.16: y^+ values at the TRS walls

Additional figures showing the wall regions in more detail is available in Appendix A.9.

4.4 Mesh Independence Study

A mesh independence study was conducted to determine the optimal level of mesh refinement for achieving accurate and reliable results. The study includes three different mesh refinements: a coarse mesh with 2.7 million cells, a fine mesh with 8 million cells, and a very fine mesh with 20 million cells. The sensitivity analysis focuses on C_{P0} and swirl, with contour plots illustrating the total pressure coefficient for each mesh. The following sections present the findings and discuss the impact of mesh density on the results.

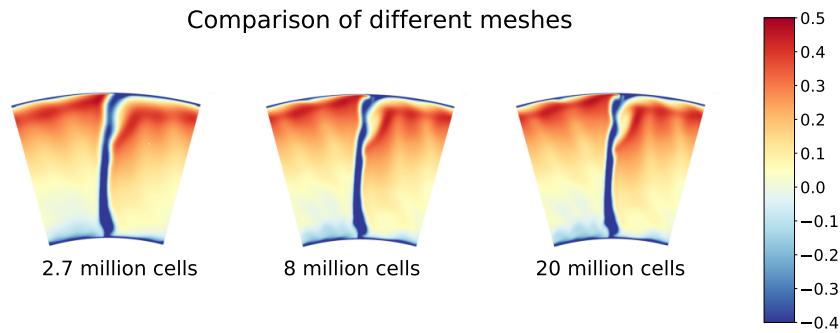


Figure 4.17: The figure is showing contour plots over the total pressure coefficient, C_{P0} downstream for three different refinements on the mesh.

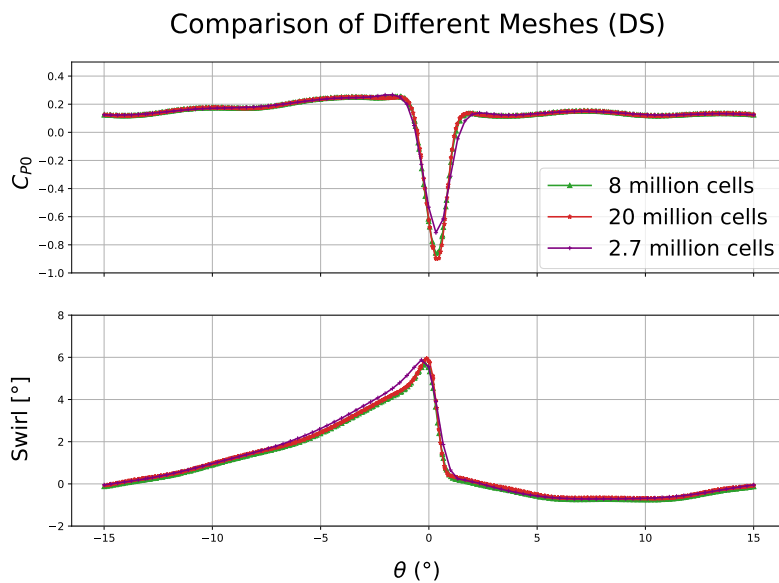


Figure 4.18: The figure above is showing C_{P0} for three different refinements of the mesh at a blade span of 50%. The graph below is instead showing the swirl.

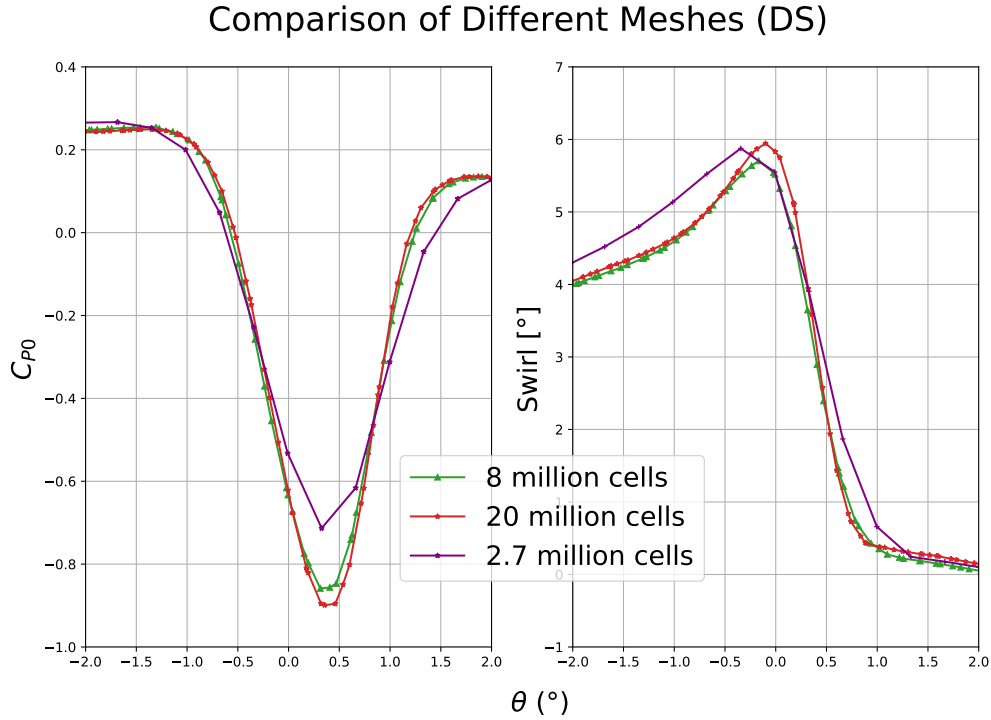


Figure 4.19: Zoomed in figure around 0 for figure 4.18

4.5 Cases 6 & 7

As mentioned in the method section describing the boundary conditions 3.4.1, the measure of whether the simulation matched the test data was to compare tangentially averaged plots upstream of the OGV and change the back pressure at the outlets until a match had been obtained. For cases 6 and 7 the pressure at both outlets were not set as identical as an additional back pressure had been applied to the HX outlet. This meant that in addition to changing the magnitude of the back pressure to match the tests, the difference in pressure between the HX outlet and the large outlet had to be adjusted to match the measured split of airflow. the resulting outlet pressure settings for cases 6 and 7 is displayed in table 4.1

Table 4.1: Pressure outlet setting and resulting splits

Case	$P_{out,top}$ [Pa]	$P_{out,bot}$ [Pa]	Split [%] (Data)	Split [%] (CFD)
6	45	190	9.1812	9.2052
7	40	190	9.3689	9.4078

With the settings in the table above the resulting tangentially averaged US plots shown in figures 4.20 and 4.21 could be obtained.

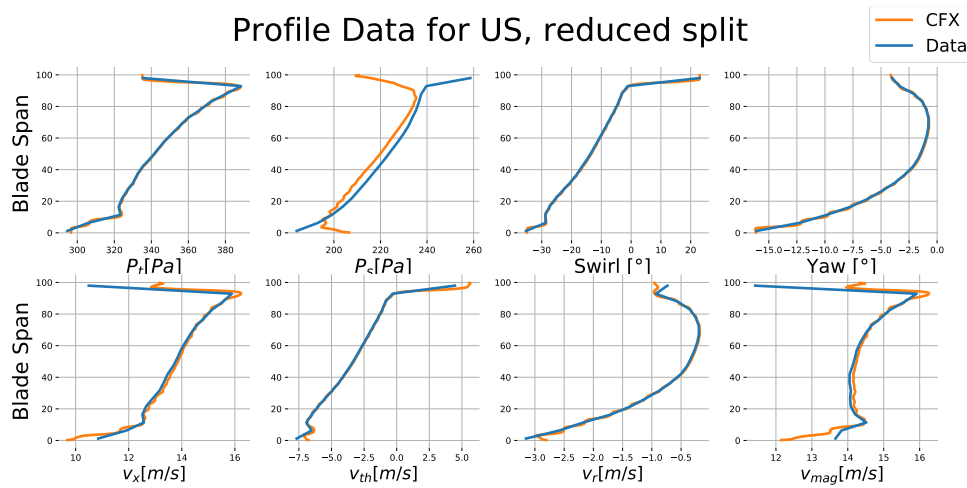


Figure 4.20: tangentially averaged data for US, case 6

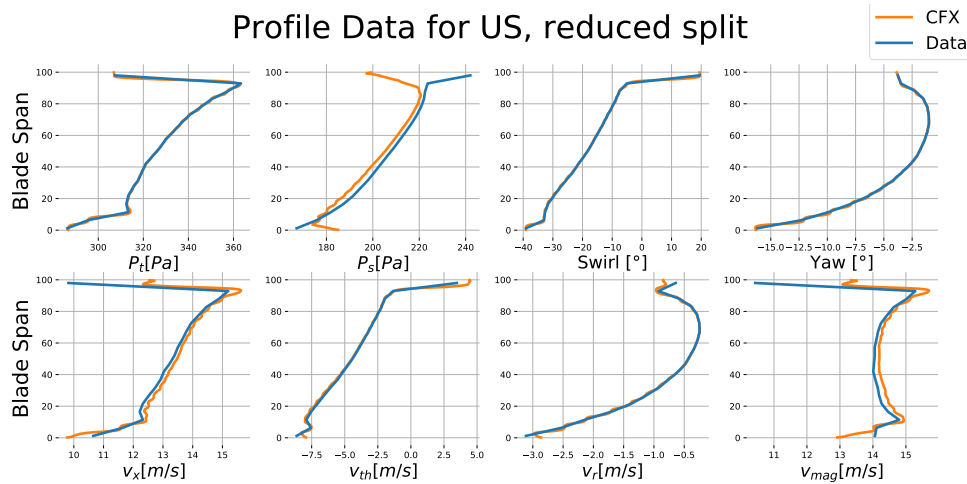


Figure 4.21: tangentially averaged data for US, case 7

The resulting contour plots of the total pressure coefficient DS of the OGV for cases 6 and 7 both for test data and simulations can be seen in figure 4.22.

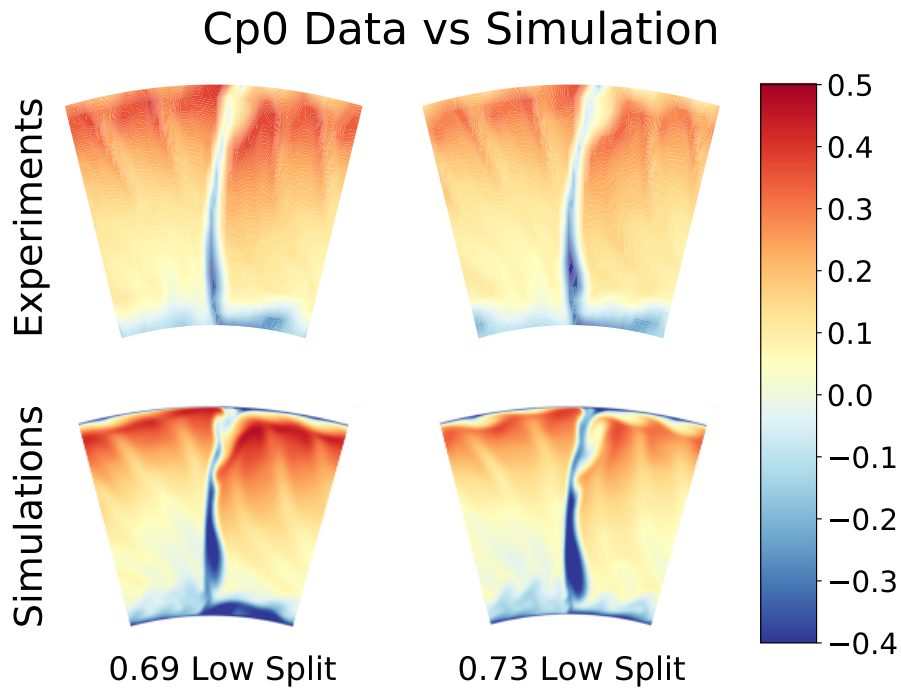


Figure 4.22: Contour plots for C_{p0} at the outlet of the TRS. The figure is presenting results for both measured data and data from simulation results, cases 6 and 7

Tangentially averaged plots were also measured at the DS profile and some of the plots are presented in figures 4.23 and 4.24.

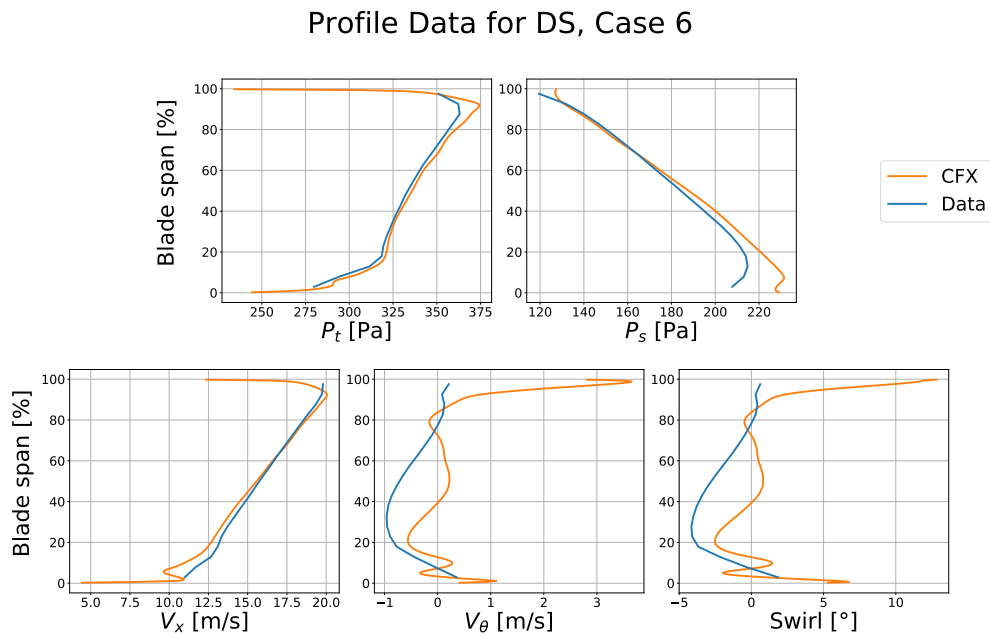
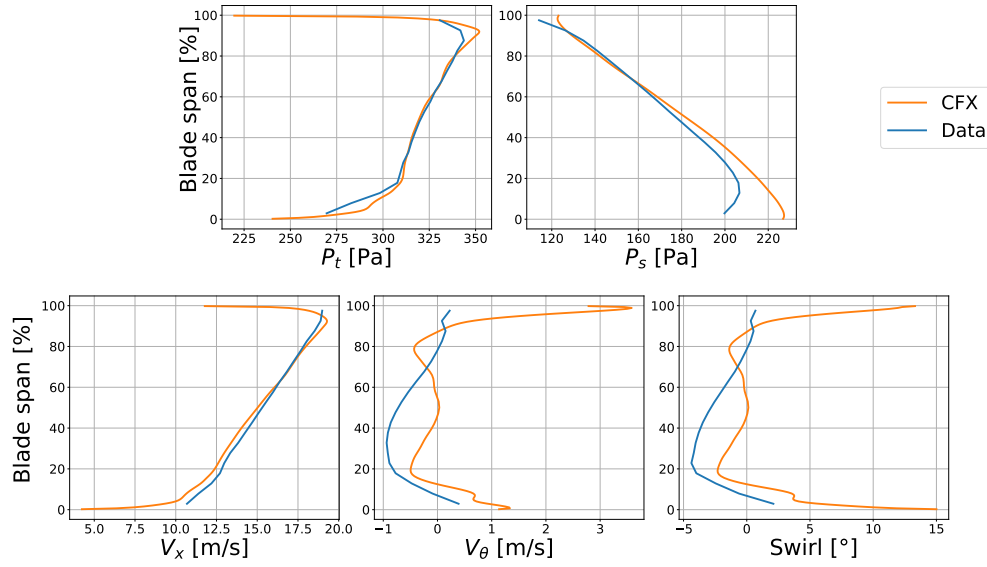
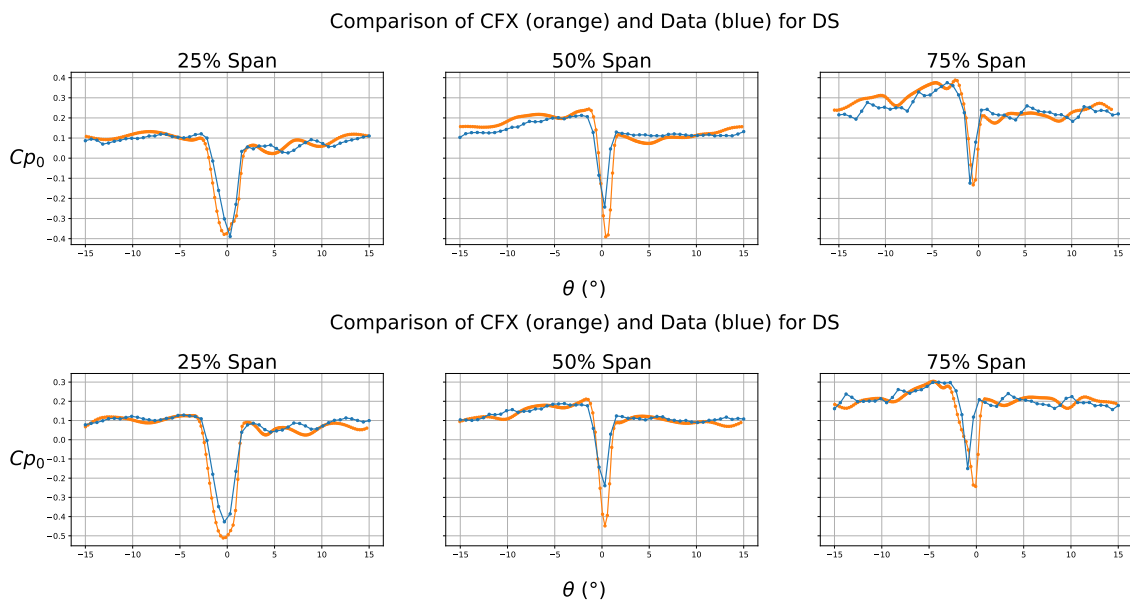


Figure 4.23: tangentially averaged data for DS, case 6

Profile Data for DS, Case 7

**Figure 4.24:** tangentially averaged data for DS, case 7

In addition to the tangentially averaged plots DS of the OGV the value of C_{P0} and swirl at certain radial spans of the DS profile was measured and are shown in figure 4.25 and 4.26.

