MASTER THESIS

ANALYTICAL PREDICTION OF TURBOCHARGER COMPRESSOR PERFORMANCE: A COMPARISON OF LOSS MODELS WITH NUMERICAL DATA

Author:
Sergio Sanz Solaesa

Coordinator at KTH:
Bertrand Kerres

Coordinator at UPM:
Rubén Abbas Cámara

September 2016
ABSTRACT.

One-dimensional models predict the performance of centrifugal compressor in short time, being a helpful design tool in the early design stages. They assume uniform flow through the compressor. Conservation of mass, momentum and energy and some empirical loss correlations are applied to estimate the real outputs.

In this thesis, this one-dimensional approach is applied to model a turbocharger compressor. Two different models are implemented. They consist of an impeller, a vaneless diffuser and a volute. The model stage outputs, pressure ratio and efficiency, are compared with experimental data. Then, both models are further investigated by comparing their losses prediction with validated Reynolds-Averaged Navier-Stokes (RANS) data.

The implemented models are taken from literature. They use the same vaneless diffuser and volute approach, but different impeller loss sets. The next impeller losses are studied: incidence, skin friction, choking, jet-wake mixing, blade loading, hub to shroud, tip clearance, shock and distortion losses. The vaneless diffuser outlet is calculated using a one-dimensional numerical solution to the underlying differential equations. For the volute, a set of empirical losses is used. The losses from the CFD are also measured by entropy rise calculations. Due to the complexity of this model, not all the losses can be independently extracted. Incidence, choking, skin friction, blade loading and jet-wake mixing losses are measured along the impeller. Besides, vaneless diffuser and volute losses are also obtained.

Results show relative total pressure ratio errors less than 5% in 49 points in a total of 77 predicted operation conditions. 69 points are estimated with a relative error less than 10%. CFD still gives better predictions, especially at low tip speeds. However, at high tip speeds one-dimensional gives similar accuracy. The one-dimensional and CFD losses comparison shows largest differences in the vaneless diffuser and volute models. Some strengths and weaknesses of the impeller losses are revealed, being possible future improvements.
ACKNOWLEDGEMENTS.

First, I would like to thank Bertrand Kerres for giving me this opportunity and for his support in all the way. I have learned a lot working with you. Also, I would like to thank Rubén Abbas for supervising my thesis from Spain. Thanks for all the comments and corrections given.

I would like to express my gratitude to my family, specially thanks to my parents Adolfo and Mónica and my sister Maríá. Without you I could not have lived this experience and your support helped me in those moments I needed.

How can I forget all the people I have met in Stockholm. Thank you all for being here. However, there are some people that deserve to have an special mention. Lappis Roots people, I will never forget the amazing times I have lived with all of you. This is just the beginning. Mention apart demand Juan, as I have been with you almost every minute I have spent in Sweden. Thank you for all your support, for hosting me and helping me to make the format of this Thesis.

By last, I would like to thanks my Spanish friends, as this master thesis is just the last step of a career that started long time ago, where you have been always with me.

For those of you that helped me directly or indirectly, many thanks.
# TABLE OF CONTENTS

**ABSTRACT** .................................................. I

**ACKNOWLEDGEMENTS** ....................................... III

**TABLE OF CONTENTS** ....................................... V

**LIST OF FIGURES** .......................................... VII

**LIST OF TABLES** ............................................ IX

**NOMENCLATURE** ............................................. X

## 1 INTRODUCTION.

1.1 Background. .................................................. 1

1.2 Purpose. ..................................................... 2

1.3 Methodology. ................................................ 2

## 2 FRAME OF REFERENCE.

2.1 Turbochargers: operating principle and components. ....... 3

2.2 Principles of compressor analysis. .......................... 4

2.2.1 Blade work input coefficient. ........................... 8

2.3 One-dimensional centrifugal compressor model. ............ 10

## 3 IMPLEMENTATION.

3.1 One-dimensional analysis. ................................... 11

3.2 Impeller model. .............................................. 12

3.2.1 Aungier set of losses. .................................. 14

3.2.2 Oh et al.’s set of losses. ............................... 19

3.3 Vaneless diffuser model. .................................... 20

3.4 Volute model. ................................................ 22

3.5 CFD losses. ................................................... 24

3.5.1 Impeller losses. ......................................... 24

3.5.2 Vaneless diffuser and volute losses. .................... 28

## 4 ANALYSIS AND RESULTS. ................................. 31
# LIST OF FIGURES

2.1 Turbocharger (Adapted from NASA). ............................................. 3
2.2 Centrifugal compressor geometry. ................................................. 4
2.3 Velocity triangles at the impeller inlet and outlet. .......................... 5
2.4 Impeller control volume (Adapted from Kerres et al. (2016)). ............... 5
2.5 h-s diagram of a centrifugal compressor. ......................................... 7
2.6 Compressor map (Adapted from Kerres (2014)). ............................... 8

3.1 Flow chart of the iterative calculation. ........................................... 12
3.2 Impeller calculation order applying Aungier pressure loss coefficients. .... .. 14
3.3 Impeller calculation order applying Oh et al. enthalpy loss coefficients. .... 14
3.4 Volute geometry. ................................................................. 23
3.5 Entropy field function. ............................................................ 25
3.6 h-s diagram along an impeller part. .............................................. 26
3.7 h-s diagram between impeller inlet and outlet. ............................... 27
3.8 Entropy field function. ............................................................ 28
3.9 Vaneless diffuser and volute sections. ......................................... 29

4.1 Total pressure ratio. ............................................................... 32
4.2 Relative TPR error between experimental measurements and 1D results. .... 33
4.3 Total-to-total adiabatic efficiency. Red dashed lines = ±10% ................. 34
4.4 Dimensionless blades work coefficient prediction. ................................ 35
4.5 Slip factor. ................................................................. 36
4.6 Internal loss coefficient. ...................................................... 37
4.7 Incidence loss coefficient. ....................................................... 38
4.8 Inlet flow and blade angle. .................................................... 39
4.9 Skin friction loss coefficient. .................................................... 39
4.10 Blade loading loss coefficient. .................................................. 40
4.11 Blade loading loss coefficient predicted by the one-dimensional models (without adding hub-to-shroud losses). ................................. 41
4.12 Mixing loss coefficient. ....................................................... 42
4.13 Entropy field at impeller tip. Negative values are reached since entropy is calculated by comparing to a reference point. ......................... 43
4.14 Wake area ratio. .......................................................... 43
4.15 Choking loss coefficient. ............................................. 44
4.16 Tip clearance loss coefficient. ................................. 45
4.17 Shock loss coefficient. ................................................ 46
4.18 Distortion loss coefficient. ........................................ 46
4.19 External loss coefficient. .......................................... 47
4.20 External losses in one-dimensional models. ............... 48
4.21 Reverse mass flow percentage. ................................. 48
4.22 Vaneless diffuser loss coefficient. ......................... 49
4.23 Volute funnel recovery coefficient. ......................... 50
4.24 Volute cone loss coefficient. .................................. 51
4.25 Pressure loss coefficients in 1D volute funnel models. .... 51
4.26 Losses breakdown ($U_2 = 0.62N_0$). ..................... 53
4.27 Losses breakdown ($U_2 = 1.14N_0$). ..................... 53
4.28 Losses breakdown ($U_2 = 1.41N_0$). ..................... 53

A.1 Turbocompresor (Adaptado de NASA). ....................... 60
A.2 Geometría de un compresor centrífugo. ...................... 60
A.3 Algoritmo de cálculo en cada componente. .................. 63
A.4 Diagrama h-s para cálculo de pérdidas del rotor en el modelo CFD. ............. 65
A.5 Volúmenes de control tomados en el rotor. ................... 66
A.6 Ratio de presión total en cada modelo y experimental. .... 67
A.7 Suma de pérdidas internas a lo largo de cada componente. ...... 68
A.8 Pérdidas en el rotor. .................................................. 70
## LIST OF TABLES

2.1 Compressor geometrical properties (Kerres et al., 2016). ........................................ 4

3.1 Loss mechanisms included in each set. ................................................................. 15
3.2 CFD model properties (Adapted from Kerres et al. (2016)). ................................. 24
3.3 Impeller parts and related losses. ................................................................. 25

4.1 Inlet conditions and gas used in the simulations. ............................................... 31
4.2 Internal loss mechanisms studied at each model. .............................................. 36

A.1 Datos geométricos compresor, modelo CFD y condiciones de entrada. ................. 62
A.2 Pérdidas implementadas en cada componente del compresor. ......................... 64
**NOMENCLATURE**

**Latin letters.**

<table>
<thead>
<tr>
<th>Latin Letter</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Area</td>
</tr>
<tr>
<td>$A_R$</td>
<td>Area ratio</td>
</tr>
<tr>
<td>B</td>
<td>Fractional area blockage</td>
</tr>
<tr>
<td>C</td>
<td>Absolute velocity</td>
</tr>
<tr>
<td>$C_r$</td>
<td>Throat contraction ratio</td>
</tr>
<tr>
<td>D</td>
<td>Divergence parameter</td>
</tr>
<tr>
<td>$D_f$</td>
<td>Diffusion factor</td>
</tr>
<tr>
<td>I</td>
<td>Work input term</td>
</tr>
<tr>
<td>L</td>
<td>Length</td>
</tr>
<tr>
<td>M</td>
<td>Mach number</td>
</tr>
<tr>
<td>PR</td>
<td>Pressure ratio</td>
</tr>
<tr>
<td>$Q_c$</td>
<td>Corrected flow coefficient</td>
</tr>
<tr>
<td>R</td>
<td>Gas constant</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds numbers</td>
</tr>
<tr>
<td>Ro</td>
<td>Rothalpy</td>
</tr>
<tr>
<td>SP</td>
<td>Sizing parameter</td>
</tr>
<tr>
<td>T</td>
<td>Temperature</td>
</tr>
<tr>
<td>U</td>
<td>Tangential velocity</td>
</tr>
<tr>
<td>W</td>
<td>Relative velocity</td>
</tr>
<tr>
<td>Z</td>
<td>Number of blades</td>
</tr>
<tr>
<td>a</td>
<td>Speed of sound</td>
</tr>
<tr>
<td>b</td>
<td>Width</td>
</tr>
<tr>
<td>$c_f$</td>
<td>Skin friction coefficient</td>
</tr>
<tr>
<td>$c_p$</td>
<td>Specific heat capacity at constant pressure</td>
</tr>
<tr>
<td>d</td>
<td>Diameter</td>
</tr>
<tr>
<td>$d_h$</td>
<td>Hydraulic diameter</td>
</tr>
<tr>
<td>e</td>
<td>Wall roughness</td>
</tr>
<tr>
<td>$f_c$</td>
<td>Correction factor</td>
</tr>
<tr>
<td>$f_{DF}$</td>
<td>Disk friction factor</td>
</tr>
<tr>
<td>h</td>
<td>Specific enthalpy</td>
</tr>
</tbody>
</table>
$m$ Meridional coordinate
$m$ Mass flow
$p$ Pressure
$r$ Radius
$s$ Specific entropy
$s_{cl}$ Blade-shroud clearance
$t$ Bale width
$y$ Distance from the wall
$w$ Specific work

**Greek letters.**

$\Omega_c$ Corrected angular speed coefficient
$\alpha$ Absolute flow angle to tangential direction
$\alpha_c$ Flow angle to axial direction
$\beta$ Relative flow angle to tangential direction
$\beta_b$ Blade angle to tangential direction
$\gamma$ Heat capacity ratio
$\delta$ Boundary layer thickness
$\varepsilon$ Radius ratio
$\varepsilon_{wake}$ Wake area ratio
$\kappa_m$ Streamline curvature
$\lambda$ Enthalpy loss coefficient
$\lambda_{dis}$ Distortion factor
$\mu$ Viscosity
$\eta$ Adiabatic efficiency
$\rho$ Density
$\sigma$ Slip facor
$\phi$ Flow coefficient
$\chi$ Wake mass flow ratio
$\omega$ Rotational speed
$\bar{\omega}$ Dimensionless pressure loss coefficient

**Subscripts.**

0 Total quantity
1 Impeller inlet
2 Impeller outlet
3 Diffuser outlet
4 Volute cone inlet
5 \hspace{1cm} \text{Volute cone outlet}
\infty \hspace{1cm} \text{Perfect guidance case}
\text{FB} \hspace{1cm} \text{Full Blade}
\text{SB} \hspace{1cm} \text{Splitter Blade}
\text{cl} \hspace{1cm} \text{Blade-shroud clearance}
\text{ext} \hspace{1cm} \text{External}
\text{h} \hspace{1cm} \text{Hub}
\text{int} \hspace{1cm} \text{Internal}
\text{m} \hspace{1cm} \text{Meridional component}
\text{u} \hspace{1cm} \text{Tangential component}
\text{r} \hspace{1cm} \text{Relative quantity}
\text{s} \hspace{1cm} \text{Isentropic condition}
\text{t} \hspace{1cm} \text{Blade tip}
\text{th} \hspace{1cm} \text{Throat parameter}
0 \hspace{1cm} \text{Total quantity}

\textbf{Superscripts.}
\bar{()} \hspace{1cm} \text{Average}
()^* \hspace{1cm} \text{Sonic conditions}
()^{dc} \hspace{1cm} \text{Design conditions}
()^L \hspace{1cm} \text{Loss}

\textbf{Acronyms.}
\textit{CFD} \hspace{1cm} \text{Computational Fluid Dynamics}
\textit{RANS} \hspace{1cm} \text{Reynolds-averaged Navier–Stokes}
Chapter 1

INTRODUCTION.

1.1 Background.

Since the first turbocharger was invented in 1925 by the Swiss engineer Alfred Büchi, their importance in engines has been always growing. Nowadays, turbochargers are assembled in a lot of engines, from small commercial engines to racing F1 engines. The reason why turbochargers have become so important is the downsizing trend.

Engine downsizing means making the engines smaller but keeping the same or even increasing the power output. Higher efficiencies are obtained by downsizing due to the reduction of thermal and friction losses. These higher efficiencies reduces the fuel consumption and the emissions which are the main goals behind the downsizing process. To do it, modern technologies, such as turbocharging, direct injection or variable timing valve, are added in the engine. This trend is very present nowadays and it can be seen how six-cylinder engines are replacing the eight ones, four-cylinders are replacing the six ones and, recently, the new three-cylinder engines are replacing fours (Squatriglia, 2016).

Turbocharging is one of the most common techniques used by manufacturers to achieve the engine downsize. The aim of turbocharging is to increase the mass of air introduced into the cylinder during the intake stroke. More air means that more fuel can be burnt, increasing the power output per cycle. Furthermore, efficiencies are higher using turbocharges since the exhaust gas energy is used instead of being lost as in atmospheric engines.

A design challenge is to design a engine-turbocharger system with a high level of matching in all range of different regimes. Therefore it is always desirable to obtain a mathematical model of it. Since a turbocharger consists of a turbine and a compressor, a model of each component need to be developed and coupled. These mathematical models allow to understand the system behaviour and performance, without needing too many expensive and long experiments.

This thesis describes an example of compressor model implementation. Nowadays, Computational Fluid Dynamics (CFD) software can run complex 3D models, getting accurate predictions. However, these simulations require long computational times and very expensive hardware. Since several decades ago, analytical models have been developed. These models make some assumptions that allows to predict whole compressor maps in short times. Moreover, they need a relatively low number of input parameters. Therefore, they are a useful tool during the design process of the centrifugal compressor, and furthermore, in the design of the turbocharger and turbocharger-engine system.
Thanks to the studies carried out in the last decades, in the open literature it is possible to find different methodologies and correlations to develop analytical compressor models. In this thesis, two of those analytical models are implemented and compared with experimental and numerical data.

1.2 Purpose.

The main goal of this thesis is the implementation of an analytical centrifugal compressor performance prediction and to compare its results with numerical and experimental data. To do it a turbocharger compressor is used as reference. It consists of a impeller, a vaneless diffuser and a volute. Therefore, those are the modelled components.

To verify the validity of the model, its results are compared with experimental data, measuring the quality of the compressor outputs predictions.

Finally, to further understand the model, another goal of the thesis is to compare the model losses with validated RANS data. The CFD model and simulations are already done and they are not a part of this study. However, it is a task of this study to extract the losses from the CFD. This comparison should yields some conclusions about the strengths and weaknesses of the model.

1.3 Methodology.

Among the possible one-dimensional models that will be described in Chapter 2, the single-zone approach is chosen since in the open literature it is possible to find several methodologies and correlations to carry it out. Two different single-zone models are implemented in this thesis.

- The methodology described by Aungier (2000), which has been validated by hundreds of different compressor stages. Approaches and losses sets for all the compressor components are provided and, therefore, it is a very complete model.

- The impeller losses set provided by Oh et al. (1997), who provides this set after comparing 144 different combinations of impeller losses with experimental data. The impeller loss correlations used in this comparison are based on the work done by many authors.

Both models are implemented in MATLAB® owing to its proper features to programme and calculate engineering and scientific problems. They are deeper explained in Chapter 3.

Once the model are implemented, the comparison with experimental data is done by calculating the relative error of the total pressure ratio and adiabatic efficiency in all the predicted points.

