A CFD ANALYSIS TOWARDS FLOW CHARACTERISTICS OF THREE PRE-SWIRLER DESIGNS

ADRIEN DULAC

Master of Science Thesis
Stockholm, Sweden 2012
A CFD ANALYSIS TOWARDS FLOW CHARACTERISTICS OF THREE PRE-SWIRLER DESIGNS

Adrien Dulac

MSc Thesis, EGI-2012-033 MSC EKV 886

Department of Energy Technology
Division of Heat and Power Technology
Royal Institute of Technology
100 44 Stockholm, Sweden
ABSTRACT

Although pre-swirlers play a determinant role in the transport of air from stationary parts to rotating holes, knowledge about their actual performance is limited. Therefore, this paper aims to relate how the pre-swirler pressure drop affects the performance of different pre-swirlers in terms of discharge coefficient, adiabatic pre-swirl effectiveness, and swirl ratio. The results are extracted from numerical simulations carried out on three different designs, two guide vanes, and a nozzle. When available, the results are compared to experimental data.

The guide vanes have shown similar responses to the pressure drop variations. Their discharge coefficients remain relatively insensitive with an average value of 97%. The swirl ratio range from 0.704 to 1.013 and 0.703 to 1.023 respectively for a pressure drop varying from 3 to 7 bars. The adiabatic pre-swirl effectiveness is of 96% and 94%, respectively, under steady state operation.

The nozzle design has shown inferior performance as compared to the guide vane designs. Its discharge coefficient remains around 91% and the swirl ratio varies between 0.678 and 1.121 for a pressure drop ranging from 3 to 10 bars. Under steady state operation, the adiabatic pre-swirl effectiveness is 1.22.

The influence of through-flows on the aforementioned parameters was also analyzed. It was observed that the through-flow deteriorates the performance of the pre-swirlers, whether in terms of dimensionless pre-swirl effectiveness, or swirl ratio. The discharge coefficient was however not affected.
TABLE OF CONTENT

ABSTRACT .......................................................................................................................... III

TABLE OF CONTENT ......................................................................................................... IV

LIST OF FIGURES ............................................................................................................... VI

LIST OF TABLES ................................................................................................................ VII

NOMENCLATURE .............................................................................................................. VIII

1. INTRODUCTION ........................................................................................................ 1

2. AIM AND OBJECTIVES ............................................................................................ 2

3. METHODOLOGY ........................................................................................................ 3

4. STATE OF THE ART ................................................................................................... 4

5. GOVERNING EQUATIONS .......................................................................................... 6

5.1. Fundamental equations of fluid dynamics ................................................................. 6

5.1.1. Continuity equation .............................................................................................. 6

5.1.2. Navier-Stokes equations ....................................................................................... 6

5.1.3. The energy equation ............................................................................................ 8

5.2. Transition to turbulent flows ................................................................................... 9

5.2.1. What is turbulence? ............................................................................................ 9

5.2.2. The Reynolds Averaged Navier Stokes equations ................................................. 9

5.3. Modeling turbulence ............................................................................................... 11

5.3.1. The Boussinesq Approach ................................................................................ 12

5.3.2. The k-ε Model ................................................................................................... 13

5.3.3. The k-ω Models ................................................................................................. 14

5.4. Turbulent Boundary Layer ..................................................................................... 16

5.5. Stator-rotor cavity flow .......................................................................................... 18

5.5.1. Vortex, vorticity and rotation .............................................................................. 18

5.5.2. Rotating flow due to a free disc ......................................................................... 18

5.5.3. Enclosed wheelspace ......................................................................................... 21

5.5.4. Stator-Rotor cavity with superposed mass flow ................................................. 22

6. MODELING ASPECTS ................................................................................................. 24

6.1. Geometrical considerations .................................................................................... 24

6.1.1. Pre-swirlers ........................................................................................................ 24

6.1.2. Pre-swirl Systems .............................................................................................. 25

6.2. Numerical Setup ..................................................................................................... 28

6.2.1. Boundary conditions ......................................................................................... 28

6.2.2. Turbulence Model ............................................................................................. 29
LIST OF FIGURES

Fig. 6.1: The Free Disc, adopted from Childs (2011) ................................................................. 18
Fig. 6.2: Velocity profiles for the free disc, adopted from Childs (2011) ...................................... 19
Fig. 6.3: The four flow regime, adopted from Childs (2011) .......................................................... 22
Fig. 6.4: Stator-Rotor cavity with superposed mass flow, adopted from Childs (2011) ................. 22
Fig. 6.5: Isometric view of Vane-1 .................................................................................................. 24
Fig. 6.6: Top view of Vane-1 ......................................................................................................... 24
Fig. 6.7: Isometric View of Vane-2 .................................................................................................. 25
Fig. 6.8: Top view of Vane-2 ......................................................................................................... 25
Fig. 6.9: Isometric View of Nozzle ............................................................................................... 25
Fig. 6.10: Top view of Nozzle ....................................................................................................... 25
Fig. 6.11: Numerical Setup in CFX 13.0 ......................................................................................... 28
Fig. 6.12: Extraction of results on an axial line (in red) ................................................................. 31
Fig. 6.13: Swirl Ratio vs. axial length for a Frozen Rotor and a Stage mixing option .................... 31
Fig. 6.14: Plane located at z*=0.3 .................................................................................................. 32
Fig. 6.15: Circumferential averaged Swirl Ratio at z*=0.3 ........................................................... 32
Fig. 6.16: Swirl ratio vs. axial length for different locations of the mixing plane ......................... 33
Fig. 6.17: 2D hexa-mesh (adopted from Jiyuan, Guan Heng, and Chaojun (2008)) ....................... 34
Fig. 6.18: O-Grid Structure on receiver holes ................................................................................ 38
Fig. 6.19: Hexa mesh on Vane-1 .................................................................................................... 38
Fig. 6.20: Tetra-mesh on Vane-2 .................................................................................................... 39
Fig. 6.21: Finer mesh at the nozzle inlet ....................................................................................... 39
Fig. 6.22: Evaluation of y+ values for all three models ................................................................. 42
Fig. 6.23: Evaluation of Vane-1 mesh quality ............................................................................... 43
Fig. 6.24: Evaluation of vane-2 mesh quality ................................................................................. 43
Fig. 6.25: Evaluation of nozzle mesh quality .................................................................................. 44
Fig. 6.26: Swirl ratio profile along a line located at z*=0.3 and θ=91.02° with relative errors ........ 46
Fig. 6.27: Circumferentially averaged swirl ratio at z*=0.3 .......................................................... 46
Fig. 7.1: Definition of surfaces ..................................................................................................... 48
Fig. 7.2: Total relative temperature throughout the cavity for vane-1 ........................................... 50
Fig. 7.3: Dimensionless pre-swirl effectiveness as a function of the pressure ratio π ................. 50
Fig. 7.4: Swirl ratio vs. axial length for the nozzle – π=1.6276 ..................................................... 51
Fig. 7.5: h-s diagram from the pre-swirler inlet 1 to outlet 2 ......................................................... 52
Fig. 7.6: Definition of surfaces 1 and 2 ......................................................................................... 53
Fig. 7.7: Location of surface 2 on the three models ....................................................................... 54
Fig. 7.8: Discharge Coefficient as a function of the pressure ratio .............................................. 55
Fig. 7.9: Isentropic Efficiency as a function of the pressure ratio .................................................. 55
Fig. 7.10: Experimental and numerical discharge coefficients for vane-1 .................................... 56
Fig. 7.11: Experimental and numerical discharge coefficients for nozzle .................................... 56
Fig. 7.12: Analysis of the tangential velocity in the axial direction ............................................... 57
Fig. 7.13: Surface plot of the swirl ratio on the radial band at z*=0.3 ........................................... 58
Fig. 7.14: Tangential band for extraction of the swirl ratio values ................................................. 58
Fig. 7.15: Averaged swirl ratio vs. axial length for Vane-1 ............................................................ 59
Fig. 7.16: Tangential Averaging Process ................................................................. 60
Fig. 7.17: Swirl Ratio versus tangential coordinate for different pressure drops .......... 60
Fig. 7.18: Averaged swirl ratio as a function of the pressure ratio ............................... 61
Fig. 7.19: Averaged swirl ratio vs. axial length for vane-2 ....................................... 61
Fig. 7.20: Swirl Ratio versus tangential coordinate for different pressure drops .......... 62
Fig. 7.21: Averaged swirl ratio as a function of the pressure ratio ............................... 63
Fig. 7.22: Averaged swirl ratio vs. axial length for nozzle ........................................ 63
Fig. 7.23: Swirl Ratio versus tangential coordinate for different pressure drops .......... 64
Fig. 7.24: Averaged swirl ratio as a function of the pressure ratio ............................... 65
Fig. 7.25: Comparative analysis of the swirl ratio for the three designs ......................... 65
Fig. 7.26: Introduction of through-flow for vane-1 ................................................ 66
Fig. 7.27: Swirl ratio vs. axial length for vane-2 ..................................................... 68
Fig. 7.28: Averaged swirl ratio vs. axial length for vane-1 ......................................... 68
Fig. 7.29: (a) Averaged swirl ratio vs. axial length for vane-2 (b) Averaged swirl ratio vs. axial length for the nozzle ................................................................. 69
Fig. 7.30: Swirl Ratio vs. tangential coordinate for vane-1 ......................................... 69
Fig. 7.31: Swirl Ratio vs. tangential coordinate for (a) vane-2 and (b) the nozzle .......... 70
Fig. 7.32: Influence of through-flow on the averaged swirl ratio for vane-1 .................. 71
Fig. 7.33: Influence of through-flow on the averaged swirl ratio for (a) vane-2 and (b) the nozzle ................. 71
Fig. 7.34: Influence of the through-flow on the dimensionless pre-swirl effectiveness ...... 72

LIST OF TABLES
Table 5.1: Phenomological Interpretation of the Reynolds Stress Equation ................ 11
Table 5.2: Phenomological Interpretation of the Turbulent Kinetic Energy Equation ....... 13
Table 6.1: Pre-swirl-system dimensions ................................................................... 27
Table 6.2: Boundary Conditions setup ...................................................................... 29
Table 6.3: Set of pressure drops and corresponding inlet total pressures .................. 29
Table 6.4: Solver control settings ............................................................................ 34
Table 6.5: Estimation of discretization error based on ASME Standards ................... 45
Table 7.1: Through-flow Boundary conditions setup .............................................. 67
Table 7.2: Mass flows for the different designs ...................................................... 67
## NOMENCLATURE

### Greek Symbols

- $\alpha$: k-ω model coefficient
- $\beta$: k-ω model coefficient, Swirl Ratio
- $\beta^*$: k-ω model coefficient
- $\beta_1$: k-ω model coefficient
- $\beta_2$: k-ω model coefficient
- $\beta_\omega$: k-ω model constant
- $\gamma$: Ratio of specific heats
- $\delta_{ij}$: Kronecker delta function
- $\delta$: Boundary layer thickness
- $\varepsilon$: Viscous dissipation term
- $\eta$: Isentropic efficiency
- $\lambda$: Lamé’s constant
- $\lambda_\ell$: Turbulent flow parameter
- $\mu_t$: Eddy viscosity
- $\mu$: Dynamic viscosity
- $\nu$: Kinematic viscosity
- $\zeta^*$: k-ω model constant
- $\pi$: Ratio of pressures
- $\varrho$: Density
- $\sigma_\varepsilon$: Inverse effective Prandtl number for $\varepsilon$ (k-\varepsilon Model)
- $\sigma_k$: Inverse effective Prandtl number for $k$ (k-\varepsilon Model), Turbulent Prandtl number for $k$ (k-ω Model)
- $\sigma_\omega$: Turbulent Prandtl number for $\omega$ (k-ω Model)
- $\tau_{ij}$: Stress Tensor
- $\Gamma_k$: Effective diffusivity of $k$
- $\Gamma_\omega$: Effective diffusivity of $\omega$
- $\Phi$: Dissipation function
- $\chi_k$: k-ω model coefficient
- $\chi_\omega$: k-ω model coefficient
- $\omega$: Specific rate of dissipation
- $\Omega$: Angular velocity
- $\Omega_{ij}$: Mean rate of rotation tensor
- $\nabla$: Nabla Operator

### Latin Symbols

- $a$: Speed of sound
- $A$: Surface
- $C_m$: Moment coefficient
- $C_p$: Heat capacity at constant pressure
- $C_w$: Dimensionless mass flow rate
- $C_\varepsilon$: k-\varepsilon model constant
- $C_{1\varepsilon}$: k-\varepsilon model constant
- $C_{2\varepsilon}$: k-\varepsilon model constant
- $e$: Internal energy
- $E$: Total energy
- $f$: Surface forces
- $F$: Body forces, compressibility correction
- $f_\beta$: k-ω model coefficient
- $f_\beta^*$: k-ω model coefficient
g  Gravitational constant
G  Gap ratio
h  Enthalpy
k  Turbulent kinetic energy, von Kármán constant
m  Mass
M_t  Turbulent Mach number
M_{10}  k-ω model constant
p^*  Apparent order, pressure
p'  Dimensionless pressure
P_b  Buoyancy production term
P_k  Production of k
P_{\omega}  Production of \omega
Q  Heat transfer
r  Radius
r^*  Dimensionless radius
R  Residual
Re  Reynolds Number
Re_t  Turbulent Reynolds number
R_\beta  k-ω model constant
Re_\phi  Rotational Reynolds Number
R_{\omega}  k-ω model constant
s  Entropy
S  Axial length of the wheelspace
S_{\delta}  Strain deformation
S_k  k-ε model source term for k
S_\varepsilon  k-ε model source term for \varepsilon
t  Time
T  Temperature
T_q  Torque
u  Velocity
u^*  Dimensionless Velocity
u_r  Shear stress velocity
V  Volume
W  Work
y^+  Dimensionless wall distance
\gamma_k  Dissipation terms of k
\gamma_{\omega}  Dilatation dissipation term
\gamma_{\omega}  Dissipation term of \omega

Abbreviations
ASME  American Society of Mechanical Engineers
CFD  Computational Fluid Dynamics
HOT  Higher Order terms
RNG  Re-Normalization Group
SAS  Secondary Air System
SST  Shear Stress Transport
1. INTRODUCTION

In gas turbine engines, the Secondary Air System (SAS) aims to cool critical components of the machine such as turbine discs and blades, and prevent the ingestion of mainstream hot gas into peripheral parts. Air is extracted from appropriate stages of the compressor and discharged into the turbine cavities in order to reduce the component thermal load. Although a lack of cooling air will lead to reduced component lifetime or machine failure, use of this air adversely affects the engine performance and thermodynamic efficiency. Engineers have therefore paid more and more attention to the Secondary Air System in order to extract the least amount of pressurized air from the compressor, thereby optimizing engine performance, while ensuring adequate cooling and sealing.

Often, before a stator-rotor cavity (also referred to as wheelspace), part of the cooling air is expanded through stationary pre-swirlers into the stator-rotor cavity and delivered to blade receiver holes on the rotating disc. The pre-swirler aims to swirl the air in order to increase its tangential velocity component. The effects are a reduction of the total temperature in the rotating frame of reference together with a more efficient and smoother transport of air into the rotating disc. Designs of pre-swirlers include simple drilled hole directed along the rotating path and guide vanes. Although pre-swirlers have been subject to many studies, rotating flow induces a level of complexity that often outperforms intuition. Thereby, knowledge about their actual performance always implies theoretical and experimental considerations combined with numerical simulations.
2. AIM AND OBJECTIVES

This study aims to improve pre-swirl correlation formulas by means of Computational Fluid Dynamics (CFD) analysis on three different pre-swirler designs. The main objective of the present report is to clarify how the pre-swirler pressure drop affects the pre-swirler performance in terms of discharge coefficient, adiabatic pre-swirl effectiveness and swirl ratio. The results will be extracted from CFD simulations and evaluated against empirical considerations in order to draw conclusion regarding the reliability of the simulations. Discrepancies between the three designs will be analyzed through a comparative analysis. Also, the influence of through-flows on the aforementioned parameters will be analyzed for the three designs.
3. METHODOLOGY

Computational Fluid Dynamics (CFD) has become a mature discipline and is now broadly adopted within the corporate setting for engineering applications. Nonetheless, selection of parameters such as boundary conditions, turbulence model, numerical schemes and others requires a thorough understanding of the fluid mechanics involved in the problem. The first part of this report therefore provides a theoretical description of flow characteristics through pre-swirlers.

The second part of the report more specifically focuses on numerical aspects. Geometrical considerations and assumptions will be explained for all three designs. Meshes will be described together with a grid convergence study. Other important aspects such as boundary conditions, numerical schemes, turbulence model, and numerical errors will be clarified.