**Figure 4.25:** C_{P0} data, case 6 and 7

4. Results

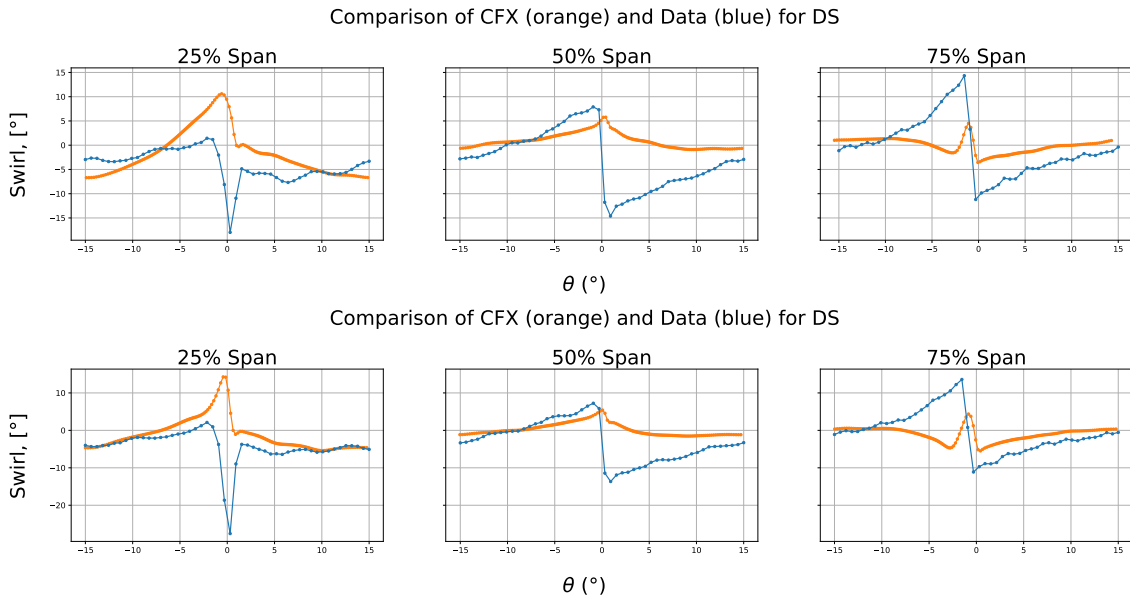


Figure 4.26: Swirl data, case 6 and 7

Tangentially averaged plots for the radial velocity and the resulting yaw angle (yaw, $\beta = \frac{V_r}{V_\theta}$) for case 6 and 7 are shown in figure 4.27.

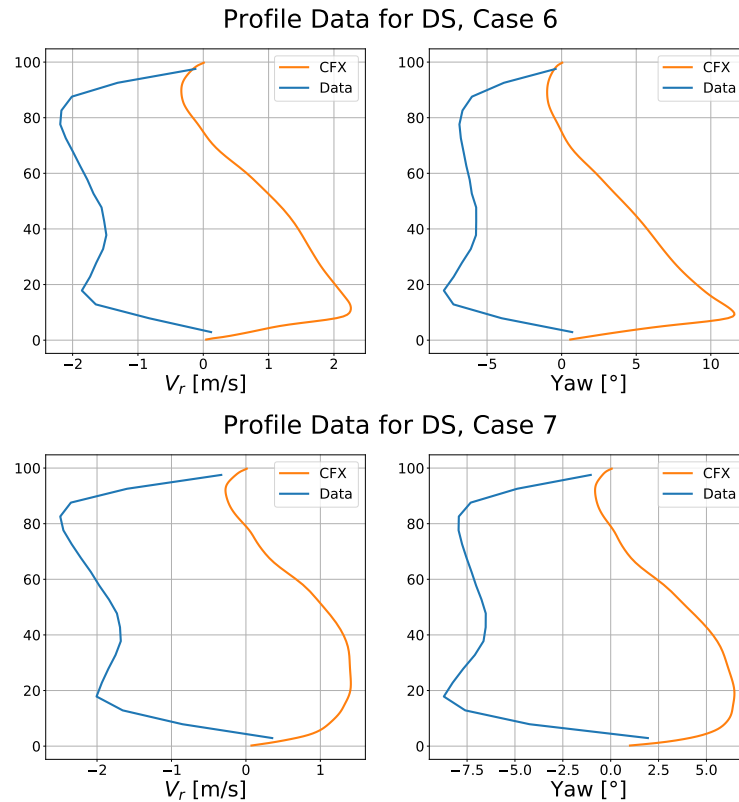


Figure 4.27: Tangentially averaged DS plots for radial velocity and yaw, case 6 and 7

In addition to these results the streamlines at the PS and SS of the OGV blades are

reported in Appendix, A.20 and A.24.

4.6 Cases 1-3

Due to the similar nature of many of the plots for cases 1-3 most of the plots presented in this section are only for case 1, corresponding plots for case 2 and 3 are included in Appendix. The cases with a blocked MHP had their US tangentially averaged plots matched in the same way as cases 6 and 7 and these plots can be found in Appendix (A.6a, A.10a, and A.14a). The required outlet pressure settings are seen in table 4.2.

Table 4.2: The presented table shows the back pressure applied in the simulation to achieve inlet profiles that correspond to the experimental data for specific cases.

Case	$P_{out,top}$ [Pa]	$P_{out,bot}$ [Pa]	Split [%] (Data)	Split [%] (CFD)
1	40	40	[N/A]	14.0723
2	40	40	[N/A]	14.2370
3	40	40	[N/A]	14.0855

The corresponding DS results for case 1 can be seen in figure 4.28, the DS plots for case 2 and 3 are in Appendix (A.10b, A.14b).

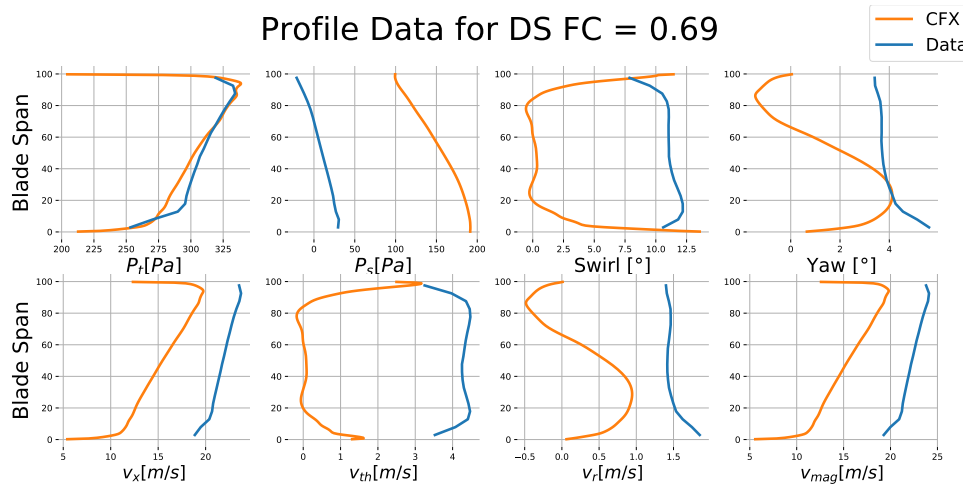


Figure 4.28: Circumstantially averaged data for DS, case 1

Contour plots of the total pressure coefficient for simulations and test data for both US and DS are presented in figures 4.29 and 4.30.

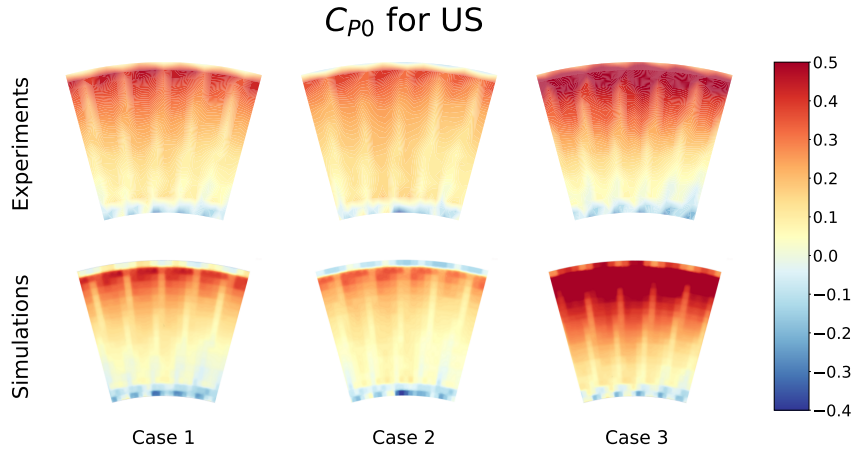


Figure 4.29: Contour plots for C_{p0} at the inlet of the TRS. The figure is presenting results for both measured data and data from simulation results, cases 1-3

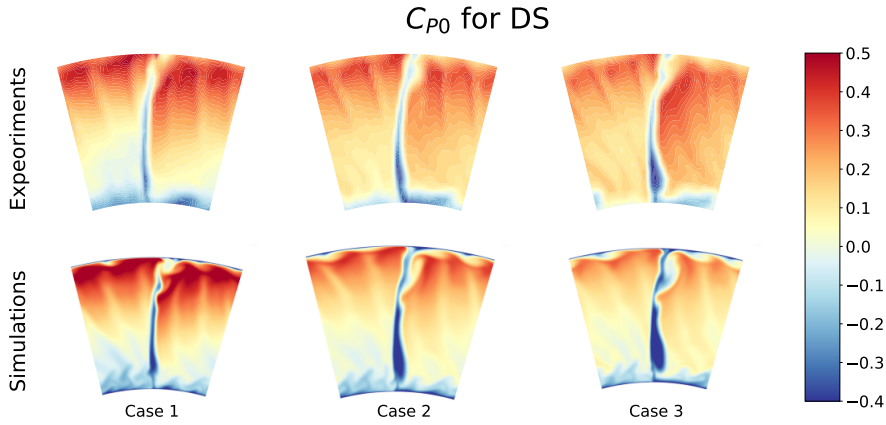


Figure 4.30: Contour plots for C_{p0} at the outlet of the TRS. The figure is presenting results for both measured data and data from simulation results, cases 1-3

Figure 4.31 shows velocity contours as well as streamlines at a span of 0.335m (equivalent to about 7 percent blade span)

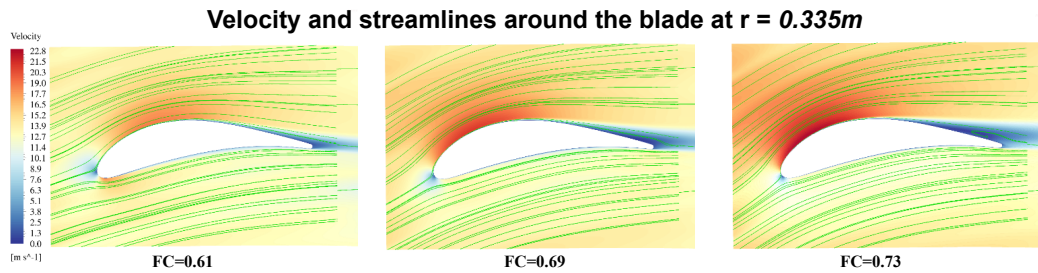


Figure 4.31: Figure showing streamlines and velocity contours around the blade for cases 1, 2 and 3

Wake swirl and C_{p0} data for case 1 is shown in figure 4.32

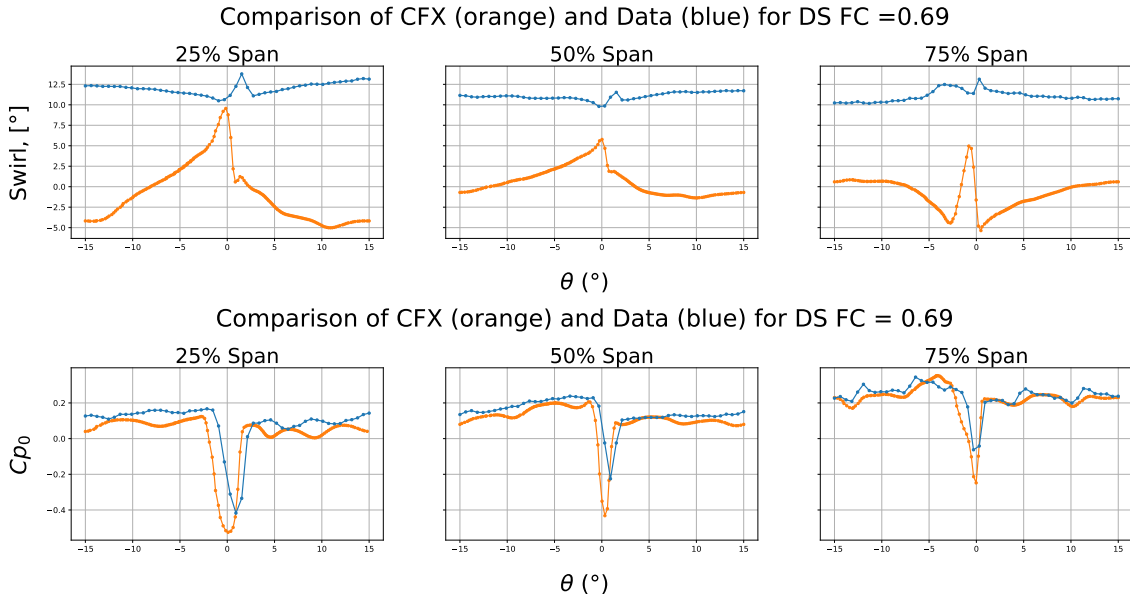


Figure 4.32: Wake data for swirl and C_{p0} downstreams for case 1 where $Re=235k$ and $Fc=0.69$

This section also presents figures that visualize computed streamlines over the OGV, showing streamlines over both the suction-side and pressure-side. The presented streamlines over the blade reveal fluid flow patterns in the steady state solution..

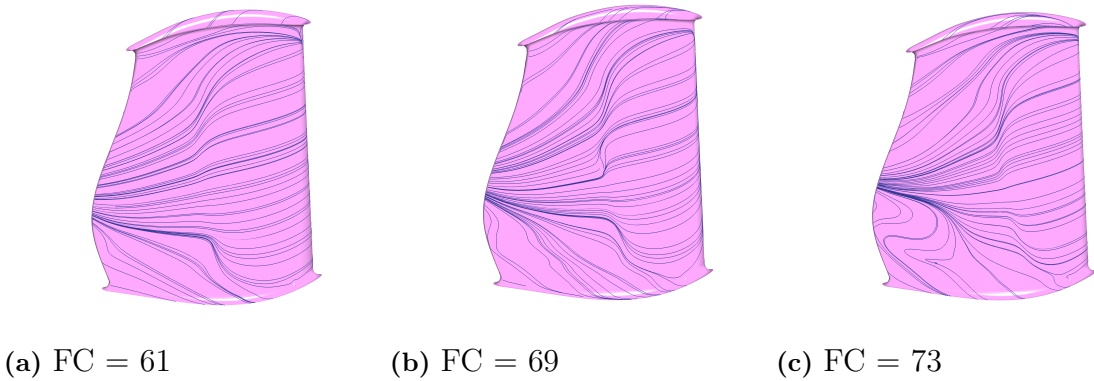


Figure 4.33: Streamlines of the airflow at the OGV, suction-side

The figure above, figure 4.33 is showing the computed streamlines over the OGV for three different cases. The streamlines in this figure is illustrated at the suction side of the OGV. The figure below, figure 4.34 is instead illustrating the streamlines on the pressure side of the OGV for the three different cases.

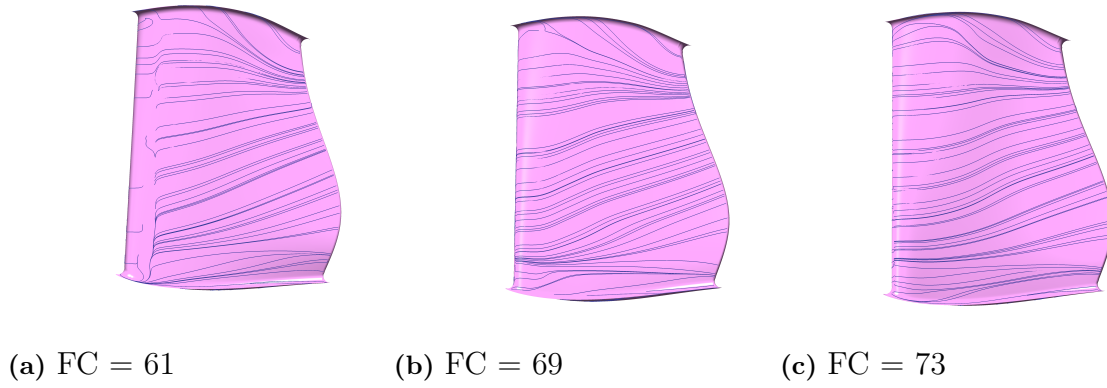


Figure 4.34: Streamlines of the airflow at the blade, pressure-side

4.7 Split Variation

As described in section 3.7 simulations without corresponding test data were carried out in order to measure the effect of the split on the effectiveness by varying the back pressure at the air outlet of the heat exchanger. As this was the only change the inlet temperature and mass flow of water remained constant. If in addition the heat capacities are assumed constant the value of C_{min} can be plotted as a function of the split, shown in figure 4.35, according to equation 2.10.

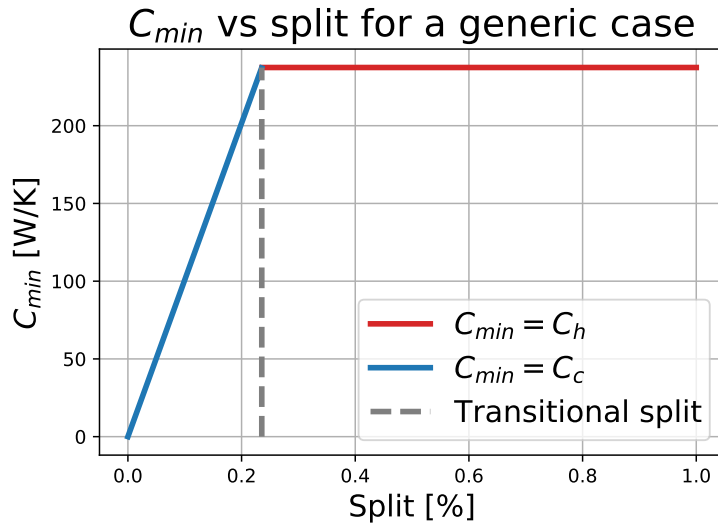
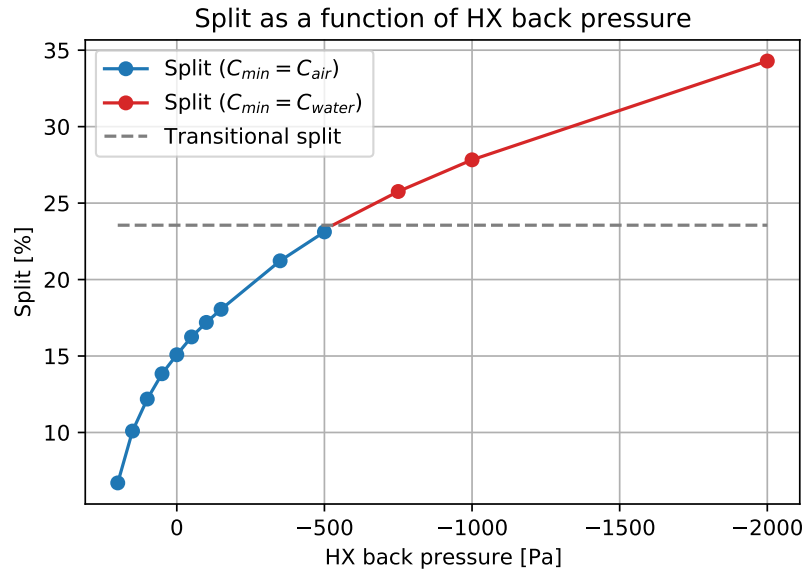


Figure 4.35: C_{min} as a function of split

Note that in the figure above a constant inlet mass flow is assumed (1 kg/s) although this varies somewhat as the specified inlet conditions are pressure and flow direction rather than mass flow. Table 4.3 shows the back pressure, the split and the C_{min} obtained when the back pressure was varied and figure 4.36 show the same values plotted.