Finally, the model losses are compared with a validated CFD model. This model is done in STAR-CCM+®. Due to the complexity of the flow, some assumptions are done to independently measure some losses. Then, the comparison is done by analysing the trend and values range of each loss across the whole compressor map.
Chapter 2

FRAME OF REFERENCE.

2.1 Turbochargers: operating principle and components.

A brief description of how turbocharger works is explained in the next paragraphs. In the literature can be easily found many books about them, such as Payri and Desantes (2011).

Turbochargers are turbine-driven forced induction devices that increase engine efficiency and power output. A turbocharger consists of a centrifugal compressor and a radial turbine joined by a common shaft (Figure 2.1). Exhaust gasses of the engine flow through the turbine. The turbine uses the thermal fluid energy to drive the compressor through the common shaft. Then, the compressor takes atmospheric air, increases its pressure and sends it to the engine’s intake. Since the air pressure and density are higher the amount of air that is introduced in the engine is larger. Therefore, the engine can generate more power through the combustion of more air and fuel. Efficiency also increases due to the higher compression ratio reached.

![Figure 2.1: Turbocharger (Adapted from NASA).](image-url)
Because of this thesis is focused on the centrifugal compressor, just this component is described in depth. Figure 2.2 shows the main parts of a compressor. First the impeller is rotating thanks to the power applied on the shaft and the fluid flows across it. The impeller blades have a specific passage between them that increases the fluid pressure and reduces its relative velocity. After leaving the impeller, the flow goes into the diffuser which further increases its pressure and decelerates it more. The diffuser can be vaneless or vaned, although vaneless diffusers are more common in turbochargers due to their wider operation range. Finally, the flow goes through the volute where pressure rises more and the velocity is adapted to the exit requirements.

![Centrifugal compressor geometry](image)

The compressor analysed in this thesis is a compressor for a passenger car turbocharger. Its main geometrical properties are summarized in Table 2.1.

<table>
<thead>
<tr>
<th>Compressor geometry</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Exducer diameter</td>
<td>52.5mm</td>
</tr>
<tr>
<td>Number of full blades</td>
<td>6</td>
</tr>
<tr>
<td>Number of splitter blades</td>
<td>6</td>
</tr>
<tr>
<td>Diffuser Area Ratio $A_R$</td>
<td>0.708</td>
</tr>
<tr>
<td>Mass flow coefficient $\phi_{dc}$</td>
<td>0.094</td>
</tr>
<tr>
<td>Work coefficient $\phi_{dc}$</td>
<td>0.622</td>
</tr>
<tr>
<td>Blade Mach number $M_{2}$</td>
<td>1.09</td>
</tr>
</tbody>
</table>

### 2.2 Principles of compressor analysis.

In this section, a brief study of the principles of compressors is carried out. This analysis can be found in many turbomachinery book. The books of Cumpsty (2004) and Korpela (2011) have been the main theoretical references used in this study.

The interaction between the blades and the flow is the essential process of the compressor operation. By this interaction, the work is transferred from the blades to the flow. It is convenient to study the flow velocity at the inlet and at the outlet in order to obtain the work applied on the shaft. To do it, velocity triangles are developed examining the flow in a stationary frame and in a moving one that rotates at the...
same angular speed as the impeller. Fig. 2.3 shows these velocity triangles of a backswept centrifugal impeller assuming that the inlet flow has no radial component and, at the outlet, it has no component in the axial direction. The absolute velocities in the stationary frame are denoted by C, while relative velocities by W and the speed due to the rotation of the moving frame by U. Besides, the angles that the absolute and relative velocities form with the tangential direction are denoted by $\alpha$ and $\beta$ respectively. These flow angles are called absolute and relative flow angles. Subscripts $u$ and $m$ respectively denote tangential and meridional components.

Considering now the flow through the impeller as shown in Fig. 2.4 and taking a control volume that includes both the impeller and the fluid, it is possible to calculate the work transferred. To obtain it, angular momentum balance is applied and the torque input on the shaft is

$$\tau = \dot{m}(r_2C_{u2} - r_1C_{u1})$$  \hspace{1cm} (2.1)

$\dot{m}$ being the mass flow, $r_2$ and $r_1$ the mean outlet and inlet radius and $C_u$ the tangential velocity. Multiplying the torque by the angular speed $\omega$ and dividing it by the mass flow, the power input per unit mass is

$$w = \omega(r_2C_{u2} - r_1C_{u1})$$  \hspace{1cm} (2.2)
Finally, taking into account that the blade speed ($U$) is equal to $U = r\omega$, Formula 2.2 can be recasted and it leads to Euler equation of turbomachinery obtained.

$$w = U_2 C_{u2} - U_1 C_{u1}$$  

(2.3)

Experiments show that some phenomena, such as slip or inlet distortion, decrease the real blade input work comparing to the Euler’s formula. Analytical approaches introduce some empirical factor to model them. They are explained subsequently in Section 2.2.1.

It is important to continue the analysis going into the thermodynamic involved. It will allow to relate the power input and flow velocities with the fluid properties (mainly temperature and pressure).

The first law of thermodynamics is applied to the same volume control than before. Since turbomachinery flows are almost adiabatic and the potential energy variation in gas flows can be neglected, the work input per unit mass is

$$w = h_2 - h_1 + \frac{1}{2} C^2$$  

(2.4)

being $h$ the static enthalpy. Defining the stagnation enthalpy ($h_0$) as the sum of the static enthalpy and the kinetic energy ($h_0 = h + \frac{1}{2} C^2$), Equation 2.4 can be simplified as follows.

$$w = h_{02} - h_{01}$$  

(2.5)

It is seen how the work applied on the shaft is absorbed by the fluid increasing its stagnation enthalpy. Through the Euler equation and Equation 2.4, the next equation is obtained.

$$R_o = h_2 - U_2 C_{u2} = h_1 - U_1 C_{u1}$$  

(2.6)

The parameter $R_o$ is called rothalpy. It is seen that is constant across the impeller. Rearranging Formula 3.2 leads to

$$h_2 - h_1 = \frac{1}{2} (U_2^2 - U_1^2) + \frac{1}{2} (W_2^2 - W_1^2)$$  

(2.7)

Equation 2.7 shows how static enthalpy and pressure rise can be achieved across the impeller by two methods: a) increasing the centrifugal effect (first term) or b) decelerating the relative flow (second term) (Korpela, 2011). While the deceleration process produces some losses, the pressure rise by the centrifugal effect ($U_2 > U_1$) is a loss-free process. This fact has favoured the developed of centrifugal compressors due to reasonable large efficiencies can be reached even without high aerodynamic performance. The flow at the exit of the impeller has usually large absolute velocity and that is why a diffuser is needed afterwards. The diffuser decelerates the flow further increasing the pressure more. Finally, the volute continues decelerating and rising the pressure. It is convenient to analyze the whole process in a h-s diagram. Figure 2.5 shows it. Label 1 and 2 represent the impeller inlet and outlet, while label 3 indicates the diffuser outlet and label 5 the volute outlet. Label 4 is reserved in this thesis to a intermediate volute point.

The diagram shows both the adiabatic process line (from 1 to 5) and the isentropic process line (from 1 to 5_s). Comparing the work input with the necessary work to reach the same exit pressure but following an isentropic process, the total-to-total efficiency can be defined as

$$\eta_{tt} = \frac{\text{work}}{w} = \frac{h_{03s} - h_{01}}{h_{03} - h_{01}}$$  

(2.8)
CHAPTER 2. FRAME OF REFERENCE.

This efficiency in centrifugal compressor is usually between 0.70-0.85 (Campbell, 1992).

Apart from the h-s diagram, the compressor map is also useful to characterize the compressor performance. It usually shows the pressure ratio as a function of the flow. Efficiency and constant speed curves are also plotted. Corrected coefficients of the flow and angular speed are used and they are defined below.

\[
Q_c = Q \sqrt{\frac{T_{0r}}{T_{01}}} \quad \Omega_c = \Omega \sqrt{\frac{T_{0r}}{T_{01}}}
\]

These corrected coefficients come from the flow coefficient \((\phi_d = \frac{\dot{m} \sqrt{R T_{01}}}{p_{01} d^2})\) and the blade Mach number \((\frac{\omega d}{\sqrt{\gamma R T_{01}}})\) for compressive flows. These last non-dimensional coefficients are used in the similitude studies of turbomachinery, which are powerful tools to compare the performance of two turbomachines or to forecast the performance of a turbomachine from its prototype’s outputs (Korpela, 2011). Corrected coefficients are not dimensionless since they are derived from the flow coefficient and blade Mach number but particularizing for a specific fluid and compressor design. Therefore, corrected coefficient are used to compare different working conditions of the same compressor.

Figure 2.6 shows an example of a compressor map. It is seen that there are two regions where the compressor can not operate in normal conditions. Above the surge line, the blades will stall and a phenomena called surge may appear. Even though there are different types of stall, their consequences are much less severe than the surge ones. In centrifugal compressors it is possible to operate under stable conditions with stall present, although the efficiency and pressure rise are lower (Cumpsty, 2004). On the contrary, surge generally produces an unstable and unacceptable operation with high flow and pressure oscillations and reverse flows. Apart from this oscillations, surge usually produces vibration and an audible sound that make it easily detectable. However, this vibration may reach excessive amplitudes and frequencies causing catastrophic effects on the whole system (Boyce et al., 1983). After many years of research on this field, there is still a great controversy about the cause and relation between stall and surge. Moreover, divergence of these results is increased due to surge and stall patterns have been
demonstrated to be quite diverse from one compressor to another (Zheng and Liu, 2015). In the other side of the compressor map, when the mass flow reaches very high values, the flow is choked and it is not possible to increase further the mass flow causing a sharp drop of the constant speed lines.

Figure 2.6: Compressor map (Adapted from Kerres (2014)).

2.2.1 Blade work input coefficient.

$I_B$ denotes the dimensionless blade work input coefficient. It is derived from Euler’s formula (Equation 2.3).

$$I_B = C_{u2}/U_2 - U_1 C_{u1}/U_2^2$$  \hspace{1cm} (2.9)

If the flow was perfectly guided by the blades, $C_{u2}$ would be equal to $C_{u2\infty} = U_2 - C_m \cot \beta_{2b}$ (see Figure 2.39). However, the real tangential flow velocity is slightly less than that. This loss is called slip. It is caused by the relative eddy rotation in a direction opposite to the impeller that appears to remains the flow irrotational in the absolute frame of reference. Therefore, a perfect blade guidance can not be considered and a slip factor $\sigma$ is introduced in the analytical study. It is described as

$$\sigma = \frac{C_{u2}}{C_{u2\infty}}$$  \hspace{1cm} (2.10)
This slip factor has been investigated for several decades and a large number of correlations have been
developed, such as the provided by Stodola (1945), Busemann (1928), Wiesner (1967) or the recent
model done by Qiu et al. (2007). Wiesner’s model is used in this thesis as it is done by Aungier (2000).
Following this model, the slip factor is described as

$$\sigma = 1 - \frac{\sin \alpha_c \sqrt{\sin \beta_{2b}}}{Z^{0.7}}$$

(2.11)

where \( z \) denotes the number of vanes. The impeller usually has splitter blades. In those cases, the number
of blades are corrected by \( Z = Z_{FB} + Z_{SB}L_{SB}/L_{FB} \).

Following the work done by Busemann (1928) and Wiesner (1967), (Aungier, 2000) introduces a cor-
rection procedure that considered the slip factor as constant up to a certain radius ratio \( \varepsilon_{LIM} \). At larger
\( \varepsilon \), slip ratio sharply falls. Therefore, slip ratio is calculated as follows.

$$\varepsilon = \frac{r_1}{r_2}$$

$$\sigma^* = \sin(19^\circ + 0.2 \sin \beta_{2b})$$

$$\varepsilon_{LIM} = \frac{\sigma - \sigma^*}{1 - \sigma}$$

$$\sigma = 1 - \frac{\sin \alpha_c \sqrt{\sin \beta_{2b}}}{Z^{0.7}}; \quad \varepsilon \leq \varepsilon_{LIM}$$

$$\sigma_{COR} = \sigma \left[ 1 - \left( \frac{\varepsilon - \varepsilon_{LIM}}{1 - \varepsilon} \right)^{\sqrt{\beta_{2b}/10}} \right] \quad \varepsilon > \varepsilon_{LIM}$$

(2.12)

On the other hand, (Aungier, 2000) also provides a distortion factor \( \lambda^{dis} \) that accounts for the impeller
tip blockage \( B_2 \). Both parameters are related as follows.

$$\lambda^{dis} = \frac{1}{1 - B_2}$$

(2.13)

Aungier (1995) offers an empirical correlation to calculate \( B_2 \) that fits a wide range of compressor per-
formance. This correlation is

$$b_2 = \delta_{BF} \frac{p_{\nu 1}}{p_{\nu 2}} \sqrt{\frac{W_1 b_h}{W_2 b_{th}}} + 0.3 + \frac{b_2^2}{L_B^2} A_R^2 + \frac{S_{CL}}{2b_2}$$

(2.14)

where \( p_\nu = p_0 - p \) and \( A_R = A_2 \sin \beta_{2b}/(A_1 \sin \beta_{th}) \).

Therefore, taking into account Equation 2.9, the slip factor \( \sigma \) and the tip distortion factor \( \lambda^{dis} \),
the blade work input coefficient is described as (Aungier, 2000).

$$I_B = \sigma(1 - \lambda^{dis} \phi_2 \cot \beta_{2b}) - U_1 C_{a1}/U_2^2$$

(2.15)
2.3 One-dimensional centrifugal compressor model.

Thanks to the researches done in the last decades, several one-dimensional approaches have been developed to estimate centrifugal compressor performance. These analytical models make different assumptions to provide a whole compressor map prediction in short time. Depending on the assumptions made, one-dimensional models can be classified by the number of flow zones studied, as shown in Harley et al. (2013), Schneider et al. (2015) or Kerres et al. (2016). Therefore, one-dimensional models can be divided into the next three categories: zero-zone, single-zone and two-zone analyses.

Zero-zone approach can be applied at the earliest design stage but it shows the lowest level of detail. An example of this model is the recent analysis implemented by Casey and Robinson (2013). It assumes that compressors for the same purpose have similar design and performance. Therefore, it compares the dimensionless flow coefficient, work coefficient and blade Mach number at the design point with the coefficients of another similar compressor whose map is completely known. Then, the whole map of the studied compressor is estimated by extrapolating all the off-design conditions from the reference compressor map and the relation between the dimensionless coefficients.

Single-zone models go further and they focus on the main stream line. Assuming uniform flow at every section, conservation of mass, momentum and energy are applied. Rothalpy conservation is also applied in the impeller in order to estimate the work input. Finally, some experimental correlation losses are introduced to calculate the real flow conditions. Therefore, this approach needs more geometry information than zero-zone models but the level of detail reached is higher, being possible to separately study each component performance. In order to implement single-zone models, several loss sets have been developed during the last decades, such as the sets of Galvas (1974), Aungier (1995), Oh et al. (1997) or (Gravdahl and Egeland, 1999).

Finally, two-zone approach is based on the jet-wake flow profile at the impeller tip. The reference is the study done by Dean and Senoo (1960). Jet flow appears at the pressure blade surface and it is assumed to be loss-free. Therefore, all the losses are concentrated in the high entropy wake flow that appears close the suction surfaces. Both flows are separately calculated, applying conservation of mass, momentum and energy to each one. Loss correlations similar than the single-zone sets are applied to the wake. Finally, a jet-wake mixing process is computed. Additional input data is required in these models comparing to the zero- and single-zone approaches. Specially, some relations to estimate the jet and wake areas are needed. The most common methodologies are based on the definition of the mass flow fractions or the velocity ratio as proposed by van den Braembussche (2013). Schneider et al. (2015) provides a two-zone turbocharger compressor model by applying a constant velocity ratio of 0.2. On the other hand, Oh et al. (2001) develops a two-zone model based on an empirical correlation that relates the wake mass flow fraction to the wake area fraction.

As it was said in Section 1.3, the single-zone models carried out by Aungier (2000) and Oh et al. (1997) are the ones implemented in this thesis. The reasons are already explained in Section 1.3.
Chapter 3

IMPLEMENTATION.

3.1 One-dimensional analysis.

Due to the development of one-dimensional compressor’s model that has been carried out during the last decades, some different procedures and losses set have been built up. As it was said before, this thesis is focused on the methodology developed by Ronald H. Aungier (2000) and Oh et al. (1997). They have been implemented in MATLAB© owing to its proper features to programme and calculate engineering and scientific problems.

Following the modularity that is characteristic of these procedures, each component is separately modelled. To develop the full compressor’s model they are connected in series afterwards. Thus, the outlets of one component are the inlets of the next one as in a real compressor. This modularity leads into a large flexibility since several different compressor’s setups can be analyzed just changing the components added and the order in which they are connected.

To model each component, the geometry data and the model of the losses that affect along it are needed. The losses models are usually experimental correlations due to the difficulty to explain them by theoretical equations. Besides, it will be seen in the next sections how these experimental correlation are usually a function of the conditions at the component outlet. Since the outlet conditions are obviously a function of the losses too, an iterative procedure is needed. Therefore, each component’s performance is calculated as follows.