In the third part of the report, the performance of the pre-swirlers will be evaluated in terms of discharge coefficient, dimensionless pre-swirl effectiveness and swirl ratio for different pressure drops. This procedure will be applied for all three designs and some results will be assessed against experimental data. A comparative analysis will be carried out to analyze the discrepancies between the different designs.
4. STATE OF THE ART

Although rotating flow is extremely important across a wide range of scientific applications, the subject induces a level of complexity that often outperforms simple explanations (Childs, 2011). In this context, use of CFD has naturally become a common practice for the gas turbine SAS designers. But the accuracy of such computational models for prediction of velocity, vorticity, temperature, pressure and other variables must be assessed against experimental observations. Therefore, several experimental and numerical investigations of pre-swirlers have been reported and published by ASME (American Society of Mechanical Engineers) and other organizations.

Two different designs of pre-swirl system are established in the literature. In a direct transfer system, the radius of the pre-swirler is similar or equal to the receiver holes radius. This is also termed as a high-radius pre-swirl system. In an alternative design, the radius of the pre-swirler is located lower than that of the receiver holes. This design is referred to as low-radius pre-swirl system. Theoretically, a direct transfer system can deliver lower cooling air temperature to the receiver holes (Kakade et al., 2011). The following picture describes the different components of a direct transfer pre-swirler system:

Meierhofer and Franklin (1981) carried out the first experiments in a direct transfer system in order to measure the total temperature in the rotating frame of reference. They also defined and quantified the effectiveness of such a system in terms of ratio of delivery air velocity to disc speed. The same experience was repeated by Dittmann and Dullenkopf (2004). The temperatures in the receiver holes were measured by means of temperature probes. The results pointed out divergences between experimental observations and theoretical isentropic models.

The performance of a nozzle/direct transfer system assembly was also reported by Benim, Brillert and Cagan (2004). Further researches (Chew et al., 2005) were also achieved on the same subject. Among other things, the discharge coefficient of such systems was estimated via numerical means. Javiya et al. (2011) also reported the results of CFD simulations applied on a low-radius system. They investigated the flow structure and heat transfer features inside the stator-rotor cavity, and compared the results with experimental data.
The performance of both nozzles and vanes were analyzed and compared by Javiya et al. (2011) with respect to some performance indicators such as the discharge coefficient, the velocity coefficient and the delivery cooling air temperature. Guide vanes have shown a slight superiority as compared to simple nozzles.

Recently, Karnahl et al. (2011) investigated the influence of the mixing interface on the flow structure and heat transfer features in a cover plate system. Three mixing options were under evaluation: frozen rotor, stage and transient. The three models have shown a fairly good agreement regarding the circumferentially averaged tangential velocity. However, investigation of the heat transfer has shown greater variations locally. The author recommends using a frozen rotor interface if the user is interested in studying the heat transfer in its circumferential rather than local behavior.

In the same manner, Javiya et al. (2012) quantified the influences of different turbulence models, near wall functions, and CFD commercial codes by establishing a comparison of the flow structures and heat transfer characteristics with experimental data. They pointed out some discouraging results from the RNG k-ε model. Other turbulence models and CFD codes have shown fairly good agreements.
5. GOVERNING EQUATIONS

Numerical simulations have become a common practice in the corporate setting but lack of theoretical knowledge can lead to hasty conclusions. Therefore, predictions of numerical models must be related to theoretical considerations. The first part of this report aims to provide insights on the fundamental equations of fluid dynamics. Latter in this section, the reader will be more specifically guided towards rotating flows. The objective is to acquire a more specific understanding of the flow characteristics in a stator-rotor cavity in order to better draw conclusions regarding the reliability of CFD simulations. This chapter refers to the excellent works established by White (2006), and Pope (2001) in their respective book. When appropriate, the reader will be referred to these authors or other sources for more detailed information.

5.1. Fundamental equations of fluid dynamics

5.1.1. Continuity equation

Sections 5.1.1 to 5.1.3 are based on White’s book (2006). The reader is invited to refer to this book for further information.

The continuity equation relates the idea that mass can neither be created nor destroyed in a flow, although it might be reorganized in space. Therefore, for a control volume, the rate of change of mass must be equal to the net flow of mass. Mathematically, this statement can be written as follow:

\[
\frac{dm}{dt} = \dot{m}_{in} - \dot{m}_{out} \tag{5.1}
\]

The partial differential form of the continuity equation can be obtained by expanding and re-organizing the different terms of the previous equation in a Cartesian coordinate system. Note that this method can be applied in any types of coordinate system. However, a Cartesian system leads to a simpler form of the equation:

\[
\frac{\partial \rho}{\partial t} + u_x \frac{\partial \rho}{\partial x} + u_y \frac{\partial \rho}{\partial y} + u_z \frac{\partial \rho}{\partial z} + \rho \left( \frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} + \frac{\partial u_z}{\partial z} \right) = 0 \tag{5.2}
\]

This continuity equation is more often given in its conservation form, applicable in any frame of reference:

\[
\frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{u}) = 0 \tag{5.3}
\]

Or,

\[
\frac{D\rho}{Dt} + \rho \nabla \mathbf{u} = 0 \tag{5.4}
\]

The term \( \frac{D\rho}{Dt} \) is called the material derivative and is further explained in the subsection below.

5.1.2. Navier-Stokes equations

The Eulerian Description

In its simplest interpretation, the Navier Stokes equation is a direct application of Newton’s second law of motion. When considering a continuum, this principle is applied to a particle of fluid, defined by its original position at an instant of time. According to Lai, Rubin and Krempl (1999), a momentum in motion can be described in two different manners:

- Either by following the particles: This formulation is known as the Lagrangian description, named after the mathematician J.S. Lagrange;
- Either by observing the particles from a fixed position. This is known as the Eulerian description (named after L. Euler).
When using the Eulerian approach, the dependency of the function with respect to time will be expressed as a function of both time and space. For the sake of example, considering a scalar \( \Phi(t, x, y, z) \):

\[
\frac{D\Phi}{Dt} = \frac{\partial \Phi}{\partial t} + \frac{\partial \Phi}{\partial x} \frac{dx}{dt} + \frac{\partial \Phi}{\partial y} \frac{dy}{dt} + \frac{\partial \Phi}{\partial z} \frac{dz}{dt}
\]  

(5.5)

The term \( \frac{Df}{Dt} \) is known as the material derivative of the function \( f \).

However,

\[
\frac{dx}{dt} = u_x, \quad \frac{dy}{dt} = u_y, \quad \frac{dz}{dt} = u_z
\]

(5.6)

Substituting equations (5.6) into equation (5.5) gives:

\[
\frac{D\Phi}{Dt} = \frac{\partial \Phi}{\partial t} + \frac{\partial \Phi}{\partial x} u_x + \frac{\partial \Phi}{\partial y} u_y + \frac{\partial \Phi}{\partial z} u_z = \frac{\partial \Phi}{\partial t} + \mathbf{u} \cdot \nabla \Phi
\]

(5.7)

where \( \nabla \) is the Nabla operator.

For a vector \( \Phi(t, x, y, z) \),

\[
\frac{D\Phi}{Dt} = \begin{pmatrix} \frac{D\Phi_x}{Dt} \\ \frac{D\Phi_y}{Dt} \\ \frac{D\Phi_z}{Dt} \end{pmatrix} = \begin{pmatrix} \frac{\partial \Phi_x}{\partial t} + \mathbf{u} \cdot \nabla \Phi_x \\ \frac{\partial \Phi_y}{\partial t} + \mathbf{u} \cdot \nabla \Phi_y \\ \frac{\partial \Phi_z}{\partial t} + \mathbf{u} \cdot \nabla \Phi_z \end{pmatrix}
\]

(5.8)

**The momentum equation**

The Navier-Stokes equation or momentum equation is described as a balance between inertia force and other forces. In fluid dynamics, this principle is applied on an infinitesimal particle of fluid. The equations are expressed in newton per unit volume of fluid.

The velocity of the fluid particle is a function of both time and space. Therefore, the inertia force acting on the element is expressed using the material derivative notation:

\[
\textit{Inertia force} = \rho \frac{Du}{Dt} = \rho \left( \frac{\partial u_x}{\partial t} + \mathbf{u} \cdot \nabla u_x \right)
\]

(5.9)

Equation (5.9) can be re-expressed using the Nabla operator:

\[
\textit{Inertia force} = \rho \frac{Du}{Dt} = \rho \left( \frac{\partial \mathbf{u}}{\partial t} + \left( \mathbf{u} \cdot \nabla \right) \mathbf{u} \right)
\]

(5.10)

The terms \( \rho \frac{\partial \mathbf{u}}{\partial t} \) and \( \rho (\mathbf{u} \cdot \nabla) \mathbf{u} \) are respectively referred to as local acceleration and convective acceleration. The local acceleration describes the variation of the velocity with respect to time, while the convective acceleration describes the variation of the velocity with respect to space. This term is non-linear and therefore considerably increases the complexity of the Navier-Stokes equation.

The forces acting on the fluid can be divided into two groups:

- The body forces \( \mathbf{F} \) such as gravity, electromagnetic, or Coriolis, and centrifugal for rotating frame of reference.
- The surface forces \( \mathbf{f} \), which result from external stresses applied on the surfaces of the fluid particle. These stresses are described by the stress tensor \( \tau_{ij} \), composed by 3 normal stresses and 6
tangential stresses (or shear stresses). The surface forces are expressed as a function of the stress tensor through equation (5.11):

\[ f = \mathbf{V} \cdot \mathbf{\tau}_{ij} \]  

(5.11)

Assuming that the fluid is Newtonian, it has been shown that the stresses are a function of the local strain deformations \( S_{ij} \), and the pressure \( p \):

\[ \tau_{ij} = -p \delta_{ij} + 2\mu S_{ij} + \delta_{ij} \lambda \ast \text{div}(\mathbf{u}) \]  

(5.12)

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

(5.13)

where \( \delta_{ij} \) is the Kronecker delta function (\( \delta_{ij} = 1 \) when \( i=j \), 0 otherwise), \( \mu \) is the dynamic viscosity and \( \lambda \) the Lamé’s constant.

If the flow is incompressible, \( \text{div}(\mathbf{u}) \) vanishes and the previous equation reduces to:

\[ \tau_{ij} = -p \delta_{ij} + 2\mu \xi_{ij} \]  

(5.14)

For compressible flows, the mean pressure \( p_{\text{mean}} \) acting on the fluid element is defined by the arithmetic averaging of the normal shear stresses:

\[ p_{\text{mean}} = -\frac{1}{3}(\tau_{xx} + \tau_{yy} + \tau_{zz}) = p - \left( \lambda + \frac{2}{3} \mu \right) \ast \text{div}(\mathbf{u}) \]  

(5.15)

In equation (5.15), the mean pressure is not equal the conventional pressure \( p \), as defined in thermodynamic. To resolve this discrepancy, the quantity \( \left( \lambda + \frac{2}{3} \mu \right) \) is neglected. This statement is known as the Stokes’ hypothesis.

Further assuming that the viscosity and density are constant, we finally get a simpler expression of the surface forces:

\[ f = -\mathbf{V} p + \mu \mathbf{V}^2 \mathbf{u} \]  

(5.16)

The Navier-Stokes equation is finally obtained by balancing the effects of inertia with the body and surface forces:

\[ \rho \left( \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \mathbf{V}) \mathbf{u} \right) = -\mathbf{V} p + \mu \mathbf{V}^2 \mathbf{u} + \mathbf{F} \]  

(5.17)

### 5.1.3. The energy equation

The total energy equation is based on the well-known fist law of thermodynamics, described by equation (5.18):

\[ \Delta E = \Delta W + \Delta Q \]  

(5.18)

where \( E \), \( W \) and \( Q \) respectively represent the total energy of the system, the work done, and the heat transfer. This equation is here applied to a particule of fluid and the variables are therefore expressed per unit volume.

Using the material derivative, equation (5.18) can be written as:

\[ \frac{DE}{Dt} = \frac{DW}{Dt} + \frac{DQ}{Dt} \]  

(5.19)
Using Fourier’s law, the heat transfer can be expressed as a function of the temperature:

\[
\frac{DQ}{Dt} = -\text{div}(q) = \text{div}(k \nabla T) \tag{5.20}
\]

where \(q\) is the heat flux and \(k\) refers to as the thermal conductivity.

The total derivative of the work done on the system is expressed as:

\[
\frac{DW}{Dt} = \nabla \cdot (u \cdot \tau_{ij}) = \varrho \left( \frac{Du}{Dt} - g \cdot u \right) + \tau_{ij} \frac{\partial u_i}{\partial x_j} \tag{5.21}
\]

Furthermore, by breaking the total energy into the addition of the internal energy, the kinetic energy and the potential energy, the derivative of the total energy can be expressed as:

\[
\frac{DE}{Dt} = \varrho \left( \frac{Du}{Dt} - g \cdot u + \frac{De}{Dt} \right) \tag{5.22}
\]

where \(e\) is the internal energy per unit mass.

Substituting equation (5.20), (5.21) and (5.22) into equation (5.19) leads to

\[
\varrho \frac{De}{Dt} = \text{div}(k \nabla T) + \tau_{ij} \frac{\partial u_i}{\partial x_j} \tag{5.23}
\]

This equation refers to as the energy equation. Another form of this equation can be obtained via decomposing the stress tensor into a pressure and a viscous term. For further information regarding this approach, the reader is invited to read White (2006). The result is exposed through equation (5.24):

\[
\varrho \frac{Dh}{Dt} = \frac{Dp}{Dt} + \text{div}(k \nabla T) + \Phi \tag{5.24}
\]

where \(\Phi\) is named the dissipation function, and \(h\) represents the fluid enthalpy.

## 5.2. Transition to turbulent flows

### 5.2.1. What is turbulence?

According to White (2006), turbulence is a flow pattern characterized by chaotic and unpredictable property changes. The notion of turbulent flow is closely related to the notion of stability, although the distinction is to be explained. A flow is described as stable when it can endure a disturbance and still return to its initial state. The Reynolds number is an indicator of stability. At high Reynolds number, a laminar flow becomes unstable and will therefore show a high disposition for turbulence.

More specifically, the Reynolds number is defined by the ratio of convective to viscous forces. At low Reynolds number, the viscous force outweighs the inertia force, therefore damping the effect of disturbances. At high Reynolds number, the inertia force becomes more important than its counterpart.

When the difference becomes sufficiently large, energy is absorbed from the laminar flow and small disturbances are amplified.

### 5.2.2. The Reynolds Averaged Navier Stokes equations

The fundamental equations of fluid dynamics are always valid whether the flow is laminar or turbulent. When turbulent, the different variables are usually decomposed into two components: a mean component and a fluctuating component. This approach is known as Reynolds Averaging.

\[
\begin{align*}
\mathbf{u} &= \bar{\mathbf{u}} + \mathbf{u}' \\
p &= \bar{p} + p' \\
T &= \bar{T} + T'
\end{align*} \tag{5.25}
\]
Substituting equations (5.25) into the continuity and Navier-Stokes equations and averaging the equations with respect to time lead to:

\[
\frac{D\rho}{dt} + \nabla \bar{u} = 0 \tag{5.26}
\]

\[
\rho \frac{D\bar{u}}{dt} = -\nabla p + \mu \nabla^2 \bar{u} + F - \rho \frac{\partial}{\partial x_j} (\bar{u}_i \bar{u}_j) \tag{5.27}
\]

For incompressible flows:

\[
\tau_{ij} = -p \delta_{ij} + 2\mu S_{ij} \tag{5.28}
\]

And so,

\[
\rho \frac{D\bar{u}}{dt} = -\nabla p + F + \frac{\partial}{\partial x_j} (\bar{T}_{ij} - \rho \bar{u}_i \bar{u}_j)
\]

where \( T_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) = 2\mu S_{ij} \)

Equations (5.26) through (5.29) are commonly known as the Reynolds Averaged Navier-Stokes (RANS) equations. They have the same general forms as the continuity and momentum equations. However, the Navier-Stokes equation is characterized by the apparition of the so-called Reynolds stresses -\( p\bar{u}_i \bar{u}_j \) (or turbulent stresses). For turbulent flow, these terms are significant and can never be neglected. Due to the unpredictable and chaotic nature of turbulent flow, the analytic form of turbulent stresses is not known. Thereby, additional equations are required in order to close the system. Two methods have raised much attention from the scientific community and are known as the Reynolds stress approach and the Boussinesq approach. The latter one will be further described in a subsequent section.

**Reynolds stress equation**

The following paragraphs are based on the work performed by Pope (2001) and George and Arndt (1988) in their respective book.