Table 4.3: Data for the cases where the split was varied by changing back pressure.

Back pressure [Pa]	Split [%]	C_{min}
+200	6.7013	C_{air}
+150	10.0923	C_{air}
+100	12.1849	C_{air}
+50	13.8403	C_{air}
0	15.0827	C_{air}
-50	16.2484	C_{air}
-100	17.1966	C_{air}
-150	18.0532	C_{air}
-350	21.2251	C_{air}
-500	23.1172	C_{air}
-750	25.7586	C_{water}
-1000	27.8296	C_{water}
-2000	34.2903	C_{water}

**Figure 4.36:** Split as a function of the HX back pressure

The data obtained from these simulations is also used to calculate the effectiveness at the different split values, which will be presented later.

4.8 Comparison of Simulations for Different Geometries

The results in this section only result for the case 1 are presented ($Re=235$, $FC=0.69$), and the result are from 3 different cases, simulations using only the OGV geometry described in section 3.7, simulations with only air and simulations where both air and water are enabled.

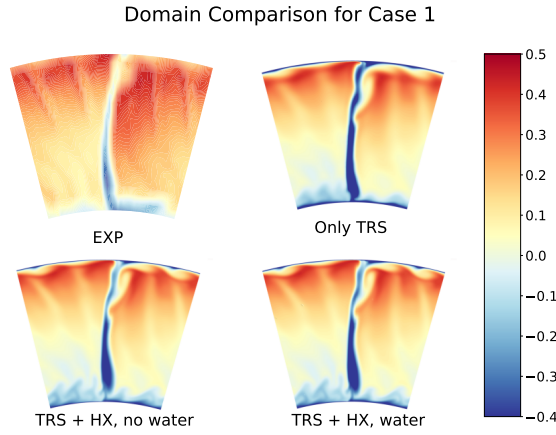


Figure 4.37: DS profiles for different geometries

Figure 4.38 shows the comparison between test data and CFD for simulations only using the extended OGV domain as well as simulations for the entire domain. All simulations are done with identical boundary conditions except for the pressure at the respective outlets, and identical turbulence settings.

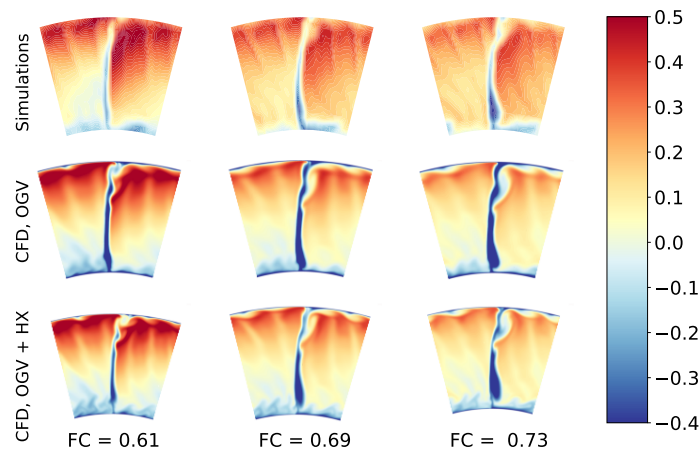


Figure 4.38: DS profiles for experiments and CFD, both only using the TRS geometry and the entire domain (TRS + HX) for cases 1-3

The following figures, 4.39 and 4.40 shows streamlines at the OGV, both for PS and SS, for case 1 using different domains.

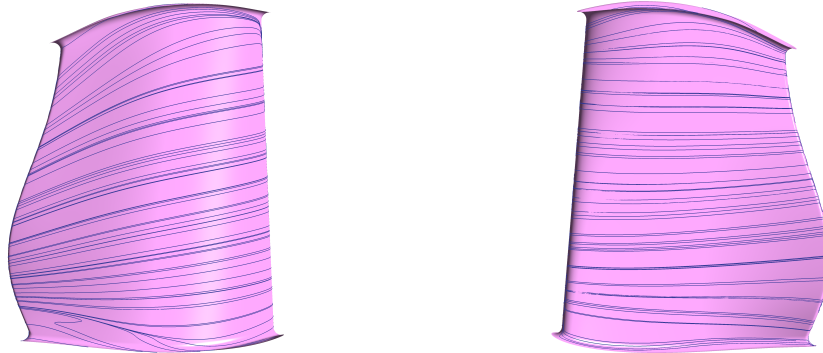


Figure 4.39: Streamlines of the airflow at the OGV for the geometry containing only the OGV

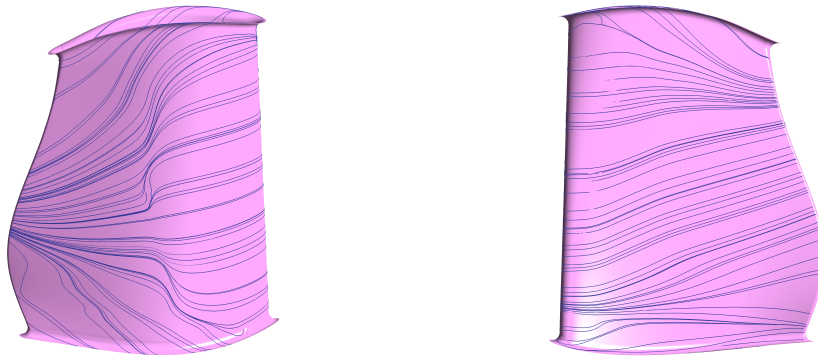


Figure 4.40: Streamlines of the airflow at the OGV for the geometry containing OGV and HX, Water enabled

In table 4.4 typical solution times for the investigated cases are presented.

Table 4.4: Solution time for various simulations

Simulation	Solution Time [s] (200 cores)
Only OGV	3.59×10^2
OGV+HX, no water	1.36×10^4
OGV+HX, water	1.39×10^5

4.9 Effectiveness of the HX

Using equation 2.10 as well as the obtained values for the splits, as presented in table ?? the effectiveness values for the HX at the different operating points was calculated using measured temperatures and mass flows from CFX presented in table 4.5, the data is further visualized in figure 4.41 The effectiveness values themselves are presented in table 4.8.

Table 4.5: Air temperatures from CFX

Case	$T_{air,top}$ [$^{\circ}C$]	$T_{air,bot}$ [$^{\circ}C$]	ΔT [$^{\circ}C$]
1	19.3030	33.4560	14.1530
2	19.7090	33.8900	14.1810
3	19.3030	33.4840	14.181
6	19.3210	36.1860	16.8650
7	18.6660	36.7050	18.0390

Table 4.6: Both the inlet and outlet temperatures for the water is tabled here. The table presents values both from measurments and from simulations

Case	$T_{water,inlet}$ [$^{\circ}C$]	$T_{water,out}$ (Data) [$^{\circ}C$]	$T_{water,out}$ (CFD) [$^{\circ}C$]
1	33.4560	39.6683	39.7480
2	33.8900	39.8321	40.3753
3	33.4840	39.7518	40.1803
6	36.1860	42.2756	41.9447
7	36.705	42.7233	42.1443

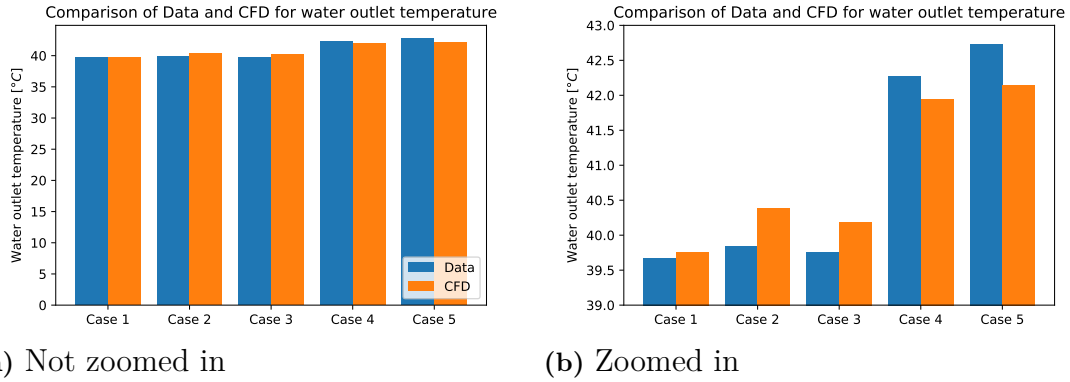


Figure 4.41: Water outlet temperature comparison

The inlet temperatures for water and air is not included in the table above as they are set as boundary conditions and thus are constant but case dependent. The mass flow in the studied 30° sector for both air and water from CFD is presented in table 4.7.

Table 4.7: Mass flow data from CFX

Case	$\dot{m}_{water}$ [kg/s]	$\dot{m}_{air,HX}$ [kg/s]	C_{water} [W/K]	C_{air} [W/K]	C_{min}
1	0.0578	0.1214	241.77	122.15	C_{air}
2	0.0578	0.1241	232.46	124.87	C_{air}
3	0.0578	0.1215	241.77	122.31	C_{air}
6	0.0457	0.0806	191.21	81.09	C_{air}
7	0.0457	0.0745	191.21	74.96	C_{air}

Table 4.8: The HX effectiveness for the different cases can be seen from this table together with a averaged values for case 1-3 and 6-7 separately

Case	Effectiveness
1	0.5357
2	0.4960
3	0.5026
Mean, cases 1-3	0.5114
6	0.5598
7	0.5874
Mean, cases 6 and 7	0.5736

The effectiveness of the cases where the split was varied more widely as reported in section 4.7 was also calculated and a plot where these are shown along with the data points can be seen in figure 4.42

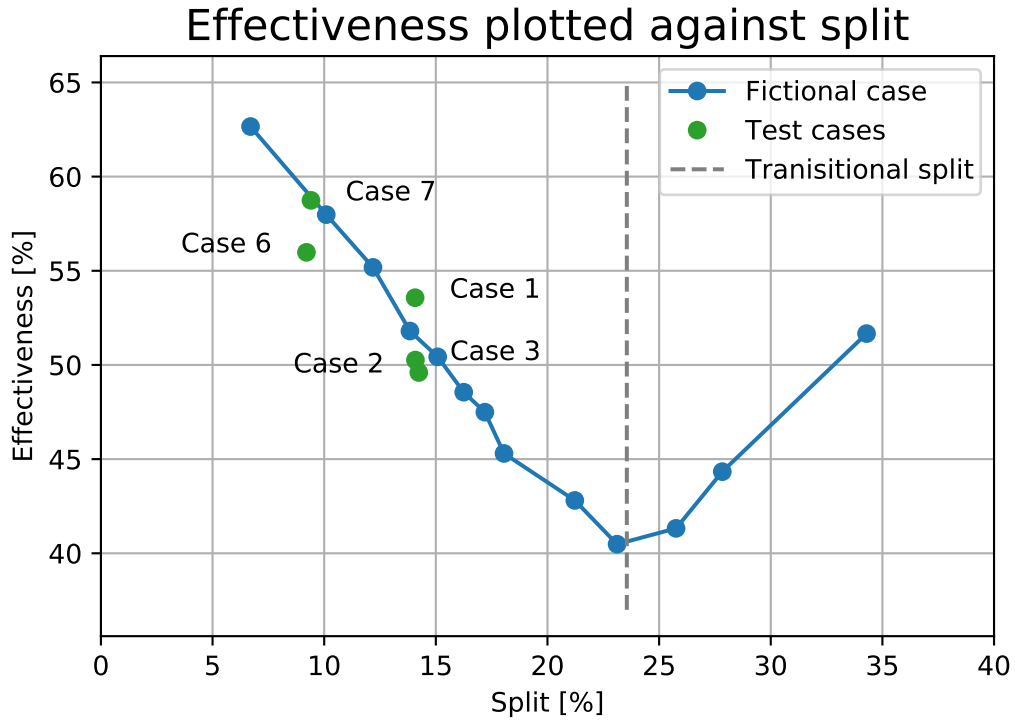


Figure 4.42: Effectiveness plotted against split

Also included in the figure above is the critical split, the approximate split where C_{min} transitions from C_{air} to C_{water} .

5

Discussion

The structure of the discussion will largely follow that of the "Results" chapter with sections that correspond to those in the results. In addition to these sections some more sections will be added such as improvements and further studies which give a more holistic discussion of the thesis in general.

5.1 Inlet Specification

When comparing the contour plots obtained when a 1D compared to a 2D inlet is used as presented in figure 4.1 the most striking difference is the removal of the NGV wakes originating from the LPT upstream of the entire TRS in the 1D case. The wake due to OGV trailing edge seems to be identical regardless of the inlet dimension with some notable differences at the hub and shroud where a more "wavy" pattern can be seen for the 2D case. The overall similitude between the two methods of specifying the inlet is further seen in figure 4.2 where only slight differences are found. Even at the more detailed level present in figure ?? the difference is almost negligible except for the aforementioned waviness. As the implementation of a 2D inlet can be quite cumbersome, for example the CSV-file that is loaded into *CFXpre* contains 21 rows of information for the 1D inlet compared to the 551 rows present in the 2D case. For the 2D case another detail to keep track of is the orientation of the plane as the 2D inlet can be mirrored which the 1D inlet can not as well as the fact that the inlet extrapolation becomes more time consuming. All the simulations in this thesis is however done using a 2D inlet as the extra detail from these simulations was deemed worth the additional time cost.

The simulations that are used when comparing the impact of the methods for inlet extrapolation are all done for case 1 and the resulting contour plots at US and DS are shown in figure 4.5. As the main extrapolation zone, per section 3.4.1 is close to the shroud the expected difference is also to take place in this region. This is also shown to be the case as both BL0 and BL2 extrapolations produce larger regions with low C_{P0} at the tip of the OGV wake. Also the shroud regions away from the blade wake are differing between the simulations although these regions are rather small. The hub regions close have more similar appearance between the extrapolation schemes. The rest of the simulations were done using a BL1 extrapolation as per a previous study of this phenomenon [9], and it would appear that this choice is somewhat validated by the simulations done in this thesis. It must be noted that none of the extrapolation measures produce results that are notably more similar

to the experimental values than the others so concluding that BL1 extrapolation is definitely more accurate than the others solely based on the results in this thesis would be unwise. Another uncertainty with the results for this comparison is when the resulting US profile is observed, see figure 4.6 the axial velocity for the BL0 extrapolation does not behave in the way expected, which would be the behavior seen in figure 3.10 where the value at the hub would be the same as the two points below it. Apart from this deviation the tangentially averaged values are very similar for the different extrapolation schemes.

The above reasoning lead to the inlet being specified by a 2D profile with a BL1 extrapolation in the hub and shroud regions.

5.2 Turbulence Modelling

Comparing the contour plots over C_{P_0} obtained for the two different turbulence models shown in figure 4.7 both similarities and differences can be noted. The largest difference can be seen from the wake due to the OGV trialling edge where it is seen that the Transition SST model predicts a thicker wake and the $k - \varepsilon$ predicts a more narrow wake. In addition to the wake due to the OGV trialling edge no differences are noted in the contour plot. As described in the theory the k-epsilon model solves equations in terms of wall distance, while the SST model uses a blending function to transition between a $k - \varepsilon$ -like formulation in the near-wall region to a $k - \omega$ -like formulation in the outer region without the use of wall distance. The Transition SST model was chosen in order to take advantage of using the $k - \varepsilon$ model in the free stream. From figure 4.8 it is seen that the two different turbulence models predicts C_{P_0} similarly with great accuracy in the free stream. Differences is seen around 0 where the wake due to the OGV trailing edge is located, C_{P_0} is predicted

The Transition SST model compared to the $k - \varepsilon$ model predicted different flow path on the OGV which can be seen in figures 4.9 and 4.10. Specifically, the Transition SST model predicts separated flow path, while the $k - \varepsilon$ model predicts a smoother and more uniform flow path along the blade, without significant variations in direction. The variations in predicted streamlines between the two turbulence models is due to their differing approaches to modeling turbulence near walls. The Transition SST $k - \omega$ model uses the $k - \omega$ model to resolve turbulence close to walls and utilizes the $\gamma - Re_\theta$ model to capture the effect of laminar-to-turbulent transition which is important in near wall regions. The $k - \varepsilon$ model's ability to predict laminar-to-turbulent transitions is not as precise as the Transition SST model, which could explain why the two models predict different types of streamlines around the OGV.

To examine how the a_1 coefficient impacts both the regular SST $k - \omega$ model and the Transition SST $k - \omega$, simulations were conducted using various values of a_1 . As described in the theory section 2.5.5 the default value of $a_1 = 0.31$ could in some cases over predict separation zones, increasing the value to 0.335 could according to the theory solve this kind of problems. Simulations were therefore conducted with

a_1 set to 0.335 and 0.35 which for the regular SST $k - \omega$ model predicted a smaller separation zone compared with a case where $a_1 = 0.31$. Hence, increasing the coefficient did as the theory suggested and affected the separation zone which can be seen in the figure down below, figure 4.11. Increasing a_1 beyond 0.335 did not affect the solution, as $a_1 = 0.335$ and $a_1 = 0.35$ produced identical results. Interestingly, the Transition SST $k - \omega$ model was unaffected by the increase in the a_1 coefficient, regardless of the chosen value

The shear stress limiter in the SST k - ω turbulence model is based on a modification of the turbulent viscosity function. The model adjusts the turbulent viscosity near walls, where strain rates are high, to avoid producing nonphysical predictions such as excessively large values of turbulent viscosity. This is accomplished by using a wall-normal distance-dependent limiter on the turbulent viscosity, the F_2 -function which is given by equation 2.51 in section 2.5.5. The second blending function, F_2 is illustrated around the OGV at $r = 0.335m$ in figure 4.13 for both the default value of a_1 which is $a_1 = 0.31$ and $a_1 = 0.35$. It is seen that F_2 is thicker around the trailing edge for the case where the $a_1 = 0.31$ compared to when $a_1 = 0.35$. Although the standard SST model is recognized for yielding less precise results than the Transition SST model, it was intriguing to explore how the a_1 coefficient affected the forecast of the separation zone in the standard model. The study found that the accuracy of the separation zone prediction was significantly influenced by the a_1 coefficient.