- First, the thermodynamic conditions and velocity triangles at the inlet are obtained. In the case of the impeller, inlet conditions are equal to inlet compressor conditions. In the other components, as it was said before, inlet conditions are equal to the conditions at the outlet of the component before.

- Then, outlet meridional velocity is guessed. In the case of the impeller, flow coefficient \( \Phi = \frac{m}{\rho_2 A_2 U_2} \) is guessed.

- After that, thermodynamic, fluid and turbomachinery equations are applied to calculate all the losses coefficients. Once the pressure losses are known, outlet conditions are obtained.

- Then, it is checked if there is convergence of mass flow. If the error is not small enough, the outlet meridional velocity or the flow coefficient is updated and the process is done again until convergence is reached.

- Finally, in the case of the impeller, external losses are applied as an enthalpy increment at constant pressure and the outlet conditions are updated.
Figure 3.1 shows this iterative procedure, similar to the one presented by Aungier (2000). It can be seen how the analysis finishes when ‘choke’ condition is reached. This condition is more general than real aerodynamic choke, e.g. when impeller efficiency is lower than 10% (Aungier, 2000). Even though vaneless diffuser can be calculated by the same iterative process, Aungier (1993) proposes a one-dimensional performance analysis based on four governing differential equations. Therefore, this iterative solution will be applied at the impeller and volute. The vaneless diffuser calculation will be explained later.

**3.2 Impeller model.**

Since Aungier describes the losses by pressure loss coefficients while Oh et al. described them by enthalpy losses there is a slight different between both calculations. On one hand, Aungier’s impeller analysis is mostly carried out on a rotating frame of reference that rotates at the rotor speed. Therefore, several relations between the relative and static conditions must be known. The first one is based on the fact that static enthalpy is the same in both frames of reference. It leads to Equation 3.1.

\[
h = h_{0r} - \frac{W^2}{2} = h_0 - \frac{C^2}{2}
\]  
(3.1)

The subscript \( r \) means relative property. Defining the relative Mach number as \( M_r = \frac{W}{a} \) and taking into account Equation 3.1 and isentropic relations between the static and total relative conditions, the following equations are obtained (Aungier, 1995).

\[
T_{0r} = T[1 + \frac{\gamma - 1}{2}M_r^2]
\]  
(3.2)

\[
p_{0r} = p[1 + \frac{\gamma - 1}{2}M_r^2\frac{\gamma}{\gamma - 1}]
\]  
(3.3)
Through the last equations, thermodynamic equations and velocity diagrams all the impeller conditions at the inlet can be easily known.

To calculate the outlet properties, once a flow coefficient is guessed, the isentropic outlet relative conditions are computed by the proper isentropic relations, equation of state, velocity diagrams and conservation of rothalpy, Ro, across the impeller. Rothalpy is defined as the difference between the total enthalpy and angular momentum and its conservation in a relative frame of reference leads to the next equation.

\[ Ro = h_0 - \omega r C_u = \text{const}. \]

\[ h_{02r} = h_{01r} + \left( U_2^2 - U_1^2 \right)/2 \] (3.4)

Non-isentropic outlet properties are obtained afterwards. To do it, the outlet relative total pressure is calculated by Equation 3.5 (Aungier, 1995).

\[ p_{02r} = p_{02r,s} - f_c (p_{01r} - p_1) \sum_i \Phi_i \] (3.5)

where \( \Phi_i \) are the different pressure loss coefficients that will be explained later. \( f_c \) is a correction factor needed to remain the entropy rise unchanged. It is defined as follows.

\[ f_c = \frac{\rho_{02r} T_{02r}}{\rho_{01r} T_{01r}} \] (3.6)

The discharge relative total temperature is easily calculated by Equation 3.4. Besides, the outlet total temperature is obtained by Euler’s formula (Equation 2.3) that leads to

\[ T_{02} = T_{01} + \frac{I_B U_2^2}{c_p} \] (3.7)

Once the outlet relative total conditions are known, total and static absolute conditions are computed by applying isentropic relations, equation of state and velocity diagrams.

On the other hand, the Oh et al. model is carried out mainly in the stationary frame of reference. Therefore, the isentropic outlet total enthalpy is calculated first by guessing an outlet flow coefficient, computing the blades work coefficient and applying the sum of all the enthalpy losses as follows.

\[ h_{02,s} = h_{01} + I_B U_2^2 - \sum_i \Delta h_i^f \] (3.8)

Isentropic and non-insentropic outlet total temperature are directly computed by Equations 3.8 and 3.7 respectively, dividing them by the specific heat capacity. Discharge total pressure is then given by

\[ p_{02} = p_{01} \left( \frac{T_{02,s}}{T_{01}} \right)^{\frac{\gamma}{\gamma - 1}} \] (3.9)

The rest of total and static outlet conditions are calculated by isentropic relations, equations of state and velocity diagrams.

Once mass flow convergence is reached, parasitic work terms (\( I_i = \frac{\Delta h_i^{ext}}{U_2^2} \)), such as the disk friction work or the recirculation work, are computed and applied at constant pressure in both models. Finally,
tip conditions are adjusted to this new enthalpy. Figure 3.2 and 3.3 show the different thermodynamic points that are involved within these processes and the order in which they are calculated. Not all the

\[ \Delta h_{b} = I_{b} U_{2}^{2} \]

\[ \Delta h_{\text{ext}} = \Delta h_{r} = (U_{2}^{2} - U_{1}^{2}) \]

\[ \sum \Delta h_{\text{int}} \]

\[ \bar{\omega}_{\text{inc}} = 0.8 \left[ 1 - C_{m1} / (W_{1} \sin(\beta_{1h}))^{2} + Z_{FB} t_{b1} / (2 \pi r_{1} \sin(\beta_{1h}))^{2} \right] \]

\[ Z_{FB} \text{ being the number of full blades and } t_{b1} \text{ the blade thickness at the inlet.} \]

The overall incidence loss is calculated by a weighted average of its value at the hub, mean and shroud. To calculate the values at the hub and shroud, \( C_{m1} \) must be corrected applying Equations 3.11. The mean
CHAPTER 3. IMPLEMENTATION.

Table 3.1: Loss mechanisms included in each set.

<table>
<thead>
<tr>
<th>Internal Losses</th>
<th>Aungier’s set</th>
<th>Oh et al.’s set</th>
</tr>
</thead>
<tbody>
<tr>
<td>Incidence</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Skin friction</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Blade-loading</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Wake-mixing</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Clearance</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Hub-to-shroud</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Choking</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Supercritical Mach</td>
<td>x</td>
<td></td>
</tr>
<tr>
<td>Distortion</td>
<td>x</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>External Losses</th>
<th>Aungier’s set</th>
<th>Oh et al.’s set</th>
</tr>
</thead>
<tbody>
<tr>
<td>Disk friction</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Recirculation</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Leakage</td>
<td>x</td>
<td>x</td>
</tr>
</tbody>
</table>

value is weighted 10 times as heavy as the hub and shroud values (Aungier, 2000).

\[
C_{mb1} = C_{m1}[1 + \kappa_{m1}b_1/2] \\
C_{mt1} = C_{m1}[1 - \kappa_{m1}b_1/2]
\] (3.11)

where \( \kappa_m \) denotes the streamline curvature.

**Choking loss.** When the Mach number at the throat approximates values of 1.0, some losses are caused because of the proximity to the choke conditions. To model these losses, first a contraction ratio correlation is calculated in order to model the aerodynamic blockage at the throat (Aungier, 2000).

\[
C_r = \sqrt{(A_1 \sin \beta_1 b_1/A_{th})}; \quad C_r \leq 1 - (A_1 \sin \beta_1 / A_{th} - 1)^2
\] (3.12)

The choking loss is modelled as

\[
X = 11 - 10C_rA_{th}/A^* \\
\omega_{CH} = 0; \quad X \leq 0 \\
\omega_{CH} = 0.5(0.05X + X^7); \quad X \geq 0
\] (3.13)

where \( A^* \) denotes the area for which sonic conditions would be reached. Dixon and Hall (1961) provides the next equation to calculate the impeller critical area.

\[
\frac{\dot{m}}{A^*} = \rho_{01}a_{01} \left( \frac{2 + (\gamma - 1)U_2^2/a_{01}^2}{\gamma + 1} \right)^{\gamma + 1} (\gamma - 1)
\] (3.14)
Skin friction loss. The friction between the wall and the flow causes a loss that is modelled as (Aungier, 2000)

\[
\bar{o}_{sf} = 4c_f(\bar{W}/W_1)^2L_b/d_h
\]

\[
\bar{W} = (W_1^2 + W_2^2)/2
\]

\[
\bar{W} \geq (W_{1b}^2 + W_2^2)/2
\]

where \( c_f \) denotes the skin friction coefficient, \( d_h \) the hydraulic diameter and \( L_{FB} \) the full blades length. \( c_f \) depends on the Reynolds number (\( Re = \rho C_d \mu \)), the wall roughness (\( e \)) and the hydraulic diameter. To estimate it, Aungier (2000) built up a series of experimental correlations based on several previous studies, such as Schlichting (1979). These correlations depend on the laminar or the turbulent nature of the flow. Therefore, for laminar flow, skin friction coefficient is calculated by

\[
c_{fl} = 16/Re_d; Re_d < 2000
\]

For \( Re_d \) values larger than 2000 the flow is considered as turbulent and other correlations must be applied. First, if the surface roughness is not very relevant and the wall can be considered as smooth wall, \( c_f \) is obtained by

\[
\frac{1}{\sqrt{4c_{fs}}} = -2\log_{10}\left[\frac{2.52}{Re_d\sqrt{4c_{fs}}}\right]
\]

and if the wall is fully rough, the turbulent skin coefficient friction is calculated by

\[
\frac{1}{\sqrt{4c_{fr}}} = -2\log_{10}\left[\frac{e}{3.71d}\right]
\]

The condition that estimates when the surface roughness becomes relevant is

\[
Re_e = (Re_d - 2000)e/d > 60
\]

For \( Re_e \) values greater than 60 the roughness starts to become significant, therefore the turbulent skin friction coefficient is calculated by

\[
c_{ft} = c_{fts}; Re_e < 60
\]

\[
c_{ft} = c_{fts} + (c_{ftr} - c_{fts})(1 - 60/Re_e); Re_e \geq 60
\]

\( c_{ft} \) must be used when \( Re_d \) is greater than 4.000 and the flow can be considered fully turbulent. Between 2000 and 4000, skin friction coefficient depends on both the laminar and the turbulent value and it is calculated by

\[
c_f = c_{fl} + (c_{ft} - c_{fl})(Re_d/2.000 - 1)
\]

The Reynolds number depends on the characteristic diameter of the problem. Since this non-dimensional parameter came from the study of fluids that flow through pipes, this diameter use to be equal to the pipe diameter. However, compressor analysis requires to denote others diameters. Specifically, for the impeller it is used the average of the hydraulic diameter at the throat and at the tip. Hydraulic diameter
is defined as

$$d_h = 4 \frac{(cross-sectional \ area)}{(wetted \ perimeter)}$$  \hspace{1cm} (3.22)

**Blade loading loss.** Due to the pressure gradient in the blade-to-blade direction, secondary flows are produced. These flows cause losses and they also may yield to impeller stall. The blade loading loss is modelled by (Aungier, 2000)

$$\bar{\omega}_{bl} = \left( \frac{\Delta W}{W_1} \right)^2 / 24$$  \hspace{1cm} (3.23)

where $\Delta W$ is

$$\Delta W = \frac{2 \pi d_2 U_B}{Z_{FB}}$$  \hspace{1cm} (3.24)

**Hub-to-shroud loading loss.** This loss is similar to the blade loading loss, but it is caused by the pressure gradient in the hub-to-shroud direction. Its effects are similar and it is modelled by (Aungier, 2000)

$$\bar{\omega}_{HS} = \left( \frac{\bar{\kappa} m \bar{b} W}{W_1} \right)^2 / 6$$  \hspace{1cm} (3.25)

$$\bar{\kappa} m = \frac{C_m^2 - C_m^1}{L_{FB}}$$

$$\bar{b} = \frac{(b_1 + b_2)}{2}$$

$$\bar{\omega} = \frac{(W_1 + W_2)}{2}$$

**Distortion loss.** Accounts for the mixing of the distorted meridional flow and the free stream flow. Following an abrupt expansion loss (Benedict et al., 1966), this loss is modelled by

$$\bar{\omega}_d = \left( \lambda^{dis} - 1 \right) C_m^2 / W_1$$  \hspace{1cm} (3.26)

**Wake mixing loss.** Aungier (1995) proposes that this loss caused by the mixing of the blade wake flow and the free steam flow can also be predicted by an abrupt expansion. First, the velocity at which the flow is separated must be predicted. It is calculated by Equations 3.27 where $D_{f,eq}$ denotes an equivalent diffusion factor and it is defined as $D_{f,eq} = W_{max} / W_2$. $W_{max}$ denotes the maximum relative reached in the blade passage and it is equal described as $W_{max} = (W_1 + W_2 + \Delta W) / 2$. $\Delta W$ is the average blade velocity difference and it is estimated by $\Delta W = 2 \pi d_2 U_B / (Z_{FB})$.

$$W_{SEP} = W_2; \quad D_{f,eq} < 2$$

$$W_{SEP} = W_2 D_{f,eq} / 2; \quad D_{f,eq} \geq 2$$  \hspace{1cm} (3.27)

The velocities before and after mixing are calculated as follows.

$$C_{m,wake} = \sqrt{W_{2_{SEP}}^2 - W_0^2}$$

$$C_{m,mix} = C_m^2 A_2 / (\pi d_2 b_2)$$  \hspace{1cm} (3.28)

Finally, the loss coefficient is estimated by

$$\bar{\omega}_{mix} = \left( \frac{(C_{m,wake} - C_{m,mix})}{W_1} \right)^2$$  \hspace{1cm} (3.29)
Clearance gap leakage loss. Loss caused in open impeller due to the fluid that flows though the clearance between the blades and the shroud. It is modelled as (Aungier, 2000)

\[ \bar{\omega}_{cl} = 2 \dot{m}_{cl} \Delta p_{cl} / (\dot{m} \rho L_1 W_1^2) \]  \hspace{1cm} (3.30)

where \( \dot{m}_{cl} \) and \( \Delta p_{cl} \) are the mass leakage flow and the pressure gradient across the gap respectively. To calculate both parameters, Aungier (1995) provides the next correlations. First, the leakage flow velocity is estimated as

\[ U_{cl} = 0.816 \sqrt{2 \Delta p_{cl} / \rho_2} \]  \hspace{1cm} (3.31)

where the factor 0.816 is derived by assuming an abrupt expansion and an abrupt contraction later when the fluid flows through the blades clearance. The leakage flow is predicted by

\[ \dot{m}_{cl} = \rho_2 s_{CL} ZL_B U_{cl} \]  \hspace{1cm} (3.32)

s being the blade-shroud clearance. The pressure gradient is estimated as

\[ \Delta p_{cl} = \frac{\dot{m} (r_2 C_{u2} - r_1 C_{u1})}{Z \bar{r} \bar{b} L_B} \]

\[ \bar{r} = (r_1 + r_2)/2 \]

\[ \bar{b} = (b_1 + b_2)/2 \]  \hspace{1cm} (3.33)

Supercritical Mach number loss. Shocks appears on the blade suction surface when the flow reaches supersonic conditions. When it happens, losses are generated and boundary layer separation may happen. Supercritical Mach number loss coefficient is described as (Aungier, 2000)

\[ \bar{\omega}_{cr} = 0.4 [ (M'_{1} - M'_{cr}) W_{max} / W_1 ]^2 \]  \hspace{1cm} (3.34)

where \( M'_{cr} \) is the inlet Mach number when sonic conditions are reached at the midpassage suction surface. It is predicted by (Aungier,1995)

\[ M'_{cr} = M'_{1} W^* / W_{max} \]  \hspace{1cm} (3.35)

Disk friction loss. The friction between the impeller and the fluid that flows through the clearance gaps causes an external loss. This leads in a increment of input work which is not used to raise the fluid pressure. Nece and Daily (1960) provides the next equation to predict disk friction losses.

\[ I_{DF} = \frac{\Delta h_{DF}^L}{U_2^2} = f_{df} \bar{\rho} r_2^2 U_2^2 / 4m \]

\[ \bar{\rho} = (\rho_1 + \rho_2)/2 \]

\[ f_{df} = \begin{cases} 2.67 / Re_{df}^{0.5} & \text{if } Re_{df} < 3 \cdot 10^5 \\ 0.0622 / Re_{df}^{0.2} & \text{if } Re_{df} \geq 3 \cdot 10^5 \end{cases} \]  \hspace{1cm} (3.36)

\[ Re_{df} = U_2 r_2 / \nu_2 \]

Recirculation loss. Some impellers show a significant raise in work input when working at low mass
flow. This is caused by fluid that flow back into the impeller tip (Aungier, 2000). Aungier (1995) models this loss by an empirical correlation as follows.

\[
I_{RC} = \frac{(D_{f,eq}/2 - 1)(W_{a2}/C_{m2} - 2 \cot \beta_2)}{W_{\text{max}}/W_2}; \quad I_{RC} \geq 0
\]

\[
D_{f,eq} = W_{\text{max}}/W_2
\] (3.37)

\(D_{f,eq}\) is the equivalent diffusion factor provided by Lieblen (1959). It is developed for axial compressor but it can be generalized to impellers. Lieblen predicts blade stall and thus flow recirculation when \(D_{f,eq} > 2\).