The Reynolds stress approach is based on the Reynolds stress equation which is obtained by multiplying consecutively the averaged Navier Stokes equation for incompressible flow by the scalars \( u_i' \) and \( u_k' \) and summing the results. \( k \) and \( i \) are two free indices that can take the value 1, 2 and 3. Einstein summation notation is here adopted:

\[
\frac{D\bar{u}_i' u_k'}{Dt} = \left( \frac{\partial u_i' u_k'}{\partial t} + \bar{u}_j \frac{\partial u_i' u_k'}{\partial x_j} \right)
\]

\[
- \frac{1}{\rho} \left( u_i \frac{\partial p'}{\partial x_k} + u_k \frac{\partial p'}{\partial x_i} \right) - \left( u_i u_j \frac{\partial u_k'}{\partial x_j} + u_k u_j \frac{\partial u_i'}{\partial x_j} \right)
\]

\[
+ \frac{1}{\rho} \left( u_i \frac{\partial T'_{kj}}{\partial x_j} + u_k \frac{\partial T'_{ij}}{\partial x_j} \right) - ((u_i' u_j') \frac{\partial \bar{u}_k}{\partial x_j} + (u_k' u_j') \frac{\partial \bar{u}_i}{\partial x_j}) \tag{5.30}
\]
Equation (5.30) is however more commonly presented in the following form:

\[
\begin{align*}
\left( \frac{\partial u'_i u'_k}{\partial t} + \bar{u}_j \frac{\partial u'_i u'_k}{\partial x_j} \right) &= - \left( \frac{p'}{\rho} \left( \frac{\partial u'_i}{\partial x_k} + \frac{\partial u'_k}{\partial x_i} \right) \right) \\
&+ \frac{\partial}{\partial x_j} \left( \left( -p' u'_k \delta_{ij} + p' u'_i \delta_{kj} \right) - u'_i u'_k u'_j \right) \left( \frac{\partial \bar{u}_k}{\partial x_j} + \bar{u}'_k \frac{S'_{ij}}{\bar{u}'} + \frac{u'_k S'_{ij}}{\bar{u}'} \right) \\
&- \left( u'_i u'_j \right) \frac{\partial \bar{u}_k}{\partial x_j} + \left( u'_k u'_j \right) \frac{\partial \bar{u}_i}{\partial x_j} + 2\nu \left( S'_{ij} \frac{\partial u'_k}{\partial x_j} + S'_{kj} \frac{\partial u'_i}{\partial x_j} \right)
\end{align*}
\] (5.31)

where \( \nu \) is the kinematic viscosity, \( \bar{u} = \frac{u}{\rho} \).

Since the Reynolds stresses constitute nine additional unknowns, closing the system of equations would require nine additional equations. \textit{A priori}, the Reynolds stress equation constitutes a promising approach since this equation actually confines nine new equations. However, this approach introduces additional unknowns. Closing the problem therefore requires additional hypothesis which are not described in this report. Nonetheless, the Reynolds stress equation constitutes a key equation to understand the physics driving turbulences and \textit{Erreur! Source du renvoi introuvable.} provides a phenomenological interpretation of its different terms:

<table>
<thead>
<tr>
<th>Mathematical forms</th>
<th>Phenomenological Interpretation</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \frac{\partial u'_i u'_k}{\partial t} + \bar{u}_j \frac{\partial u'_i u'_k}{\partial x_j} )</td>
<td>This is simply the total derivative of the Reynolds stress.</td>
</tr>
</tbody>
</table>

**Rate of change of Reynolds Stress**

\[ - \left( \frac{p'}{\rho} \left( \frac{\partial u'_i}{\partial x_k} + \frac{\partial u'_k}{\partial x_i} \right) \right) \]

This term relates the exchange of energy between the different components of stress.

**Pressure-Strain effects**

\[ \frac{\partial}{\partial x_j} \left( \left( -p' u'_k \delta_{ij} + p' u'_i \delta_{kj} \right) - u'_i u'_k u'_j \right) \left( \frac{\partial \bar{u}_k}{\partial x_j} + \bar{u}'_k \frac{S'_{ij}}{\bar{u}'} + \frac{u'_k S'_{ij}}{\bar{u}'} \right) - \left( u'_i u'_j \right) \frac{\partial \bar{u}_k}{\partial x_j} + \left( u'_k u'_j \right) \frac{\partial \bar{u}_i}{\partial x_j} + 2\nu \left( S'_{ij} \frac{\partial u'_k}{\partial x_j} + S'_{kj} \frac{\partial u'_i}{\partial x_j} \right) \]

It represents the spatial redistribution of the Reynolds Stresses.

**Diffusion (transport) of Reynolds Stresses**

\[ \text{It describes the production of Reynolds stresses due to mean velocity gradients.} \]

**Production of Reynolds Stresses**

\[ 2\nu \left( S'_{ij} \frac{\partial u'_k}{\partial x_j} + S'_{kj} \frac{\partial u'_i}{\partial x_j} \right) \]

This term quantifies the destruction of Reynolds stresses due to viscous effects.

**Dissipation of Reynolds Stresses**

Table 5.1: Phenomenological Interpretation of the Reynolds Stress Equation

### 5.3. Modeling turbulence

Nowadays, turbulence modeling continues to raise much interest from the scientific community. Although it is still the subject of many research programs, turbulence modeling has become a mature discipline. A number of approaches have been developed in order to cover a wide range of technical applications. None of the models has however shown to be superior in all applications to other models and the choice of turbulence model requires a thorough and skillful method. Wilcox (2006) provides a holistic view of turbulence modeling. The different methods are usually divided into three categories:

- The Reynolds Averaged approach: Based on RANS equations, new transport equations are introduced into the system in order to calculate the Reynolds stresses. Reynolds Averaged
approach integrates a number of methods that range from very simple model to more complicated and accurate models. In term of performances, this approach significantly reduces computational cost while withstanding a fairly good level of accuracy. Its core interest is engineering applications.

- The Large Eddy Simulation (LED): The full Navier Stokes equation is resolved for motion scales in the order of the grid size while turbulence modeling is adopted for smaller motion scales. It significantly increases computing resources, and requires high accuracy for spatial and temporal discretization (Wilcox, 2006).
- Direct numerical simulation: This approach resolves the full unsteady Navier-Stokes equation for all turbulence scales. It requires significant computing resources but provides a high level of accuracy. It is mainly used for research purposes.

Since Reynolds-Averaged approach is particularly suitable for engineering applications, the following chapter will focus mainly on this method. Other approaches are otherwise very well documented in the literature.

5.3.1. The Boussinesq Approach

Sections 5.3.1 to 5.3.3 are based on Fluent user’s guide (Fluent, 2006), Pope (2001) and George and Arndt (1988).

In the Boussinesq approach, the Reynolds stresses are calculated from a turbulent viscosity model:

\[-\rho u_i' u_j' = 2\mu_t S_{ij} - \frac{2}{3} k \delta_{ij}\] (5.32)

where \( \mu_t \) is referred to as the eddy (or turbulent) viscosity and \( k \) the turbulent kinetic energy. The eddy viscosity can then be derived from turbulent parameters such as \( \varepsilon \), \( k \) or \( \omega \) (dissipation rate). In such a case, additional transport equations are required to determine these parameters. In the \( k-\varepsilon \) and the \( k-\omega \) models, two additional transport equations (two-equation models) are introduced to determine \( k \) and \( \varepsilon \) and \( k \) and \( \omega \) respectively.

The Boussinesq approach constitutes a simple hypothesis that cannot capture the full complexity of the flow. The \( k-\varepsilon \) and \( k-\omega \) models, however sophisticated, will therefore always suffer from the weaknesses of the Boussinesq approach. This approach is however appreciated for its simplicity and low computational cost.

The following subsection provides additional information about the turbulent kinetic energy.

The turbulent kinetic energy equation

The turbulent kinetic energy equation is described by equation (5.33):

\[ k = \frac{1}{2} \left( u_x'^2 + u_y'^2 + u_z'^2 \right) \] (5.33)

The transport equation used in the \( k-\varepsilon \) and \( k-\omega \) models is derived from the previous equation. The result is more commonly known as the turbulent kinetic energy equation:

\[ \frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left( -\frac{1}{\rho} \left( p' u_i' \right) \delta_{ij} - \frac{1}{2} \left( q^2 u_i' u_j' \right) + 2\nu S'_{ij} S'_{ij} \right) \]

\[-\overline{u_i'} u_j' \frac{\partial \overline{u_i}}{\partial x_j} - 2\nu \overline{S'_{ij} S'_{ij}} \] (5.34)

where \( q^2 = u_i' u_i' \).
As for the Reynolds stress equation, Table 5.2 provides a phenomenological interpretation of the different terms.

<table>
<thead>
<tr>
<th>Mathematical forms</th>
<th>Phenomenological Interpretation</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \frac{Dk}{Dt} )</td>
<td>Total derivative of the kinetic energy.</td>
</tr>
<tr>
<td>( \frac{\partial}{\partial x_j} \left(-\frac{1}{\rho}(\bar{p}'u_i')\delta_{ij} - \frac{1}{2}(\bar{q}^2u_i')\right) + 2\nu\bar{S}_{ij}'u_i' )</td>
<td>Transport of kinetic energy due to pressure fluctuations, turbulence, and viscous stress respectively.</td>
</tr>
<tr>
<td>(-u_i'u_j'\frac{\partial \bar{u}_i}{\partial x_j} )</td>
<td>Production of kinetic energy from mean velocity gradients.</td>
</tr>
<tr>
<td>2( \nu(\bar{S}<em>{ij}'\bar{S}</em>{ij}') )</td>
<td>This term is more commonly noted ( \varepsilon ). It represents the dissipation of kinetic energy due to the effects of viscous stresses on turbulence.</td>
</tr>
</tbody>
</table>

Table 5.2: Phenomenological Interpretation of the Turbulent Kinetic Energy Equation

### 5.3.2. The k-\( \varepsilon \) Model

According to Fluent (2006), the k-\( \varepsilon \) turbulence model is a semi-empirical model based on theoretical considerations and observations. Due to its robustness, simplicity, and economy in terms of computational cost, the model has become popular within the field of engineering applications. In terms of accuracy, the model performs well for a wide range of industrial flows. However, the model behavior deems well for low Reynolds numbers, close to the walls. A couple of alternatives were proposed to enhance its behavior. As previously mentioned, the model is a two-equation model which signifies that two additional transport equations are introduced, one for the turbulent kinetic energy \( k \) and one for the viscous dissipation term \( \varepsilon \). The transport equation for \( k \) is derived from the exact solution while the transport equation for \( \varepsilon \) was obtained through theoretical and empirical considerations. The eddy viscosity is then expressed as a function of \( k \) and \( \varepsilon \).

For the turbulent kinetic energy \( k \):

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + P_k + P_b - \rho \varepsilon - Y_M + S_k \tag{5.35}
\]

For the viscous dissipation \( \varepsilon \):

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left( \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (P_k + C_{3\varepsilon} P_b) - C_{2\varepsilon} \frac{\varepsilon^2}{k} + S_\varepsilon \tag{5.36}
\]

where \( P_k \) and \( P_b \) represent the production of kinetic energy due to mean velocity gradients and buoyancy respectively. \( Y_M \) accounts for the effect of compressibility on turbulences. \( S_k \) and \( S_\varepsilon \) are two source terms.
The kinetic energy production term is derived from the turbulent kinetic energy equation:

\[ P_k = -\bar{u}_i^\prime \bar{u}_j^\prime \frac{\partial \bar{u}_i}{\partial x_j} \]  

(5.37)

When the effects of buoyancy are taken into consideration, the turbulent kinetic energy is coupled with the energy equation through the term \( P_b \):

\[ P_b = \beta g_i t \frac{\mu_t}{\text{Pr}_t} \frac{\partial T}{\partial x_i} \]  

(5.38)

where \( \beta \) is the thermal expansion factor, \( g_i \) the component of the gravitational constant in the \( i \)th direction, and \( \text{Pr}_t \) the turbulent Prandtl number. The default value of the latter is taken as 0.85.

Effects of compressibility on turbulence cannot be neglected for high Mach numbers. They are referred to as dilatation dissipation. In the \( k-\varepsilon \) model, \( Y_M \) is modeled via equation (5.39), proposed by Sarkar (1993):

\[ Y_M = 2 \rho \varepsilon M_t^2 \]  

(5.39)

Where \( M_t \) represents the turbulent Mach number (\( a \) is the speed of sound, \( \sqrt{\gamma RT} \) for ideal gas):

\[ M_t = \sqrt{\frac{k}{\alpha^2}} \]  

(5.40)

The eddy viscosity is computed from equation (5.41):

\[ \mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon} \]  

(5.41)

The values of the model constant have been determined through empirical considerations and are often defined as follow:

\[ C_{1\varepsilon} = 1.44, \quad C_{2\varepsilon} = 1.92, \quad C_{\mu} = 0.09 \]  

(5.42)

\[ \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3 \]

According to Fluent (2006), additional \( k-\varepsilon \) models have been developed in order to improve the model response to some specific applications. These models are referred to as RNG (Re-Normalization Group) \( k-\varepsilon \) and Realizable \( k-\varepsilon \). The RNG \( k-\varepsilon \) model was built using the renormalization group approach (mathematical method). This results in a different form of the transport equation for \( \varepsilon \), attempting to take into consideration the effects of all turbulence length scales on the flow. Therefore, while the eddy viscosity is computed from a single length scale in the standard \( k-\varepsilon \) model, it is computed according to the eddy scale in the RNG approach. This improves the response of the model for low Reynolds number and near wall regions.

The realizable \( k-\varepsilon \) model is a recent model that is more consistent with the physics of turbulent flows. The main difference remains in the introduction of a variable \( C_{\omega} \) in order to satisfy additional mathematical constraints. This results in a more realistic model for flows involving vortices and rotations. However, its application to rotating frame of reference is not recommended.

### 5.3.3. The \( k-\omega \) Models

The \( k-\omega \) model is also a two equations model for which one of the transport equations is derived from the turbulent kinetic energy \( k \) while the other is derived from the specific dissipation rate \( \omega \) defined as the ratio of \( \varepsilon \) to \( k \). Due to modifications of equations for low Reynolds number, the \( k-\omega \) model has proven robustness and accuracy for near wall regions and low Reynolds number flows. The model is described below:
For the turbulent kinetic energy:

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + P_k - Y_k + S_k
\]  

(5.43)

For the specific rate of dissipation:

\[
\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + P_\omega - Y_\omega + S_\omega
\]  

(5.44)

Where \( \Gamma_k \) and \( \Gamma_\omega \) represent the effective diffusivity of \( k \) and \( \omega \) respectively, given by:

\[
\Gamma_k = \mu + \frac{\mu_t}{\sigma_k}
\]  

(5.45)

\[
\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega}
\]  

(5.46)

\( \sigma_k \) and \( \sigma_\omega \) represents the turbulent Prandtl numbers, both taken equal to 2.0 in the default configuration.

The eddy viscosity is derived from equation (5.47):

\[
\mu_t = \alpha^* \frac{\rho k}{\omega}
\]  

(5.47)

\( \alpha^* \)'s role is to reduce the turbulent viscosity for low Reynolds number. At high Reynolds number, it takes the value 1.

\( P_k \) and \( P_\omega \) describe the production of turbulent kinetic energy and specific dissipation rate respectively:

\[
P_k = -u'_i u'_j \frac{\partial \bar{u}_i}{\partial x_j}
\]  

(5.48)

\[
P_\omega = \alpha^* \frac{\omega}{k} P_k
\]  

(5.49)

\( \alpha^* \)'s role is also to reduce the production of \( \omega \) for low Reynolds number.