The impact of the eddy length scale can be seen from figure 4.14 where four different values have been used and simulated for. It seems that a larger value on the eddy length scale yields in that the wakes due to the NGVs disappear. Setting the eddy length scale to a large value also affects the airflow near the hub and shroud which can be seen in the figure. The wake due to the OGV is also affected by the eddy length scale, comparing the contour plots for when the value is set to $40mm$ compared to $0.1mm$ it is shown that the large wake in the middle shrinks when the value is low. The presence or lack thereof can also be seen in figure 4.15 as indicated by the smaller peaks and valleys next to the central OGV wake. The straightness of the red and green lines is indicative of the lack of NGV wakes found in these cases. As for the OGV wake the test data is most closely matched using an eddy length scale of $3mm$ as the others over predict its depth. No matter the eddy length scale the CFD over predicts the width of the OGV wake at 25% span where as all wakes are similar in width for the 75% span case.

The eddy length scale in the simulations conducted in this thesis was based on previous papers at GKN, the value was recommended to be set to $3mm$ before the investigation of the impact of the eddy length scale had been made. Luckily, by doing this analysis it is seen that this value seems to agree the best with the test data although there might be an even better value that has not been studied. The eddy length scale can also be affected by which turbulence model is used, this study is done for the Transition SST model only.

5.3 Near Wall Modelling

As for the value for $y+$ obtained from CFD it is presented that the maximum value of the $y+$ in the TRS section presented in figure 4.16 is slightly larger than two. This is above the desired value of unity although the area where such large $y+$ values is measured is small, the majority of the TRS area showing a $y+$ value substantially lower than unity. Of importance is however to note that the studied case presented in this figure is a case where the Reynolds number is 235.000. If a case with a higher Re, such as case 4 where Re=350.000, were simulate it is possible that the increased velocity would lead to a larger $y+$, possibly larger than unity at a substantial portion of the area. Especially the leading edge of the OGV would obtain exceedingly large $y+$ as the value is close to unity for Re=235,000. Having a $y+$ value of unity is however only a guideline value and not an absolute rule as this value is arbitrarily chosen as this value usually gives the width where the viscous sub-layer is fully resolved while giving enough resolution at wall to where the shear stress can be determined. In order to improve the $y+$ value to guarantee it's value to be less than unity a measure is to increase the resolution of the mesh, which was not done as increasing the mesh resolution would mean re-meshing the entire geometry, something that would be tiresome as the original mesh took approximately a month to obtain. In some cases the CAD model has to be redesigned with finer tolerance in order to obtain good $y+$ values. The unity value of $y+$ is a target value for the transition SST and not necessarily for other turbulence models such as the k- ε or k- ω models. In these models the viscous sub-layer is not fully resolved and needs to be model-ed, usually by employing wall functions up to a $y+$ of around 30. This means that when comparing the results from different turbulence models the type of mesh used for the simulations has to be considered, a mesh suitable for the SST-transport model is not necessarily optimal for other models as other inflation sizes are required. In this thesis the same mesh is used regardless of turbulence model which is not optimal.

5.4 Mesh Independence Study

From the mesh independence study conducted it can be seen that the numerical solution of what happens in the free stream is very accurate independent of which mesh is used. From figure 4.18 it is seen that C_{p0} and swirl in the free-stream is predicted very similar for all three mesh refinements. The mesh independence study reveals that the most significant differences occur in the wake region due to the OGVs trailing edge. However, when comparing the contour plots displayed in Figure 4.17 for the three different mesh refinements, no substantial differences are observed. A closer examination of the first contour plot, which corresponds to the case with 2.7 million cells, indicates a lower resolution compared to the other two plots. Looking closer into the central wake some larger differences can be noted, this can be seen from figure 4.19. It is clear that the coarser mesh predicts a lower magnitude of C_{p0} which is reasonable due to the lower number of cells and the larger cell size of the elements. It is also seen that the differences between the fine mesh

and the very fine mesh are almost negligible. It is important to consider that the region housing the central wake is particularly sensitive due to the crucial role of the OGV. In contrast to the use of tetrahedral elements in the mesh independence study, the meshes employed in this thesis were constructed using hexa elements. The total number of cells in the mesh used for the thesis was 3.5 million. Despite the variation in mesh type compared to the mesh independence study, it is concluded that the numerical solutions obtained in this thesis remain independent of the mesh and are considered reliable.

5.5 Cases 6 & 7

In order to match the initial conditions of the CFD simulations with the test data for cases 6 and 7 the two back pressures were adjusted. As both the split and tangentially averaged plots (as reported in table 4.1 and figure 4.20 respectively) were desired values the setting of the back pressure was cumbersome as a certain ratio of back pressure between the outlets did not produce the same split if the magnitude was adjusted. This in combination with the fact that accuracy of the inlet conditions only (with the authors' knowledge) could be evaluated after a simulation had been completed contributed to the accepted discrepancy between the test result and the simulations. The deviation of the split is approximately 0.3-0.4 percent and the static pressure in figures 4.20 and 4.21 can be seen to be slightly lower for the CFD. This indicates that a better outlet pressure combination could be obtained, although the used combination is fairly accurate to the test data. As for the C_{P0} DS contour plots, figure 4.22, the typical CFD over prediction of the separation can be seen at both the central OGV wake and the NGV wakes. This over prediction is commonly seen for this type of simulations [7] [8] [9] and one can note that over predicting occurrences such as separation is better than under predicting due to the fact that the over prediction would mean that if a real aircraft component were designed from the CFD results the margins of error would be greater than required.

As for the DS tangentially averaged plots shown in figures 4.23 and 4.24 the total pressure is similar for the simulations and the test data except for a point around 90% of the blade span where the CFD shows a slight peak giving a divergence of around 20 Pascal. One has to bear in mind that the MHP does not produce data for the very tips of the hub or shroud and that the clear boundary layer effects form the CFD plots cannot be compared to the test values. A similar divergence to that of the total pressure plots can be seen at the hub for the static pressure plots. These pressure plots are very similar for the two cases, which also is the case for the axial velocity, the tangential velocity and the resulting swirl plots. Overall the test data produce a smoother curve and the CFD plots show more radical changes. This could be due to the resolution of the plots, the MHP only travels over 20 different radii where as the simulations reports 100 different radii (an arbitrarily chosen value). This resolution in measured data can not explain the difference in tangentially velocity (by definition of swirl) and swirl at the midspan of the DS profile where the CFD shows a almost completely axial flow and the tests show a non-zero tangential velocity.

The plots for C_{P0} , figure 4.25, show very similar results for CFD and tests which one might expect seeing the tangentially averaged total pressure plots above. These plots are however not tangentially averaged and therefore show more detail at the chosen radii. The biggest outlier in this plot is the substantial drop seen at the center for 50% span, which essentially corresponds to the propensity of the CFD to exaggerate the separation due to the OGV as seen in figure 4.22. In figure 4.25 one can also see the progressive increase in pressure loss coefficient at higher spans corresponding to the transition from yellow to red tint in figure 4.22.

The swirl at these radii, as presented in figure 4.26 shows quite significant discrepancy between the test data and the simulations. This is most clear at the zero degrees, where the OGV is placed upstream. The OGV gives rise to separation zones where the velocity is low and not necessarily axial. As the velocity is low in separation zones the swirl value becomes highly variable and it is likely this that gives peaks at zero degree angle. Beyond the central peak the largest difference is at positive angles for the 50 % span case where the CFD gives higher values than the tests. This is the reason for the aforementioned midspan difference for the swirl in figures 4.23 and 4.24.

The final comparison of tests and CFD for cases 6 and 7 that is presented is the tangentially averaged DS plots of the radial velocity and the corresponding yaw angle. These plots are separated from the other plots of this type as these show seemingly substantial divergence to the point where the CFD predicts a positive velocity (and yaw) and the test show negative values. The magnitudes of the radial velocity is small, the maximum difference in test data and simulations being four m/s. After consultation with the test conductors at Chalmers it was concluded that the discrepancy likely was a result of the nature of the probe. The MHP is quite narrow and long and is therefore prone to swaying in the wind created in the rig which would effect the radial velocity measurements in particular.

5.6 Cases 1-3

For cases 1-3 the back pressure is identical at the two outlets which means that matching the US data simply is done by changing the pressure magnitude until a sufficient match had been reached. The resulting pressure values and the corresponding splits can be seen in table 4.2 and the US tangentially averaged plots indicating the validity of the outlet conditions are presented in Appendix, A.6, A.10 and A.14. The similitude of the simulations and test downstream of the OGV can however not be determined by the tangentially averaged plots such as figure 4.28. The reason for this is the partial blockage of the DS MHP described in section 3.2 which produces swirl of about ten degrees after the OGV. Such a high swirl is strange since the purpose of the OGV is de-swirling the flow to maximize the thrust and is therefore highly unlikely to occur during on-design conditions such as case 1, and can thus be attributed to the inhibited MHP. Since the swirl is a function of the velocity components nonphysical swirl means nonphysical axial or tangential veloc-

ity components, and both of these components are seen to deviate from the CFD results. What seems to match between the CFD and the tests is the total pressure meaning that the contour plots in figures 4.29 and 4.30 show are clear similitude when comparing test and CFD. For the US profiles in figure 4.29 are almost identical with the exception of case 3 where the simulations give higher C_{P0} close to the shroud and the fact that the contour plots from CFD seem to be lower resolution. This low resolution of the CFD contours is unexpected since all DS contours are high resolution but could be a consequence of the discrete nature of the data, and although this would imply the same resolution for the "Experminents" plots the plotting used in *Python* could be better at interpolation and thus smoothening the plots.

The DS contour plots for CFD show the expected over prediction of separation when compared to the test data as well as the extended survival of the NGV wakes. The experimental contour plots for cases 2 and 3 show a larger hub separation at the suction side (the right side in the presented figure) which is not present in the simulations. Figure 4.31 show the physical changes that correspond to the change in flow coefficient with the green streamlines highlighting the larger US swirl, an increased angle of attack, which causes the separation as seen in the dark blue both in this plot and in figure 4.30.

The swirl seen in the upper portion of figure 4.32 is expectedly totally different for the test data and the simulations, but what is more unexpected is the difference compared to case 6. The only significant difference between cases 1 and 6 is the back pressure at the HX outlet and one might expect similar swirl characteristics. Especially at 25% span case 6 shows a significant dip in swirl at $\theta = 0$ where case 1 shows a peak. This could just be a consequence of the low velocities and separation but the difference is interesting. The lower part of figure 4.32 shows the C_{P0} plots where the CFD and the tests seem to be more in accordance although not quite to the degree of the C_{P0} plots for case 6 or 7.

The change in flow characteristic due to a change in the flow coefficient can also be visualized using streamlines at the surface of the OGV blades as presented in figures 4.33 and 4.34. The plots on the suction side are most interesting as this is the side where the separation occurs and turbulent regions can be seen. Also apparent in these figures is the effect of the flow splitter downstream of the blade as the flow is pushed over or under the splitter which will be more apparent later.

5.7 Split Variation

For the instances where the split was varied outside of the test data range the aim was to study the effect of the split on the HX effectiveness, especially as the C_{min} switches from air to water. To determine where this change would happen figure 4.35 was plotted with an assumed mass flow rate of 1 kg/s. This assumed mass flow rate is not constant as the mass flow is a function of the boundary conditions but since these results are intended to be qualitative rather than quantitative and the only change the mass flow does is the value of the transitional split. The HX

back pressure and their corresponding splits are shown in table 4.3 and figure 4.36 where the most notable feature is the logarithmic look of the figure. This behavior indicates the fact that decreasing the back pressure gives a smaller return in split increase the larger the magnitude becomes. In addition to this it can be noted that simulations were attempted at even lower back pressure but this caused the simulations to crash without producing a result.

5.8 Comparison of Simulations for Different Geometries

In order to investigate the effects of the heat exchanger on the TRS, simulations were conducted solely on the TRS. This allowed for a comparison of simulation results for both scenarios, enabling the analysis of any differences. Figure 4.37 is showing contour plots for the total pressure coefficient downstream of the OGV. The two lower plots show that there is no difference for the DS contour plot if the water circuit is enabled or not, which is expected as this only ought to effect the behaviour of air in the HX. This is nevertheless an important result as this shows that the simulations without water that are far more computationally demanding can be used to make conclusions. The contour plot showing the DS profile for the "Only TRS", the extended domain described in section 3.7, show some noteworthy differences to the regular cases where the domain includes the HX. The central wake is more pronounced, especially in the shroud region for the "Only TRS" simulations, as well as a reduction in the separation on the SS close to the hub. These characteristic differences between using and not using a HX are also present in cases 2 and 3 as depicted in figure 4.38. This figure additionally show that the CFD simulation with different geometries give more similar results than the experimental test data indicating shortcomings in the CFD problem specification.

Figure 4.39 displays the streamlines present on the OGV surface when only the TRS is employed, while Figure 4.40 illustrates the streamlines for the case where both the TRS and HX are used. Notably, differences can be observed between the two scenarios. In the scenario where only the TRS is utilized, the streamlines appear to follow a consistent and continuous path along the blade, without any significant variations in direction. This trend is noticeable on both the suction and pressure sides of the blade. This is however not the case for when the HEX is installed downstream of the TRS. The predicted streamlines from the CFD in figure 4.40 shows that the airflow on both the suction and pressure side appears to differ, especially for the suction side. The streamlines on the lower left corner of the OGV do also show a larger separation zone which also matches with the DS profile comparison from figure 4.38 for the two different cases. The most probable cause for these differences is the presence of the splitter which deforms the uniformity flow and forcing the air to either enter the HX or continuing to the atmospheric conditions above it. As for the suction side plots the air is pushed down by the blade continually when no splitter is present (only TRS geometry). If the splitter on the other hand is present the downwards force created by the blade and initial conditions is disturbed causing

a notable change in the streamlines at the suction side. At the pressure side this downward force is not present as the only TRS case shows almost axial streamlines, a characteristic also interrupted when the flow is split.

As a final note it might be added that the "Only TRS" domain is the domain that has been used in previous studies which investigate this domain much more extensively [7] [8] [9]. These previous studies does not produce entirely similar plots to the ones present in this thesis, the most notable difference being that the OGV wake close to the shroud tends to decrease much more than found in this thesis. This indicates some difference in the problem formulation within the CFD program.

5.9 Effectiveness of HX

In section 2.2.3 the fact that effectiveness is the final presented HX performance metric is explained, but several other aspects of the HX has been investigated despite not being included in the report. What can be concluded is that the CFD predicts water outlet temperatures very similar to the temperatures measured during the tests, as per figure 4.41. The variable other than temperature that is used to calculate the effectiveness is the mass-flow, see equation 2.10 (assuming that the heat capacity is fairly constant).

The difference in effectiveness between cases 1-3 and 6 & 7 is notable although a larger sample size would be necessary to definitely state that a lower split has a higher effectiveness than a higher split no matter the circumstance. What can be stated that given a constant mass flow of water, constant water and air inlet temperature and constant heat capacities the equation for effectiveness reduces to $\varepsilon = K \frac{\Delta T_h}{C_{min}}$ (these are in fact not constant for the test but they could be kept constant as they are independent of each other). For all the studied cases the air is the fluid that constitutes C_{min} which is expected since the heat capacity of air is a quarter of the heat capacity of water, meaning that the effectiveness is limited by the heat capacity and mass flow of the air.

Fictional cases where C_{min} changed were however constructed as presented in figure 4.42. As predicted by figure 4.35 the transitional split that indicates the switch from $C_{min} = C_{air}$ to $C_{min} = C_{water}$ occurs at a split slightly above 20 percent. This critical split corresponds well to the minimum value for effectiveness in figure 4.42. That the effectiveness drops as the transitional split is approached from a lower split is due to the fact that the mass flow of air is the limiting factor in this region so having less impact and therefore lower mass flow of air is advantageous in order to obtain a higher effectiveness. When the water is the limiting factor, the split is above the transitional, the equation for effectiveness becomes

$$\epsilon = \frac{C_h(T_{h,in} - T_{h,out})}{C_{min}(T_{h,in} - T_{c,in})} = \frac{C_{water}(T_{h,in} - T_{h,out})}{C_{water}(T_{h,in} - T_{c,in})} = \frac{(T_{h,in} - T_{h,out})}{(T_{h,in} - T_{c,in})} \quad (5.1)$$

and as the inlet temperatures are constant the effectiveness increases as the outlet temperature of water decreases which it does as the split grows. Also included in

figure 4.42 are the test cases and it can be seen that the these cases give similar results to that of the fictional case. The slight discrepancies are likely due to variation in the inlet temperature and mass flow of water as well as well as the inlet air characteristics.

5.10 Possible Improvements and Further Research

This section entails some possible improvements that could have been done in the thesis as well as topics that is of interest in further research.

5.10.1 Further Turbulence Modelling Details

As may be evident from the discussion of turbulence modelling there are several aspects of turbulence modelling that give significant impact on the obtained results, especially on eventual separations. As the bulk of the simulations were carried out prior to the investigation of the turbulent eddy length scale value the most obvious improvement in regards to turbulence modelling would be to redo the simulations with a more appropriate value for this parameter. There are however several more parameters employed in the CFD software which could impact the simulations (such as the turbulent Schmidt numbers, the intermittency coefficients and other coefficients shown in table 3.4) which have not been investigated in this thesis and studying these would of course also be central to the results.

5.11 Compilation of Results

For every CFD simulation an enormous amount of data is produced meaning that a central part of CFD simulation is choosing the appropriate data and the amount of it to disclose as results. In this thesis the main focus has been the total pressure coefficient for the contour plots and several quantities for the x-y plots. Presenting plots of the total pressure coefficients is the norm for aerospace applications as a non-dimensional way to measure pressure is very useful, but this does not mean that this is the only way to extrapolate interesting results from the data. The Authors are confident that the results presented are clear and concise although it is possible that some results of interest have been omitted due to the selection of results.

5.11.1 Data for Reynolds Number = 350000

Unfortunately, the measurements conducted at Chalmers did not cover the US data for $Re = 350000$. The study could not explore the impacts of higher Reynolds numbers since simulations for different Reynolds numbers were not available. Consequently, the analysis of the flow characteristics in the study could not be extended, limiting its depth. Having data for other Reynolds numbers would be of interest since a variation in Re somewhat corresponds to a variation in flight speed and extrapolating conclusions about the validity of the integrated HX for aircraft in general using a constant flight speed is dubious at best.