**Leakage loss.** The leakage flow that passes through the blade-shroud gap losses energy and it is reenergized again. Aungier (2000) describes this loss assuming that half of the leakage flow suffers this process and provides the next correlation.

\[
I_L = \frac{\dot{m}_{cl} U_{cl}}{(2U_2 \dot{m})} \tag{3.38}
\]

where \(\dot{m}_{cl}\) and \(\Delta p_{cl}\) can be calculated by Equations 3.31 and 3.32.

### 3.2.2 Oh et al.’s set of losses.

As shown in Table 3.1, all the losses of this set are also implemented in Aungier’s set. That is why each loss origin is not be explained again and just the formulas are shown below.

**Incidence loss.** Conrad et al. (1980) provides the next formula which is used in this set.

\[
\Delta h_{\text{inc}} = f_{\text{inc}} \frac{W_{a1}^2}{2} \tag{3.39}
\]

where \(f_{\text{inc}}\) is a value between 0.5 and 0.7 as recommended by the author.

**Skin friction loss.** This loss model is based on the work done by Jansen (1967) and it is described as follows.

\[
\Delta h_{\text{sf}} = 2 c_f \frac{L_{FB}}{d_h} \bar{W}^2
\]

\[
\bar{W} = \frac{C_{1t} + C_2 + W_{1t} + 2W_{1h} + 3W_2}{8} \tag{3.40}
\]

where the subscripts \(t\) and \(h\) denote the values at the impeller inlet tip and hub respectively.

**Blade loading loss.** This loss is modeled following the next formula developed by Coppage et al. (1956).

\[
\Delta h_{\text{bl}} = 0.05 D_f^2 U_2^2 \tag{3.41}
\]

\(D_f\) being the diffusion factor described as

\[
D_f = 1 - \frac{W_2}{W_1} + \frac{0.75 \Delta h_{\text{euler}}/U_2^2}{(W_{1t}/W_2) [(Z/\pi)(1 - d_{1t}/d_2) + 2(d_{1t}/d_2)]} \tag{3.42}
\]
Clearance gap leakage loss. Jansen (1967) described this loss as:

$$
\Delta h_{0cl}^L = 0.6 \frac{scL}{b_2} C_{m2} \sqrt{\frac{4\pi}{b_2 Z} \left( \frac{r_2^2 - r_{1b}^2}{(r_2 - r_{1t})(1 + \rho_2/\rho_1)} \right)} C_{u2} C_{m1}
$$

(3.43)

Wake mixing loss. Wake and jet mixing loss is modeled following the work done by Johnston and Dean (1966).

$$
\Delta h_{0mix}^L = \frac{1}{1 + \tan^2 \alpha_2} \left( \frac{1 - \varepsilon_{wake} - b^*}{1 - \varepsilon_{wake}} \right)^2 C_s^2
$$

(3.44)

\(\varepsilon_{wake}\) is the wake area ratio \((A_{2wake}/A_{2geo})\) at the impeller tip. To estimate it, two-zone modelling literature has been used. Specifically, the methodology described by Oh et al. (2001) has been implemented. \(\varepsilon_{wake}\) is estimated by an iterative solution adopting Japikse’s diffusion factor ((Japikse, 1996)) to calculate the jet relative velocity and the empirical correlation \(\chi = 0.93\varepsilon_{wake}^2 + 0.07\varepsilon_{wake}\), where \(\chi\) denotes the wake mass flow fraction. First, \(\chi\) is guessed to be a value from 0.15 to 0.25 as recommended by the author. Then, jet conditions are calculated by the estimated diffusion factor and assuming conservation of rothalpy. Finally, \(\varepsilon_{wake}\) is computed and \(\chi\) after it by the empirical correlation. If the difference between the initial guessed \(\chi\) and this last \(\chi\) is lesser than a certain tolerance, solution is declared as converged. If it is not, the process is repeated again starting from the last value of \(\chi\) obtained.

Disk friction loss. This external loss is the same than in Aungier’s set being described by Equation 3.36.

Recirculation loss. Oh et al. (1997) describes this external loss as follows.

$$
I_{RC} = 8 \cdot 10^{-5} \sinh(3.5\alpha_2^3) D_f^2
$$

(3.45)

where \(D_f\) is calculated by Equation 3.42

Leakage loss. This external loss is modeled by the correlation presented by Aungier (2000) which has been explained before (Equation 3.38).

3.3 Vaneless diffuser model.

Aungier (1993) provides a one-dimensional analysis for vaneless diffusers. It is also valid for any other annular passage that may be used in a compressor stage or passages used to connect different stages. It is based on typical one-dimensional analysis that apply the main fluid conservation equations. However, this analysis was improved by the author by adding some terms based on an comparison with experimental data. Aungier (1993) claims that the result is an analysis that reach even better results that some previous three-dimensional method predictions (e.g. Aungier (1988)).

This analysis consists of applying the next four differential equations (Equations 3.46 to 3.47) along the vaneless diffuser, therefore, iterative solutions are not needed.

$$
2\pi\rho bC_m(1 - B) = \dot{m}
$$

(3.46)

$$
\frac{bC_m d(rC_u)}{dm} = -rC_u c_f
$$

(3.47)

$$
\frac{1}{\rho} \frac{dm}{dm} = \frac{C_s^2 \sin(\alpha_c)}{r} - C_m \frac{dC_m}{dm} - \frac{Cc_u c_f}{b} - \frac{dI_D}{dm} - I_c
$$

(3.48)

$$
h_0 = h + 0.5C_s^2
$$

(3.49)
where \( m \) is the meridional coordinate, \( B \) the area blockage, \( I_D \) the diffusion loss term and \( I_C \) denotes the curvature losses.

Diffusion losses are based on the work done by Reneau et al. (1967) in classical diffusers. Aungier (1993) uses this work describing the vaneless diffuser as an analogy of classical diffuser but adding some empiric factors. The following equations are from this source.

\[
d\frac{I_D}{dm} = -2(p_0 - p)(1 - E) \frac{1}{\rho C} \frac{dC}{dm}
\]

(3.50)

\( E \) is a empirical diffusion factor described as

\[
E = \begin{cases} 
1 ; & D \leq 0 \\
1 - 0.2(D/D_m)^2 ; & 0 < D < D_m \\
0.8 \sqrt{D_m/D} ; & D \geq D_m
\end{cases}
\]

(3.51)

\( D_m \) and \( D \) are divergence parameters taken from the analogy with classical diffuser analysis. They are calculated by

\[
D = -\frac{b}{C} \frac{dC}{dm}
\]

(3.52)

\[
D_m = 0.4 \left( \frac{b_2}{L} \right)^{0.35}
\]

(3.53)

If at any point the diffuser divergence angle exceeds the value of 9°, a second estimation of \( I_D \) is carried out related to this excessive meridional gradient.

\[
I_D = 0.65(p_0 - p)[1 - (rb)_m/(rb)]/\rho
\]

(3.54)

where \((rb)_m = (rb)_1[1 + 0.16m/b] \). The larger diffusion local loss between Equation 3.50 and 3.54 is applied at tha point. On the other hand, curvature loss term is empirically described as (Aungier, 1993)

\[
I_C = \kappa_m(p_0 - p)C_m/(13\rho C)
\]

(3.55)

where \( \kappa_m \) denotes the stream sheet curvature which is given by (Aungier, 1993)

\[
\kappa_m = -\frac{\partial \alpha_C}{\partial m}
\]

(3.56)

Curvature loss is not significant in vaneless diffuser but it is always important to take into account when modelling other components such as crossover bends.

Finally, \( c_f \) is calculated as it is done in the impeller (Equations 3.16 to 3.21) and the area blockage \( (B) \) is estimated by guessing a 1/7th power law for the boundary layer velocity profile. This boundary layer model defines the boundary layer velocities as follows

\[
C_m = C_{me}(y/\delta)^{1/7} \\
C_u = C_{ue}(y/\delta)^{1/7}
\]

(3.57)

where the subscript \( e \) denotes the values at the boundary layer edge, \( y \) means the distance from the wall.
and $\delta$ is the boundary layer thickness. Taking into account these velocity profiles, blockage area is estimated then by integrating the mass flow across the passage. It leads to (Aungier, 1993)

$$\int_0^b \rho C_m dy = \rho b C_{me}[1 - (2\delta)/(8b)] = \rho b C_{me}(1 - B)$$

(3.58)

$$B = 2\delta/(8b)$$

Therefore, the boundary layer thickness must be known to estimate $B$. Aungier (1993) provides a method to predict $\delta$. Integrating the angular momentum flux across the passage (similarly than Eq.3.58) leads to

$$rC_u = rC_{ue}[1 - (2\delta)/(4.5b)]$$

(3.59)

If $rC_{ue}$ is known, Equation 3.59 provides a method to estimate $\delta$ along the vaneless diffuser since $rC_{ue}$ must remain constant (Aungier, 1993). At the diffuser inlet, conditions are known since they are equal than at the impeller outlet that have been computed before. Taking again the 1/7th power law and assuming a simple flat plate turbulent boundary layer for the impeller boundary layer (Aungier, 1993), $\delta$ at the impeller outlet and thus at the diffuser inlet is estimated by

$$\delta_2 = 0.373Re_{2}^{-1/5}L_{FB}$$

(3.60)

$Re_2$ being the Reynolds number at the impeller outlet. Therefore, once $\delta_2$ is known, $rC_u$ and thus $\delta$ and $B$ can be calculated along the passage during the calculation process based on integrating the Equations 3.46-3.49.

### 3.4 Volute model.

Volute performance is calculated following an iterative process as shown in the flow chart above (Figure 3.1). Aungier (2000) provides several pressure loss coefficient to calculate the non-isentropic conditions. These loss coefficient are based on previous volute analysis, mainly the work presented by Weber and Koronowski (1986). However, Aungier (2000) added some improvements to these last works. Figure 3.4 shows a general volute geometry. Point 3 corresponds to both the diffuser outlet and volute inlet and point 4 and 5 respectively denote the cone inlet and outlet.

Once the diffuser analysis is done, point 4 is calculated by

$$p_{04} = p_{03} - (p_{03} - p_3)\sum_i \bar{\omega}_i$$

(3.61)

$$T_{04} = T_{03}$$

(3.62)

$\bar{\omega}_i$ denotes the different pressure loss coefficients that deviate the solution from the ideal conditions that would be $p_{04} = p_{03}$ since there is no work input. The loss models implemented by Aungier are explained as follows.

**Meridional velocity loss.** It accounts for the transformation of the inlet velocity into a “swirl” component along the volute. Therefore, meridional inlet velocity is assumed to be lost and this loss is modeled as ((Weber and Koronowski, 1986)).

$$\bar{\omega}_{m} = \left(\frac{C_{3m}}{C_3}\right)^2$$

(3.63)
**Tangential velocity loss.** Ideal conditions must verify conservation of angular momentum. If angular momentum decreases across the volute, a tangential velocity loss is implemented that is described as ((Weber and Korowski, 1986)).

\[
\bar{\omega}_t = \frac{r_3 C_{u3}^2}{r_4 C_{u4}^2} \left[ 1 - \frac{1}{SP^2} \right] \quad (SP \geq 1)
\]

\[
\bar{\omega}_t = \frac{r_3 C_{u3}^2}{r_4 C_{u4}^2} \left[ 1 - \frac{1}{SP} \right]^2 \quad (SP < 1)
\]  

being \(SP\) the sizing parameter, given by

\[
SP = \left( \frac{r_3 C_{u3}}{r_4 C_{u4}} \right)
\]  

**Skin friction loss.** It accounts for the losses caused by the frictional losses and it is described as ((Aungier, 2000))

\[
\omega_{sf} = 4c_f(C_4/C_3)^2L/d_h
\]

\[
L = \pi(r_5 + r_6)/2
\]

\[
d_h = \sqrt{4A_4/\pi}
\]

\[
\omega_{sf} = 4c_f(C_4/C_3)^2L/d_h
\]

\[
L = \pi(r_5 + r_6)/2
\]

\[
d_h = \sqrt{4A_4/\pi}
\]

\[
(3.66)
\]

\(L\) being the average flow path length. Finally, once point 4 is known, outlet cone conditions are computed by ((Aungier, 2000)).

\[
C_5A_5 = C_4A_4
\]

\[
p_{05} = p_{04} - (p_{03} - p_3)\omega_{EC}
\]

\[
T_{05} = T_{04}
\]

\[
(3.67)
\]

where \(\omega_{EC}\) is the exit cone loss described as follows (Aungier, 2000).
Exit cone loss.

\[ \omega_{ec} = \left[ \frac{(C4 - C5)}{C3} \right]^2 \]  

(3.68)

### 3.5 CFD losses.

The CFD simulations are run in STAR-CCM+ and they use steady-state RANS calculations, see Sundström et al. (2015). Table 3.2 shows the main properties of the numerical setup.

<table>
<thead>
<tr>
<th>Numerical setup</th>
</tr>
</thead>
<tbody>
<tr>
<td>Domains</td>
</tr>
<tr>
<td>Grid type, size</td>
</tr>
<tr>
<td>Walls</td>
</tr>
<tr>
<td>Turbulence</td>
</tr>
<tr>
<td>Rot.-stat. interf.</td>
</tr>
<tr>
<td>Solver</td>
</tr>
</tbody>
</table>
| Boundary cond.                      | Inlet: Mass flow  
|                                     | Outlet: Pressure |

#### 3.5.1 Impeller losses.

After several decades of research studies, impeller loss mechanisms are not fully understood yet. The complex flow involved and the wide range of different working conditions that may appear (from highly unsteady conditions as surge to supersonic conditions that leads to choking) make very hard to independently analyze each internal loss. For example if the flow is supersonic at the impeller inlet, shock waves may appear at the same place than the losses caused by the incidence angle and the well-known frictional loss. Therefore, to independently estimate the impeller losses from the CFD some assumptions need to be done. Besides, not all the implemented losses in the one-dimensional models have been investigated. The present study is focused on incidence, skin friction, choking, blade-loading and wake-mixing losses. Future works could go further and investigate other losses such as tip clearance losses or recirculation losses.

The main assumption of this study is to divide the impeller passage into four different parts and relate each loss to a single part. Though this assumption is clearly reasonable with some losses, it is not very accurate to others. For example incidence loss is related to the inlet part, where it must only appear and thus results may not diverge a lot from reality. On the other hand, even blade loading loss is supposed to be more significant after the throat due to the larger pressure gradients, it must also appear before the throat. However, all assumptions will be motivated.

Figure 3.5 (a) shows how impeller passage is divided into four different parts (A-D) limited by five different sections (i-v) where the flow properties are measured. Figure 3.5 (b) shows those sections in the CFD model. Though each part will be explained below in depth, Table 3.3 summarizes where each one is located and which losses are measured along it.

Harley (2013) measures the total losses along the impeller but as a sum, without distinguishing between each internal loss and external losses. However, this study follows a methodology similar to it, measuring just the inlet and outlet properties and then calculating the losses as the deviations from the ideal conditions. Each loss, either internal or external loss, is described as an entropy rise. Figure 3.6 shows how it...
is done. First, points \( i \) and \( o \) are fully determined from the CFD by computing the mass flow averages of total absolute pressure and temperature at the inlet and outlet planes. Point \( b \) denotes the conditions the flow would have if external losses were null. To locate it, Euler’s formula is applied. Mass flow average values of \( rwC_u \) are calculated at the inlet and outlet planes. The difference between both values is the total blades work (\( \Delta h_{ib} = h_b - h_i \)). Assuming that external losses are applied at a constant pressure as done in the one-dimensional models, the entropy rise associated to the external losses is obtained by the thermodynamic relation

\[
T ds = dh - vdp \tag{3.69}
\]

Since pressure is constant,

\[
\Delta s_{ext} = c_p \ln \left( \frac{T_o}{T_b} \right) \tag{3.70}
\]

On the other hand, the entropy rise between \( b \) and \( i \) is caused by the internal losses. Applying relation 3.69 from \( i \) to \( b \), it leads to

\[
\Delta s_{int} = c_p \ln \left( \frac{T_b}{T_i} \right) - R \ln \left( \frac{p_b}{p_i} \right) \tag{3.71}
\]

However, internal and external losses in centrifugal compressors and pumps are mainly described as adiabatic head (enthalpy) loss coefficients (\( \lambda_i = \frac{\Delta h_{ib}^2}{U_i^2} \)), where subscript 2 refers to the impeller tip.
Therefore, these entropy rises are converted into head loss coefficient by

$$\lambda_i = \frac{T_0^2 \Delta s_{\text{int}}}{U^2}$$  \hspace{1cm} (3.72)

Figure 3.6: h-s diagram along an impeller part.