The analytic forms of the dissipation terms \( Y_k \) and \( Y_\omega \) are given by the following equations:

\[
Y_k = \varphi \beta^* f_{\beta^*} \rho k \omega
\]  

(5.50)

Where

\[
f_{\beta^*} = \begin{cases} 
1 \text{ if } \chi_k \leq 0 \\
1 + 680 \chi_k^2 \text{ if } \chi_k > 0 
\end{cases}
\]  

(5.51)

With

\[
\chi_k = \frac{1}{\omega^2} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
\]  

(5.52)

\[
\beta^* = \beta_i^* (1 + \zeta^* F)
\]  

(5.53)

\[
\beta_i^* = \beta_\omega \frac{4}{15} \left( \frac{Re_k}{Re_\rho} \right)^4 + \frac{\left( \frac{Re_k}{Re_\rho} \right)^4}{1 + \left( \frac{Re_k}{Re_\rho} \right)^4}
\]  

(5.54)
\[ Re_t = \frac{\rho k}{\mu \omega} \]  
\[ Y_\omega = q \beta f_\beta \omega^2 \]  
Where

\[ f_\beta = \frac{1 + 70 \chi_\omega}{1 + 80 \chi_\omega} \]  
\[ \chi_\omega = \left| \frac{\Omega_{ij} \Omega_{ik} S_{kl}}{(\beta_\omega \omega)^3} \right| \]  
\[ \Omega_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \]  
And

\[ \beta = \beta_i (1 - \frac{\beta_i}{\lambda_i} \zeta^* F) \]  

The compressibility correction function F is defined by equation (5.54):

\[ F = \begin{cases} 0 & \text{if } M_t \leq M_{t0} \\ (M_t^2 - M_{t0}^2) & \text{otherwise} \end{cases} \]  

Where

\[ M_{t0}^2 = \frac{2k}{a^2} \]  

The default values of the model constants are given as follow:

\[ \beta_\omega^* = 0.09, \beta_i = 0.072, R_\beta = 8, \zeta^* = 1.5 \]
\[ M_{t0} = 0.25, \sigma_k = 2.0 \text{ and } \sigma_\omega = 2.0 \]  

Another version of the k-\( \omega \) model is known as the Shear Stress Transport (SST) k-\( \omega \) model. This model was built in a way to benefit from the robustness and accuracy of the standard k-\( \omega \) model in the near wall regions and the accuracy of the k-\( \varepsilon \) model in the free stream. This is achieved by incorporating blending functions in order to progressively switch from the k-\( \omega \) model to the k-\( \varepsilon \) model depending on the wall distance. The result is a more accurate model for a larger class of applications (Fluent, 2006).

### 5.4. Turbulent Boundary Layer

This section is based on White's book (2006).

The no-slip condition imposes a null velocity at the wall. Away from the wall, the velocity is equal to the mainstream velocity. Due to the velocity gradient, the particles of fluid are subjected to shear stress forces. This induces vorticity which is then diffused and sometimes amplified by the diffusion and production terms respectively when moving away from the wall. Therefore, vicious damping near the wall greatly affects flow patterns in the mainstream.

Equations related to turbulent boundary layers are non-linear and only a few simple cases can be analytically resolved. For more complex situations, knowledge has rather raised from empirical
considerations. It was showed that the near-wall velocity profile can be accurately described by three distinct regions: the inner layer, the overlap layer and the outer layer. Dimensionless parameters were introduced in order to generalize the equations.

- The dimensionless wall distance:
  \[ y^+ = \frac{u_r y}{v} \]  
  \[ (5.64) \]
  Where \( u_r \) represents the shear stress velocity and is defined as the square root of the ratio of the averaged wall shear stress to the density:
  \[ u_r = \sqrt{\frac{\tau_w}{\rho}} \]  
  \[ (5.65) \]
- The dimensionless velocity:
  \[ u^+ = \frac{u}{u_r} \]  
  \[ (5.66) \]

Very close to the wall, viscous damping dominates the flow. The region is described by the inner layer for which \( y^+ \) ranges from 0 to 30:

- Very close to the wall, for value of \( y^+ \) inferior to 5, the region is referred to as the viscous sublayer. It has been observed that the velocity profile is linear:
  \[ u^+ = y^+ \]  
  \[ (5.67) \]
- When \( y^+ \) ranges from 5 to 30, the region is referred to as buffer sublayer. It constitutes a smooth transition from the viscous sublayer to the overlap layer described by the log law. Spalding (1961) established an mathematical expression to describe the velocity profile:
  \[ y^+ = u^+ + e^{-kB} \left( e^{ku^+} - 1 - ku^+ - \frac{(ku^+)^2}{2} - \frac{(ku^+)^3}{6} \right) \]  
  \[ (5.68) \]

where B and k are two constants, the latter being referred to as the Von Kármán constant. This expression is known as the Spalding’s law of the wall.

The overlap layer is described by a classical relation in turbulence theory: the log-law. In this region, viscous forces progressively decrease against inertia forces. In this region, vorticity is transported from the viscous sublayer to the mainstream. The dimensionless velocity is described by the following equation:

\[ u^+ = \frac{1}{k} \ln(y^+) + B \]  
\[ (5.69) \]

The k-\( \omega \) models are usually not adopted to resolve the flow in the viscous sublayer and buffer layer. In such a case, the boundary layer is modeled using wall functions that connect the viscous sublayer to the log law region. This approach has proven to reduce computational effort, to be robust and fairly accurate when low Reynolds number effects are not significant on the mainstream.

The k-\( \omega \) models incorporate modifications of equations in the inner wall region. When the mesh is fine enough, the near wall region is entirely resolved, including the viscous sublayer. This approach requires additional computational resources but provides a more accurate solution as compared to the wall function approach.
5.5. Stator-rotor cavity flow

The following section is based on the work performed by Childs (2011). Rotating flows are also governed by the equations described above. However, the subject has its own conventions and notations and therefore requires a special attention. As an introduction to more complicated flow in a gas turbine wheelspace, three cases are investigated: the free disc, the closed wheelspace and the stator-rotor cavity with superposed mass flow. These examples represent a good initiation to rotating flow structures. Naturally, a pre-swirl system represents an extremely complex case and cannot be studied theoretically. However, the three cases described above shed new lights on flow characteristics in pre-swirler cavities as well as important dimensionless parameters such as the swirl ratio $\beta$, the rotational Reynolds number $\text{Re}_\phi$, and the turbulent flow parameter $\lambda_t$.

5.5.1. Vortex, vorticity and rotation

According to Childs (2011), a vortex is defined as a region of spinning fluid or concentrated rotation in a flow. Although vortices are sometimes not evident to observe, they can be visible in tornadoes, hurricanes, whirlpools or other natural phenomena. In the simplest definition, vorticity refers to as a spinning of an element of fluid. Although vorticity and rotation seem to be similar, they must be distinguished. Let’s introduce an example of a vortical flow without rotation. In a laminar fully developed uniform flow of a viscous fluid in a pipe, the flow moves in straight lines. However, if a cork is introduced in the flow, it would rotate around its own axis as translating in the pipe. The physical mechanism behind the vorticity is viscosity. At the wall, the fluid velocity is zero due to the no-slip condition, while a small distance from the wall, the fluid has a higher axial velocity. The non-symmetrical distribution of the velocity around the cork therefore induces an unbalanced shear stress, converted into spinning. Another extreme is when a fluid is moving in circular paths without having vorticity. These two examples show that vorticity and rotation are two different notions that are not intuitive.

5.5.2. Rotating flow due to a free disc

![Diagram of rotating disc](image)

Fig. 5.1: The Free Disc, adopted from Childs (2011)

$\Omega$ and $b$ are the angular velocity and outer radius of the rotating disc respectively.
The problem of a laminar steady flow on a free disc was investigated by Kármán (1921) and Cochran (1934). Assuming that the body forces are insignificant, they managed to reduce the Navier-Stokes and continuity equations to a set of differential equations which could be solved analytically. The solution to the free disc problem was investigated after reduction of the characteristic variables to dimensionless parameters. These parameters are described in the equations (5.70):

\[
\begin{align*}
    z^* &= z \sqrt{\frac{\Omega}{\nu}} \\
    u_r^* &= \frac{u_r}{r\Omega} \\
    u_\phi^* &= \frac{u_\phi}{r\Omega} \\
    u_z^* &= \frac{u_z}{\sqrt{\nu\Omega}} \\
    p^* &= \frac{p}{\rho\nu\Omega}
\end{align*}
\]  

(5.70)

where \((u_r, u_\phi, u_z)\) are the components of the velocity in the cylindrical coordinate system, and \(\nu\) is the kinematic viscosity. The dimensionless velocity profiles are presented in Fig. 5.2.

![Fig. 5.2: Velocity profiles for the free disc, adopted from Childs (2011)](image)

Due to the no-slip condition, a boundary layer appears on each side of the rotating disc. Within the boundary layer, the fluid is forced to rotate due to the shearing force from the disc surface. The centrifugal forces caused by the rotation of the disc constrain the fluid to travel in a radially outward path. In order to satisfy the conservation of mass, an axial mass flow takes place from the surrounding towards the disc center. This effect is well-known as the “free disc pumping effect”, and can be observed through the dimensionless axial velocity profile.
To describe the flow regime inside the boundary layers, it is interesting to introduce the local rotational Reynolds number:

\[ Re_{\phi,local} = r^* \times Re_{\phi} = \frac{\rho \Omega r^2}{\mu} \]  

(5.71)

where \( x \) is the dimensionless radius \( r^* = \frac{r}{b} \) and \( Re_{\phi} \) represents the rotational Reynolds number \( (Re_{\phi} = \frac{\rho \Omega b^2}{\mu}) \). \( q \) refers to as the density, \( \mu \) the dynamic viscosity and \( r \) is the distance from the z-axis.

From this equation, it can be seen that the local rotational Reynolds number is proportional to the square of the distance from the z-axis. The flow tends therefore to be laminar near the rotational axis while it becomes turbulent as \( r \) increases. For larger value of the angular velocity, the flow will tend to become turbulent for smaller value of \( r \). The transition will occur when the local rotational Reynolds number attains a value of around \( 2 \times 10^5 \), while the flow is fully turbulent for values above \( 3 \times 10^5 \).

The boundary layer for the laminar flow is commonly defined as the axial distance away from the disc at which the tangential velocity is 1% of the disc speed. From this definition and since the boundary layer thickness is independent on the radius, it comes:

\[ \delta = 5.5 \times \frac{\sqrt{v}}{\Omega} \]  

(5.72)

where \( \delta \) is the laminar boundary layer thickness.

The rotational Reynolds number increases as the radius increases, indicating a transition from laminar to turbulent flow. When turbulent, the flow breaks down into a set of vortices, and the boundary layer becomes larger.

**Mass flow rate:**

It is of interest to calculate the mass flow entrained towards the rotating disc. Among other things, this mass flow will influence the heat transfer on the disc as well as the mass flow required to avoid hot gas ingestion from the main annulus.

Due to the conservation of mass, the mass flow can be calculated at any axial distance from the disc. The mass flow is equal to the axial flow rate into the boundary layer.

\[ m = \int \rho \times u_{z \rightarrow \infty} \, dA = \int_{0}^{b} \rho \times u_{z \rightarrow \infty} \times 2\pi r \, dr = 2.779\rho b^2 \sqrt{\Omega \nu} \]  

(5.73)

Since \( u_{z \rightarrow \infty} = -0.8845 \).

At this stage, it is convenient to introduce a dimensionless mass flow rate:

\[ C_{\mu} = \frac{m}{\mu b} \]  

(5.74)

For the free disc:

\[ C_{\mu} = 2.779 \times \sqrt{Re_{\phi}} \]  

(5.75)

**Moment coefficient:**

Following the same procedure as for the mass flow rate, a moment coefficient is a dimensionless parameter which refers to as the magnitude of the torque due to the viscous drag on one side of the disc:

\[ C_{m} = \frac{T_{q}}{0.5\rho \Omega^2 b^5} \]  

(5.76)

where \( T_{q} \) is the torque due to viscous drag.
By integrating the tangential component of the fluid stress at the disc surface, Childs (2011) has shown that:

\[ Cm = \frac{1.935}{\sqrt{Re\phi}} \]  

(5.77)

### 5.5.3. Enclosed wheelspace

An enclosed wheelspace is composed by a rotating and a stationary disc without flow supply or extraction. As for the free disc example, the no-slip condition at the rotating disc surface causes the apparition of a boundary layer. A small distance from this disc, outside the boundary layer, the flow is still rotating. This rotation is described by the swirl ratio \( \beta \):

\[ \beta = \frac{u_\theta}{r\Omega} \]  

(5.78)

On the stationary disc, there is also a boundary layer due to movement of the fluid around the stationary surface. Due to the no-slip condition at the stationary disc, the swirl ratio of the rotating core is about 0.4. At this stage, it is important to introduce a new dimensionless parameter to describe the influence of the geometry on the flow. The gap ratio \( G \) is defined as the ratio of the wheelspace axial length to the outer radius of the discs.

\[ G = \frac{s}{b} \]  

(5.79)

When the cavity is relatively thin, for small values of \( G \), the boundary layers on either side of the wheelspace tend to merge while for larger value of \( G \), the boundary layers remain separate. When describing these different flow regimes, it is common to refer to the scientific work accomplished by Daily and Nece (1960). They carried out experimental measurements in order to identify the flow regimes in the cavity based on the gap ratio \( G \) and the rotational Reynolds number. They were able to identify four different flow regimes:

- **Regime 1**: For low rotational speeds, the rotational Reynolds number is relatively low which indicates a laminar flow. The boundary layers on either side of the cavity are however relatively thick (inversely proportional to the root square of the rotational speed). If the cavity is relatively thin, the boundary layers will merge. Regime 1 therefore refers to as laminar flow, merged boundary layers. For this case, the moment coefficient is given by:

  \[ Cm = \frac{\pi}{G \ast Re\phi} \]  

(5.80)

- **Regime 2**: For higher rotational speeds, the boundary layers tend to become thinner. If the gap ratio \( G \) is large enough, the boundary layers remain separate. Regime 2 refers to as laminar flow, separate boundary layers. For regime 2, the moment coefficient is given by:

  \[ Cm = \frac{1.85 \ast G^{0.1}}{\sqrt{Re\phi}} \]  

(5.81)

- **Regime 3**: For turbulent flow, the boundary layers are relatively thin. Their fusion will therefore only occur for small cavities. This regime refers to as turbulent flow, merged boundary layers. The moment coefficient is given by:

  \[ Cm = \frac{0.040}{G^{0.167} Re\phi^{0.25}} \]  

(5.82)
• Regime 4: The last case refers to as turbulent flow, separate boundary layers. For this regime, the moment coefficient is given by:

\[
C_m = \frac{0.051 \times G^{0.1}}{Re^{0.2}}
\]  

(5.83)

These four flow regimes can be appropriately distinguished in accordance with the gap ratio and the rotational Reynolds number. This is represented in Fig. 5.3.

Fig. 5.3: The four flow regime, adopted from Childs (2011)

5.5.4. Stator-Rotor cavity with superposed mass flow

Fig. 5.4: Stator-Rotor cavity with superposed mass flow, adopted from Childs (2011)

When the stator-rotor cavity is open, the radial outward flow induced by the rotation of the disc exits the cavity. When the superposed mass flow does not equalize the outflow at the outer radius of the disc, an inflow of fluid will occur on the stator side.
In general, the rotation of the core is adversely affected by the presence of a superposed mass flow. The swirl ratio therefore indicates values below 0.4. The superposed mass flow is balanced by the net outflow at the outer radius, as indicated by equation (5.84):

\( \dot{m}_{\text{superposed}} = \dot{m}_{\text{inflow}} + \dot{m}_{\text{outflow}} \)  

(5.84)

By applying a set of assumptions (these assumptions are not described here but the reader is invited to refer to Childs (2011) for further information), the dimensionless inflow and outflow of fluid are expressed respectively by:

\[ \frac{\dot{m}_{\text{inflow}}}{\mu r} = -r^* \beta 0.8 (r^*^2 \text{Re}_\phi)^0.8 \]  

(5.85)

\[ \frac{\dot{m}_{\text{outflow}}}{\mu r} = r^* 0.13955(1 - \beta)1.6 (r^*^2 \text{Re}_\phi)^0.8 \]  

(5.86)

By dividing equation (5.84) by \( \mu r \),

\[ \frac{C_w \text{Re}_\phi^{-0.8}}{r^*^{2.6}} = 0.13955[(1 - \beta)^1.6 - \beta^{0.8}] \]  

(5.87)

The turbulent flow parameter is expressed by:

\[ \lambda_t = C_w \text{Re}_\phi^{-0.8} \]  

(5.88)

This parameter reflects the relative strength of rotation to throughflow in the cavity.
6. MODELING ASPECTS

Chapter 6 provides further information regarding modeling aspects. More specifically, information regarding geometry assumptions, boundary conditions, turbulence model, and discretization schemes are disclosed to the reader. The mesh quality is also evaluated for the three different designs, and a grid dependency analysis is carried out.

6.1. Geometrical considerations

6.1.1. Pre-swirlers

The pre-swirler designs considered include simple drilled holes directed along the rotating path and other more complex airfoil shaped geometries. The latter design type *a priori* offers more interesting aerodynamic performance as compared to drilled holes. The distinction *a priori* is used here because it is difficult to evaluate how recirculating flows in the stator-rotor cavity can affect the flow structure. Although the flow characteristics inside the pre-swirler can sometimes be predicted in a theoretical way, the flow patterns further away in the cavity are difficult to predict other than by numerical means. Therefore, the study puts highlights on three different designs, two of them being composed by airfoil profiles, and one of them being a simple drilled hole.