5.11.2 Further Investigations

As mentioned several times in the thesis no details of the HX geometry has been divulged due to a pending patent. This does not mean that no HX details has been examined, several studies of HX characteristics such as pressure drop and heat transfer coefficient has been done and reported internally to GKN. There are however many more aspects of the HX that have to be investigated, especially relating the test data and simulations of the HX model to a HX sized up to real engine size. These investigations are also related to the economic validity of actually integrating such a HX into a high BPR engine. As an engine operating with liquid hydrogen would require pre-heating some sort of heat exchange would be required although it is not necessarily done in the same way as presented in this thesis.

In the thesis seven operation points are presented (see table 3.1) but due to the faulty MHP for cases 1-3 and the lack of US data for cases 4 and 5 there were only two cases with the sufficient amount of test data with which the simulations could be extensively compared. Having more cases would naturally mean that more substantiated conclusions could be drawn, especially cases with a different Reynolds number than 235000 would be of interest. A deviation in flow speed.

To further validate the application of a HX in a hydrogen engine a natural step would be to perform tests with hydrogen. This is however quite a large step up in scope of the physical testing so one could imagine that CFD simulations with hydrogen might be a intermediate step in the process of implementing a HX in a hydrogen engine.

6

Conclusion

In conclusion it can be stated that it is possible to model the behaviour of the flow in a TRS with an integrated HX using CFD simulations, although no exact matches has been obtained in this thesis. The central theme of the CFD simulations is the overprediction of separation due to the central OGV. There are however several factors employed in the setup of the problem that impact the appearance and magnitude of separation zones. One of these is the specification of the inlet which is central to the appearance of turbulent regions. It was found that the simulations that most closely resemble the test data utilized a 2D inlet with a BL1 extrapolation at the hub and shroud which is supported by previous CFD studies made on the TRS component.

The impact of turbulence modelling on the studied DS plane was also shown to be significant. In this thesis different approaches and constants for turbulence modelling have been compared with each other and test data to determine the most appropriate settings for this application. As for the turbulence model in it self the Transition SST showed similar results to the $k - \epsilon$ in terms of the DS contour plots although the Transition SST in theory should be an improvement on the $k - \epsilon$ model. The most notable difference between the two models is the streamlines at the blade which is expected as the near-wall treatment differs greatly between the two. Ultimately the Transition SST was chosen due to it being more tested in similar cases and for the fact that the mesh was made with this model in mind. A couple of the parameters, the a_1 coefficient and the eddy length scale, were investigated albeit the latter was investigated after the test cases had been simulated. The results of these investigations was that the a_1 coefficient had an impact on the thickness of the OGV wake, although only up to a value of 0.335. As for the eddy length scale a value of 3 mm showed the largest degree of similitude to the test data of the tried values. The value of the eddy length scale was shown to impact the presence of NGV wakes as well as the OGV wake and is therefore of large importance. The independence of the numerical solution on the mesh size for the TRS domain was also established from a qualitative mesh independency study.

Using the above described settings the 5 different test data cases were simulated and results was obtained. The two central cases, case 6 and 7, were the most central as the test data was the most optimal for these cases. These simulations gave a close resemblance to the test data both in terms of the measured split and the tangentially averaged plots, with exception to the radial velocity and yaw angle which likely is caused by the probes behavior in the rig. The contour DS plots show the

characteristic overprediction of the OGV and NGV wakes.

Test cases 1-3 operated with a somewhat blocked MHP yielding large discrepancies in the tangentially averaged DS plots. This makes the comparison between tests and CFD largely irrelevant for these cases although the CFD still is deemed trustworthy due to the nature of the other cases as well as the fact that the total pressure plots are matching and hence the C_{P0} contour plots are quite similar. These cases showcase the impact of the flow coefficient, showing increased separation at increased flow coefficient.

Simulations done using a domain without the HX showed similar results, in terms of the measured DS quantities, as with the HX further indicating the importance of specifying the problem in the CFD program. Previous studies of the TRS component have ostensibly employed a domain similar to the "Extended domain", albeit with slightly altered flow conditions but the results in this study are substantially different further indicating variations in the numerical setup.

As for the effectiveness of the HX it was shown that the effectiveness followed the expected trend in regards to the amount of air entering the HX. The CFD also predicted the outlet temperature of the water to a value very similar to that of the test data. This could and was used to further determine the validity of the HX even though this could not be included in the thesis.

This thesis has established the foundation for the required studies in order to validate the implementation of a heat exchanger in a hydrogen engine. Although a hydrogen engine has not been specifically addressed in this thesis, it still serves as a starting point for future research on the implementation of a heat exchanger in such an engine. The CFD simulations conducted in this thesis have undergone successful validation and are deemed reliable for accurately predicting aerodynamic and thermal behavior in a TRS and for analyzing the performance of a heat exchanger.

Bibliography

- [1] A. Contreras, S. Yiğit, K. Özay, and T. Veziroğlu, “Hydrogen as aviation fuel: a comparison with hydrocarbon fuels,” *International Journal of Hydrogen Energy*, vol. 22, no. 10-11, pp. 1053–1060, 1997. [Online]. Available: [https://doi.org/10.1016/S0360-3199\(97\)00008-6](https://doi.org/10.1016/S0360-3199(97)00008-6)
- [2] I. Dincer and C. Acar, “A review on potential use of hydrogen in aviation applications,” *International Journal of Sustainable Aviation*, vol. 2, no. 1, pp. 74–100, 2016. [Online]. Available: <https://doi.org/10.1504/IJSA.2016.076077>
- [3] “Flightpath 2050 : Europe’s vision for aviation : maintaining global leadership and serving society’s needs,” *European Commission and Directorate-General for Mobility and Transport and Directorate-General for Research and Innovation*, 2012. [Online]. Available: <https://data.europa.eu/doi/10.2777/15458>
- [4] D. L. Daggett, S. T. Brown, and R. T. Kawai, “Ultra-efficient engine diameter study,” NASA, Tech. Rep. NASA/CR—2003-212309, 2003. [Online]. Available: <https://ntrs.nasa.gov/citations/20030061085>
- [5] B. Rojo, D. Kristmundsson, V. Chernoray, C. Arroyo, and J. Larsson, “Facility for investigating the flow in a low pressure turbine exit structure,” *11th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics, ETC 2015*, 01 2015. [Online]. Available: https://www.researchgate.net/publication/282956605_Facility_for_investigating_the_flow_in_a_low_pressure_turbine_exit_structure
- [6] B. Rojo, “Design of an experimental facility for investigating the flow in a low pressure turbine exit structure,” 2015. [Online]. Available: <https://research.chalmers.se/en/publication/224747>
- [7] V. Vikhorev, V. Chernoray, O. Thulin, S. Deshpande, and J. Larsson, “Detailed experimental study of the flow in a turbine rear structure at engine-realistic flow conditions,” *Journal of Turbomachinery*, vol. 143, pp. 1–11, 03 2021. [Online]. Available: <https://doi.org/10.1115/1.4050451>
- [8] V. Vikhorev, P. Nylander, V. Chernoray, J. Larsson, and O. Thulin, “Experimental and numerical flow analysis of an engine realistic state-of-the-art turbine rear structure,” *Journal of Engineering for Gas Turbines and Power*, vol. 144, 03 2022. [Online]. Available: <https://doi.org/10.1115/1.4054075>
- [9] P. Nylander, S. Deshpande, and J. Larsson, “Influence From Nozzle Guide Vane Wakes and Inlet End-Wall Boundary Layers on Turbine Rear Structure

- Aerodynamics,” *Turbo Expo: Power for Land, Sea, and Air*, vol. Volume 10C: Turbomachinery — Design Methods and CFD Modeling for Turbomachinery; Ducts, Noise, and Component Interactions, 06 2022. [Online]. Available: <https://doi.org/10.1115/GT2022-81677>
- [10] I. Jonsson, V. Chernoray, and R. Dhanasegaran, “Infrared thermography investigation of heat transfer on outlet guide vanes in a turbine rear structure.” *International Journal of Turbomachinery, Propulsion and Power*, vol. 5, no. 23, p. 23, 2020. [Online]. Available: <https://doi.org/10.3390/ijtp5030023>
- [11] G. Corchero and J. Montanes, “An approach to the use of hydrogen for commercial aircraft engines,” *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, vol. 219, no. 1, pp. 35–44, 2005. [Online]. Available: <https://doi.org/10.1243/095441005X9139>
- [12] I. Jonsson, “Experimental aerothermal study of internal jet engine structures,” Ph.D. dissertation, 2021. [Online]. Available: <https://research.chalmers.se/en/publication/527645>
- [13] K. Bhupendra, S. Paulas, K. Adam, S. Vishal, and S. Riti, “A review of hydrogen as a fuel for future air transport,” *American Institute of Aeronautics and Astronautics*, vol. 48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, 30 July 2012 - 01 August 2012, Atlanta, Georgia, 2012. [Online]. Available: <https://doi.org/10.2514/6.2012-4267>
- [14] H. Abedi, C. Xisto, I. Jonsson, T. Grönstedt, and A. Rolt, “Preliminary analysis of compression system integrated heat management concepts using lh2-based parametric gas turbine model,” *Aerospace*, vol. 9, p. 216, 04 2022. [Online]. Available: <https://doi.org/10.3390/aerospace9040216>
- [15] F. P. Incropera, D. P. DeWitt, T. L. Bergman, A. S. Lavine *et al.*, *Fundamentals of heat and mass transfer*. Wiley New York, 1996, vol. 6.
- [16] O. Kunz and W. Wagner, “The gerg-2008 wide-range equation of state for natural gases and other mixtures: An expansion of gerg-2004,” *Journal of Chemical & Engineering Data*, vol. 57, p. 3032–3091, 10 2012. [Online]. Available: <https://doi.org/10.1021/je300655b>
- [17] B. Andersson, R. Andersson, L. Håkansson, M. Mortensen, R. Sudiyo, and B. van Wachem, *Computational Fluid Dynamics for Engineers*. Cambridge University Press, 2011. [Online]. Available: <https://doi.org/10.1017/CBO9781139093590>
- [18] J. Anderson, *Governing Equations of Fluid Dynamics*. Springer Berlin Heidelberg, 2009, pp. 15–51. [Online]. Available: https://doi.org/10.1007/978-3-540-85056-4_2
- [19] L. Davidson, “Fluid mechanics, turbulent flow and turbulence modeling,” Division of Fluid Dynamics, year = 2015, Chalmers University of Technology, Göteborg Sweden 2023 [Online]. [Online]. Avail-

- able: http://www.tfd.chalmers.se/~lada/postscript_files/solids-and-fluids_turbulent-flow_turbulence-modelling.pdf
- [20] L. Könözy, *The k - ω Shear-Stress Transport (SST) Turbulence Model*. Cham: Springer International Publishing, 2019, pp. 57–66. [Online]. Available: https://doi.org/10.1007/978-3-030-13543-0_3
 - [21] F. R. Menter, M. Kuntz, and R. Langtry, “Ten years of industrial experience with the sst turbulence model,” *Turbulence, heat and mass transfer*, vol. 4, no. 1, pp. 625–632, 2003. [Online]. Available: https://www.researchgate.net/publication/228742295_Ten_years_of_industrial_experience_with_the_SST_turbulence_model
 - [22] S. Evans and S. Lardeau, “Validation of a turbulence methodology using the sst k - ω model for adjoint calculation,” in *54th AIAA Aerospace Sciences Meeting*, 2016, p. 0585. [Online]. Available: <https://doi.org/10.2514/6.2016-0585>
 - [23] R. B. Langtry, “A correlation-based transition model using local variables for unstructured parallelized cfd codes,” *Journal of Turbomachinery*, 2006. [Online]. Available: <https://doi.org/10.1115/1.2184352>
 - [24] M. Rahman, “Predicting transition with algebraic intermittency function,” *Physics of Fluids*, vol. 34, no. 3, p. 034113, 2022. [Online]. Available: <https://doi.org/10.1063/5.0077513>
 - [25] “Ansys cfx-solver theory guide,” 2011, only available through an ANSYS account.
 - [26] F. Menter, R. Langtry, and S. Völker, “Transition modelling for general purpose cfd codes,” *Flow, Turbulence and Combustion*, vol. 77, pp. 277–303, 10 2006. [Online]. Available: <https://doi.org/10.1007/s10494-006-9047-1>
 - [27] A. F. Molland and S. R. Turnock, “Chapter 3 - physics of control surface operation,” in *Marine Rudders, Hydrofoils and Control Surfaces (Second Edition)*, 2nd ed., A. F. Molland and S. R. Turnock, Eds. Oxford: Butterworth-Heinemann, 2022, pp. 21–56. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/B9780128243787000068>
 - [28] L. Eça, F. Pereira, and G. Vaz, “Viscous flow simulations at high reynolds numbers without wall functions: Is $y^+ \simeq 1$ enough for the near-wall cells?” *Computers & Fluids*, vol. 170, pp. 157–175, 2018. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S0045793018302299>
 - [29] F. R. Menter, R. B. Langtry, S. R. Likki, Y. B. Suzen, P. G. Huang, and S. Völker, “A Correlation-Based Transition Model Using Local Variables—Part I: Model Formulation,” *Journal of Turbomachinery*, pp. 413–422, 03 2004. [Online]. Available: <https://doi.org/10.1115/1.2184352>
 - [30] F. Menter, J. Ferreira, T. Esch, and B. Konno, “The sst turbulence model with improved wall treatment for heat transfer predictions in gas turbines,” *Proceedings of the International Gas Turbine Congress*, pp. 2–7, 01 2003. [Online]. Available: https://www.researchgate.net/publication/305083505_

The_SST_turbulence_model_with_improved_wall_treatment_for_heat_transfer_predictions_in_gas_turbines

- [31] E. S. Johansen, O. K. Rediniotis, and G. Jones, “The compressible calibration of miniature multi-hole probes,” *J. Fluids Eng.*, vol. 123, no. 1, pp. 128–138, 2001. [Online]. Available: <https://doi.org/10.1115/1.1334377>

A

Appendix 1

A.1 Experimental data

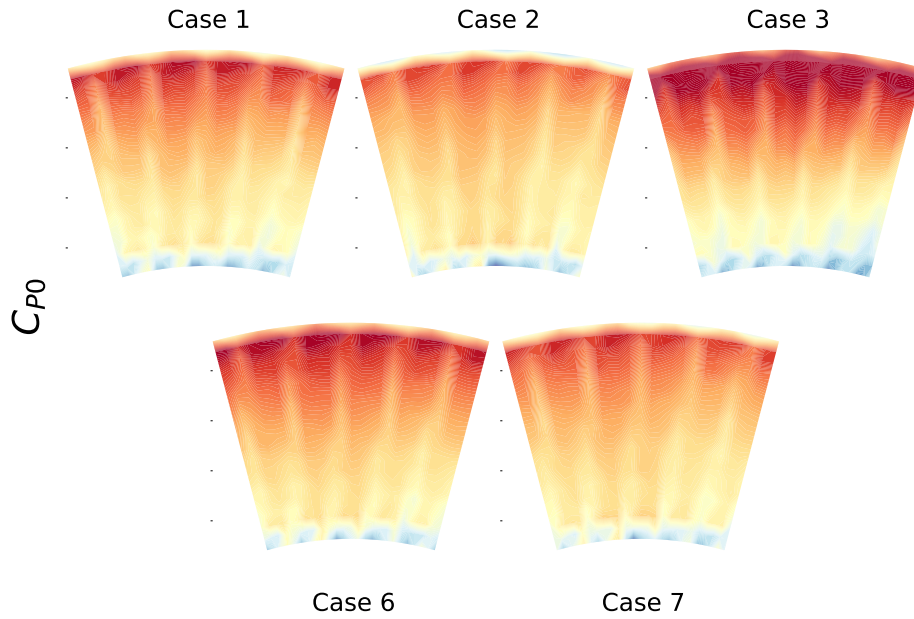


Figure A.1: cp_0 for the test data, US, (set to scale of -0.4 to 0.5)

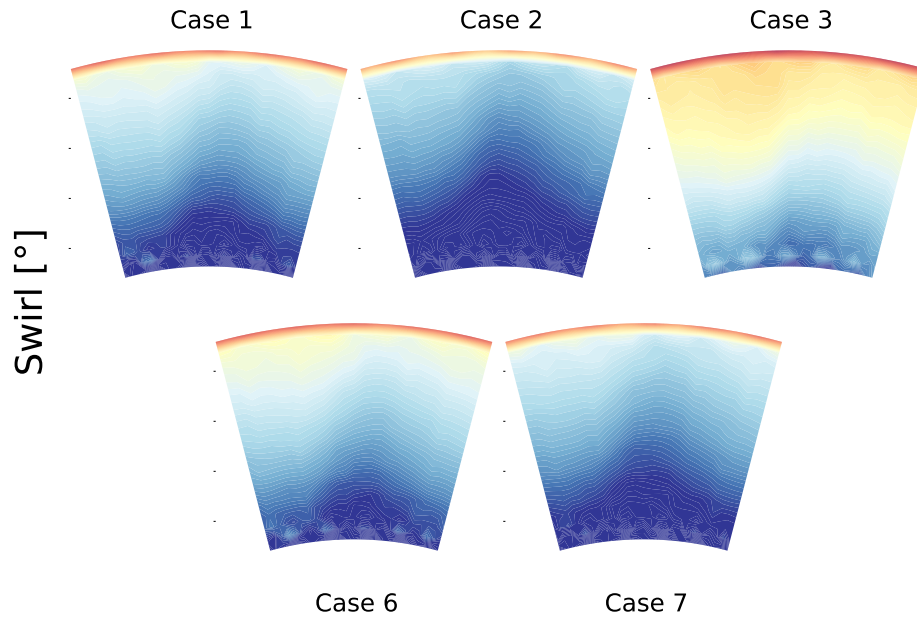


Figure A.2: Swirl for the test data, US, (set to scale of -30° to 30°)