Figure 3.7 shows the h-s diagram at the impeller tip. Since compressive effects are present in centrifugal compressor and head loss is not a state point parameter, it would be an error to directly calculate the enthalpy loss coefficient in each part (Aungier, 1995). However the entropy rise must remain unchanged along the impeller passage. Thus, working with entropy rises is more consistent than working with enthalpy losses. Each part will be deeper explained as follows, motivating why only some losses are measured in each part and describing how they are computed.

**Part A.** It starts at the impeller inlet and ends slightly before the throat. Though several internal losses could appear along this part (tip clearance, blade-loading, shock waves,...) just incidence and skin friction losses have been taken into account. One-dimensional models describes the incidence loss as the largest one. It is reasonable to consider it as the main loss here due to its guessed larger value and since this loss must only appear in this part, close to the inlet. Skin friction is also measured in part A because it is independently calculated by computing the skin friction factor ($c_f$) from the CFD and it is thus easily separable from the incidence loss.

To calculate the skin friction loss, it is applied the well-known equation to calculate the frictional enthalpy losses (Equation 3.73). The correction factor ($f_c$) provided by Aungier (Equation 3.6) is included in order to calculate the equivalent head loss at the impeller tip. Then, the equivalent entropy rise is obtained by Equation 3.74 that is based on the definition of head loss.

$$\Delta h_{0sf}^A = f_c (0.5 \rho_{ref} V_{ref}^2)(4c_f \frac{L_A}{D_h})$$  \hspace{1cm} (3.73)
\[ \Delta s_f^A = \Delta h_{0sf}^A / T_{02} \]  
where \( A \) denotes part \( A \), \( L_A \) is the blade length along this part and \( D_h \) is the hydraulic diameter that is estimated as the average between the inlet and the throat hydraulic diameters.

Once skin friction loss is known, the entropy rise caused by the incidence loss can be computed as follows.

\[ \Delta s_{\text{inc}} = \Delta s_{\text{int}}^A - \Delta s_{sf}^A \]  
where \( \Delta s_{\text{int}}^A \) is calculated by applying Equation 3.71 to this part.

**Part B.** It is located at the throat, being limited by two planes very close to each other (d=2mm). Due to the small size of this part, it is reasonable to consider that just choking losses (if choke exists) have a significant effect in it. However, external losses have been calculated too. \( \Delta s_{ch} \) is calculated by Equation 3.71 that leads to

\[ \Delta s_{ch} = c_p \ln \left( \frac{T_{0b}}{T_{0ii}} \right) - R \ln \left( \frac{p_{0b}}{p_{0ii}} \right) \]  
being \( ii \) the inlet plane of this part and \( b \) the flow conditions just applying the blade work as it was explained before (Figure 3.6).

**Part C.** It extends from the throat until the impeller tip. Skin friction and blade loading are the only internal losses measured in it. Skin friction is measured due to the same reason than in part A: it is easy to calculate independently. Why blade-loading is also related to this part instead of other internal losses that may appear, is motivated by:

- One-dimensional results point blade-loading as the second largest loss after incidence loss that has been already measured before.

---

**Figure 3.7:** h-s diagram between impeller inlet and outlet.
• It is reasonable to say that blade loading may mostly appear along this part since pressure gradients are higher and the diffusion process after the throat make the boundary layer to grow. Therefore, the two origins of blade-loading losses mostly happen after the throat while other losses may not be as concentrated in a single region as this loss.

How each loss is computed is similar than in Part A. First, the entropy rise caused by the external losses and frictional forces are computed and then, blade loading loss is obtained.

**Part D.** It is related to wake-mixing loss. It starts at the impeller tip, where the jet and wake are clearly identified. It ends where the flow can be considered mixed-out. Since it is not at the same point at every stage point, some approximation needs to be done. Jhonson and Dean (1966) investigate about the jet-wake flow. One of their conclusions was wake disappears in most of the cases at a radius 1.1 times larger than the impeller tip radius. Therefore, plane 5 is defined as a cylindrical section with a radius equal to $1.13 \cdot r_2$. Figure 3.8 are taken from the CFD and they show the entropy field at sections $iv$ and $v$ at a random stage point. It is seen how at plane 4 (impeller tip) wake is easily identified being an area of higher entropy while at plane 5 flow this entropy variation is much lesser. Once part D is fully defined, wake-mixing loss can be calculated. Since there are no blades along part D, work input is null and thus stagnation temperature must be constant. Applying Equation 3.69 to this conditions and assuming there are no external losses, entropy rise is calculated by

$$\Delta s_{mix} = -R \ln \left( \frac{p_v}{p_{iv}} \right)$$

where $iv$ and $v$ respectively denote the inlet and outlet planes of this part.

![Entropy field function.](a) Section $iv (r_2)$ (b) Section $v (1.13r_2)$

---

**3.5.2 Vaneless diffuser and volute losses.**

Both vaneless diffuser and volute are components where there is no work input. Therefore, the stagnation temperature is constant in them and entropy rise thus can be calculated by applying relation 3.69 that, under these conditions, leads to

$$\Delta s_i = -R \ln \left( \frac{p_o}{p_i} \right)$$

where subscripts $i$ and $o$ denote the inlet and outlet plane of each component.
Figure 3.9 shows the planes where properties have been measured. Sections 2 and 3 are the diffuser inlet and outlet sections 3 and 5 are the volute inlet and outlet. Plane 4 has been added in order to independently study the losses along the volute scroll and volute cone.

Figure 3.9: Vaneless diffuser and volute sections.
Chapter 4

ANALYSIS AND RESULTS.

It is seen in Figure 3.1 that for all the models it is needed the next input data: compressor geometry, inlet conditions and working conditions. The studied compressor geometry is shown in Table 2.1. The inlet conditions and the gas used in this analysis are summarized in Table 4.1. Finally, the working conditions are seven different speed lines and the mass flows within the range between the surge and the choke line at each tip speed.

Table 4.1: Inlet conditions and gas used in the simulations.

<table>
<thead>
<tr>
<th>Inlet Conditions</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Gas</td>
<td>Air</td>
</tr>
<tr>
<td>Gas model</td>
<td>Ideal</td>
</tr>
<tr>
<td>Total pressure</td>
<td>1bar</td>
</tr>
<tr>
<td>Total temperature</td>
<td>300K</td>
</tr>
</tbody>
</table>

4.1 Overall stage predictions.

To analyze and validate any theoretical model the results must be compared with experimental data. Even though the CFD model is already validated (Sundström et al., 2015), both the one-dimensional and the CFD model results are compared with the experimental compressor performance as follows. Though there are several parameters that analyse the overall performance such as static-to-static pressure ratio or polytropic efficiency, this study is focused on the total pressure ratio and the total-to-total adiabatic efficiency, that are given by

\[
PR_{i\rightarrow f} = \frac{P_{i5}}{P_{i1}}
\]

\[
\eta_{i\rightarrow f} = \frac{PR_{i\rightarrow f}^{(\gamma-1)/\gamma} - 1}{T_{i5}/T_{i1} - 1}
\]

1 denoting the impeller inlet and 5 the impeller outlet. Figure 4.1 shows both the total pressure ratio predicted by all the models and the experimental data. Tip speeds are normalized with the third lowest speed (\(N_0\)) and mass flow are normalized with the choke mass flow at the highest tip speed.
Figure 4.1: Total pressure ratio.

It can be seen the high accuracy of the CFD model. The lowest speed lines show negligible differences when comparing to the experimental curves. When increasing the angular speed, some deviation appears, mainly at the medium high flows of each speed line.

On the other hand, it is seen the potential of the one-dimensional models. Though the results are slightly less accurate than the CFD results, one-dimensional prediction is also reliable, estimating the total pressure ratio with negligible deviations. To better analyze the accuracy of one-dimensional models, Figure 4.2 shows the relative error of the total pressure ratio between the models and the experimental measurements. Relative error is calculated by

$$
\varepsilon_{\text{TPR}} = \frac{TPR_{\text{Model}} - TPR_{\text{Experimental}}}{TPR_{\text{Experimental}} \cdot 100}
$$

(4.3)

Most points are within the range of $\varepsilon_{\text{TPR}} = \pm 5\%$ that verifies the reliability of the one-dimensional analysis implemented. Besides, some characteristics and trends are shown.
• At the lowest speed lines (0.62N₀ – N₀) both set models results are very similar. An underestimation (\( \varepsilon_{TPR} < 0 \)) of the total pressure ratio is shown on those speed lines. Moreover, it can be seen how this deviation increases towards the surge and the choke side of the map, reaching the highest deviation (\( \sim -15\% \)) next to the choke line. A minimum close to the design points is shown, reaching error values of \( -3\% \).

• The predictions are reached at 1.14N₀ and 1.24N₀. Deviations are mostly minor than 5%. Underestimations still predominate.

• At the highest speeds (1.34N₀ and 1.41N₀), high accuracy is still reached towards the surge line. However, prediction becomes worse towards the choke side of the map. It could be caused by a high underestimation of choking losses that leads to higher total pressure ratios. While Aungier provides a model for this losses, Oh et al.’s set does not even model it. Therefore, it is reasonable the even worse prediction of Oh et al.’s set. Besides, at the highest speed (1.41N₀) both models overestimate (\( \varepsilon_{TPR} > 0\% \)) the pressure ratio along the whole line.

Total-to-total adiabatic efficiency is shown in Figure 4.3. The relative error of all the theoretical models are plotted too. CFD model results show almost a perfect match again. Only at the lowest speed line, deviations greater than 10% (red dashed lines) are reached. In general, it appears a slight overestimation that is caused by the assumption of adiabatic walls introduced in the CFD model (Sundström et al., 2015)

One-dimensional predictions show larger deviation than the CFD model again. Both models still have similar results and an underestimation predominates. At the lowest speed lines (0.62N₀ – N₀), the efficiency prediction worsens towards the choke line and error values of 40 – 50% are reached. On the other hand, results are overestimated at high speed lines and close to the choke line. It could be caused by a wrong choking loss prediction again.

The correct estimation of all the models show the reliability of these theoretical models. However, there are some deviation trends that can be investigated. In the next sections, several internal and external loss is investigated and the reason of some of the weakest predictions are explained.
Figure 4.3: Total-to-total adiabatic efficiency. Red dashed lines = ±10%
4.2 Impeller performance and losses prediction.

4.2.1 Blades work.

The imparted work from the impeller to the fluid determines the compressor performance and limits the achievable total pressure ratio. Therefore, it is one of the main characteristics that must be investigated in order to understand the compressor performance. Hence, theoretical models must correctly predict it in order to achieve accurate predictions.

It is reminded that dimensionless blades work coefficient is theoretically described by Euler’s formula as

\[ I_B = \frac{U_2 C_{u2} - U_1 C_{u1}}{U_2^2} \]  \hspace{1cm} (4.4)

and taking into account the slip factor (\(\sigma\)) and the tip distortion factor (\(\lambda_{dis}\)), one-dimensional models describe it as

\[ I_B = \sigma (1 - \lambda_{dis} \Phi_2 \cot \beta_2 b) - \frac{U_1 C_{u1}}{U_2} \] \hspace{1cm} (4.5)

Figure 4.4 shows the results of \(I_B\) in all the models.

![Figure 4.4: Dimensionless blades work coefficient prediction.](image)

It is seen how both one-dimensional losses set yield almost equal results as it was expected since total pressure ratios estimated are very similar (Fig. 4.1). However, some differences between CFD and one-dimensional results are evident. Slip factor estimation might be a source of deviations. Wiesner’s (1967) slip factor is used in the one-dimensional models. It is constant since it only depends on geometry parameters. Figure 4.5 shows this estimated slip factor and the value from the CFD model. Only the results of some speed lines are plotted to make it more readable. At the speeds that are not plotted, slip factor takes similar values and trends.
It can be seen how CFD slip factor takes values from 0.82 to 0.97. Therefore, one-dimensional models are overestimating the slip effect (\(\sigma = 0.81\)). The same Wiesner model overestimation is shown at Schneider et al. (2015). At the lowest speed lines (0.62\(N_0\), 0.84\(N_0\) and \(N_0\)), the \(I_B\) differences between CFD and one-dimensional models are mainly caused by this slip factor variations. Towards the surge line, slip factor difference is largest and thus \(I_B\) difference too. There is a minimum difference of \(\sigma\) next to the design point, being at that point minimum the variation of \(I_B\) between CFD and one-dimensional models. Towards the choke line, slip factor slightly grows and so does \(I_B\).

![Figure 4.5: Slip factor.](image)

However, at the highest speed lines, results are very different. Although CFD slip factor increases towards the choke side of the map, showing a sharp growth close to the choke line and Wiesner model still overestimates the slip effect, \(I_B\) is now larger in the one-dimensional analysis than in the CFD model.

### 4.2.2 Internal losses.

Several internal losses are studied in each theoretical model. Similarly to Table 3.1, Table 4.2 summarizes the internal losses predicted in each model. Though just five losses are studied in the CFD model, all the other losses should be included in those five in a certain way due to the calculation procedure followed.

<table>
<thead>
<tr>
<th>Incidence</th>
<th>Aungier’s set</th>
<th>Oh et al.’s set</th>
<th>CFD model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Skin friction</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Blade-loading</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Wake-mixing</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Clearance</td>
<td>x</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Hub-to-shroud</td>
<td>x</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Choking</td>
<td>x</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Supercritical Mach</td>
<td>x</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Distortion</td>
<td>x</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Figure 4.6 shows the sum of all the internal losses in each model. Overall, a good match between all the models is achieved. Specially, between CFD and Aungier model since it can be seen that losses are within the same range and they follow the same trend. In those models, along each speed line internal losses increase toward the surge and choke line and a minimum appears close to the design point as expected due to its higher efficiency. Besides, the increment close to the choke line is higher at the highest speed lines. It is explainable by the fact that choking losses are more significant when the impeller works at higher Mach rotational numbers. On the other hand, though Oh et al. set yields not very different values, it underestimates the losses in relation to the other models. Besides, the increment towards the choke side of the map does not appear since choking losses are not implemented in this set.

Moreover, taking the CFD as the reference model due to its better overall predictions (Fig. 4.1 and 4.3), Aungier set slightly overestimates the internal losses, mainly at the highest speed lines. Each internal loss is independently investigated as follows.

![Figure 4.6: Internal loss coefficient.](image)

**Incidence loss.** Figure 4.7 shows the incidence loss coefficient in each model. A good match between all the model is again shown, mainly between Aungier and CFD models. Oh et al. gives less weight to incidence loss leading to the lower loss values.

Trend is similar for all the models: decreasing towards the choke side of the map. Since incidence losses are caused by the flow adjustment to the inlet blade angle, they must follow the same trend than the absolute difference between the inlet flow angle ($\beta_1$) and the blade angle ($\beta_{ib}$). This difference is called the incidence angle. $\beta_1$ mainly depends on the inlet conditions, inlet geometry and impeller speed.
Therefore, it is expected to take similar values for all the models. Effectively, Figure 4.8 shows $\beta_1$ and $\beta_{1b}$. It can be seen how $\beta_{1b} - \beta_1$ decreases towards the choke line motivating the incidence loss trend. Oh et al. and Aungier models are shown together in one line since $\beta_1$ takes exactly the same values.

Moreover, CFD incidence loss shows larger values. As it was explained in Chapter 3, other losses that may appear in impeller part A could be overestimating incidence losses. Besides, CFD incidence loss coefficients show a slight stabilization close to the choke line. At the highest speed lines, that stabilization appears but there is a drop after it. Incidence losses should not stabilize until incidence angle was zero, when losses must be null since there is no flow adjustment. Figure 4.8 shows how incidence angle is not close to zero at any operation point. Therefore, the stabilization must be caused by the growth of some other loss at this side of the map. Due to the fact that this effect happens close to the choke line, choking losses are probably the cause. Shock waves does not have to appear at the same point at every operation point as it happens in a converging-diverging nozzle (Korpela, 2011). CFD throat region was located by analyzing mainly the highest speed and highest mass flow points where shock waves are strongest. Therefore, it is reasonable that at lower speeds and even lower mass flows at a constant speed, shock waves may start before than what it is considered as throat area and thus the losses produced by them are included in the incidence loss coefficient, being the reason of the shown stabilization.
**Skin friction loss.** Figure 4.9 shows the skin friction loss coefficient for all the models. One-dimensional analysis defines this loss as a function of the relative velocity squared. It influences the trend shown in the curves. Since relative velocity increases when the mass flow grows, 1D loss coefficients quadratically increase from the surge line to the choke side. The weighting of this loss is larger in Aungier set than in Oh et al. model.

Besides, values between both one-dimensional model results are reached in the CFD model. However, at each speed line, CFD shows a minimum near the design point that it is not estimated in Aungier and Oh et al. models. Although CFD loss are not exact predictions since the skin friction coefficient ($c_f$) was assumed to be constant and equal to its average along all the impeller walls, it is understandable to say that this is a weakness of the one-dimensional predictions.
Since the CFD minimum comes from a minimum of $c_f$, one-dimensional results could be refined by improving the $c_f$ estimation.