The pre-swirler dimensions are extracted from common designs used in gas engines but they will not be disclosed for confidentiality reasons. For convenience, the pre-swirlers will be referred to as Vane-1, Vane-2 and Nozzle throughout the discussion. They are represented through the following pictures.

![Isometric view of Vane-1](image1)

![Top view of Vane-1](image2)

Vane-1 is a complex geometry designed to achieve the best aerodynamic performances. It is composed by airfoil profiles on either side of the geometry in order to reduce pressure losses inside the pre-swirler.
Vane-2 is slightly less complex as compared to vane-1. An airfoil profile is used on the suction side of the geometry while the pressure side is composed by a flat surface. The inlet cross-sectional surface is characterized by a narrowing towards the pre-swirl corps while the outlet surface is characterized by an enlargement towards the stator-rotor cavity. It is \textit{a priori} difficult to draw conclusions regarding the influence of these features as compared to vane-1. The numerical simulations are expected to shed new lights on these features.

The nozzle is composed by two simple drilled holes of different diameters. The nozzle is oriented in the direction of the rotating disc. The inlet diameter is reduced to accelerate the flow towards the tangential direction.

\subsection*{6.1.2. Pre-swirl Systems}

The pre-swirlers were assembled in a common direct-transfer system. The radius of the pre-swirlers, and consequently this of the receiver holes, was set to 327 millimeters. Since a full 360° model would be too demanding in terms of computational effort, the simulations were conducted on partial models using rotational periodicities. The receiver holes were located every 5° in the rotating domain. Symmetrical properties of the different pre-swirlers are however not identical:

\begin{itemize}
  \item Vane-1 is symmetric by group of two vanes. The pre-swirl system is therefore represented by a 8° stationary domain composed by two pre-swirlers and a 10° rotating domain composed by two receiver holes. Fig. 6.7 represents an isometric view of Vane-1.
\end{itemize}
Vane-2 is symmetric by group of one vane. The model is therefore represented by a 10° stationary domain composed by one vane and a 10° rotating domain composed by 2 receiver holes. This is represented in Fig. 6.8.

Nozzle: As for Vane-2, the pre-swirl system is symmetric by group of one nozzle. The numerical model is established on a 6° stationary domain and 5° rotating domain, composed by one nozzle and one receiver hole respectively. This is represented in Fig. 6.9.
To enable comparison among the three models, geometrical dimensions of the inlet cavity, the wheelspace and the receiver holes remain identical for all three designs. These dimensions were defined in order to ensure realistic conditions, close to those found in a gas turbine. These dimensions are described in Table 6.1:

<table>
<thead>
<tr>
<th>Notation</th>
<th>Names</th>
<th>Dimensions</th>
</tr>
</thead>
<tbody>
<tr>
<td>b1</td>
<td>Inlet-Cavity axial length</td>
<td>20 mm</td>
</tr>
<tr>
<td>S1</td>
<td>Inlet-Cavity radial length</td>
<td>10 mm</td>
</tr>
<tr>
<td>b2</td>
<td>Rotor-Stator-cavity axial length</td>
<td>40 mm</td>
</tr>
<tr>
<td>S2</td>
<td>Rotor-Stator-cavity radial length</td>
<td>20 mm</td>
</tr>
<tr>
<td>L</td>
<td>Receiver hole length</td>
<td>40 mm</td>
</tr>
<tr>
<td>D</td>
<td>Receiver hole diameter</td>
<td>12 mm</td>
</tr>
</tbody>
</table>

Table 6.1: Pre-swirl-system dimensions
6.2. Numerical Setup

6.2.1. Boundary conditions

Numerical simulations were conducted using the commercial code CFX 13.0. A specific attention was given to the boundary condition specifications. A total pressure was adopted at the inlet while a static pressure was defined at the outlet. This combination is known to be sensitive to the initial guess. Special care was therefore given to the initial conditions in order to ensure consistent comparisons between the different models.

A combination of mass flow at the inlet/static pressure at the outlet is more appropriate when privileging robustness. However, given the known parameters for the simulations, the actual boundary conditions were more adequate. Table 6.2 provides additional details regarding the boundary conditions adopted to vane-1, vane-2 and the nozzle:

![Diagram of Numerical Setup in CFX 13.0](image)

**Table 6.2: Boundary Conditions**

<table>
<thead>
<tr>
<th>Fluid model</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid</td>
<td>Air Ideal Gas</td>
<td></td>
</tr>
<tr>
<td>Heat transfer model</td>
<td>Total Energy (including viscous work)</td>
<td></td>
</tr>
<tr>
<td>Turbulence Model</td>
<td>Shear Stress Transport (SST)</td>
<td></td>
</tr>
<tr>
<td>Motion of rotating domain</td>
<td>+/- 9500 rpm (the sign depends on the orientation of the pre-swirler)</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Mixing Plane Interface</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Mixing Model</td>
<td>Frozen rotor</td>
<td></td>
</tr>
<tr>
<td>Mesh connection</td>
<td>General Grid Interface</td>
<td></td>
</tr>
<tr>
<td>Location of the mixing Plane</td>
<td>Middle of the cavity</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Rotational periodicity</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh connection</td>
<td>General Grid Interface</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Inlet Boundary Condition</th>
<th>To be defined for each simulation</th>
</tr>
</thead>
</table>

Fig. 6.11: Numerical Setup in CFX 13.0

28
Flow direction: Perpendicular to surface
Turbulence intensity: Medium Intensity (5%)
Total Temperature: 420°C

**Opening Boundary Condition**

<table>
<thead>
<tr>
<th>Opening type</th>
<th>Entrainment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Static Pressure</td>
<td>15 bars</td>
</tr>
<tr>
<td>Turbulence intensity</td>
<td>Zero gradient</td>
</tr>
<tr>
<td>Static temperature</td>
<td>450°C</td>
</tr>
</tbody>
</table>

**Wall Boundary Condition**

<table>
<thead>
<tr>
<th>Heat transfer</th>
<th>Adiabatic</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slip Walls</td>
<td>Top and bottom walls of the stator-rotor cavity</td>
</tr>
<tr>
<td>No-Slip Walls</td>
<td>All the rest</td>
</tr>
</tbody>
</table>

Table 6.2: Boundary Conditions setup

This study aims to draw conclusions regarding the pressure drop influence on pre-swirler’s performance. Therefore, a set of different pressures was adopted at the inlet.

Under steady state operation of the gas turbine, the static pressure drop across the pre-swirler system (from the cavity inlet to receiver hole outlet) is 7 bars for vane-1 and vane-2. The nozzle is part of a larger gas turbine. Its steady state pressure drop is 10 bars. When starting the engine, the pressure drop progressively increases to finally reach the steady state conditions. Therefore, the set of different pressure drops were carefully defined for each model, ranging from 3 bars to 7 and 10 bars for vane-1 and 2, and nozzle respectively. Table 6.3 provides further information regarding the different pressure drops and their corresponding inlet total pressure for all three models:

<table>
<thead>
<tr>
<th>Model</th>
<th>Pressure Drops</th>
<th>Corresponding inlet total pressure</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vane-1</td>
<td>3, 4, 5, 6, 7 bars</td>
<td>18, 19, 20, 21, 22 bars</td>
</tr>
<tr>
<td>Vane-2</td>
<td>3, 4, 5, 6, 7 bars</td>
<td>18, 19, 20, 21, 22 bars</td>
</tr>
<tr>
<td>Nozzle</td>
<td>3, 4, 5, 6, 7, 8, 10 bars</td>
<td>18, 19, 20, 21, 22, 23, 25 bars</td>
</tr>
</tbody>
</table>

Table 6.3: Set of pressure drops and corresponding inlet total pressures

Note that the temperature at the outlet is actually defined for backflows only, which did not occur in any of the simulations since the receiver holes were defined sufficiently long. The actual temperature is calculated by resolving the total energy equation during the solving process.

**6.2.2. Turbulence Model**

A special care was given to the different turbulence models. Such a model should be able capture the full complexity of the flow, from the boundary layer to the mainstream, while ensuring a moderate computational demand. Given the number of simulations (17 simulations for the different pressure drops) to conduct, the Large Eddy Simulation (LES) models or Direct Numerical Simulations (DNS) were not considered. These models are extremely resource consuming, and they provide a level of accuracy that is not required for this study.

Since the realizable k-ε model is not recommended for rotating frames of reference, the choice was reduced to the standard k-ε, RNG k-ε, standard k-ω and SST k-ω model. In addition, according to Javiya et al. (2012), the RNG k-ε model presented discouraging results when applied to 2D pre-swirler numerical models and is therefore not recommended for 3D pre-swirl simulations. This model was sidelined from the discussion.

The standard k-ε, standard k-ω and SST k-ω models are known to be equivalent in terms of computational demand. However, the SST k-ω model presents a heavy advantage over the two other models: by combining the accuracy and robustness of the standard k-ω models in the near wall regions and the accuracy of the k-ε model in the free stream, the SST k-ω model is more likely to capture the full
complexity of the flow, from the boundary layer to the free stream. The choice therefore naturally fell on this turbulence model.

### 6.2.3. Interface options

**General Grid Interface**

When using an interface such as a mixing plane, a translational periodicity or a rotational periodicity, meshes are to be connected along the interface. Two options are available in CFX 13.0. The first one is referred to as direct (one-to-one) connection. This is only suitable when there is a perfect correspondence between the nodes across the interface. When the grids on either side of the interface do not match, a second alternative known as General Grid Interface (GGI) is employed. The calculation is achieved based on an intersection algorithm that enables relative flexibility in terms of grid distribution differences.

**Mixing Models for the mixing plane**

The mixing plane interface is a common approach within numerical simulations to connect a stationary domain to a rotating domain. Three optional mixing models are available in CFX 13.0: frozen rotor, stage and transient rotor-stator. The latter is used for transient simulations and will not be further developed in the following section.

According to Karnahl et al. (2011), a frozen rotor is the least expensive in terms of computational resource. Across the interface, the frame of reference is changed but the orientation of the components remains the same. In other words, the receiver hole position relative to this of the pre-swirler remains constant throughout the simulation.

Stage constitutes an interesting alternative to frozen rotor although this approach is more demanding in terms of computational cost. This method performs a circumferential averaging of the variables through bands of constant radius. This option is particularly useful when accounting for the effect of recirculating flow inside the cavity. The latter option seems therefore more appropriate to capture the flow characteristics inside the cavity. A comparative analysis was carried out between a frozen rotor and a stage option in order to evaluate if computational time should be prioritized over numerical accuracy. The analysis was based on Vane-1 (intermediate mesh), using a mixing plane located at the middle of the cavity. The total pressure at the inlet was set to 22 bars. Special care was given to the initial guess in order to avoid discrepancies due to the model’s sensitivity to the initial conditions.

The swirl ratio variation was first analyzed in the axial direction. The result was extracted using a Matlab code on a line located in the middle of the cavity, from the stationary disc to the rotating disc as shown in Fig. 6.12.
The results are exposed in Fig. 6.13.

On these curves, three regions can be appropriately distinguished. When $z^*$ ranges from 0 to 0.1, the effects of the no-slip conditions at the stationary wall imposes low values of the swirl ratio. Further away in the cavity, when $z^*$ ranges from 0.1 to 0.4, the pre-swirler effects dominate the flow. The maximum swirl ratio is observed at $z^*=0.22$. The no-slip condition at the rotating disc imposes a swirl ratio of 1 when $z^*=1$. The effect of the rotating disc is observed from somewhere around $z^*=0.5$. When using a stage option, the velocity is not continue across the mixing plane due to the averaging algorithm.

A good agreement is observed between the two mixing options. The relative discrepancy ranges from 0.01% to 3.19% with an average of 1.83%. The maximum discrepancy is observed when $z^*=0.32$. 

31
Circumferential averaging of the swirl ratio was also computed on a plane located at $z^*=0.3$ (this location is subject to further analyses in the third part of this report) for both mixing options. This was achieved using radial bands along the radial length, from the bottom of the cavity to the top. The radial band length was set to $r^*=0.01$, which corresponds to the average length of the mesh cells in this direction.

The results are exposed in Fig. 6.15. A good agreement between the two mixing options is observed in the central region, where the effects of the pre-swirler dominate. The relative discrepancy ranges from 0.04% to 3.90%, with a maximal value observed at $r^*=0.82$, in the upper part of the cavity.
It is difficult to draw conclusions regarding the reliability of the mixing options since no experimental data are available for comparison. However, the two models show good agreements in the radial and axial directions. Due to the relative small discrepancy between the mixing options, the decision was rather established based on the computational effort. Since a stage option is more resource demanding, a frozen rotor option was therefore adopted for the simulations.

Location of the mixing plane
Throughout the literature review, it was observed that the mixing plane is often located in the middle of the cavity but this choice is not explained. It is difficult to evaluate how this location can affect the accuracy and robustness of the simulations other than by numerical means. In this paragraph, results from two simulations with different mixing plane locations are exposed and described. These simulations are conducted on vane-1 (intermediate mesh) and the mixing option is set to frozen rotor. The mixing planes are located at one half and three fourth of the cavity, respectively. The inlet total pressure was set to 22 bars for both simulations. Data were extracted on a line located in the middle of the cavity as described in Fig. 6.16.

![Fig. 6.16: Swirl ratio vs. axial length for different locations of the mixing plane](image)

Despite high velocity gradients, there is a good agreement between the two models in the upstream part of the cavity. Differences are however observed when moving away from the pre-swirler, towards the receiver holes. When the mixing plane is located at $z^*=0.75$, the model seems to over-estimate the value of the swirl ratio at the rotating disc since the value is different from 1, despite a high gradient very close to the wall. A finer near-wall treatment in this region would certainly improve the predictions. Without experimental data, it is difficult to conclude which of the two models is the most accurate. On the other hand, instabilities were also induced when using a mixing plane located at 75% of the cavity during simulations. Therefore, our choice naturally fell on a mixing plane located at the middle of the cavity.
6.2.4. Solver Controls

Table 6.4 provides further information regarding solver control settings implemented for the simulations:

<table>
<thead>
<tr>
<th>Accuracy</th>
<th>Convergence control</th>
<th>Others</th>
</tr>
</thead>
<tbody>
<tr>
<td>Advection Scheme</td>
<td>High Resolution</td>
<td></td>
</tr>
<tr>
<td>Turbulence option</td>
<td>Second order</td>
<td></td>
</tr>
<tr>
<td>Residual Type</td>
<td>RMS</td>
<td>Steady State</td>
</tr>
<tr>
<td>Residual Target</td>
<td>0.00001</td>
<td>Double Precision</td>
</tr>
</tbody>
</table>

Spatial discretization of governing equations

The following section is based on the work performed by Jiyuan, Guan Heng and Chaoqun (2008). During a numerical simulation, the transport equations are calculated at a specific point of space and time. The actual values of pressure, temperature, velocity and other variables are evaluated through resolving the governing equations via approximation of the derivatives. In fluid mechanics, this is often achieved via the finite difference method and the finite-volume method, described below. For the sake of example, these methods are here applied on a 2-dimensional hexa-mesh represented in Fig. 6.17.

![Fig. 6.17: 2D hexa-mesh (adopted from Jiyuan, Guan Heng, and Chaoqun (2008))](image)

N, S, E and W stand for North, South, East, and West respectively. A capital letter indicates a value at the center of the cell while a small letter indicates the value at the face.

In the finite difference method, the values of the derivatives are often calculated at the node \((i,j)\) using Taylor series:

\[
\phi(x + \Delta x) = \sum_{i=1}^{n} \frac{\phi^{(i)}(x)}{i!} \Delta x^i + O(\Delta x^{n+1})
\]

(6.1)

The previous expression is \((n+1)\)-order accurate. Since Taylor series can be expanded endlessly, this approach enables a reduction of the truncation error.
The numerical value of the derivative at the node \((i,j)\) can be calculated using the values of the scalar \(\Phi\) at the neighboring nodes. In the x-direction:

\[
\phi_{i+1,j} = \phi_{i,j} + \left(\frac{\partial \phi}{\partial x}\right)_{i,j} \Delta x + O(\Delta x^2) \quad (6.2)
\]

\[
\phi_{i-1,j} = \phi_{i,j} - \left(\frac{\partial \phi}{\partial x}\right)_{i,j} \Delta x + O(\Delta x^2) \quad (6.3)
\]

Equation (6.4) is obtained by subtracting equations (6.2) and (6.3):

\[
\left(\frac{\partial \phi}{\partial x}\right)_{i,j} = \frac{\phi_{i+1,j} - \phi_{i-1,j}}{2\Delta x} + O(\Delta x^2) \quad (6.4)
\]

This finite-difference approximation presented above is referred to a central difference scheme and is second-order accurate. Using the same method, other finite-difference forms can be obtained with a lower or higher order of accuracy.