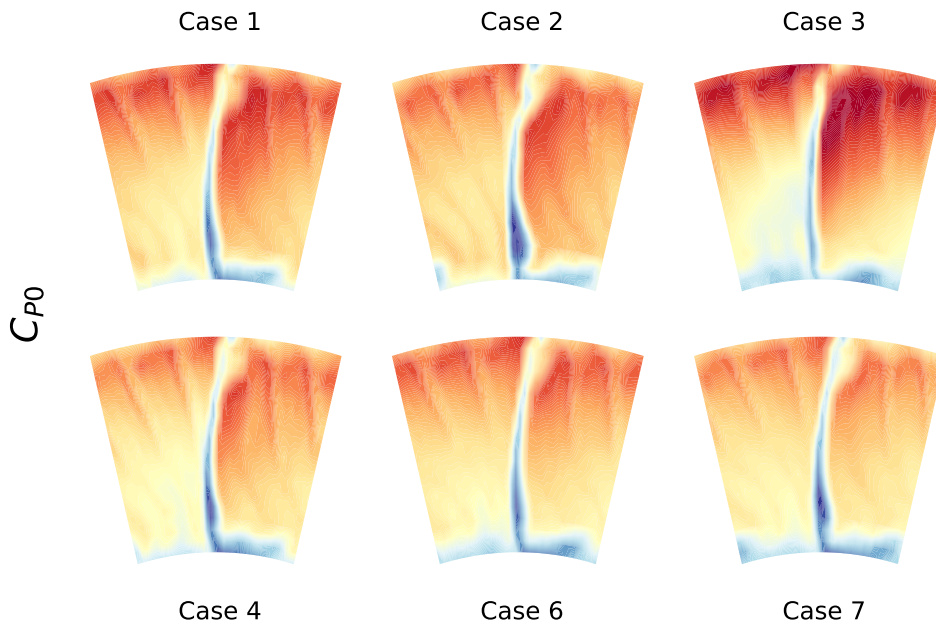


Figure A.3: cp_0 for the test data, DS, (set to scale of -0.4 to 0.5)

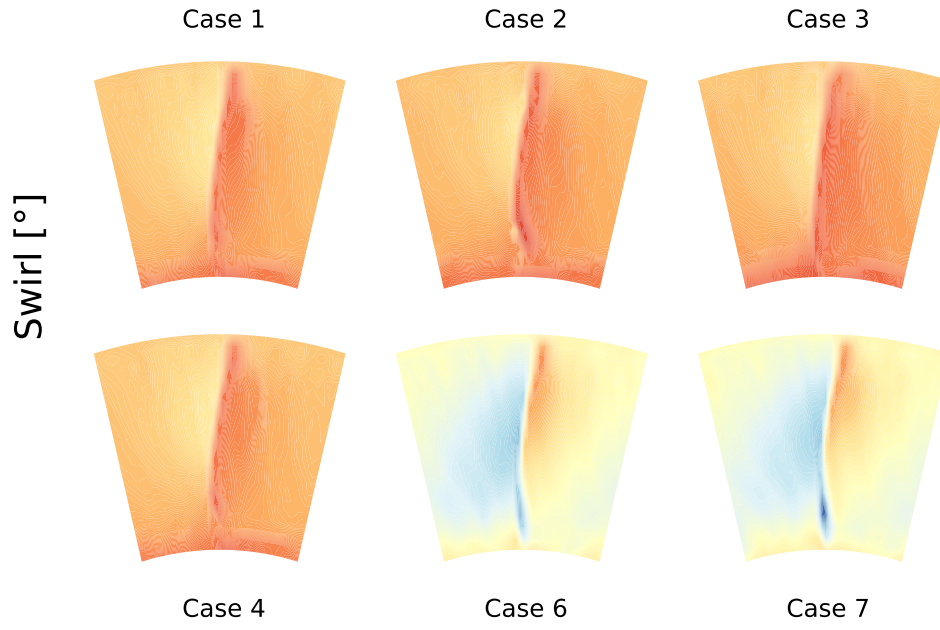


Figure A.4: Swirl for the test data, DS, (set to scale of -30° to 30°)

A.2 Case 1 Re235FC0.69

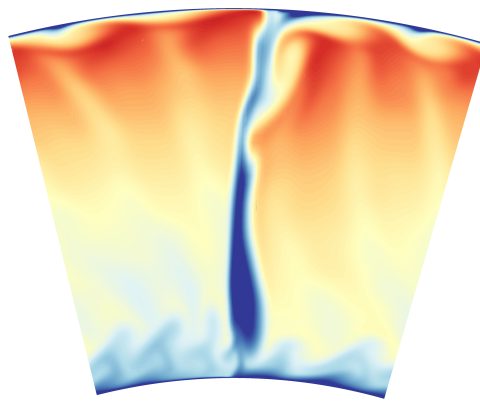


Figure A.5: cp_0 at DS

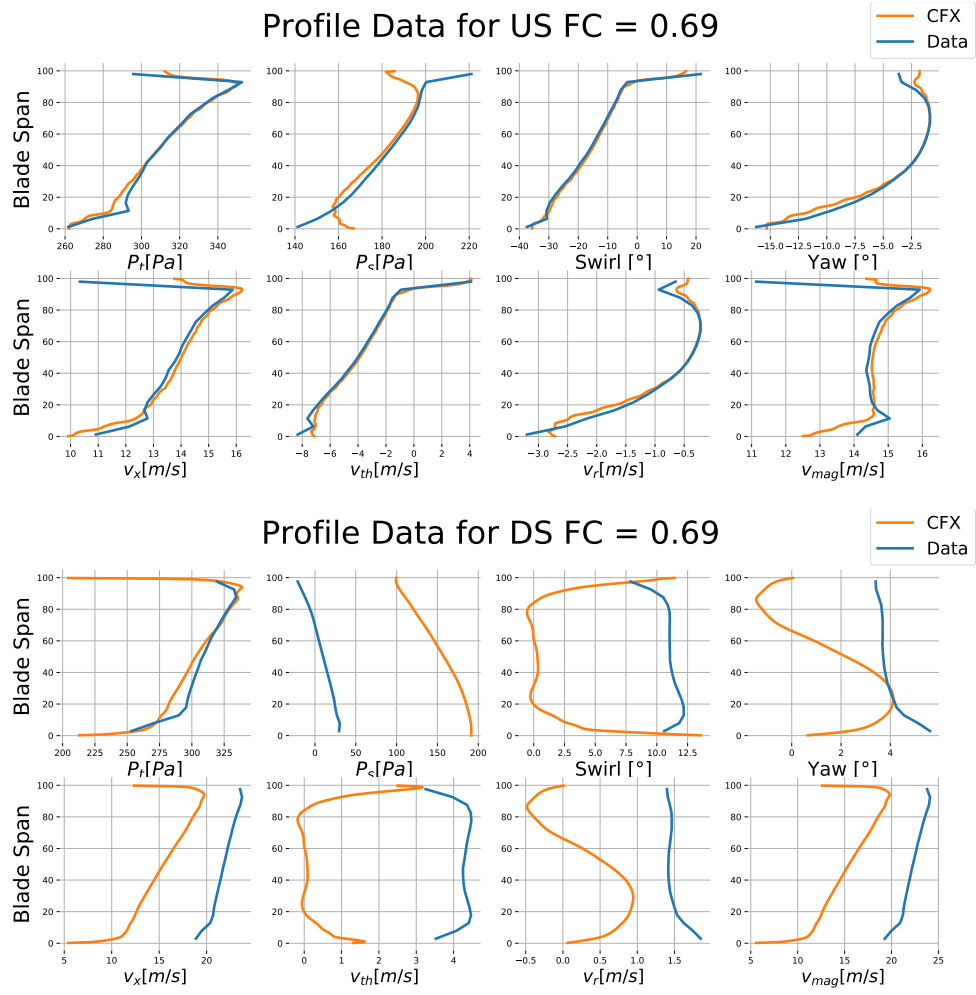


Figure A.6: tangentially averaged data for US and DS, case 1

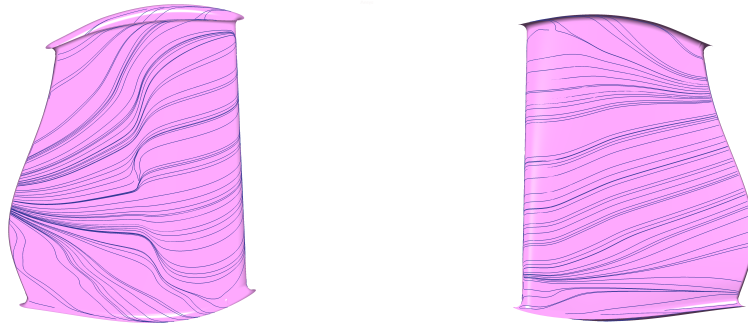


Figure A.7: Streamlines of the airflow at the OGV for case 1

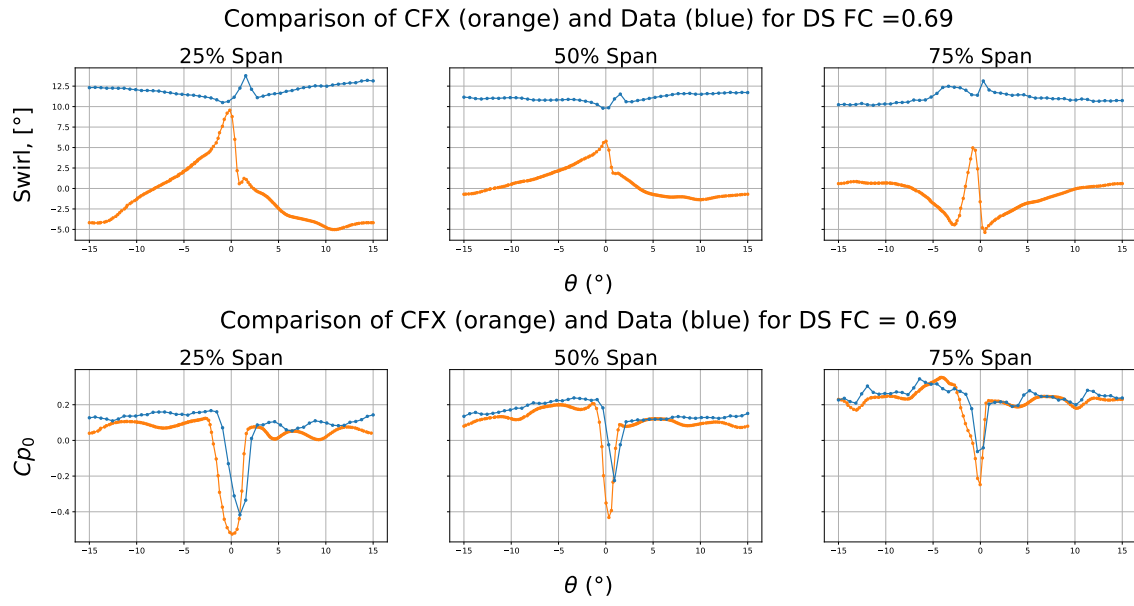
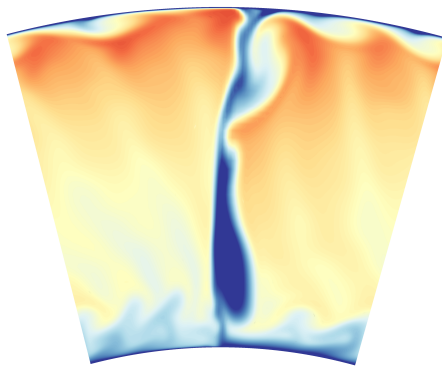


Figure A.8: Wake data, case 1

A.3 Case 2 Re235FC0.73

Figure A.9: cp_0 at DS

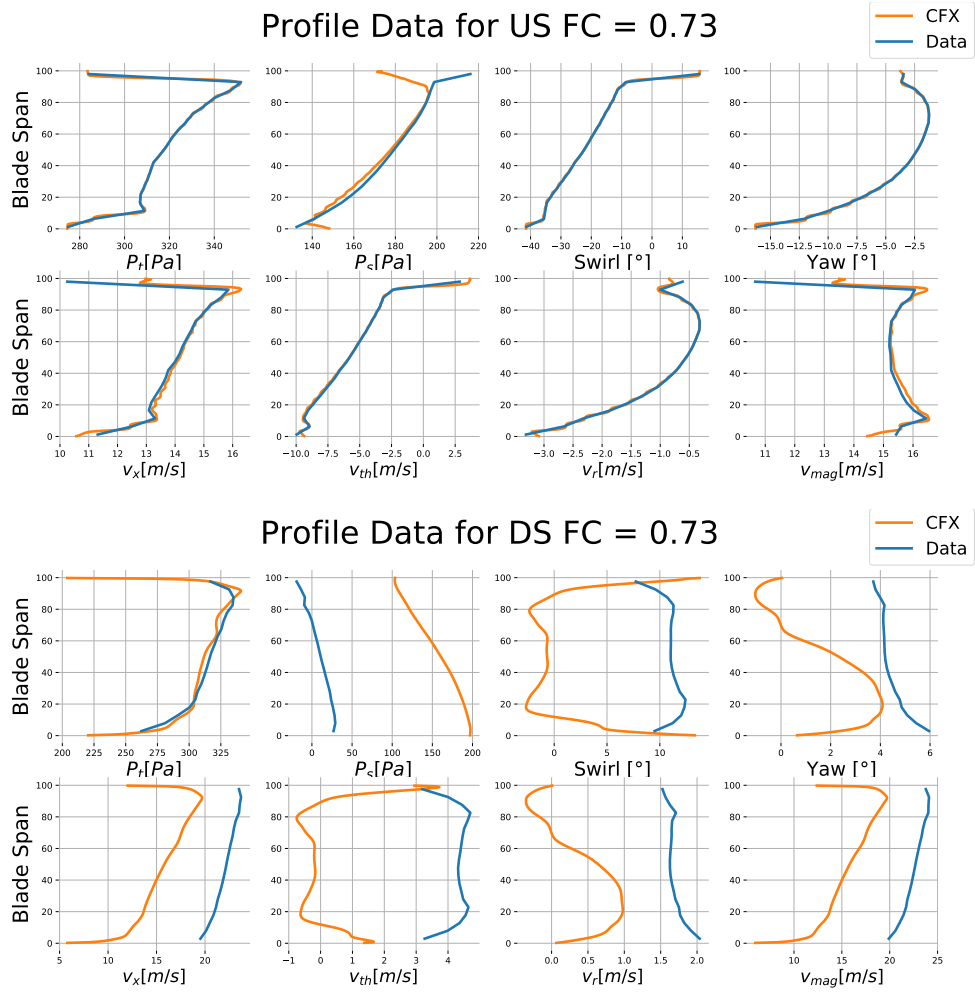


Figure A.10: tangentially averaged data for US and DS, case 2

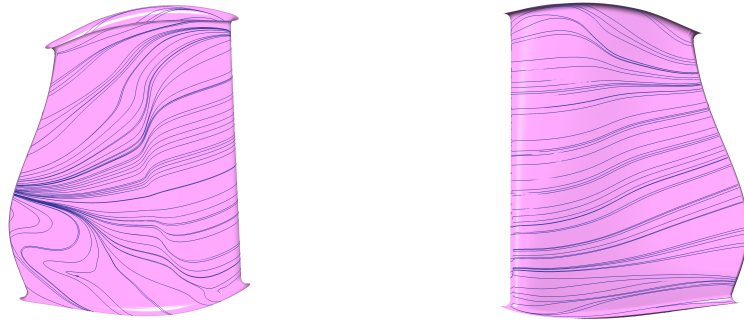


Figure A.11: Streamlines of the airflow at the OGV for case 2

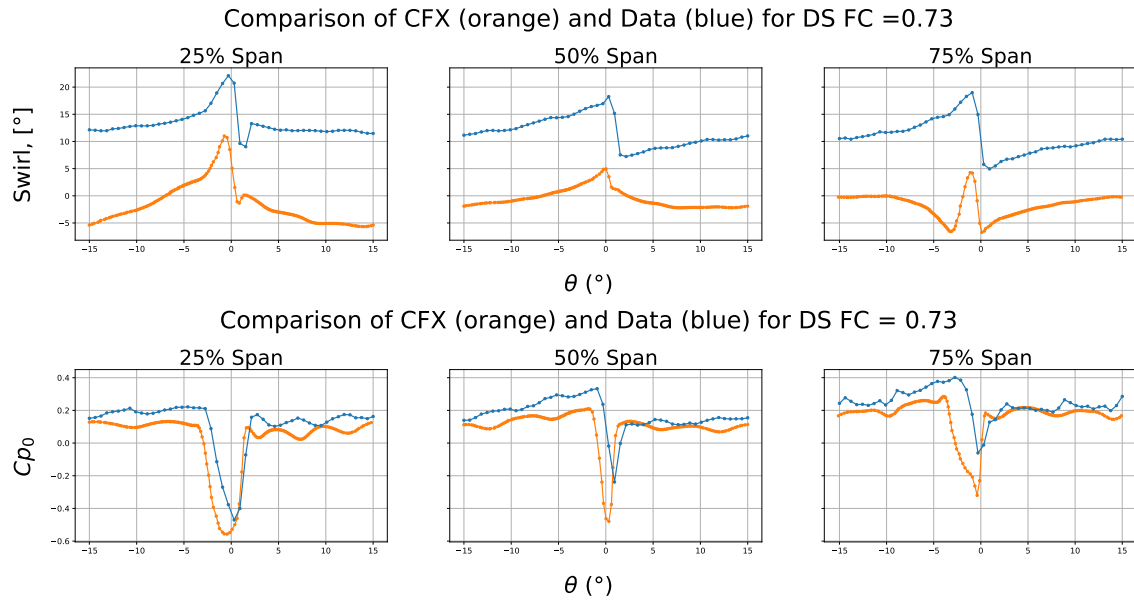
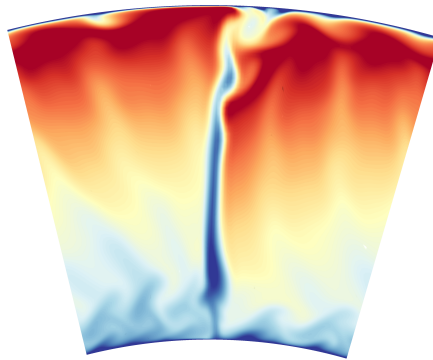