**Blade loading loss.** The worst agreement is found in the blade loading losses. Figure 4.10 shows the blade-loading loss coefficient for all the models. Aungier curve is the sum of the blade-loading and the hub-to-shroud loss. Since the cause of both losses are similar, it is understandable to sum them. Besides, hub-to-shroud losses should be included in the CFD blade-loading losses as well, since it includes all the losses that may appear after the throat except the skin friction loss. Oh et al. set does not model the losses related to the pressure gradients in the hub-to-shroud direction.

![Figure 4.10: Blade loading loss coefficient.](image)

Between the one-dimensional predictions, it is seen a large difference. Oh et al. losses monotonically decrease from the surge line towards the choke line. Aungier losses show a decrement from the surge line to a point close to the design point where a minimum is found. Then, losses slightly increase towards the choke line. Besides, Aungier values are always higher than Oh et al. predictions. The reason of this difference is just the addition of the hub-to-shroud losses in the Aungier results. Figure 4.11 shows both one-dimensional blade loading losses without taking into account the hub-to-shroud losses. It can be seen how both models would have the same trend, though it is shown that Aungier predicted higher losses than Oh et al set. The decreasing trend towards the choke line is caused since the diffusion process after the throat increases when decreasing the mass flow. This leads to higher deceleration of the flow that causes larger momentum losses since the boundary layer along the blades grows. The reason of this thicker boundary layer is the higher adverse pressure gradient (Botha and Moolman, 2005).
On the other hand, CFD losses (Figure 4.10) show a very irregular trend. Besides, even negative values are reached at the lowest tip speeds close to the surge line. These unrealistic values may be caused by back flow that alters the values. Later (Figure 4.21) it will be motivated the presence of this back flow. At those low speeds, the loss at the design point is similar than the value predicted by the Oh et al. model. However, the trend is the opposite: increasing from the surge line to the choke side of the map. At the highest speeds the trend is different. From the surge line until close to the design point, losses slightly decrease. Then, a minimum is reached and finally losses sharply increase towards the choke line. This trend is similar than Aungier prediction but with a much higher raise close to the choke line. Therefore some conclusions arise from all these facts.

- At the lowest speed, CFD losses do not appear to be realistic. Therefore, the trend at the highest tip speeds may be considered the reference. However, the sharp increment close to the choke line may be unrealistic as well. It could be produced because some shock waves may appear at the suction surface and produce boundary layer separations. This is another source of losses and Aungier models it separately.

- Focusing on those regimes, Aungier estimates a similar trend but values are overestimated except at the highest mass flow points where shock waves appear to be altering CFD values.

- Since Oh et al. gives very weak predictions, it is seen the importance of modeling the hub-to-shroud losses as well.
**Mixing loss.** The jet-wake flow profile at the impeller tip leads to losses due to the later mixing process. Figure 4.12 shows the mixing loss coefficients predicted by the three models.

![Figure 4.12: Mixing loss coefficient.](image)

Oh et al. model yields the lowest results for this loss. Aungier gives more weight to mixing losses than Oh et al. but the trend it is the same: decreasing from the surge line towards the choke side of the map. On the other hand, though CFD results are similar than Aungier’s towards the surge side of the map, they present a minimum close to the design point. Then, CFD losses increase towards the choke line but without reaching as high values as in the surge side. This trend is explainable by taking into account the next facts:

- Assuming constant wake area ratio ($\varepsilon$), the higher mass flow, the larger wake-mixing losses are (Botha and Moolman, 2005).
- However, it is shown in compressors that the wake area ratio decreases when increasing the mass flow (Johnston and Dean, 1966)

To further study this loss, wake area ratio is measured at the impeller tip from the CFD. It is done by studying the entropy at the impeller tip plane. Wake area must be a high entropy region while jet should show lower values. Figure 4.13 (similar than Figure 3.8) shows this entropy variation from wake to jet. Following the work done by Schneider et al. (2015), wake area ratio is calculated from the CFD by assigning to the wake area the cells where $s > \bar{s}$, being $\bar{s}$ the mass flow entropy average at the plane. Due to the large entropy gradients between wake and jet, this method becomes not very sensitive to variation...
of $\tilde{\sigma}$ and, therefore, it can be considered as valid enough to estimate wake and jet areas (Schneider et al., 2015). Figure 4.14 shows the wake area ratio obtained from the CFD.

Figure 4.13: Entropy field at impeller tip. Negative values are reached since entropy is calculated by comparing to a reference point.

It can be seen a large drop from the surge line towards the design point. However, towards the choke side, the wake is stabilized and it slightly decreases or it even marginally increases. This wake stabilization is caused by blockage from tip clearances flow and separated boundary layers that are more relevant close to choking conditions (van den Braembussche, 2013). Schneider et al. (2015) already included this minimum wake area in his two-zone model.

Therefore, this explains the CFD wake loss trend. Towards the surge line, the losses increases due to the wake growth is more significant than the mass flow decrement. Towards the choke side of the map, losses grows again since the wake area is almost constant but the mass flow is increasing.

At the lowest speed lines, Oh et al. loss shows a minimum next to the design point too. However, at high speed lines it does not appear and the overall values are very low comparing to the other models. Figure 4.14 shows the wake ratio estimation in this model. It can be seen how they are very low comparing to the CFD results. Therefore, a better estimation of the wake area ratio from two-zone modelling studies or even taking the CFD values could be a way to improve this loss. On the other hand, Aungier loss does
not depend on the wake area ratio and that explains why it correctly estimates this loss towards the surge line comparing to the CFD results but not towards the choke side of the map. Thus, it could be a correct refinement of this loss to add some dependency of the wake area.

Choking loss. Figure 4.15 shows the losses coefficient related to the shock waves that appear when some portion of the flow reaches supersonic conditions at the throat or close to it. Oh et al. does not model this loss.

It can be seen that Aungier predicts only shock losses at the highest tip speeds whereas CFD results show losses at all the tip speeds towards the choke line. At $1.24N_0$, Aungier reaches values similar to the CFD results. However, at $1.32N_0$ and $1.41N_0$ Aungier predicts lower losses even though the tip speed is larger. It does not appear to be very realistic, since the faster tip speed the larger relative Mach number and thus higher choking losses may be expected.

On the other hand, at the highest tip speeds CFD choking loss increase towards the choke line. However, at lower speeds they present a minimum before the highest mass flow point. It is not expected and it may be caused because not all the shock is included in the region taken as the throat, as it was explained before (Incidence losses).

![Figure 4.15: Choking loss coefficient.](image-url)
**Clearance loss.** Flow leakage through the clearance between the blades and the shroud causes a loss that is estimated in both one-dimensional models. Figure 4.16 shows the clearance loss coefficients. It can be seen the largest importance that Oh et al. gives to this loss comparing to Aungier. However, the trend is equal in both models. At the highest tip speeds, loss increases when increasing the mass flow. At lower speeds a maximum appears before the choke line. This maximum moves towards the surge line when reducing the tip speed. This trend is explained by Botha and Moolman (2005) as follows.

- Clearance loss increases when the difference between the main flow velocity and the leakage flow velocity grows. Since the larger mass flow, the larger main velocity is, clearance loss tends to increase when mass flow grows.
- Besides, clearance loss also increases when the pressure difference over the blades is larger.

First point explains why the larger mass flow, the larger clearance losses are as it is shown at the highest tip speeds. However, after the design point, when increasing the mass flow, pressure ratio decreases. Taking into account the second point, towards the choke line leakage should decrease when increasing the mass flow. Therefore, when the pressure difference variation is more significant than the increment in the velocity difference, a maximum appears as it shown at lower tip speeds.

![Figure 4.16: Tip clearance loss coefficient.](image)

**Shock losses.** Shock waves may appear in the blade suction at high rotating Mach number and high mass flows. These shocks produce losses and boundary layer separations (Aungier, 1995). These losses are not studied neither in the CFD model nor in the Oh et al. set. Figure 4.17 shows the results given by the Aungier model.
It can be seen that shock losses raises when increasing the tip speed and the mass flow as it was expected. However, it appears that this loss might be overestimated in our compressor. It is in the same order of magnitude of some other relevant losses such as tip clearance and skin friction losses whereas other studies such as Harley et al. (2013) and Schneider et al. (2015) describe this loss as negligible comparing to the others.

Distortion loss. Only in Aungier model it is studied the loss related to the distorted flow and the mainstream mixing at the impeller tip. Figure 4.18 shows the predicted loss coefficient. Losses increase towards the choke side of the map. A quadratically trend is shown due to the directly relation between the flow distortion and the skin friction (Aungier, 1995). However, the reached values are negligible, being 10 or 100 times lesser than the other losses.
4.2.3 External losses.

Different types of parasitic works appear in centrifugal compressor. These external losses increase the work input but without raising the flow pressure. As it was explained in Chapter 3, one-dimensional models study three different types of parasitic works: leakage, disk friction and recirculation. On the other hand, external losses are taken from the CFD as a whole.

Figure 4.19 shows the total external losses in all the models and Figure 4.20 shows each external loss estimated by one-dimensional models.

Both one-dimensional models yield similar predictions and their curves are overlapped in Figures 4.19 and 4.20. However, they do not match the CFD results. Towards the surge side of the map and specially at low tip speeds, one-dimensional analyses highly underestimate the losses. At those operation points, it is expected that recirculation losses predominate over the others. Figure 4.21 shows the back flow measured in the CFD at the impeller inlet. It is calculated as a percentage by dividing by the total mass flow that goes downstream.
It can be seen that the relation between this back flow and the external losses. Therefore, the largest difference between one-dimensional models and CFD appears to be caused by the high underestimation of the recirculation losses in our compressor.

On the other hand, towards the choke side of the map one-dimensional external losses are higher than in the CFD. It may be caused by an overestimation of the leakage loss since it is by far the most relevant external loss in one-dimensional models (Figure 4.20).
4.3 Vaneless diffuser performance and losses prediction.

Vaneless diffuser performance is analyzed by its pressure recovery capacity and the losses that appear along it. Both values are obviously related. The larger the losses, the less pressure recovery reached. Figures 4.22 show the losses for all the models.

![Figure 4.22: Vaneless diffuser loss coefficient.](image)

Both one-dimensional models give very similar results since they use the same approach. Losses increase towards the surge side of the map. This is caused by a longer path in the diffuser that leads to higher friction losses (Kerres et al., 2016). CFD losses also show a raise towards the surge line. However, the values are much lower than in the one-dimensional models. This is the main reason for the lower TPR prediction (Figure 4.1) of the one-dimensional models in these operation points. On the other hand, CFD losses show a slight raise towards the choke line. At low tip speeds, CFD losses reach higher values than one-dimensional models in this regimes. However, at high tip speeds, one-dimensional models give higher losses also close to the choke line.

Therefore, it is seen how an improvement of the vaneless diffuser model would be necessary to enhance the predicted overall compressor performance by the one-dimensional models, mainly close to the surge line.
4.4 Volute performance and losses prediction.

As it was explained in Chapter 3, volute is divided into two parts: volute funnel and volute cone. Figure 4.23 and 4.24 show the losses of both parts in all the models. Both one-dimensional models use the same volute funnel and cone model. Hence, both results are very similar and their curves are almost overlapped in all the figures.

![Volute funnel recovery coefficient](image)

Figure 4.23: Volute funnel recovery coefficient.

Volute cone losses are correctly predicted by the one-dimensional models (same trend and very similar values in overall) whereas funnel losses are highly overestimated. To better explain these differences, Figure 4.25 show all the estimated losses along the volute funnel model. Pressure loss coefficient are shown instead of $\lambda$ coefficients. It is reminded that pressure loss coefficients are described as

$$\bar{\bar{w}}_i = \frac{\Delta P_i}{p_{03} - p_3}$$

Skin friction appears to be negligible comparing to the other losses. CFD funnel loss shows a similar trend than tangential velocity loss coefficient. It is shown a minimum that coincide with the "optimal volute sizing point", where the angular momentum is better conserved. Towards the surge line, CFD losses increase due to diffusion losses and towards the choke line they also raises owing to the meridional velocity loss (Kerres et al, 2016). However, even though one-dimensional losses also increase towards the choke line, their increment is much higher, showing the largest differences at those regimes. Therefore, taking into account Figure 4.25 again, the overestimation of the meridional velocity loss appears to be the reason of the weak volute funnel prediction.
CHAPTER 4. ANALYSIS AND RESULTS.

Figure 4.24: Volute cone loss coefficient.

(a) Meridional velocity loss coefficient.

(b) Tangential velocity loss coefficient.

(c) Skin friction loss coefficient.

Figure 4.25: Pressure loss coefficients in 1D volute funnel models.
4.5 **Losses breakdown.**

A common way to further study all the losses is by breaking down them. Figures 4.26, 4.27 and 4.28 show the losses breakdown for the lowest tip speed ($U_2 = 0.62N_0$), the medium speed ($U_2 = 1.14N_0$) and the highest one ($U_2 = 1.41N_0$). They summarize all the losses studied in the previous sections. However, these graphs give a better understanding of which losses are more relevant at each regime. Besides, it is also seen the difference between the total work input and the final useful work.

The white area below the coloured areas denotes the useful work, i.e. the work that is fully used to increase the fluid pressure. Then, each coloured area shows the loss produced by each different source. Finally, the upper line (above External Losses in the CFD graphs or Leakage in the one-dimensional models) denotes the total work consumed by the compressor. Moreover, the blades work is also visible as the area below the last external loss. The losses that are negligible at each model, such as recirculation or distortion losses, are not discernible and therefore they are not pointed.

Even though all the losses have been already explained, some conclusion are extracted from these plots.

- Overall, most relevant losses are the external, incidence, vaneless diffuser and volute losses. Besides, largest differences between CFD and one-dimensional models are found in the external losses, vaneless diffuser and volute.

- At low tip speeds, external losses show a sharp increase towards the surge large due to the back flow. This increment is not predicted by the one-dimensional models. This is the reason why at low tip speeds and close to the surge line the work input calculated by the CFD is much larger than the predicted by the one-dimensional models. However, efficiency is similar at those regimes (Figure 4.3). This is caused by a lower useful work in the one-dimensional models. The high overestimation of the vaneless diffuser loss is the main reason of it.

At higher tip speeds, external losses and work input become more similar in all the models, being even higher in the one-dimensional models at the highest tip speed.

- Incidence loss is the largest impeller loss. As it was seen in Figure 4.7, it follows the same trend in all the models but being higher in the CFD. The weight of this loss slightly decreases when working at higher tip speeds.

- The total pressure ratio reached is defined by the area below the last impeller loss. This is in agreement with Figure 4.1. At low tip speeds, one-dimensional models underestimate this area giving lower total pressure ratios but at the highest tip speed it is the opposite.

- Largest differences are found in the vaneless diffuser. Close to the surge large, it is the highest loss predicted by the one-dimensional losses at all the tip speeds. However, it is not the most relevant loss in the CFD model. It is even almost negligible at the highest tip speeds.

- Volute losses are in big disagreement between the models. Towards the surge line, it is similar in all the models at every speed. However, when increasing the mass flow at low tip speeds, one-dimensional models predict a sharply increment that leads to become this loss in the most relevant. CFD also shows a loss increment but much lower. At higher tip speeds, this increment is not as sharp as before and volute losses are more similar between all the models.
CHAPTER 4. ANALYSIS AND RESULTS.

Figure 4.26: Losses breakdown ($U_2 = 0.62N_0$).

Figure 4.27: Losses breakdown ($U_2 = 1.14N_0$).

Figure 4.28: Losses breakdown ($U_2 = 1.41N_0$).
Chapter 5

CONCLUSIONS AND FUTURE WORK.

5.1 Conclusions.

Centrifugal compressors are still a engineering challenge due to the complexity of the flow and the phenomena involved. Numerical solutions carried out in CFD software give very accurate performance predictions, at least in a very wide range of regimes. However, this accuracy requires very complex models, long computation times and expensive hardware. In this thesis, it has been seen the potential of the much simpler analytical approaches to predict a turbocharger compressor performance. Specifically, two different one-dimensional models have been successfully implemented. They are based on the work done by Aungier (2000) and Oh et al. (1997). Needing much less geometric input data and much shorter computational times (∼ 60 seconds to calculate 77 operation points running in a 4 cores processor) than CFD calculations, they correctly estimate the compressor performance prediction:

- Comparing with the compressor experimental measurements, the implemented one dimensional models estimate the total pressure ratio with an absolute relative error less than 5% in 49 and 41 operation points respectively of a total of 77 calculated regimes. Expanding the range, 69 and 61 points are predicted with an absolute relative error less than 10%. The weakest total pressure ratio estimations are shown at the highest speed lines and close to the choke line, reaching a maximum relative error of ∼ 25% in both models.

- Regarding the total-to-total adiabatic efficiency, Aungier model predicts 42 and 60 points with an absolute relative error less than 5% and 10% respectively, whereas Oh et al. model estimates 40 and 55 points within those error ranges. The highest errors are reached at low tip speeds and towards the choke line, taking a maximum value of 60%.

Hereafter, to better understand the advantages and weaknesses of our one-dimensional analysis, the results were compared to numerical RANS simulations of the same compressor. Several losses were measured from the CFD by their entropy raise. The CFD impeller region was divided into four different parts to measure the incidence, choking, blade-loading, mixing and blade-loading losses. The vaneless diffuser and volute losses were directly calculated measuring the entropy rise between the inlet and the outlet of each component. Some conclusions are taken from this analysis.