The finite-volume method calculates the numerical value of the derivative at the center of the cell using Gauss’ divergence theorem. In the x-direction:

\[
\left(\frac{\partial \phi}{\partial x}\right) = \frac{1}{\Delta V} \int \frac{\partial \phi}{\partial x} dV = \frac{1}{\Delta V} \int_A \phi dA_x \approx \frac{1}{\Delta V} \sum_{i=s}^n \phi_i A_i^x \quad (6.5)
\]

Where \(\phi_i\) is the value at the face (i takes the value s,n,e and w), and \(A_i^x\) is the projection of the face surface i in the x-direction. If the mesh is orthogonal, then:

\[
\left(\frac{\partial \phi}{\partial x}\right) = \frac{1}{\Delta V} \left( \phi_e A_e^x - \phi_w A_w^x + \phi_n A_n^x - \phi_s A_s^x \right) = \frac{A}{\Delta V} (\phi_e - \phi_w) = \frac{\phi_e - \phi_w}{\Delta x} \quad (6.6)
\]

Using a central difference scheme, the value at the face can be calculated through the following expressions:

\[
\phi_e = \frac{\phi_p + \phi_E}{2} \quad (6.7)
\]

\[
\phi_w = \frac{\phi_p + \phi_w}{2} \quad (6.8)
\]

Other discretization scheme can be used and will lead to slightly different results. The order of accuracy depends on the scheme. When using the Reynolds Averaged Navier-Stokes approach, an upwind scheme is often adopted for the convective/advective terms to improve the stability of the simulation.

Substituting equation (6.7) and (6.8) into equation (6.6) finally gives the value of the derivative in the x-direction:

\[
\left(\frac{\partial \phi}{\partial x}\right) = \frac{\phi_E - \phi_w}{2\Delta x} \quad (6.9)
\]

According to Jiyuan, Guan Heng, and Chaoqun (2008), the finite-volume method can easily accommodate deformation of the mesh while the finite-difference method demands a high degree of regularity. CFX 13.0 has adopted a finite-volume approach. The user can choose between an upwind (first order) advection scheme and a high resolution advection scheme. The high resolution scheme is a second order
scheme except in region with high gradients. In such region, a first order scheme is adopted to improve the stability of the simulation. The second order advection scheme gives a more accurate result but is also less robust and requires additional computational effort.

In the first place, a first order upwind advection scheme was used until convergence was reached (see convergence target below). A second simulation was then conducted with a high resolution scheme using the results of the previous simulation as the initial condition.

**Numerical Accuracy**

Numerical accuracy should always be estimated when running a CFD simulation. Sources of errors and uncertainties are to be identified and treated separately. In their book, Jiyuan, Guan Heng and Chaoqun (2008) identify three sources of numerical errors:

- **Spatial and time discretization error**: Time discretization is also applied to governing equations when using a transient simulation. In this study, simulations are conducted assuming a steady state. Therefore, time discretization is not further developed. As seen in the previous chapter, space discretization of governing equation always carries a truncation error. This error depends upon the resolution of the mesh and the order of the advection scheme. A mesh dependency analysis is carried out in the next chapter to estimate this error.

- **Round-off error**: Processors perform calculations with a finite number of digits: 8 by default. When using double precision, the number of digits increases to 16. This option is sometimes indispensable to reach a tight convergence.

- **Iteration error**: When using the steady state option, the solver tries to find the steady state solution based on an iteration approach using a time step scheme. To reduce computational effort, this process is however stopped before the steady solution is reached. The iteration error quantifies the divergence between the fully converged steady state solution and the actual output solution.

The numerical accuracy also depends on other factors such as the convergence target or modeling errors (turbulence modeling for instance).

**Convergence target**

Using the finite-volume approach, it can be shown that any transport equation reduces to the form:

\[ a_p \phi_p = \sum_{i=1}^{N} a_i \phi_i + b \]  \hspace{1cm} (6.10)

Where \( N \) is the number of faces bounding the cell, \( a_i \) the neighboring coefficients and \( b \) the source term. In the case of the previous hexa-mesh:

\[ a_p \phi_p = a_S \phi_S + a_N \phi_N + a_E \phi_E + a_W \phi_W + b \]  \hspace{1cm} (6.11)

However, the previous balance is not satisfied at each new iteration step. A notion of residual has therefore been introduced in order to quantify this discrepancy:

\[ R_p = a_p \phi_p - \sum_{i=1}^{N} a_i \phi_i + b \]  \hspace{1cm} (6.12)

Since the residuals are calculated at every cell, it is interesting to estimate the root mean square (RMS) of the residuals, defined for the entire numerical model:

\[ R_{RMS} = \sqrt{\sum R_p^2} \]  \hspace{1cm} (6.13)
This value is a common indicator of convergence and is often used as convergence criterion. Ansys (2009) guide provides good indications regarding the residual level:

- A value of 0.0005 is a loose convergence. It is used when rough estimations are necessary.
- A value of 0.0001 is a good convergence, sufficient for most engineering applications.
- A value of 0.00005 is a tight convergence. This level of convergence is often difficult to reach and is therefore very often used for research purpose.

Some regions of the model can be subject to unsteady behaviors and turbulences. In these regions, the local residuals can remain relatively high although the RMS residual is low. Although the convergence target can be reached, the numerical model is sometimes unable to capture accurately the flow characteristics in these regions. The user can however check the level of local convergence by implementing monitoring points in strategic locations. The values of the pressure, temperature, velocity and others can be displayed all along the simulation, which gives to the user a good estimation of the convergence level despite the RMS value. Proper convergence requires checking if the actual values in these locations are stable when the convergence criterion is reached.

For the sake of this study, two monitoring points were implemented: one right after the pre-swirler outlet and one before the receiver holes. In these regions, velocity and pressure are subject to high gradients. A RMS target of 0.0001 was implemented for the simulations. This target was reached for all simulations and the local values of pressure, velocity and temperature were stable.

### 6.3. Mesh considerations

#### 6.3.1. Mesh types, near wall treatment and mesh quality

**Mesh types**

Mesh generation constitutes a fundamental part of numerical modeling. Poor quality meshes necessary lead to inaccuracy and instability. Commercial codes provide a set of performance indicators to evaluate the mesh quality. These indicators are further explained in the following section but an important distinction should be established before. Meshes are usually divided into two categories: structured mesh and unstructured mesh.

A structured mesh presents a higher degree of regularity. In 3 dimensions, it is composed by hexahedrons. For simple geometries, a structured mesh enables to reach a good level of quality while maintaining a relatively low number of hexahedrons. In addition, users can keep an entire control over the mesh generation process and refinement in specific regions can easily be implemented. However, according to Jiyuan, Guan Heng, and Chaoqun (2008), more complex geometries increase non-orthogonality and skewness of the grid, and the transport equations require additional terms to encompass the modifications. In such a case, computational effort considerably increases and the benefits of having fewer cells vanish.

Unstructured meshes are mainly composed by tetrahedron and can accommodate more complex geometries. From the user’s perspective, a tetra-mesh is easier to generate as the process is automatic. This however adversely affects the degree of controllability. A tetra-mesh is greatly robust which constitutes a heavy advantage over a hexa-mesh.

A third alternative is to use a hybrid mesh: a structured mesh is used for simple geometry parts while an unstructured mesh accommodates more complex regions.

Given the ability of a structured mesh to accommodate simple geometries, it seemed very natural to model the cavity inlet and wheelspace with a hexa-mesh. The pre-swirlers and receiver holes however constitute more complex geometries characterized by high curvatures.
For vane-1 pre-swirl system was modeled using a structured mesh. An O-grid structure was implemented in the receiver holes. This function is particularly useful to reduce the skewness of the cells when meshing a tube. This is represented in Fig. 6.18.

![Fig. 6.18: O-Grid Structure on receiver holes](image)

Vane-1 pre-swirler was modeled using 3 distinct blocks. This was achieved in order to reduce the skewness of the cells close to the pre-swirler outlet. The skewness in this region was particularly high as compared to the rest of the model, but sufficiently low to enable full convergence (as defined by the convergence target).

![Fig. 6.19: Hexa mesh on Vane-1](image)

Vane-2 was modeled using a tetra-mesh. As compared to vane-1, vane-2 is slightly more complex to mesh due to the presence of some triangular sections in the bottom part of the pre-swirler. Attempts were made to model vane-2 with a structured mesh, but this gave birth to region with very poor quality. Our choice therefore naturally fell on a tetra-mesh. Densities were used in regions where smaller cells were required. Fig. 6.20 represents the tetra-mesh applied to the pre-swirler.

The cavity inlet and wheelspace were also modeled using a tetra-mesh despite the clear advantages of using structured mesh in these regions. Such a choice would require the presence of interfaces to ensure adequate intersections between the tetra-mesh and hexa-mesh. Across these interfaces, values of the different variables are extrapolated right before and after the interface. This constitutes a source of inaccuracy, especially in regions characterized by high gradients such as at the pre-swirler outlet. This can be avoided by connecting the tetra-mesh and hexa-mesh in ICEM CFD but this alternative is time consuming.
The nozzle was also entirely modeled using a tetra-mesh. Robustness and rapidity of the mesh process were privileged for this case. At the nozzle inlet/outlet, the mesh quality significantly decreased after implementing appropriate near-wall treatments. When running the simulation, the local residuals remained very high in these regions. This problem was resolved by using a locally finer mesh in order to accommodate the high curvature shape of the nozzle. This is represented at the nozzle inlet in Fig. 6.21.

Near-wall treatment
The no-slip condition significantly affects the flow characteristics in the mainstream due to the resulting boundary layer (White, 2006). In order to capture the complex features of the boundary layer, the grid should be finer near the walls. Without adequate near-wall treatment, the ability of the simulation to converge is significantly reduced. However, a too fine mesh can also significantly increase the
computational cost of the simulation. Near-wall treatment thereby constitutes a fragile equilibrium between computational cost, robustness and ability to capture the complete features in this region. Experience is once again an important asset.

In the case of a hexa-mesh, near wall treatment is implemented using bunching functions in ICEM CFD while a prism layer algorithm is applied on the existing mesh (in the absence of near wall treatment) for a tetra-mesh. The algorithm however significantly reduces the mesh quality in regions of high curvature. Near the walls, the first cell was always located at 0.01 millimeter. This value was established using a rough estimation of the y+ value at the first cell. The growth ratio was set to 1.2. Six prim layers were implemented for the tetra meshes. The y+ value was checked during the post treatment process for a pressure drop of 7 bars for vane-1 and vane-2 and a pressure drop of 10 bars for the nozzle. This corresponds to the greatest pressure drop used for each simulation, involving the highest velocity and therefore constituting the critical case when studying the y+ values. The y+ values are represented in the set of figures below:

**Vane-1**
Evaluation of mesh quality

As previously mentioned, the mesh quality plays a significant role for simulation stability and convergence ability. Thereby, the mesh quality was systematically checked. It can however be extremely complex to keep a certain level of quality after implementing adequate near-wall treatment. Experience is once again indispensable to find the right trade-off.

When using a hexa-mesh, the quality was evaluated via the determinant of the (normalized) Jacobian Matrix and the skewness. The determinant of the Jacobian matrix is an indicator of cell deformation. A value of 1 indicates a perfect cube while a value of 0 indicates a flat hexahedron. Skewness focuses more specifically on the cell angles and indicates regions of high skew.

When using a tetra-mesh, the quality was evaluated based on the aspect ratio. As for the Jacobian Matrix, the aspect ratio is an indicator of deformation. The values are normalized between 0 and 1. Higher values indicate lower deformation.
Fig. 6.23 is a histogram representing the number of cells for which the Jacobian Matrix and skewness fall within the quality range described on the x-axis for vane-1. Values above 0.6 are not represented.

Fig. 6.23: Evaluation of Vane-1 mesh quality

The Jacobian matrix remains relatively high for the entire model, indicating low cell deformation. In the region of the pre-swirler outlet, the skewness was relatively important due to the high curvature of the geometry. However, these cells remain few as compared to the number of cells for the entire model. The simulation robustness and ability to converge were therefore not affected by this region.

In the same manner, the aspect ratio for vane-2 and nozzle is presented via histograms in the following figures:
6.3.2. Mesh dependency study

Given the increasing importance of numerical simulation within the corporate setting, or for research purposes, Celik et al. (2008) has provided a guideline for estimation of discretization error in order to enhance transparency and quality of CFD scientific publications. For further information about this procedure, the publication is referred at the end of this paper.

Estimation of the discretization error assumes to evaluate the error between the discrete solution and the exact solution (Roache, 1997):

\[ E = f_{\text{discrete}} - f_{\text{exact}} = C h^p + \text{HOT} \]  \hspace{1cm} (6.14)

where \( C \) is a constant, \( h \) is a measure of the grid size, \( p \) is the observed order of accuracy (or apparent order of accuracy) and \( \text{HOT} \) stands for higher order terms. The apparent order of accuracy is likely to be lower than the discretization scheme order due to additional uncertainties (wall functions, turbulence modeling, grid quality, round off error, residuals…). A grid convergence study gives an estimation of the observed order of accuracy.

When neglecting the higher order terms in equation (6.14), one can write:

\[ C = \frac{E}{h^p} \]  \hspace{1cm} (6.15)

A mesh is sufficiently refined when lower values of \( h \) do not reduce \( C \). The solution is then said to be in the asymptotic range of convergence.

The grid refinement study was carried out on Vane-1 with a pressure drop of 5 bars. The swirl values are taken on a line located at \( z^*=0.3 \). This location is subject to further studies in the third part of this paper.
\( \phi = \text{Swirl Ratio} \)  
At \( z^* = 0.3 \) and \( r^* = 0.3 \)

| \( N_1, h_1 \) | 7 012 592, 2.18\( \times 10^{-4} \)m |
| \( N_2, h_2 \) | 2 501 288, 3.07\( \times 10^{-4} \)m |
| \( N_3; h_3 \) | 961 398, 4.23\( \times 10^{-4} \)m |

\( r_{31} \)

1.41

\( r_{32} \)

1.38

| \( \phi_1 \) | 0.8049 |
| \( \phi_2 \) | 0.8090 |
| \( \phi_3 \) | 0.8217 |
| \( p \) | 3.68 |
| \( \phi_{21}^{\text{ext}} \) | 0.8009 |
| \( \varepsilon_{31}^j \) | 0.507\% |
| \( \varepsilon_{32}^j \) | 0.499\% |
| \( \text{GC}_{\text{fine}}^{21} \) | 0.621 \% |

Table 6.5: Estimation of discretization error based on ASME Standards

The coarse, intermediate and fine grids have 0.96, 2.5, and 7.0 million cells respectively which correspond to refinement factors of 1.38 and 1.41.

The apparent order of accuracy was calculated using the Richardson extrapolation:

\[
p = \frac{1}{\ln(r_{31})} \left| \ln \left( \frac{\varepsilon_{32}}{\varepsilon_{21}} \right) + q(p) \right| \tag{6.16}
\]

\[
q(p) = \ln \left( \frac{r_{21}^p - s}{r_{32}^p - s} \right) \tag{6.17}
\]

\[
s = \text{sign} \left( \frac{\varepsilon_{32}}{\varepsilon_{21}} \right) \tag{6.18}
\]

\[
\varepsilon_{ij} = \phi_i - \phi_j \tag{6.19}
\]

These equations were applied along interpolated curves (linear interpolation using 25 points along the radial length of the cavity) of the swirl ratio at \( z^* = 0.30 \) and \( \theta = 91.02^\circ \) (middle of the cavity). The values of the observed order of accuracy range from 0.552 to 16.84 with an average of 4.24. High values of \( p \) coupled with negative values of \( \varepsilon_{32}/\varepsilon_{21} \) in some locations indicate oscillatory convergence. However, the relative errors in these locations remain low which reduces the significance of the oscillations on the numerical accuracy. Fig. 6.26 represents the radial swirl ratio profile with relative errors (relative to fine grid) using the intermediate grid. Although a finer mesh was applied in the central region of the stator-rotor cavity, this location is affected by important variations which induce larger relative errors. A finer mesh would give better agreement but would also require additional computational efforts.
Fig. 6.26: Swirl ratio profile along a line located at $z^*=0.30$ and $\theta=91.02^\circ$ with relative errors

Circumferential averaging of the swirl ratio was also computed on a plane located at $z^*=0.3$ for the coarse, intermediate and fine grids. This was achieved using 0.05 ($r^*$, dimensionless) wide radial band along the radial length. The results are presented in Fig. 6.27. Oscillatory convergence can be observed for values of $r^*$ ranging from 0.35 to 0.60. The graph shows a good agreement between the intermediate and fine grids.