Figure A.12: Wake data, case 2

A.4 Case 3 Re235FC0.61

Figure A.13: cp_0 at DS

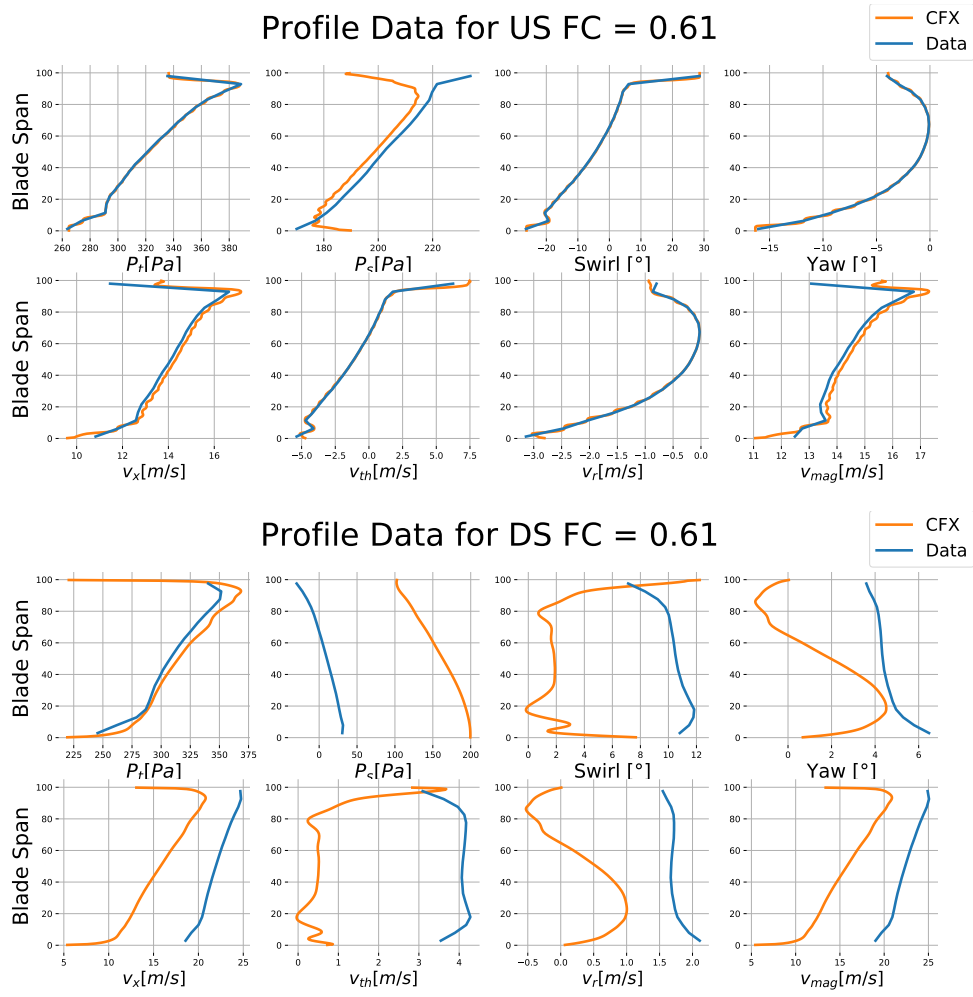


Figure A.14: tangentially averaged data for US and DS, case 3

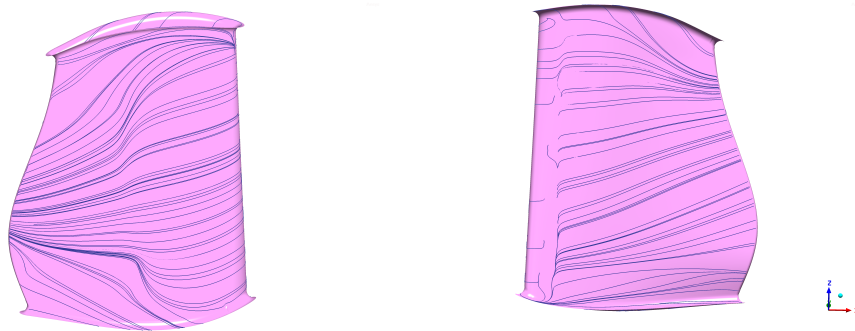


Figure A.15: Streamlines of the airflow at the OGV for case 3

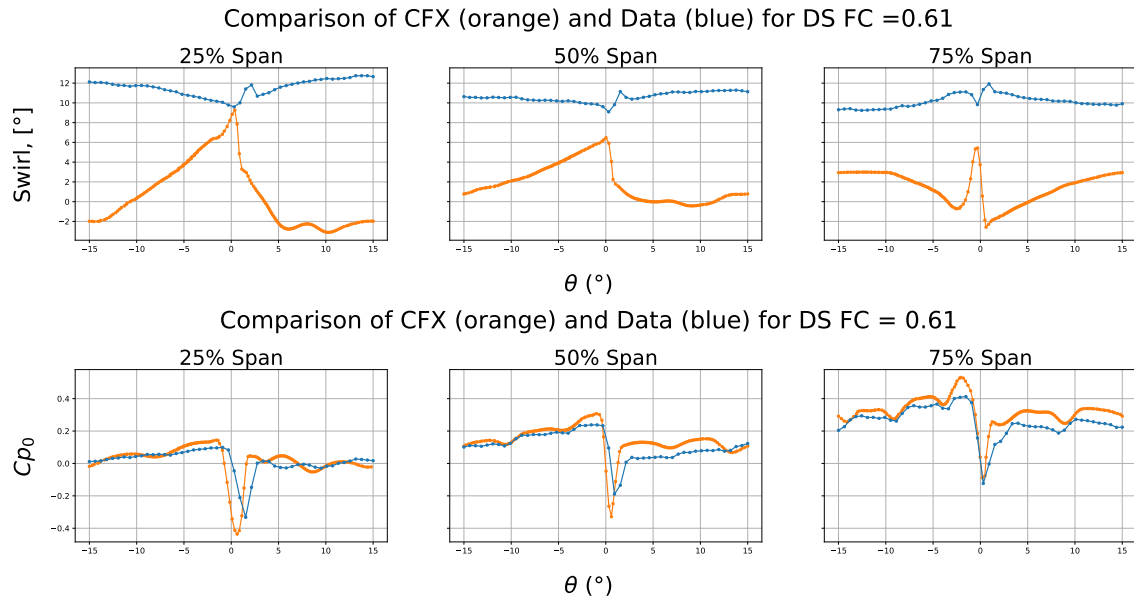
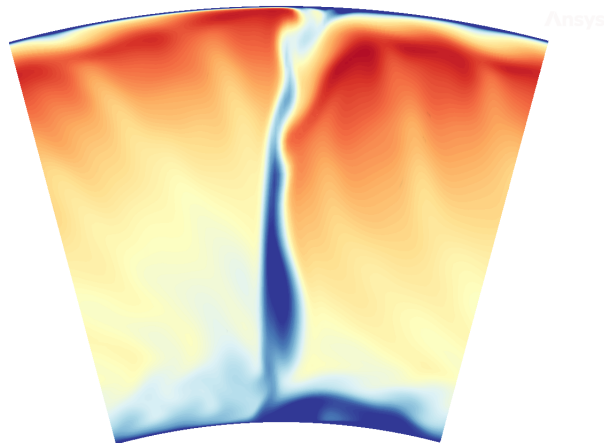