- Overall, the sum of the impeller internal losses is well predicted, specially by the Aungier model. It gives the same trend along the whole map but a slight losses overestimation is shown. Within the internal losses measured in the CFD, incidence losses appear to be the most significant losses and, at the same time, they are the best predicted losses by the one-dimensional analysis. Skin friction is also correctly estimated by the models. On the other hand, blade-loading losses shows a big disagreement between CFD and one-dimensional analysis. Choking and mixing losses do not
disagree as much as blade-loading losses do, but they differ either in trend or in magnitude at some regimes.

It is reminded that not all the impeller internal losses that are modeled in the one-dimensional analysis were measured in the CFD.

- External losses match at high tip speeds. However, at low tip speed and close to the surge line CFD losses sharply increase that appears to be related to the back flow. This increment is not predicted by the one-dimensional models, giving an almost constant value along each speed line.

- The stationary components (vaneless diffuser and volute) show the largest differences between the one-dimensional analysis and CFD results. Vaneless diffuser is highly overestimated, specially close to the choke line at every regime. On the other hand, though the volute cone losses are very similar between CFD and one-dimensional models, the losses along the volute funnel show a large difference at low speeds and close to the choke line. It appears to be caused by a wrong prediction of the meridional losses. Therefore, both components have a lot of influence in the weakest overall predictions.

Seeing these results, one-dimensional model can be considered a helpful and useful tool to investigate and design centrifugal compressor, specially in the early design stages. Low total pressure ratio and efficiency errors can be expected. Thereafter, CFD models allow to further understand all the phenomena, better predict the performance and test different three-dimensional geometries.

Finally, it has been seen how to compare “loss by loss” CFD results with one-dimensional models gives a further knowledge of the one-dimensional analysis. Even though CFD losses are very hard to be independently measured and some improvements are needed, it is a good methodology to detect the weaknesses of one-dimensional losses correlations and improve them without needing expensive experimental tests.

5.2 Future work.

Several future work could be carried out to go further in this kind of investigation. First, regarding the one-dimensional analysis, not all the components that can appear in a centrifugal compressor have been implemented (e.g. vaned diffusers or return channels). Aungier (2000) provides similar models than the ones implemented in this thesis to some of those components. To take advantage of the high flexibility and modularity that characterises these kind of analysis, all the components should be modeled. Then, a wider range of compressor could be analysed with the same code, adding all the components that appear in the real compressor setup and connecting them in the correct order.

Besides, to even cover a wider range of centrifugal compressors, it would be a good practice to implement losses set that yields well predictions when reaching total pressure ratios higher than ~4.2. Both Aungier and Oh et al. losses set are more focused on turbocharger compressor, where lower pressure ratios are reached.

The losses comparison between numerical solution and one-dimensional models yields more possible future work. On one hand, it has been seen how the CFD impeller internal losses are measured in a way that still shows some weaknesses. Therefore, some improvements could be carried out in that field such as to measure more losses (e.g. tip clearance or shock losses) or to improve the measurement of some losses that have been already investigated (e.g. a better location of the throat area).

On the other hand, the numerical and analytical comparison has also shown some one-dimensional prediction weaknesses. Specially, in the stationary components performance. The CFD losses of those
components can be considered trustworthy since no assumptions were done and the compressor performance is predicted with high accuracy at almost all the speed lines. Taking this results as the reference, it is seen a high deviation of the one-dimensional results. Therefore, future work could improve those stationary component models. Several analytical models can be found in the literature e.g. the volute model carried out by van den Braembussche et al. (1999). Finally, some weaknesses in the impeller internal losses correlations have been revealed such as a slip factor overestimation and the need of a better mixing loss model that depends on the wake ratio. Therefore, future work can also be focused on enhance the impeller internal losses implemented.
Appendix A

RESUMEN EN ESPAÑOL DEL TRABAJO.

A.1 Introducción y objetivos.

A.1.1 Introducción y marco de referencia.
Desde hace varios años, la tendencia de downsizing ha adquirido una gran relevancia en el campo de los motores de combustión interna alternativos. El efecto más claro de esta tendencia es ver cómo los motores de seis cilindros están reemplazando a los de ocho, los de seis a su vez están siendo sustituidos por los de cuatro y, muy recientemente, los motores de tres cilíndricos están desbancando a los de cuatro (Squatriglia, 2016).

El objetivo de esta tendencia es reducir la cilindrada del motor pero manteniendo la misma su potencia, o incluso aumentándola. Esta reducción de tamaño va acompañada de un aumento de eficiencia debido, entre otros factores, a las menores pérdidas térmicas y de fricción. Con este aumento de eficiencia, el motor reduce sus emisiones contaminantes, un factor de crucial importancia para mejorar la sostenibilidad y poder cumplir con las presentes normativas que establecen los máximos de contaminación posibles de cualquier motor comercial. Entre las tecnologías más implantadas para cumplir con el downsizing se encuentran la sobrealimentación del motor, la inyección directa o la distribución de válvulas variable.

El presente estudio entra en el campo de la sobrealimentación. La sobrealimentación consiste en introducir más masa de aire en el cilindro durante la etapa de admisión, y con ello aumentar la potencia a través de la combustión de más aire y combustible. Para aumentar la masa de aire, la sobrealimentación se basa en aumentar su densidad de entrada. Ese es el objetivo del turbocompresor.

El turbocompresor está formado por una turbina radial y un compresor centrífugo que giran en torno a un eje común. En la Figura A.1 se pueden ver estos componentes. Para conseguir aumentar la densidad del aire de entrada al motor, los gases de escape son conducidos a la entrada de la turbina. La turbina es impulsada por estos gases gracias a la gran energía, principalmente térmica, que aún tienen. Por tanto, la turbina transmite la potencia al eje común. El rotor del compresor recoge dicha potencia y la utiliza para comprimir el aire atmosférico. A la salida del compresor, el aire posee mayor presión y mayor densidad. A través de la canalización necesaria, este aire presurizado es conducido a la entrada del cilindro, consiguiendo el objetivo deseado.

El presente trabajo se centra en el estudio analítico del compresor, por ello sólo este componente es analizado a continuación. El compresor centrífugo está compuesto por tres componentes principales: el rotor, el difusor y la voluta. En la Figura A.2 se muestra un esquema de su geometría. El rotor, impulsado por el eje, es la única parte móvil del compresor y está compuesto por una serie de álabes. El aire es conducido a través de ellos, aumentando su presión gracias al trabajo que éstos imparten. Al
salir del rotor, el aire fluye a través del difusor, donde aumenta más aún su presión gracias a un proceso de difusión. El difusor puede tener álabe o no. Finalmente, el aire cruza la voluta, aumentando más su presión. Además, la voluta dirige al aire para que tenga una dirección de salida compatible con el elemento al que esté conectado a continuación.

Durante el diseño de un compresor centrífugo, es importante la obtención de modelos matemáticos que puedan predecir su funcionamiento, y así poder reducir el número necesario de costosos prototipos y largos ensayos. A día de hoy, mediante soluciones numéricas realizadas a través de modelos de Computational Fluid Dynamics (CFD) se pueden obtener precisas simulaciones del compresor centrífugo. Sin embargo, debido a su gran complejidad, estos modelos requieren largos tiempos de computación y costoso hardware. Por ello, gracias a la investigación realizada durante las últimas décadas, se han desarrollado diferentes modelos unidimensionales (1D) que predicen todo el mapa de funcionamiento del compresor en tiempos muy cortos. Además, estos modelos 1D requieren relativamente pocos datos acerca de la geometría del compresor. Todo ello los convierte en una potente y útil herramienta durante el proceso de diseño de compresores centrífugos, principalmente en sus primeras fases.

Varios autores (Harley et al. (2013), Schneider et al. (2015) y Kerres et al. (2016)), clasifican los modelos 1D en tres categorías dependiendo de las zonas de flujo estudiadas: a) modelos cero onas, b) modelos de una zona y c) modelos de dos zonas.
APPENDIX A. RESUMEN EN ESPAÑOL DEL TRABAJO.

Los modelos cero zonas sólo requieren los coeficientes adimensionales de flujo, trabajo y el número Mach del álabe en el punto de diseño. Estos coeficientes se comparan con los de otro compresor del cual todo su mapa de funcionamiento sea conocido. Bajo la hipótesis de que los compresores fabricados para el mismo objetivo tienen diseño y rendimiento similar, obtenen el mapa adimensional del compresor mediante la extrapolación del resto de puntos a partir del mapa conocido del compresor de referencia y la proporción entre sus coeficientes en el punto de diseño. El modelo realizado por Casey and Fesich (2010) es un ejemplo de este tipo de análisis.

Los modelos de una zona alcanzan un mayor nivel de detalle. Asumen el flujo uniforme en cada sección del compresor. Bajo esta hipótesis, se aplican las ecuaciones de conservación de masa, momento y energía al flujo a través de cada componente del compresor. Además, para obtener las condiciones reales, se aplican diferentes correlaciones empíricas de pérdidas. Ejemplos de estos modelos y diferentes conjuntos de correlaciones se pueden encontrar en los trabajos de Galvas (1974), Oh et al. (1997) o (Aungier, 2000).

Finalmente, los modelos de dos zonas se centran en el perfil de flujo central y estela característico de la salida del rotor. El flujo central, situado junto a la superficie de presión de cada álabe, posee mayor velocidad y menor entropía. Por ello, estos modelos lo asumen como flujo sin pérdidas. Todas las pérdidas se concentran en la estela, que se encuentra junto a las superficies de succión y posee mayor entropía que el flujo central. Bajo estas hipótesis, los modelos de dos zonas aplican las ecuaciones de conservación a cada parte del flujo por separado. Además, en el cálculo de las condiciones de la estela se aplican diferentes correlaciones de pérdidas provenientes de los modelos de una zona. Ejemplos de estos modelos son los análisis realizados por (Oh et al., 2001) y Schneider et al. (2015).

Debido a que los modelos de una zona han sido los más desarrollados hasta hoy, son los implementados en este trabajo. En concreto, se han desarrollado dos modelos diferentes basados en los trabajos de Aungier (2000) y Oh et al. (1997). La razón de esta elección es que (Aungier, 2000) proporciona un completo análisis en el que todos los componentes del compresor son modelados, incluyendo diferentes correlaciones de pérdidas para cada uno de ellos. Por otro lado, (Oh et al., 1997) realiza una amplia comparación entre diferentes correlaciones de pérdidas del rotor, entre las que se encuentran correlaciones de muchos otros autores, proporcionando un conjunto óptimo de modelos de pérdidas.

A.1.2 Objetivos.

Los objetivos del presente trabajo son:

- Implementación de dos modelos 1D de predicción del rendimiento de un compresor centrífugo.
- Análisis de los resultados mediante la comparación con datos experimentales del mismo compresor. En concreto, se estudiará el error relativo en las estimaciones del ratio de presión y la eficiencia adiabática del compresor.
- Comparación de las pérdidas obtenidas con los resultados obtenidos por simulaciones numéricas basadas en las ecuaciones de Reynolds-averaged Navier–Stokes (RANS) del mismo compresor. Estas simulaciones ya han sido realizadas y, por tanto, el desarrollo del modelo CFD no entra entre las tareas de este proyecto. Sin embargo, sí que entra como parte de este estudio la búsqueda de un método para extraer cada pérdida individual del modelo CFD y la obtención y análisis de estos resultados.
A.2 Implementación.

En esta sección se explican los principales aspectos de los modelos unidimensionales implementados así como la metodología seguida para calcular las pérdidas en el modelo CFD. En la Tabla A.1 se recogen los datos geométricos más relevantes del compresor modelado, así como las condiciones de entrada del aire y el modelo de gas utilizado.

Table A.1: Datos geométricos compresor, modelo CFD y condiciones de entrada.

<table>
<thead>
<tr>
<th>Geometría del compresor.</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Diámetro exterior del rotor</td>
<td>52.5mm</td>
</tr>
<tr>
<td>Número de álabe completos</td>
<td>6</td>
</tr>
<tr>
<td>Número de álabe divisor</td>
<td>6</td>
</tr>
<tr>
<td>Ratio de área de difusión</td>
<td>$A_R=0.708$</td>
</tr>
<tr>
<td>Coeficiente de flujo máximo</td>
<td>$\phi_{dc}=0.094$</td>
</tr>
<tr>
<td>Coeficiente de trabajo</td>
<td>$\phi_{dc}=0.622$</td>
</tr>
<tr>
<td>Número Mach del álabe</td>
<td>$M_{2dc}=1.09$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Propiedades del modelo numérico</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Dominios</td>
<td>Tubería de entrada, rotor, difusor, voluta y tubería de salida</td>
</tr>
<tr>
<td>Tipo de mallado, tamaño</td>
<td>Poliédrico + capa prismas, 1.1 · 10^6</td>
</tr>
<tr>
<td>Paredes</td>
<td>Adiabáticas</td>
</tr>
<tr>
<td>Turbulencia</td>
<td>RANS, SST $k – omega$</td>
</tr>
<tr>
<td>Rot.-stat. interf.</td>
<td>Plano de mezcla</td>
</tr>
<tr>
<td>Solver</td>
<td>2nd order implicit coupled</td>
</tr>
<tr>
<td>Cond. contorno.</td>
<td>Entrada: Flujo máximo</td>
</tr>
<tr>
<td></td>
<td>Salida: Presión</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Condiciones de entrada.</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Gas</td>
<td>Aire</td>
</tr>
<tr>
<td>Modelo de gas</td>
<td>Ideal</td>
</tr>
<tr>
<td>Presión total</td>
<td>1bar</td>
</tr>
<tr>
<td>Temperatura total</td>
<td>300K</td>
</tr>
</tbody>
</table>

A.2.1 Modelos unidimensionales.

Los modelos analíticos han sido implementados en MATLAB®. Siguiendo la modularidad que caracteriza a los modelos 1D de una zona, cada componente es modelado por separado y luego son conectados en serie según el orden en el que aparezcan en el compresor real analizado. Como se dijo anteriormente, a lo largo de cada componente se aplican las ecuaciones de conservación de masa, momento y energía y las diferentes correlaciones de pérdidas. En el rotor, se aplica además la conservación de rotalpía para calcular el trabajo de entrada. Debido al carácter empírico de los modelos de pérdidas, éstos dependen de las condiciones de salida que en un primer momento son desconocidas. Por lo tanto es necesario llevar a cabo un cálculo iterativo. En la Figura A.3 se presenta el algoritmo de cálculo implementado.
Aunque el difusor también puede ser calculado siguiendo este método, Aungier (2000) proporciona un camino alternativo basado en las ecuaciones de conservación en su forma diferencial, que a su vez incluyen varios términos para modelar las diferentes pérdidas. Estas ecuaciones se aplican a lo largo del compresor, resolviéndolas mediante métodos numéricos.

En la Tabla A.2 se recogen todas las pérdidas analizadas en cada componente, así como una breve descripción de su origen. También, se indica entre los dos modelos implementados cual tiene en cuenta cada pérdida y cuál no la modela. Los modelos del difusor y de la voluta son los mismos en ambos análisis ya que Oh et al. (1997) sólo proporciona el análisis del rotor. Por tanto, ambos componentes son analizados siguiendo el método de Aungier (2000).

Todas las correlaciones de las pérdidas se pueden encontrar en el Capítulo 3. Cabe destacar que Aungier (2000) proporciona las pérdidas en forma de coeficientes de pérdida de presión, mientras que en la colección de Oh et al. (1997) son descritas como pérdidas de entalpía medidas a la salida del rotor. Para hacer un estudio consistente y debido a que en la literatura es más común que las pérdidas se describan como decrementos de entalpía, las pérdidas de ambos modelos son convertidas en coeficientes adimensionales de pérdida de entalpía ($\lambda_i$), calculados con la siguiente fórmula.

$$\lambda_i = \frac{\Delta h_{fi}}{U_2^2}$$  \hspace{1cm} (A.1)

### A.2.2 Cálculo de pérdidas en el modelo CFD.

Para investigar más a fondo los modelos de pérdidas unidimensionales, se comparan las pérdidas con las del modelo CFD. Sin embargo, es imposible hacer una comparación de todas las pérdidas internas por separado, ya que en el modelo CFD es imposible valorar todas ellas independientemente debido a que muchas de ellas actúan simultáneamente y en la misma zona del flujo.

Por ello, primero se comparan las suma de pérdidas a lo largo de cada componente, lo que sí que es
Table A.2: Pérdidas implementadas en cada componente del compresor.