Fig. 6.27: Circumferentially averaged swirl ratio at $z^*=0.30$
The value of the grid size $h$ computed for the intermediate mesh was taken as the minimum value that should be used for Vane-2 and nozzle. This value is believed to engender the same numerical accuracy for the other models although one can argue that a tetra-mesh would not react in the same manner as a hexa-mesh when grid refinement is applied.
7. **ANALYSIS OF RESULTS**

The main function of the pre-swirler is to efficiently supply cooling air to critical components exposed to high thermal loads. Swirling the air inside the rotor-stator cavity induces a reduction in the total relative temperature (in the rotating frame of reference), together with a smoother transport of air into the receiver holes. Cooling will be all the more efficient as the total relative temperature is reduced, and the receiver hole mass flow is augmented. The performance of the pre-swirlers can therefore be evaluated with respect to two indicators: the dimensionless pre-swirl effectiveness is an indicator of total temperature reduction while the discharge coefficient reflects the pre-swirler performance in terms of delivered mass flow. The following chapter provides theoretical insights on these indicators, as well as results extracted from the CFD simulations.

### 7.1. Dimensionless pre-swirl effectiveness

#### 7.1.1. Theoretical considerations

The total enthalpy is defined as follow:

\[ h_0 = h + \frac{u^2}{2} \tag{7.1} \]

where the subscript 0 indicates a total quantity. Assuming that air behaves as an ideal gas,

\[ h_0 - h = C_p (T_0 - T) \tag{7.2} \]

Substituting equation (7.1) into equation (7.2) gives the expression of the total temperature for ideal gases:

\[ T_0 = T + \frac{u^2}{2C_p} \tag{7.3} \]

The total temperature is therefore expressed as a function of the static temperature T and its dynamic counterpart \( \frac{u^2}{2C_p} \).

![Fig. 7.1: Definition of surfaces](image)

The reduction in total relative temperature over the pre-swirl cavity is evaluated against a reference temperature 2, taken as the total temperature right after the pre-swirler (see Fig. 7.1).
The total temperature in the stationary frame at the pre-swirl outlet is given by:

\[ T_{0,2} = T_2 + \frac{u_2^2}{2C_p} \]  \hfill (7.4)

The total temperature in the rotating frame of reference at the receiver hole inlet is expressed by:

\[ T_{0,3,\text{rot}} = T_3 + \frac{u_{3,\text{rot}}^2}{2C_p} \]  \hfill (7.5)

The velocity \( u_{3,\text{rot}} \) in the rotating frame of reference can be related to the velocity in the stationary frame:

\[ u_{3,\text{rot}} = u_3 - \Omega \times r \]  \hfill (7.6)

The subscript rot indicates a velocity in the rotating frame of reference.

The reduction of the total relative temperature is calculated through subtracting equation (7.5) to (7.4):

\[ T_{0,3,\text{rot}} - T_{0,2} = \left( T_3 + \frac{u_{3,\text{rot}}^2}{2C_p} \right) - \left( T_2 + \frac{u_2^2}{2C_p} \right) \]

\[ = (T_3 - T_2) + \frac{u_{3,\text{rot}}^2 - u_2^2}{2C_p} \]  \hfill (7.7)

Without pre-swirler, the total temperature at the receiver hole inlet would be higher than the total temperature at the pre-swirl outlet. This is due to the work delivered by the rotating disk to bring air to the same angular velocity as the receiver hole. The dimensionless pre-swirl effectiveness is defined by the ratio of enthalpy reduction to rotational kinetic energy:

\[ \Theta = \frac{C_p |T_{0,3,\text{rot}} - T_{0,2}|}{\frac{1}{2} r^2 \Omega^2} \]  \hfill (7.8)

Note that the dimensionless pre-swirl effectiveness is equal to 1 when the swirl ratio is identically equal to 1 at surface 3.

7.1.2. Analysis of results

The first analysis consisted in studying the variations of the total relative temperature throughout the cavity for different pressure drops. This analysis was carried out on vane-1. The values of the total relative temperature were extracted on a slice line located at the middle of the cavity, from the stationary disc to the rotating disc (as shown in Fig. 6.12). The different pressure drops are described by the pressure ratio \( \pi \):

\[ \pi = \frac{p_{1,0}}{p_2} \]  \hfill (7.9)

Where \( p_{1,0} \) and \( p_2 \) represent the total pressure at the pre-swirl system inlet and the static pressure at surface 2, respectively.
The results are exposed in Fig. 7.2.

Near the stationary disc, the total relative temperature remains high to finally quickly drop as rotational effects take place. It can be observed that the swirl ratio increases as the pressure ratio increases. As the swirl ratio comes closer to 1, the total relative temperature decreases. A more efficient cooling is therefore achieved when $\pi=1.4157$.

In a second analysis, the dimensionless pre-swirl effectiveness was plotted for the three designs against the pressure ratio $\pi$. The values of the pressures, temperatures and velocities were extracted and averaged over the surface 2 and 3 as described in Fig. 7.1. The results are exposed in Fig. 7.3.
At identical pressure ratio, vane-1 offers the most promising performance, followed by vane-2 and finally the nozzle. Under steady state operation of the gas turbine, the dimensionless pre-swirl effectiveness is of 96% and 94% for vane-1 and vane-2 respectively. When the pressure ratio ranges above 1.48, the dimensionless pre-swirl effectiveness is above 1 for nozzle. The tangential air velocity at the receiver hole inlet is actually higher than the rotational speed of the disc. The energy transferred to air through swirling is higher than the kinetic energy of the disc. For the sake of this analysis, Fig. 7.4, represents the axial averaged values of the swirl ratio for a pressure ratio of 1.6276 for the nozzle. The pressure ratio corresponds to the steady state pressure drop (10 bars for the nozzle). Close to the rotating disc, the swirl ratio is around 1.1.

![Swirl ratio vs. axial length for the nozzle – π=1.6276](image.png)
7.2. Discharge coefficient

7.2.1. Theoretical considerations

The concept of entropy is mathematically defined by the differential form of the second law of thermodynamics:

\[ Tds = dh - \frac{dp}{\rho} \quad (7.10) \]

When the process is adiabatic reversible, the creation of entropy is null. Such a process is referred to as isentropic and describes an ideal reaction from state 1 to 2. Hence, for an isentropic process, \( ds = 0 \):

\[ dh = \frac{dp}{\rho} \quad (7.11) \]

Assuming that air behaves as an ideal gas and integrating equation (7.11) from state 1 to 2 will lead to a more familiar expression:

\[ \left( \frac{T_{2,is}}{T_{0,1}} \right)^{\frac{\gamma - 1}{\gamma}} = \left( \frac{p_2}{p_{0,1}} \right) \quad (7.12) \]

where \( \gamma \) represents the ratio of specific heats.

Friction, contraction and expansion inside the pre-swirler adversely affect the static pressure driving the flow. When compared to isentropic conditions, the actual mass flow is therefore reduced. The discharge coefficient is naturally defined as the ratio of actual mass flow to isentropic mass flow. This coefficient can be derived from two different methods.

The discharge coefficient is calculated between two surfaces, referred to as 1 and 2 in this report. Surface 1 is defined as the pre-swirler system inlet, while surface 2 is defined at the pre-swirler outlet, as described in Fig. 7.6. Note that surface 2 will be re-defined more specifically for each pre-swirler system.
The first approach is based on the definition of the isentropic efficiency:

\[
\eta = \frac{\Delta h_0}{\Delta h_{0,is}} = \frac{h_{0,1} - h_2}{h_{0,1} - h_{2,is}}
\]  

(7.13)

where the subscript \(is\) refers to isentropic conditions.

Using Fig. 7.5 leads to:

\[
\eta = \frac{u_2^2}{u_{2,is}^2}
\]  

(7.14)

Assuming an ideal gas behavior, the isentropic efficiency can also be expressed as a function of the temperatures:

\[
\eta = \frac{T_{0,1} - T_2}{T_{0,1} - T_{2,is}}
\]  

(7.15)

As previously mentioned, the discharge coefficient is defined as the ratio of actual mass flow to isentropic mass flow:

\[
C_d = \frac{\dot{m}_2}{\dot{m}_{2,is}} = \frac{q_2}{q_{2,is}} \frac{u_2}{u_{2,is}} = \sqrt{\eta} \frac{T_{2,is}}{T_2}
\]  

(7.16)

and according to equation (7.12):

\[
C_d = \sqrt{\eta} \frac{T_{0,1}}{T_2} \left( \frac{p_2}{p_{0,1}} \right)^{\frac{\gamma - 1}{\gamma}}
\]  

(7.17)

Substituting equation (7.12) into equation (7.15) gives a relation for the ratio of total to static temperature:

\[
\frac{T_{0,1}}{T_2} = \frac{1}{1 - \eta \left( 1 - \left( \frac{p_2}{p_{0,1}} \right)^{\frac{\gamma - 1}{\gamma}} \right)}
\]  

(7.18)

Combining equation (7.17) and (7.18) finally leads to equation (7.19):

\[
C_d = \frac{\sqrt{\eta}}{\eta + (1 - \eta) \left( \frac{p_{0,1}}{p_2} \right)^{\frac{\gamma - 1}{\gamma}}}
\]  

(7.19)

In the second approach, the discharge coefficient is expressed as a function of the actual mass flow:

\[
C_d = \frac{\dot{m}_2}{q_{2,is} u_{2,is} A_2} = \frac{\dot{m}_2}{\frac{p_2}{RT_{2,is}} \sqrt{2cp(T_0 - T_{2,is})}}
\]  

(7.20)
Substituting equation (7.12) into the previous equation and re-arranging the terms leads to the second result:

\[ C_d = \frac{m_2}{A_z p_{0,1} \sqrt{\frac{2\gamma}{\gamma - 1}} \left( \frac{p_z}{p_{0,1}} \right)^{\gamma} - \left( \frac{p_z}{p_{0,1}} \right)^{\gamma+1}} \]  

(7.21)

### 7.2.2. Analysis of results

When calculating the discharge coefficients, surface 2 needed to be redefined in a consistent way to enable comparison. Surface 1 was defined as the inlet surface for each of the pre-swirl system. Pressure and temperature at this surface were fixed by the inlet boundary condition. Surface 2 was defined as the surface characterized by the minimum cross sectional area inside the pre-swirler. When several surfaces fulfilled this condition, the surface closest to the pre-swirler outlet was chosen. These surfaces are represented in Fig. 7.7.

![Fig. 7.7: Location of surface 2 on the three models](image)

In the previous chapter, two approaches were described to calculate the discharge coefficient. The first approach is based on the definition of the isentropic efficiency, while the second approach is simply the ratio of the actual mass flow to the isentropic mass flow. The isentropic mass flow is entirely defined by the pressure and temperature conditions and was therefore calculated by averaging the values of these variables on the considered surfaces. The actual mass flow was extracted from the simulation in the post treatment process. In the same way, the isentropic efficiency was computed using the values of the averaged temperatures and pressures on the surfaces. These two methods naturally led to the same discharge coefficient.
The values of the discharge coefficient and isentropic efficiency were respectively plotted against the pressure ratio $\pi$. The results are exposed in Fig. 7.8 and Fig. 7.9 respectively.

![Fig. 7.8: Discharge Coefficient as a function of the pressure ratio](image1)

![Fig. 7.9: Isentropic Efficiency as a function of the pressure ratio](image2)

It is observed that the discharge coefficient remains relatively insensible to the pressure drop variations. Vane-1 and vane-2 present very similar performance. Their discharge coefficients vary from 96.7\% to 97.4\% and 96.9\% to 97.2\% respectively. As expected, the aerodynamic performance of the nozzle is slightly lower. Its discharge coefficient ranges from 90.3\% to 91.6\%.

### 7.2.3. Comparison with experimental data

In another project, experiments were carried out on vane-1 and the nozzle in order to evaluate the influence of the pressure ratio on the discharge coefficient. The experimental data covers a wider range of pressure ratios as compared to the numerical analysis. The analysis however focused on the pressure ratios covered by both the numerical and experimental analyses. The results for vane-1 are exposed in Fig. 7.10.
The averaged relative error between the experimental and numerical analyses is 5.5%. This discrepancy can be explained through two reasons: For the experiments, air from the pre-swirler was discharged into an open cavity at the atmospheric pressure. The experimental discharge coefficient was computed between the pre-swirler inlet and the cavity. The numerical discharge coefficient was for its part calculated between the pre-swirler inlet and the surface characterized by the minimal cross sectional area (see Fig. 7.7). Therefore, the experimental discharge coefficient additionally accounts for the losses in the final section of the pre-swirler and at the pre-swirler outlet. The experimental discharge coefficient is therefore expected to be lower than the numerical one. Also, the experimental uncertainty was evaluated at 2%.

In the same manner, experimental data was compared to numerical data for the nozzle. This is achieved through Fig. 7.11.

Air from the nozzle was also discharged into an open cavity at atmospheric pressure. The averaged relative difference between the numerical and experimental analysis is of 5.7%. This discrepancy can equally be explained by the aforementioned reasons.


7.3. Swirl ratio analysis

This analysis aimed to agree on an averaged value of the swirl ratio that represents the effects of the pre-swirler inside the cavity. This value is used as an indicator of the tangential velocity for a one dimensional flow simulator. Since the stator-rotor cavity dimensions were standardized into a common system for all three models, the swirl ratio indicator was to encompass the effects of the pre-swirler, the stationary disc, and the rotating disc but reduce the effects of the bottom and top surfaces of the cavity. These surfaces indeed represent a source of inaccuracy as compared to the actual cavity layouts, partially open cavities traversed by radial flows. This inaccuracy was reduced by first setting these two surfaces as free-slip walls in the pre-treatment process and then by choosing an appropriate location when extracting the values of the swirl ratio during the post-treatment process.

7.3.1. Vane-1

As previously mentioned, the first step in this approach was to determinate the region where the swirl ratio values should be extracted in accordance to the flow simulator requirements. For vane-1, five simulations with different pre-swirl pressure drops were carried out. The following considerations were applied on a pressure drop of 7 bars, which corresponds to the steady state operation of the gas turbine. The swirl ratio was first analyzed in the radial direction on different axial planes. As shown in Fig. 7.12, the pre-swirler footprint is clearly visible close to the stationary disc. Along the axial length, this footprint spreads radially, although the pre-swirler effects remain relatively predominant in the central radial region of the cavity.

![Surface plot of the swirl ratio on a plane located at z*=0.3 (r* ranging from 0.3 to 0.7).](image)

Fig. 7.12: Analysis of the tangential velocity in the axial direction

While the bottom and top parts of the cavity indicate a stable tangential velocity, the central region is indeed characterized by high tangential velocity gradients. These variations can be appreciated via a surface plot of the swirl ratio on a plane located at z*=0.3 (r* ranging from 0.3 to 0.7).
As previously mentioned the averaged swirl ratio should encompass the effects of the pre-swirler and the stationary and rotating discs while reducing the effects of the bottom and top surfaces of the wheelspace. Therefore, the values of the swirl ratio were only extracted on a central radial band of 12 millimeter wide, from $r^*=0.3$ to $r^*=0.7$ as shown in Fig. 7.14 (for an axial plane located at $z^*=0.3$).

The second step in this procedure consisted in studying the swirl ratio in the axial direction. To do so, values of the swirl ratio where extracted throughout the central radial band from the stationary disc to the rotating disc. Via a Matlab procedure, these values were interpolated (3D linear interpolation using...
cylindrical coordinates) on planes located every 0.02 millimeter, which corresponds to the average mesh size along the axial length of the cavity. Interpolated values of the swirl ratio in the central radial band for every plane were averaged and plotted against the dimensionless axial length as shown in Fig. 7.15.

On the previous graph, three regions can be appropriately distinguished. Close to the stationary disc, the swirl ratio remains relatively low due to the no-slip condition at the wall. On the opposite side of the cavity, the rotating disc effects dominate the flow. Note that the average swirl ratio at the rotating disc is not equal to 1 due to the presence of the receiver holes. Indeed, the tangential velocity is slightly lower at the receiver holes. In the averaging process, this will adversely affect the averaged swirl ratio. When moving away from the pre-swirler outlets, the effects of the stationary disc progressively fade against the high tangential velocity flux delivered by the pre-swirlers. The maximum value of the swirl ratio is reached at $z^*=0.22$. The swirl ratio remains relatively high despite a slight drop from $z^*=0.3$ to $z^*=0.82$.

For the sake of the flow simulator, the swirl ratio was further investigated at $z^*=0.3$ for all pressure drops. In this region, the pre-swirler effects are predominant, while recirculating flow still plays a minor role.
The first approach consisted in studying the swirl ratio in the tangential direction. The central radial band was divided into 20 sections of constant tangential angle as described in Fig. 7.16.