Figure A.16: Wake data, case 3

A.5 Case 6 Re235FC0.69, reduced split

Figure A.17: cp_0 at DS

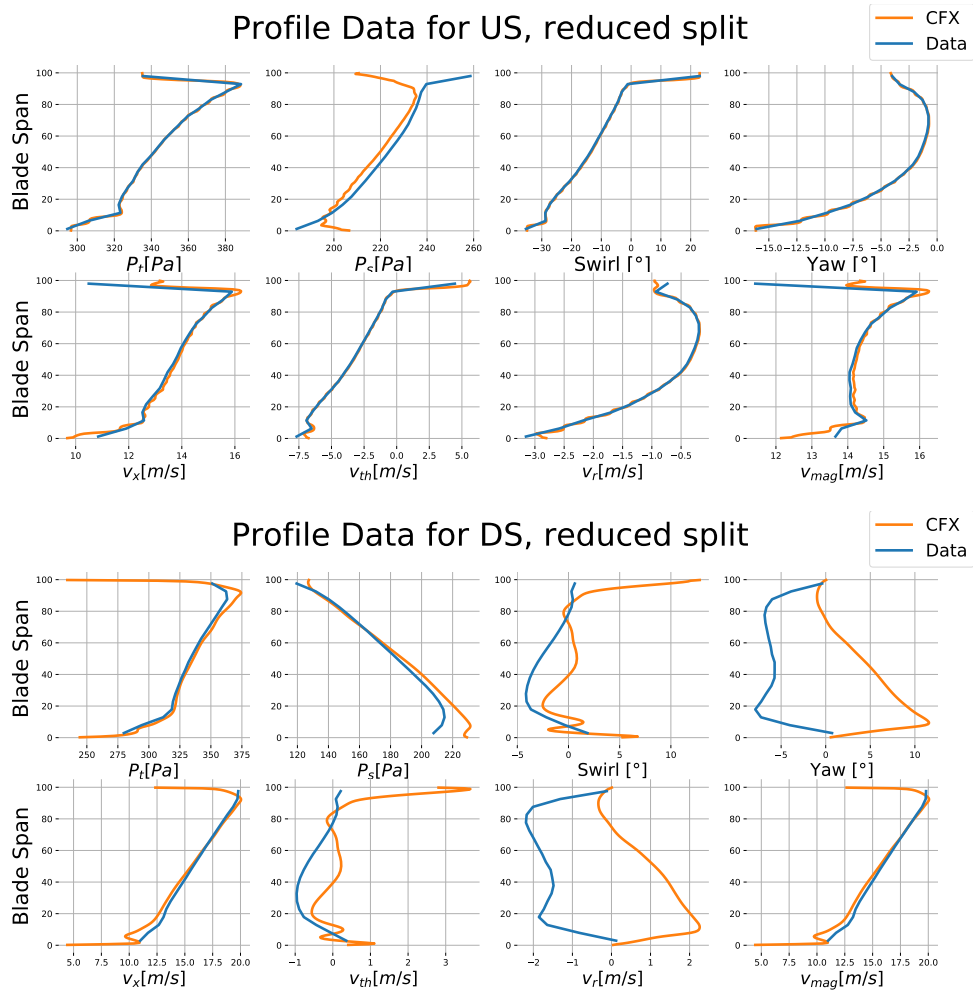


Figure A.18: tangentially averaged data for US and DS, case 6

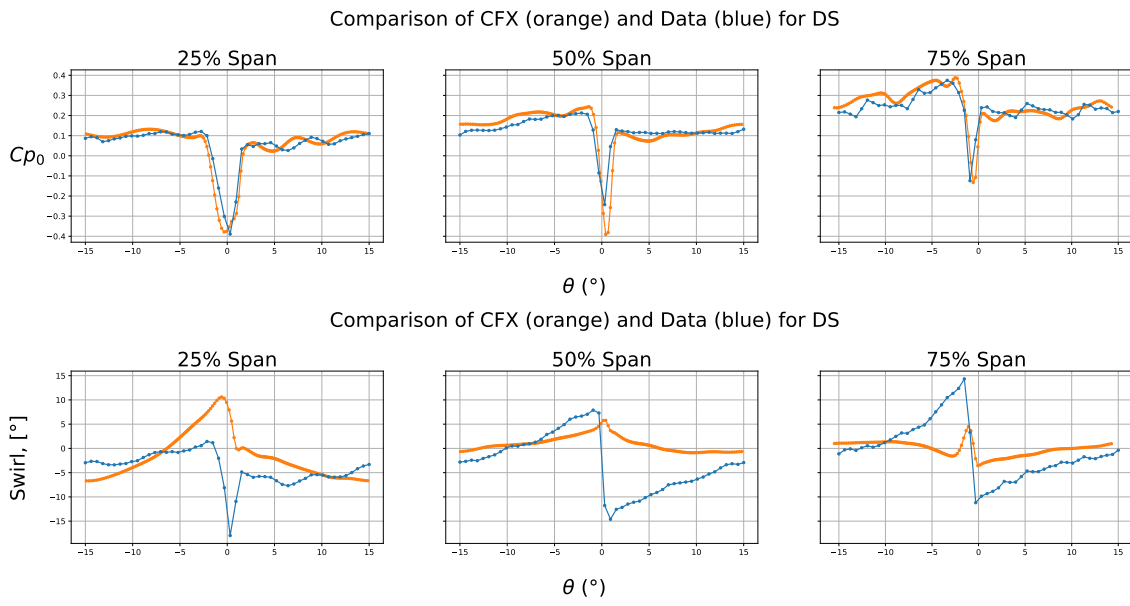


Figure A.19: Wake data, case 6

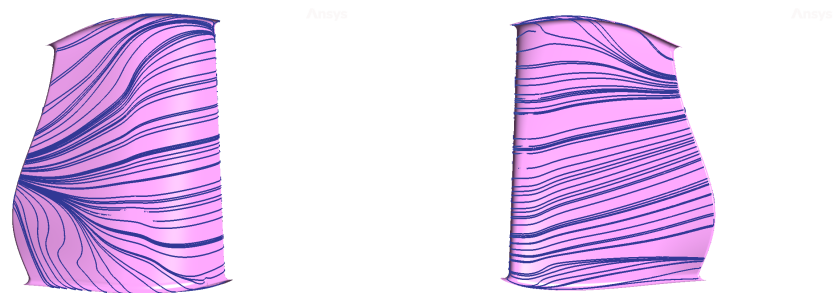


Figure A.20: Streamlines of the airflow at the OGV for case 6

A.6 Case 7 Re235FC0.73, reduced split

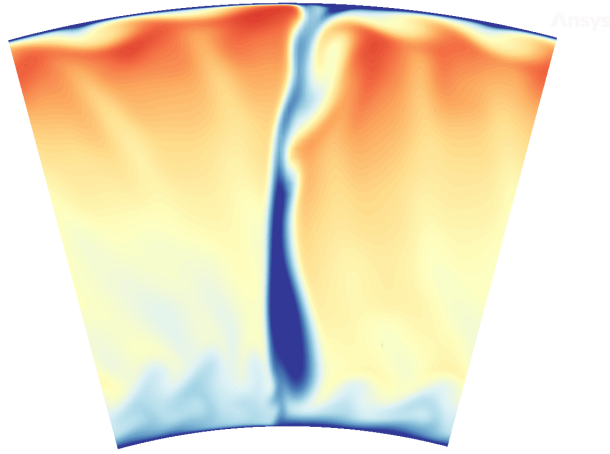


Figure A.21: cp_0 at DS

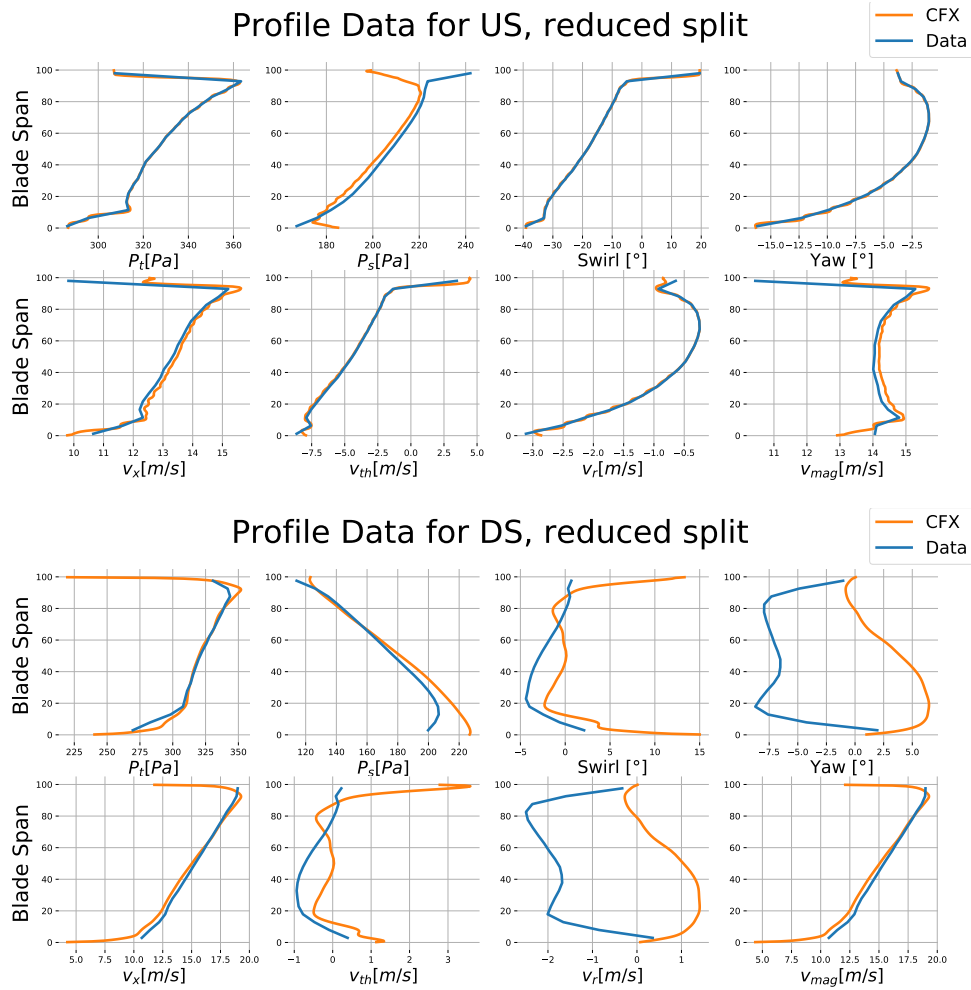


Figure A.22: tangentially averaged data for US and DS, case 7

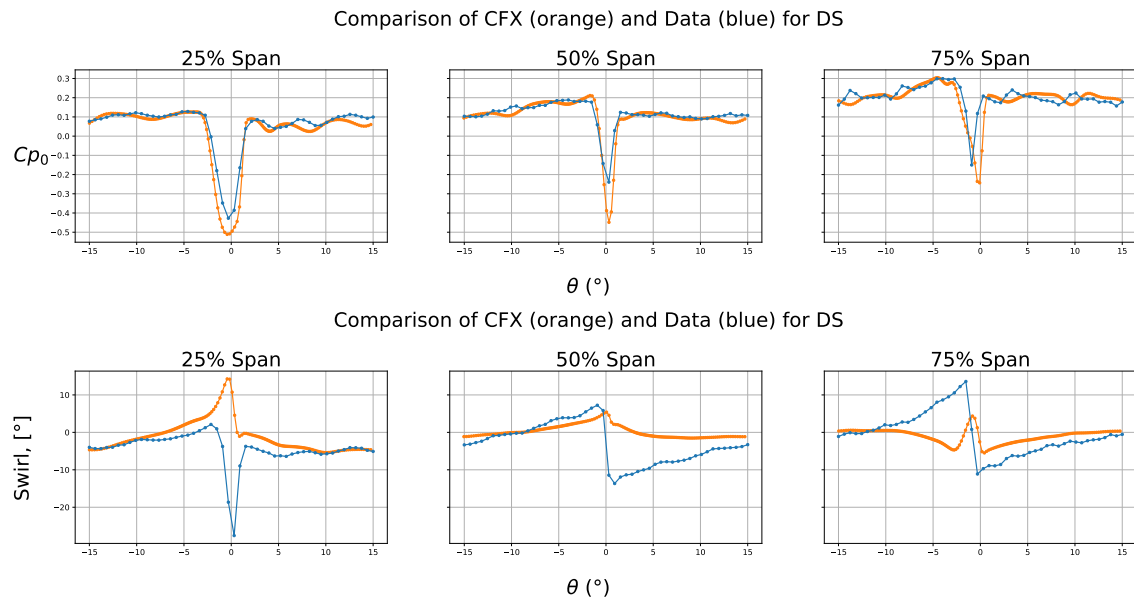


Figure A.23: Wake data, case 7

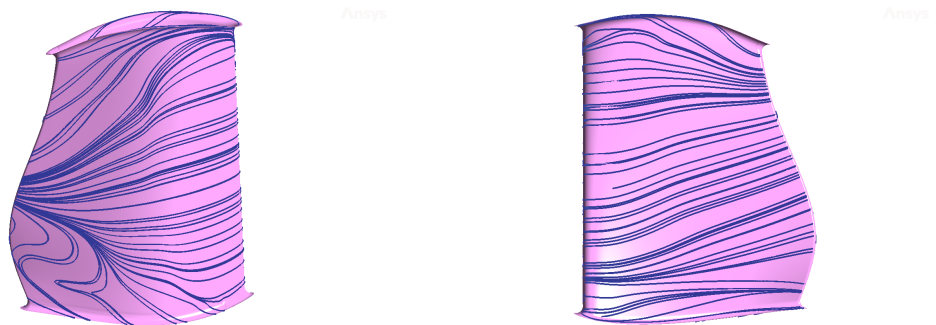


Figure A.24: Streamlines of the airflow at the OGV for case 7

A.7 Different Geometries

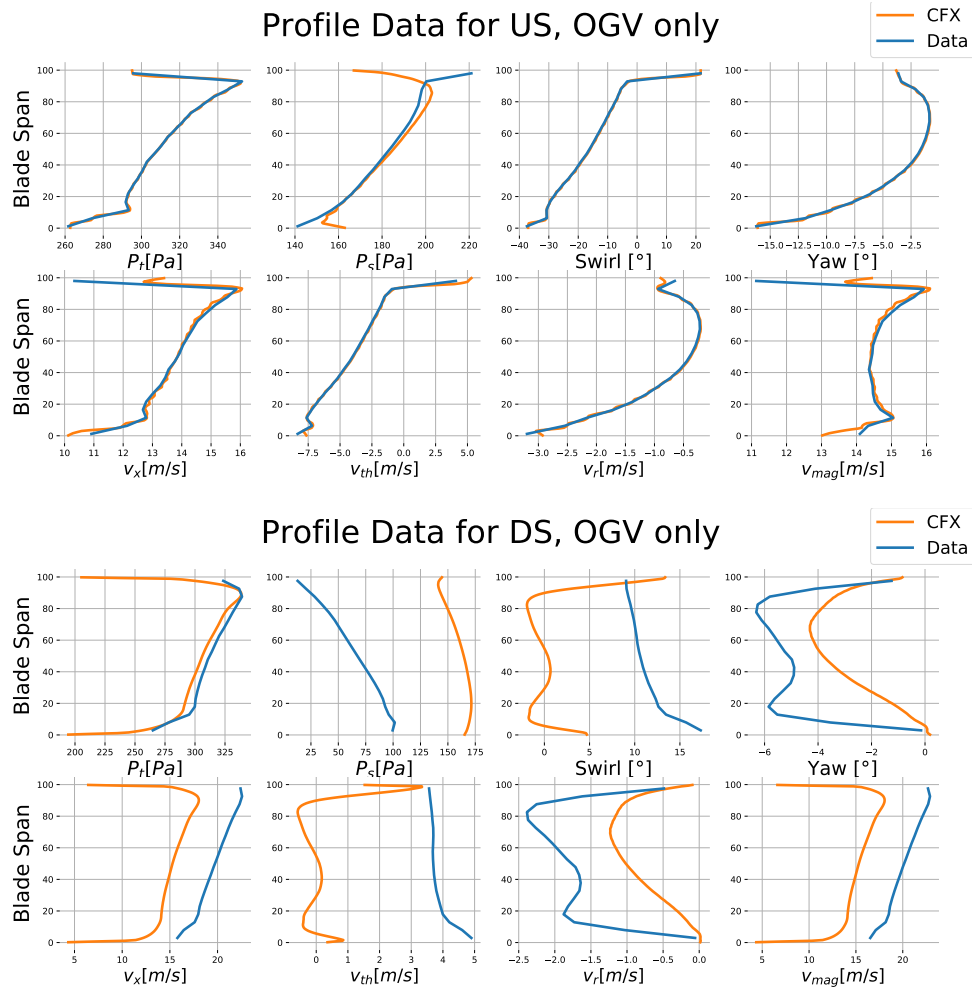


Figure A.25: tangentially averaged data for US and DS, only OGV case

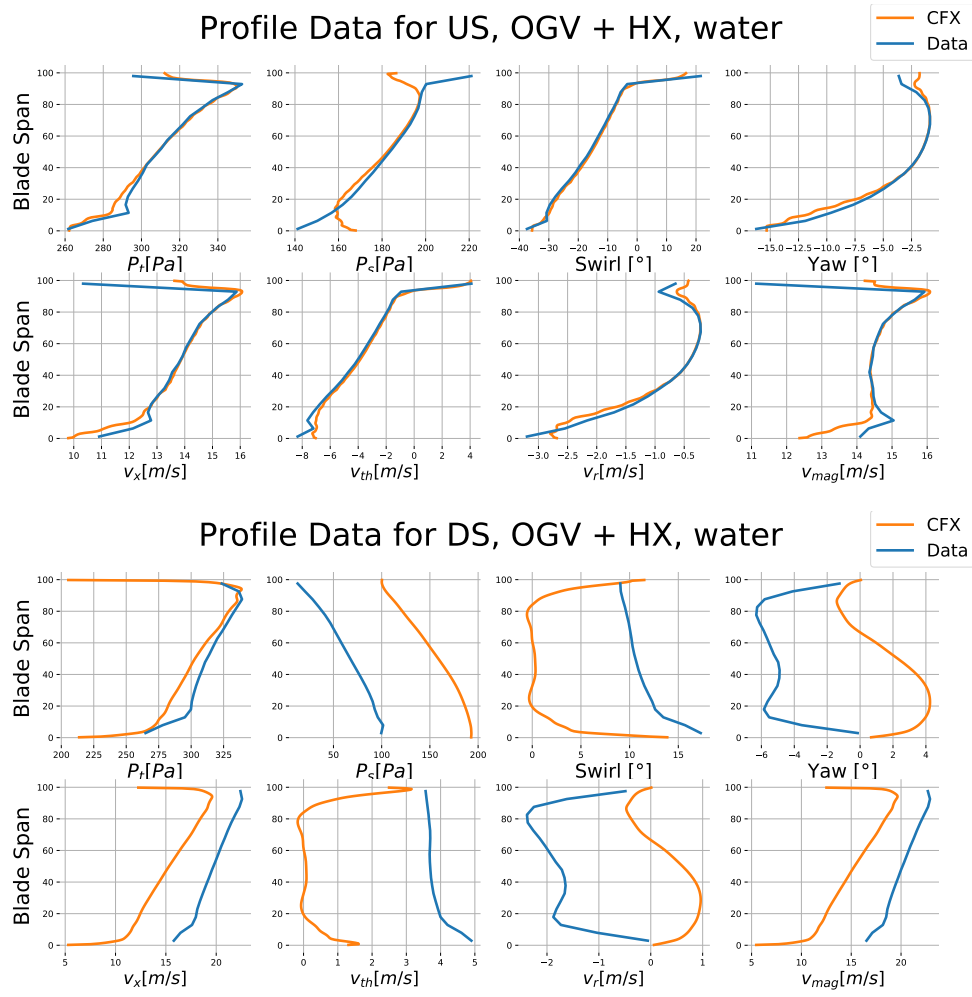


Figure A.26: tangentially averaged data for US and DS, case with TRS + HX and water enabled

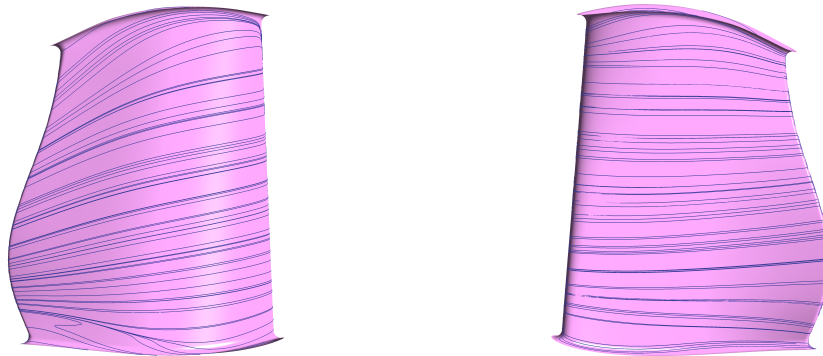


Figure A.27: Streamlines of the airflow at the OGV for radial inlet

A.8 1D inlet, Re235FC0.69

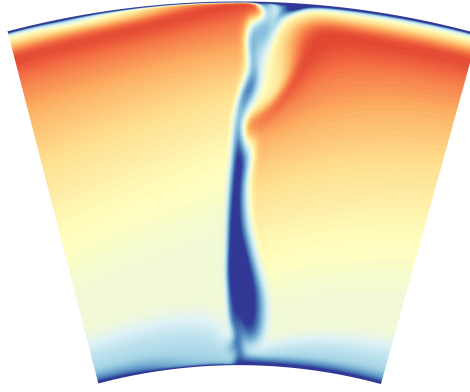


Figure A.28: cp_0 at DS

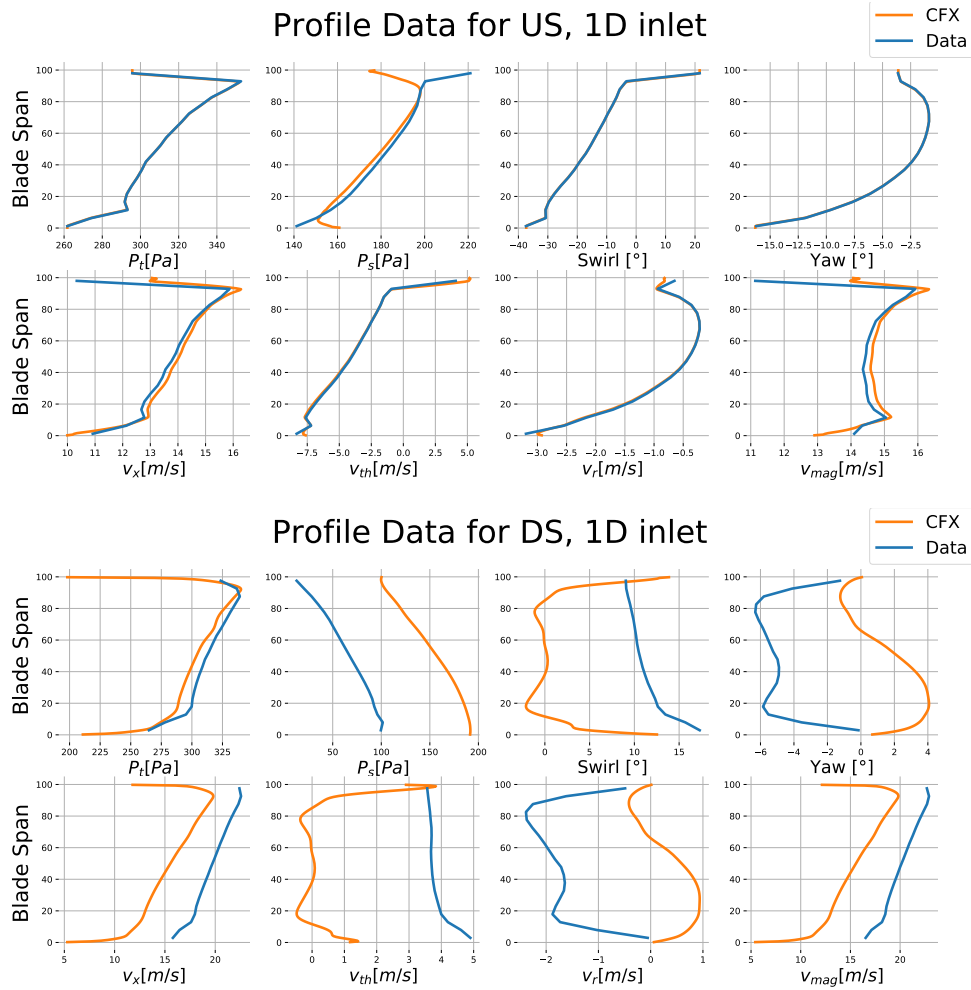


Figure A.29: tangentially averaged data for US and DS, radial inlet

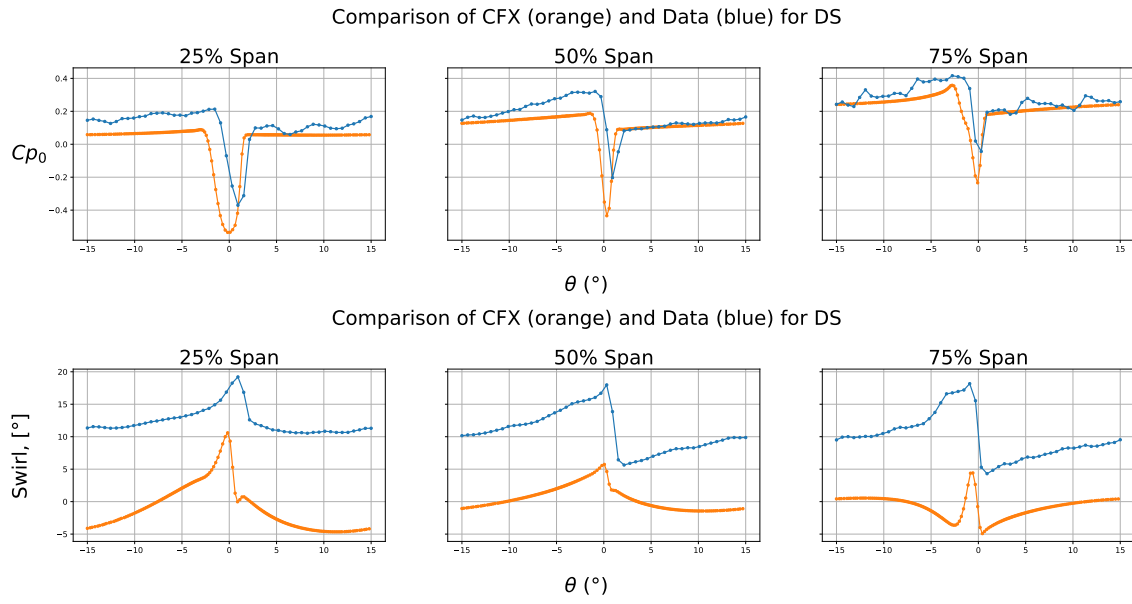


Figure A.30: Wake data, radial inlet

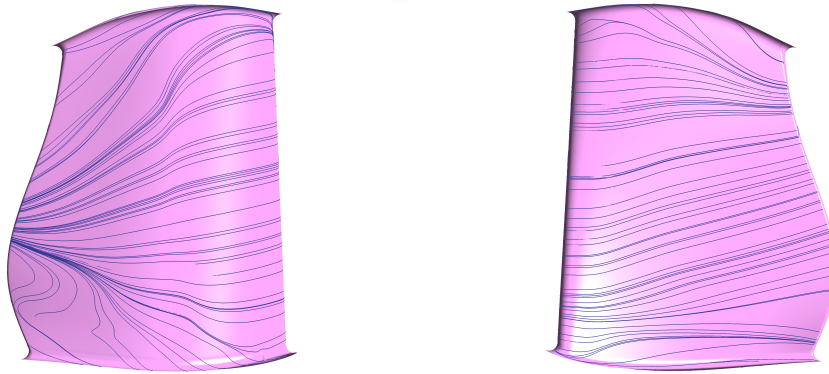
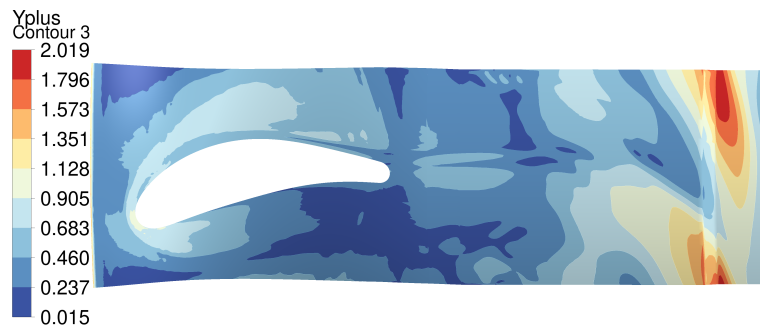


Figure A.31: Streamlines of the airflow at the OGV for radial inlet

A.9 y^+

Figure A.32: y^+ values at the hub

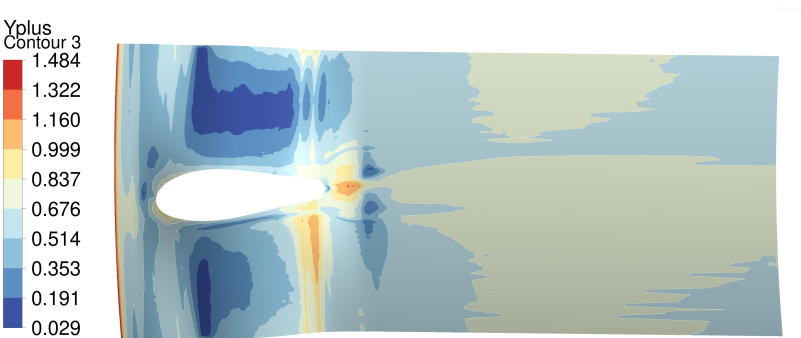


Figure A.33: y^+ values at the shroud

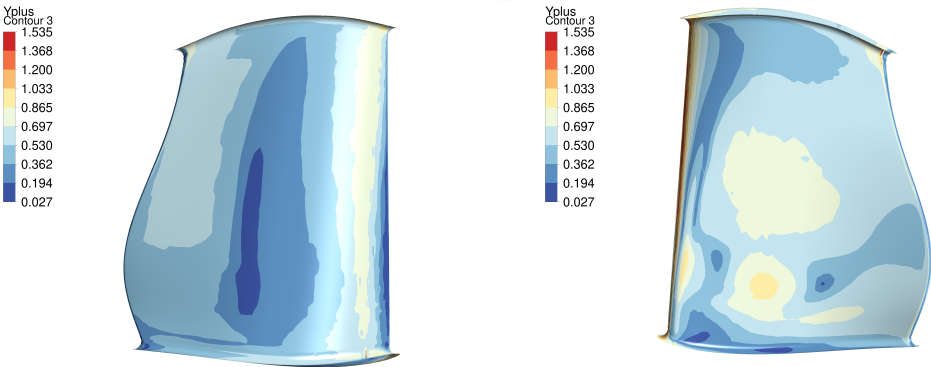


Figure A.34: y^+ values at the OGV, PS and SS

DEPARTMENT OF SOME SUBJECT OR TECHNOLOGY
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden
www.chalmers.se



CHALMERS
UNIVERSITY OF TECHNOLOGY