<table>
<thead>
<tr>
<th>Pérdida</th>
<th>Aungier Oh et al.</th>
<th>Origen</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>ROTOR</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Incidencia</td>
<td>x</td>
<td>Diferencia entre el ángulo de entrada del álabe y del flujo.</td>
</tr>
<tr>
<td>Fricción</td>
<td>x</td>
<td>Fricción entre el fluido y las paredes.</td>
</tr>
<tr>
<td>Holgura</td>
<td>x</td>
<td>Flujo que fluye por la holgura entre los álabes y la carcasa.</td>
</tr>
<tr>
<td>Mezcla</td>
<td>x</td>
<td>Mezcla entre la región central del flujo y la estela a la salida del rotor.</td>
</tr>
<tr>
<td>Perfil</td>
<td>x</td>
<td>Crecimiento de la capa límite a lo largo del álabe y gradiente de presión en la dirección de álabe a álabe.</td>
</tr>
<tr>
<td>Choque</td>
<td>x</td>
<td>Condiciones de choque alcanzadas en la garganta</td>
</tr>
<tr>
<td>Numero Mach</td>
<td>x</td>
<td>Ondas de choque locales en la superficie de succión del álabe que generan separaciones de la capa límite</td>
</tr>
<tr>
<td>Super crítico</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Distorsión</td>
<td></td>
<td>Distorsión del flujo a la salida por efectos viscosos o bloqueos</td>
</tr>
<tr>
<td>Hub to shroud</td>
<td>x</td>
<td>Grandiente de presión en la dirección del cubo hacia la carcasa</td>
</tr>
<tr>
<td>Externas</td>
<td>x</td>
<td>Pérdidas externas que incluyen pérdidas por fugas, recirculación o fricción del disco</td>
</tr>
<tr>
<td><strong>DIFUSOR</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fricción</td>
<td>x</td>
<td>Fricción entre el fluido y las paredes.</td>
</tr>
<tr>
<td>Difusión</td>
<td>x</td>
<td>Pérdidas asociadas al proceso de difusión.</td>
</tr>
<tr>
<td>Curvatura</td>
<td>x</td>
<td>Curvatura del difusor.</td>
</tr>
<tr>
<td><strong>VOLUTA</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fricción</td>
<td>x</td>
<td>Fricción entre el fluido y las paredes.</td>
</tr>
<tr>
<td>Velocidad</td>
<td>x</td>
<td>Pérdida de la componente meridional de la velocidad debido a la trayectoria en remolino que adquiere el flujo en la voluta.</td>
</tr>
<tr>
<td>meridional</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Velocidad</td>
<td>x</td>
<td>Pérdida de momento angular.</td>
</tr>
<tr>
<td>tangencial</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Salida cónica</td>
<td>x</td>
<td>Cambio de una trayectoria en remolino propia de la voluta a una trayectoria lineal a lo largo del cono.</td>
</tr>
</tbody>
</table>
fácilmente calculable. Para ello se mide el incremento de entropía a lo largo de cada componente mediante el cálculo del mass flow average de la temperatura total y presión total tanto a la entrada como salida de cada componente. Posteriormente, se aplica la siguiente relación termodinámica de Gibbs para calcular el incremento de entalpía.

\[ T ds = dh - vd p \quad \Rightarrow \quad \Delta s_i = c_p \ln \left( \frac{T_{0i}}{T_{0o}} \right) - R \ln \left( \frac{p_{0i}}{p_{0o}} \right) \]  

(A.2)

significando los subíndices i y o, entrada y salida respectivamente. Debido a que en el difusor y la voluta no hay trabajo de entrada, la entalpía total es constante y por ello el término referente al cambio de temperatura es nulo. Una vez que el incremento de entalpía es obtenido, se calcula un coeficiente equivalente de pérdida de entalpía a presión constante aplicado a la salida del rotor. Por tanto, los coeficientes \( \lambda_i \) son calculados mediante

\[ \lambda_i = \frac{\Delta h_{li}}{U_2^2} = \frac{T_{o2} \Delta s_i}{U_2^2} \]  

(A.3)

Cabe destacar que en el rotor, para diferenciar entre las pérdidas internas y externas, se calcula un punto intermedio. En la Figura A.4 se muestra el diagrama h-s a lo largo del rotor. El punto b denota las condiciones que tendría el flujo si no hubiese pérdidas externas. Es calculado aplicando la ecuación de Euler entre la entrada y la salida, es decir \( \Delta h_b = \omega (r_2 C_{u2} - r_1 C_{u1}) \). Una vez el punto b es conocido, las pérdidas internas y externas del CFD se calculan siguiendo el diagrama h-s y las ecuaciones anteriormente presentadas.

Figura A.4: Diagrama h-s para cálculo de pérdidas del rotor en el modelo CFD.

Una vez la suma de pérdidas en cada componente es conocida, las pérdidas internas del rotor son estudiadas independientemente. Para ello el rotor es dividido en cuatro volúmenes de control diferentes, tal y como se representa en la Figura A.5. Posteriormente, se realizan las siguientes hipótesis: a) En el primer volumen de control que se extiende desde la entrada del rotor hasta la garganta, el incremento de entropía está causado principalmente por las pérdidas de incidencia. b) En la garganta, el incremento de entropía es originado por las pérdidas de choque, si es que estas aparecen. c) Desde la garganta al
Por tanto, se puede ver que no todas las pérdidas internas del rotor modeladas en los análisis 1D son medidas. La comparación se realiza por tanto para las pérdidas de fricción, incidencia, choque, perfil y mezcla, así como para las pérdidas externas del rotor y la suma de pérdidas del difusor y voluta.

### A.3 Análisis de resultados.

#### A.3.1 Predicción analítica del rendimiento del compresor.

A continuación, en la Figura A.6 se muestran los resultados del ratio de presión total predichos por los modelos 1D, así como los datos experimentales y del modelo CFD. Las líneas de velocidad son normalizadas dividiendo entre la tercera velocidad más baja ($N_0$) y el flujo másico es también normalizado respecto al flujo másico de choque en la máxima velocidad.
APPENDIX A. RESUMEN EN ESPAÑOL DEL TRABAJO.

Se puede ver cómo los resultados del modelo CFD son muy precisos a bajas velocidades. Sin embargo, a mayores velocidades, la predicción es peor y se presenta una sobreestimación en los puntos cercanos al punto de diseño. Sin embargo, el mayor error relativo es de tan sólo un 6.1%, encontrado en la curva de mayor velocidad.

En cuanto a los modelos 1D, se puede ver que a bajas velocidades ambos proporcionan resultados muy semejantes, subestimando el ratio de presión experimental. Ambos modelos son menos precisos que el modelo CFD en estos regímenes. Sin embargo, a velocidades altas, el modelo de Aungier alcanza precisiones semejantes a las del modelo CFD, mientras que el modelo de Oh et al. sobreestima los resultados. En estas velocidades se puede ver la incapacidad de ambos modelos para predecir la fuerte caída de rendimiento en los puntos próximos a la línea de choque.

De los 77 puntos de operación simulados, los modelos de Aungier y Oh et al. presentan un error relativo en la estimación del ratio de presión total menor a un 5% en 49 y 41 puntos respectivamente. Aumentando el rango a errores menores del 10%, son 69 y 61 los puntos simulados por cada modelo que cumplen dicha condición.

En cuanto a la eficiencia, un estudio detallado se puede encontrar en el Capítulo 4. En resumen, el modelo CFD vuelve a obtener mejores resultados, aunque ligeramente sobreestiman los valores experimentales debido, principalmente, a la hipótesis de paredes adiabáticas incluidas en el modelo (Sundström et al., 2015). En cuanto a los modelos 1D, predomina una subestimación de la eficiencia. Sin embargo, cerca de la línea de choque a altas velocidades se observan fuertes sobreestimaciones, causadas por la mala predicción de las pérdidas de choque. El modelo de Aungier predice 42 y 60 puntos con un error relativo menor al 5% y 10% respectivamente. En cuanto al modelo de Oh et al., los puntos estimados dentro de esos límites son 40 y 55 respectivamente.
A.3.2 Suma de pérdidas en cada componente.

La Figura A.7 recoge la suma de pérdidas internas a lo largo de cada componente en los modelos 1D y las simulaciones RANS.

A la vista de los resultados en el rotor, se puede ver cómo el modelo de Aungier predice mejor sus pérdidas tomando como referencia los resultados numéricos. Tanto en el modelo CFD como en el modelo de Aungier la tendencia es la misma: los mayores valores se alcanzan cerca de la línea de bombeo. Al aumentar el flujo máscico a cada velocidad, las pérdidas se reducen hasta alcanzar un mínimo próximo al punto de diseño. Finalmente, cerca de la línea de choque las pérdidas vuelven a aumentar debido a los fenómenos supersónicos. Aunque la tendencia es la misma, Aungier sobreestima los valores de las pérdidas, especialmente a altas velocidades. Por otro lado, el modelo de Oh et al. presencia una tendencia monótona decreciente debido a que no modela las pérdidas de choque.

Figure A.7: Suma de pérdidas internas a lo largo de cada componente.
APPENDIX A. RESUMEN EN ESPAÑOL DEL TRABAJO.

En el resto de componentes (difusor y voluta) ambos modelos presentan resultados muy semejantes ya que utilizan el mismo análisis, y las pequeñas diferencias se deben sólo a las diferentes condiciones a la salida del rotor. En el difusor, se ve cómo los modelos 1D sobreestiman fuertemente las pérdidas, con valores mayores que el doble de los obtenidos en el modelo CFD cerca de la línea de bombeo. Además, las pérdidas del modelo CFD presentan un mínimo, que no es predicho por los modelos analíticos. Resultados semejantes se pueden apreciar en el estudio realizado por Harley et al. (2013).

Finalmente en la voluta, también se pueden ver importantes sobreestimaciones. Aunque las pérdidas en la salida cónica son estimadas correctamente, con ligeras sobreestimaciones cerca de la línea de línea de choque, en el primer tramo de la voluta las predicciones son débiles. Principalmente cerca de la línea de choque y a bajas velocidades los modelos 1D obtienen pérdidas muy superiores a las numéricas. Esto se debe a las elevadísimas pérdidas de velocidad meridional modeladas por los estudios analíticos. En el capítulo 4 se explica más en detalle el estudio de cada componente.

A.3.3 Pérdidas en el rotor.

Finalmente, se realiza la comparación de las diferentes pérdidas del rotor obtenidas en el modelo CFD. En este resumen no se comentan los resultados del resto de pérdidas que tienen en cuenta los modelos 1D pero que no han sido obtenidas del modelo numérico. Este análisis se puede encontrar en el Capítulo 4 del presente trabajo.

La Figura A.8 muestra los resultados de las diferentes pérdidas del rotor.

A continuación, se detallan brevemente los puntos más importantes extraíbles de esta comparación de modelos.

- Las pérdidas externas presentan una gran diferencia cerca de la línea de bombeo y especialmente a bajas velocidades. Este hecho se debe a las pérdidas por el flujo inverno presente en estos puntos. Los modelos 1D tienen en cuenta este hecho, pero se puede ver cómo lo subestiman en gran nivel.

- Las pérdidas de incidencia presentan a grandes rasgos la misma tendencia en todos los modelos: decrecimiento desde la línea de bombeo a la línea de choque. Sin embargo, los valores del CFD son los más elevados, lo que puede estar originado por la acción de otras pérdidas en el volumen de control asociado a las pérdidas de incidencia. Además, comparando los valores de estas pérdidas con el resto, se puede ver cómo son las más relevantes en los puntos próximos a la línea de bombeo, suponiendo en estos regímenes la mitad de las pérdidas totales del rotor.

- En cuanto a las pérdidas de fricción, todos los modelos toman valores semejantes, situándose las pérdidas del modelo CFD entre las de los modelos 1D. En estos últimos modelos se muestra un crecimiento cuadrático en dirección a la línea de choque, debido a la dependencia de estas pérdidas con el cuadrado de la velocidad relativa, que a su vez es proporcional al flujo másico. En el modelo CFD, los mayores valores se alcanzan también en los puntos cercanos al choque, pero en las velocidades más altas aparece un ligero mínimo próximo al punto de diseño de cada velocidad.

- En cuanto a las pérdidas de choque en la garganta, el modelo Oh et al. no las tiene en cuenta y es por ello que no es representado en la figura. El modelo de Aungier presenta sólo valores no nulos en las mayores velocidades y junto a la línea de choque, justamente donde las ondas de choque más fuerte son esperables. Sin embargo, en el modelo CFD aparecen pérdidas en todas las velocidades, y con valores siempre superiores a las del modelo de Aungier.
En las pérdidas de perfil se muestra una gran diferencia entre todos los modelos. El modelo de Aungier estima las mayores pérdidas, presentando los valores más altos cerca de la línea de bombeo y con un pequeño mínimo en el punto de diseño.
El modelo de Oh et al. muestra una tendencia monótona decreciente desde la línea de bombeo a la de choque. La causa de la diferencia entre los dos modelos es principalmente la adición de las pérdidas de *hub-to-shroud* al modelo de Aungier y no en el de Oh et al., ya que éste no las analiza. Estas pérdidas *hub-to-shroud* son crecientes desde la línea de bombeo a la línea de choque, y por ello la suma da lugar a la tendencia con un pequeño mínimo presente del modelo de Aungier.

Por otro lado, las pérdidas de CFD presentan valores y tendencias muy irregulares. A velocidades bajas se alcanzan incluso valores negativos, lo que es totalmente irreal. Sin embargo, a velocidades más altas se muestra una tendencia semejante en todas y parecida a la de Aungier. Sin embargo, el crecimiento junto a la línea de choque es mucho más drástico. Este hecho es causado por la aparición en el volumen de control asociado a las pérdidas de perfil de ondas de choque locales en la superficie de succión de los álabes. Recordemos que estas pérdidas son modeladas por Aungier (Tabla A.2) por separado.

- Finalmente, las pérdidas generadas por la mezcla de la región central del flujo y la estela a la salida presentan también ciertas diferencias. El modelo de Aungier y el modelo CFD se asemejan junto a la línea de bombeo. Sin embargo, próximo al punto de diseño las pérdidas CFD presentan un mínimo y luego crecen al aumentar el flujo másico. Este hecho es explicable por la estabilización del área ocupada por la estela en estos puntos, lo que hace que las pérdidas aumenten al aumentar el flujo másico. El modelo de Aungier no depende del área de la estela y por ello presenta una tendencia decreciente a lo largo de cada velocidad. Por otro lado, el modelo de Oh et al. sí que tiene en cuenta la cantidad de estela y por ello a las velocidades más bajas sí que se observa un mínimo. Sin embargo, los valores son muy inferiores a los otros dos modelos en todos los puntos de operación lo que es causado, precisamente, por una subestimación del área de la estela.

En el Capítulo 4 se muestra una explicación en mucha mayor profundidad de la causa de las diferencias de cada pérdida entre los diferentes modelos.

### A.4 Conclusiones y líneas futuras.

Tras la correcta implementación de los modelos y una vez realizadas las comparaciones y análisis establecidos como objetivos, se pueden extraer las siguientes conclusiones:

- Aunque a bajas velocidades la predicción del compresor obtenida por los modelos 1D sea menos precisa que la de los modelo CFD, se puede ver igualmente cómo la mayoría de los puntos son estimados con bajos errores relativos. Además el modelo de Aungier obtiene una precisión semejante a los resultados numéricos en velocidades altas.

- En cuanto a la comparación de la suma de pérdidas a lo largo de cada componente entre los modelos 1D y CFD, se puede ver que el modelo de los componentes estacionarios (difusor y voluta) son la mayor fuente de error relativo. En cuanto al rotor, la colección de Aungier estima la misma tendencia aunque con valores ligeramente sobredimensionados. La colección de Oh et al. obtiene peores resultados para el compresor estudiado.

- Finalmente, la comparación de pérdidas internas del rotor de los modelos 1D y CFD se presenta como un método que permite analizar más en profundidad los modelos unidimensionales, viendo las ventajas y debilidades de los mismos. Así se ve por ejemplo que las pérdidas de incidencia están bien modeladas en ambas soluciones analíticas, mientras que las pérdidas de mezcla presentadas por Aungier deben ser refinadas mediante la dependencia de éstas respecto del flujo de estela existente y las pérdidas de Oh et al. deben dar más peso a estas pérdidas.
Directamente de estas mismas conclusiones y de los análisis realizados se plantean varias líneas futuras posibles.

- Se han modelado únicamente el rotor, un difusor sin álalbes y una voluta. Para completar la modularidad y flexibilidad que ofrecen estos tipos de estudio, sería necesario desarrollar el modelo de difusores con álalbes y canales de retornos que pueden aparecer conectando diferentes compresores en serie. Aungier (2000) provee de modelos para ambos componentes. Una vez realizado esto, aumentaría el rango de compresores que se podrían modelar con el código implementado.

- En cuanto a la comparación de la sumas de pérdidas en cada componente, se plantea la necesidad de una mejora de los modelos estacionarios. En la literatura es posible encontrar otros enfoques unidimensionales para modelar dichos componentes, como es el ejemplo del análisis de volutas presentado por van den Braembussche et al. (1999).

- Finalmente, en cuanto a la comparación de las pérdidas internas del rotor se abren varios caminos para posibles investigaciones. Por un lado, un refinamiento en la metodología seguida para el cálculo de las pérdidas en el modelo CFD podría permitir estimar más fuentes de pérdidas, como las pérdidas por holguras o las pérdidas relacionadas a ondas de choque locales. Por otro lado, esta misma comparación muestra la imprecisión de las correlaciones de algunas pérdidas, como por ejemplo las de mezcla. Por ello, se plantea como camino una mejora de estos modelos.
REFERENCES


73