Fig. 7.16: Tangential Averaging Process

Average values of the swirl ratio was computed for each section and plotted against the tangential direction. This procedure was applied to the 5 pressure drops. For the steady state case (Pressure drop of 7 bars for Vane-1), the standard deviation in every tangential section was also computed. The tangential direction is a dimensionless angle varying from 0 to 1. These values respectively refer to the right and left extremities of the cavity as presented in Fig. 7.16. The results are exposed in Fig. 7.17.

Fig. 7.17: Swirl Ratio versus tangential coordinate for different pressure drops
In a second approach, averaged values of the swirl ratio over the entire band was calculated and plotted against $\pi$. Since five pressure drops, and therefore 5 simulations were carried out, this graph is composed by 5 points, each of them corresponding to a specific pressure drop. The results are exposed in Fig. 7.18.

![Graph showing averaged swirl ratio as a function of the pressure ratio](image)

**Fig. 7.18: Averaged swirl ratio as a function of the pressure ratio**

### 7.3.2. Vane-2

The same procedure was applied on vane-2. Values of the swirl ratio were extracted on a 12 millimeter central band. The axial variations are exposed in Fig. 7.19 for a pressure drop of 7 bars.

![Graph showing averaged swirl ratio vs. axial length for vane-2](image)

**Fig. 7.19: Averaged swirl ratio vs. axial length for vane-2**
The same behavior is observed along the axial direction for vane-1 and vane-2. The values of the swirl ratio are however slightly greater for vane-2 due to a smaller cross sectional area of the pre-swirl outlet. The tangential velocity at the pre-swirl outlet is therefore naturally greater. Due to a globally higher tangential velocity, the effects of the recirculating flow on the pre-swirler jet are relatively lower. When r* ranges from 0.4 to 0.7, the swirl ratio therefore remains constant as compared to vane-1.

Our attention focused more specifically on the plane located at z*=0.4, where the effects of the pre-swirler still overcomes the flow while recirculating effects remain low. On this plane, the swirl ratio was analyzed in the tangential direction through averaged values. For a pressure ratio of 1.4445, the error bars represents the standard deviation in each tangential band. The results are exposed in Fig. 7.20.

![Swirl Ratio versus tangential coordinate for different pressure drops](image)
For each of the pressure ratio, and on the plane located at $z^*=0.4$, the swirl ratio was averaged on the central radial band and plotted against the pressure ratio. The result can be observed in Fig. 7.21.

![Fig. 7.21: Averaged swirl ratio as a function of the pressure ratio](image)

### 7.3.3. Nozzle

As compared to vane-1 and vane-2, the nozzle is significantly smaller, therefore inducing a smaller footprint. As a consequence, the radial band width was reduced to 8 millimeters. The values of the swirl ratio were extracted for $r^*$ varying from 0.4 to 0.6. The results are exposed in Fig. 7.22 for a pressure drop of 7 bars.

![Fig. 7.22: Averaged swirl ratio vs. axial length for nozzle](image)
Despite a smaller cross-sectional area at the nozzle outlet, the swirl ratio is lower as compared to vane-1 and vane-2 due to lower aerodynamic performance. As shown in the previous chapter, the discharge coefficient is lower for the nozzle, therefore inducing a lower mass flow as compared to the isentropic case. Close to the rotating disc, recirculating flow with high tangential velocity mixes with the nozzle mainstream. The swirl ratio therefore increased to reach a maximum value at z*=0.89. This is very certainly due to an increased influence of the recirculating flows caused by a lower tangential velocity. The plane of interest is located at z*=0.3 and tangential variations of the swirl ratio are exposed in Fig. 7.23 for different pressure drops. As for the previous graphs, the errors bars represent the standard deviation of the averaged swirl ratio in each tangential section for a steady state pressure ratio, 1.6276.

Fig. 7.23: Swirl Ratio versus tangential coordinate for different pressure drops
On this plane, the swirl ratio was averaged on the central radial band and plotted against the pressure ratio. The result can be observed in Fig. 7.24.

![Graph showing averaged swirl ratio as a function of pressure ratio.](image)

**Fig. 7.24:** Averaged swirl ratio as a function of the pressure ratio

### 7.3.4. Comparative Analysis

The swirl ratio was plotted against the pressure ratio for the three designs in order to enable comparison. This is represented in Fig. 7.25:

![Graph showing comparative analysis of swirl ratio for three designs.](image)

**Fig. 7.25:** Comparative analysis of the swirl ratio for the three designs
Vane-1 and vane-2 offer very similar performance, with a slight advantage for vane-2 at higher pressure ratio. Due to a lower discharge coefficient, the values of the swirl ratio for the nozzle are always located below those of vane-1 and vane-2. Therefore, to reach a swirl ratio of 1, the nozzle requires additional flow of air, which reduces the overall engine performance. A swirl ratio of 1 is reached when \( \pi \) is equal to 1.390, 1.405 and 1.470 for vane-2, vane-1 and nozzle respectively.

It is also interesting to note that the relative differences between each of the curves remain relatively constant regardless of the pressure ratio. This is explained by a constant discharge coefficient for the three designs.

### 7.4. Influence of the through-flow

#### 7.4.1. Numerical aspects

The actual stator-rotor cavity in the SAS of the gas turbine is a partially open cavity traversed by through-flows. In the previous analyses, the through-flow was neglected and the cavity sealed. It is difficult to predict how this assumption influences the different results exposed above other than by numerical means. Thereby, a new analysis was carried out on the three designs in order to account for the effects of through-flow. The through-flow was introduced on the steady state operation of the preswirlers, corresponding to a pressure drop of 7 bars, for vane-1 and 2, and 10 bars for the nozzle. Fig. 7.26 represents the numerical model for vane-1.

![Fig. 7.26: Introduction of through-flow for vane-1](image-url)
### Inlet

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flow</td>
<td>To be defined for each design</td>
</tr>
<tr>
<td>Flow direction</td>
<td>Perpendicular to surface</td>
</tr>
<tr>
<td>Turbulence intensity</td>
<td>Zero Gradient</td>
</tr>
<tr>
<td>Total Temperature</td>
<td>420°C</td>
</tr>
</tbody>
</table>

### Through-flow inlet

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flow</td>
<td>10% of the mass flow inlet</td>
</tr>
<tr>
<td>Flow direction</td>
<td>Perpendicular to surface</td>
</tr>
<tr>
<td>Turbulence intensity</td>
<td>Zero Gradient</td>
</tr>
<tr>
<td>Total Temperature</td>
<td>420°C</td>
</tr>
</tbody>
</table>

### Through-flow outlet

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flow</td>
<td>10% of the mass flow inlet</td>
</tr>
</tbody>
</table>

Table 7.1: Through-flow Boundary conditions setup

The convergence target was set to 0.00001 on the RMS values of the residuals. The simulations have shown instabilities when using a total pressure at the inlet. A mass flow boundary condition was therefore adopted. The actual values of the inlet mass flows were extracted from the results of the simulations without through-flow when using a pressure drop corresponding to the steady state operation of the gas turbine. The through-inflow and outflow were set identical as being 10% of the pre-swirler inflow. The different mass flows are exposed in the following table:

#### Vane-1

<table>
<thead>
<tr>
<th>Mass Flow Type</th>
<th>Mass Flow Rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-swirler mass</td>
<td>0.2142</td>
</tr>
<tr>
<td>Through-inflow</td>
<td>0.0200</td>
</tr>
<tr>
<td>Through-outflow</td>
<td>0.0200</td>
</tr>
</tbody>
</table>

#### Vane-2

<table>
<thead>
<tr>
<th>Mass Flow Type</th>
<th>Mass Flow Rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-swirler mass</td>
<td>0.1883</td>
</tr>
<tr>
<td>Through-inflow</td>
<td>0.0190</td>
</tr>
<tr>
<td>Through-outflow</td>
<td>0.0190</td>
</tr>
</tbody>
</table>

#### Nozzle

<table>
<thead>
<tr>
<th>Mass Flow Type</th>
<th>Mass Flow Rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-swirler mass</td>
<td>0.1010</td>
</tr>
<tr>
<td>Through-inflow</td>
<td>0.0100</td>
</tr>
<tr>
<td>Through-outflow</td>
<td>0.0100</td>
</tr>
</tbody>
</table>

Table 7.2: Mass flows for the different designs

#### 7.4.2. Swirl ratio analysis

The axial values of the swirl ratio on a line located at the middle of the cavity, from the stationary to the rotating disc, were plotted against the axial dimensionless length. For all three pre-swirl systems, it was observed that the no-slip condition at the rotating disc was not respected since the swirl ratio value was different from 1 (see Fig. 7.27). An attempt was made on vane-2 to improve the model response by refining the near-wall treatment in this region. Thereby, a total of 12 prism layers were implemented with a fist cell located at 0.000005 meter. However, the model response remained identical. The results are exposed at such.
As for the previous analysis, the values of the swirl ratio were averaged over a central radial band and plotted against the axial dimensionless length. The results are exposed in the following figures.

Fig. 7.27: Swirl ratio vs. axial length for vane-2

Fig. 7.28: Averaged swirl ratio vs. axial length for vane-1
The introduction of the through-flow in the numerical models obviously reduces the values of the axial averaged swirl ratio. In Fig. 7.28 (vane-1), the relative difference between the two curves ranges from 8.13% to 15.81%, with an average value of 10.47%. For vane-2, a good correspondence is observed close to the stationary disc. The relative difference between the curves progressively increases when moving towards the rotating disc. The values range from 1.22% to 14.92%, with an average value of 4.20%. Nozzle is the most affected by the introduction of the through-flow. The relative difference ranges from 11.48% to 27.99%, with a mean value of 23.23%. It is important to note that the through-flow was introduced on a pressure drop of 7 for vane-1 and 2, and 10 bars for the nozzle. It can therefore be observed that at higher pressure drop, the introduction of a through-flow further affects the values of the axial swirl ratio.

In the next set of figures, the averaged swirl ratio is plotted against the tangential direction following the same procedure as described in section \textit{Erreur! Source du renvoi introuvable.}. The values were extracted on planes located at $z^*=0.3$ and $z^*=0.4$ for vane-1 and the nozzle, and vane-2 respectively. This was achieved using the models with and without through-flow.

![Fig. 7.30: Swirl Ratio vs. tangential coordinate for vane-1](image-url)
With through-flow, the shapes of the curves above differ from the cases without through-flow. This is explained by the apparition of new vortex structures due to the introduction of additional air flows. These vortices mix with the pre-swirler mainstream and affect the tangential component of the velocity. As presumed from the axial analysis of the swirl ratio, vane-2 is the least affected by the introduction of the through-flow in the numerical model. The relative difference between the two curves remains low regardless of the tangential coordinate. In some regions of the cavity, the values of the swirl ratio for vane-1 are greatly reduced due to strong vortex structures. This induces a greater overall reduction of the swirl ratio as compared to vane-2. Finally, the nozzle is the most affected by the introduction of through-flow. The values of the swirl ratio always remain lower with through-flow, regardless of the tangential coordinate.

7.4.3. Pressure drop Analysis

The discharge coefficient is expected to remain the same, with or without through-flow, and is therefore not considered in this section. Only one pressure drop was considered for each of the three designs when analyzing the influence of the through-flow. It is therefore impossible to draw an entire graph of the dimensionless pre-swirl effectiveness, or the swirl ratio against the pressure ratio. However, the values of the aforementioned indicators were placed on the graphs as a premise of a future study in which the through-flow is considered for all pressure drops. As described in section Erreur ! Source du renvoi introuvable., the values of the swirl ratio were averaged on a central band located at $z^*=0.3$ and $z^*=0.4$ for vane-1 and the nozzle, and vane-2 respectively. The results are exposed in the set of figures below.
As expected from the previous analysis, vane-1 and vane-2 are slightly affected by the introduction of through-flows while the value of the averaged swirl ratio is greatly reduced for the nozzle. It is also interesting to note that the pressure ratio $\pi$ is slightly different with and without through-flow. This is due to the non-sealing of the cavity after introduction of the through-flow.
In a second analysis, the influence of the through-flow on the dimensionless pre-swirl effectiveness was evaluated. The result is exposed in Fig. 7.34.

These results can be presumed from the previous analysis. None of the three designs transfers sufficient energy to the tangential component of the velocity in order to equalize the kinetic energy of the rotating disc. The greatest influence of the through-flow is once again and as expected observed on nozzle. Despite a greater pressure ratio as compared to the other models, the dimensionless pre-swirl effectiveness is still lower than 1. This is due to a drop in the value of the swirl ratio after introduction of the through-flow. The values of the dimensionless pre-swirl effectiveness are of 77.5%, 73.1% and 94.5% for vane-1, vane-2 and nozzle respectively, after introduction of the through-flow and under steady state operation.
8. CONCLUSIONS AND FUTURE WORK

The objective of the present report was to analyze how the pre-swirler pressure drop influences different parameters such as the dimensionless pre-swirl effectiveness, the discharge coefficient and the swirl ratio. The analysis was carried out on three different pre-swirl designs, two guide vanes and a nozzle. The vanes have shown similar response to the pressure drop variations:

- Their discharge coefficients remain relatively constant, 97%, regardless of the pressure ratio.
- The swirl ratio varies relatively linearly with the pressure ratio for both cases. For the same pressure ratio, vane-2 transfers air with a slightly higher tangential velocity, therefore providing more interesting performance. The performances of the two designs are however very similar. The values range from 0.704 to 1.013 and 0.703 to 1.023 for vane-1 and vane-2, respectively.
- Although vane-2 is slightly superior in terms of swirl ratio, vane-1 has actually shown to be superior in terms of dimensionless pre-swirl effectiveness. When moving toward the receiver holes, the swirl ratio of vane-1 is closer to 1 than that of vane-2. Therefore, the energy diffused through swirling of air into the tangential velocity component represents 96% of the rotational energy of the disc for vane-1, against 94% for vane-2 under steady state operation.

Due to different aerodynamic features, the nozzle performance stands behind those of the guide vanes:

- The discharge coefficient remains relatively insensitive to the pressure drop variations. Its value is around 91%.
- The swirl ratio also varies linearly with the pressure ratio. For the same pressure ratio, the nozzle provides air to the stator-rotor cavity with a weaker tangential velocity as compared to vane-1 and vane-2. Therefore, the swirl ratio remains lower than that of vane-1 and vane-2, regardless of the pressure ratio. The values vary from 0.6782 to 1.121 for a pressure ratio ranging from 1.1889 to 1.6276.
- The performance of the nozzle in terms of dimensionless pre-swirl effectiveness also lags behind that of vane-1 and vane-2. Therefore, in order to provide 100% of the rotational energy of the disc through swirling of air, the nozzle requires a greater value of the pressure ratio.

The influence of the through-flow on the aforementioned parameters was also determined for the steady state pressure drops. It was observed that the through-flow deteriorates the performance of the pre-swirlers, whether in terms of dimensionless pre-swirl effectiveness, or swirl ratio. The discharge coefficient is however not affected by the introduction of the through-flow. The dimensionless pre-swirl effectiveness is reduced by 19.4%, 22.3% and 22.5% for vane-1, vane-2 and the nozzle respectively after introduction of the through-flow. The swirl ratio is for its part respectively reduced by 5.65%, 1.11% and 9.73%.

As previously mentioned, the through-flow analysis constitutes a premise to a more detailed study in which the influence of the through-flow is evaluated against different pressure drops. The numerical models with through-flow have however shown to become unstable when changing the pre-swirl system pressure drop due to transient effects. Unsteady simulations might therefore be more appropriate to carry out successfully this study.

Also, the present paper put emphasis on the pre-swirler’s performance and the flow structures inside the stator-rotor cavity regardless of the heat transfer. This parameter could constitute the subject of a deeper analysis in which correlations between the swirl ratio and the heat transfer are drawn.
ACKNOWLEDGEMENT

I would like to express my fully gratitude to Dr. Philip Evegren who have challenged me all along this project and enabled me to use all the best of my knowledge. Encouragements and theoretical supports from Philip were very beneficial to this project. I would also like to extent my gratitude to Mr. Fredrik Borén whose management skills and regular feedbacks and support were fundamental in the accomplishment of this project. I am sincerely thankful to Dr. Jens Fridh for the guidance he provided throughout the project.

I am also truly grateful to Siemens Industrial Turbomachinery that provided financial and theoretical support to this project, and all the employees whom I have met over the last five months.

Theoretical comprehension in this project was also based on materials available in scientific journals and books. I owe sincere thankfulness to the authors whose names are thoroughly listed in the references.

Last but not least, many thanks to all my colleagues and dear friends who constituted a strong support in this project.
REFERENCES


