Entropy Generation in Fluid Mixing

by

François T. Le Floch-Yin

Submitted to the Department of Aeronautics and Astronautics on May 25, 2006 in Partial Fulfillment of the Requirements for the Degree of Master of Science in Aeronautics and Astronautics at the MASSACHUSETTS INSTITUTE OF TECHNOLOGY

ABSTRACT

This thesis describes the processes of viscous dissipation in two generic situations for mixing flows. The objective is to illustrate, with simple examples, details of the entropy generation processes that are captured in an overall manner by a control volume analysis. The two situations are parallel mixing flows in a duct and the evolution of a wake in a centrifugal compressor. Results are given for the evolution of the velocity profile and for the dissipation function and stagnation pressure fields.

Thesis supervisor: Edward M. Greitzer
Title: H.N. Slater Professor of Aeronautics and Astronautics

Thesis supervisor: Choon S. Tan
Title: Senior Research Engineer, Gas Turbine Laboratory
AKNOWLEDGMENTS

A la mémoire de mon grand-père,
Ingénieur technique à l’Aérospatiale

First, I wish to express my gratitude to my advisors, Professor Edward M. Greitzer, and Dr. Choon S. Tan, who granted me this great opportunity. Working with them was a very profitable and enjoyable experience, and I am very grateful for all their advice, patience, and mentorship.

I want to thank warmly my parents Mr. J.-P. Le Floch and Mrs. A.-M. Le Floch-Yin, and grandmothers Mrs. C. Le Floch-Michaud and Mrs. Yin-Ting Y.Y., who provided the necessary funding for my entire first year of studies at the Massachusetts Institute of Technology. This Master’s thesis is dedicated to them, who have always given me their unconditional support.

I also want to thank all the GTL staff and students who helped me throughout the year, especially the late Mr. Paul E. Warren, and Dr. Yifang Gong for their technical guidance, Mrs. Lori Martinez for her administrative support, Jérome Bert, Alfonso Daniel Villanueva Villarreal, Sean Nolan, Barbara Brenda Botros and Adrian Townsend for their daily friendly assistance.

Finally, a special thanks goes to M. Jackson for her encouragement.
Table of Contents

ABSTRACT

AKNOWLEDGMENTS

1. Introduction ................................................................................................................. 2

2. Previous work on mixing flows and shear layers................................................................. 3

   2.1. CONTROL VOLUME AND CONSERVATION EQUATIONS .......................................................... 3

   2.2. BACKGROUND WORK ON ENTROPY GENERATION (BEJAN, 1996).......................................4

   2.3. BACKGROUND WORK ON SHEAR LAYER MIXING (SCHLICHTING, 1968).............................5

3. Definition of the model for the case of parallel flows mixing ...................................... 9

   3.1. DEFINITION OF THE CASE OF STUDY: PHYSICAL MODEL AND RESTRICTIONS ..............9

   3.2. DEFINITION OF THE NUMERICAL MODEL: BOUNDARY CONDITIONS, ACCURACY AND CONVERGENCE ............................................................................................................................ 10

4. Results for the duct mixing simulation ........................................................................ 11

   4.1. VELOCITY PROFILES, AND LENGTH FOR MIXING TO BE ACCOMPLISHED ............................. 11

   4.2. STATIC PRESSURE FIELD .................................................................................................... 19

   4.3. DISSIPATION FUNCTION AND MASS AVERAGE STAGNATION PRESSURE ......................... 23

   4.4. SUMMARY ............................................................................................................................ 31

5. Wake in a centrifugal compressor: modeling ........................................................... 33

6. Wake in a centrifugal compressor: results ...................................................................... 36

   6.1. RELATIVE VELOCITY PROFILES, AREA OF MIXING ............................................................. 36

   6.2. REDUCED STATIC PRESSURE FIELD .................................................................................... 41

   6.3. LOSSES: DISSIPATION FUNCTION, AND REDUCED STAGNATION PRESSURE .................... 41

7. Comparison of the results, and concluding remarks .................................................. 46

Appendix A: Similarity solution of the Shear Layer Theory (Schlichting)......................... 48

Appendix B: Dissipation associated with the similarity solution ..................................... 50

Appendix C: Entropy generation in mixing flows.................................................................. 57

Bibliography............................................................................................................................. 64
List of Figures

Figure 2.1 : Control Volume definition................................................................. 4
Figure 2.2 : Similarity solution velocity profile for a shear layer between two streams .......... 6
Figure 2.3 : Definition of the width of the shear layer ............................................. 7
Figure 2.4 : Spatial evolution of the shear layer (theoretical model)......................... 8
Figure 3.1 : Sketch of the flow domain ................................................................... 9
Figure 4.1 : Contours of $u_I/U_1$ from $x/h=-1$ to $x/h=7$ (shear layer early mixing) .......... 11
Figures 4.2(a) to (n) : Longitudinal velocity component, $u_I/U_1$, at different $x/h$ stations ....... 13
Figure 4.3 : Contours of $u_I/U_1$ from $x/h=-1$ to $x=50$, with a 2% step ....................... 15
Figure 4.4 : Contours of $u_I/U_1$ from $x/h=-1$ to $x/h=7$ (shear layer early mixing) .......... 16
Figure 4.5(a) to (l) : Transversal velocity component, $u_I/U_1$, for $x/h < 1$.................. 17
Figure 4.6(a) to (f) : Transversal velocity component, $u_I/U_1$, for $x/h \geq 1$ (small scale) ........ 18
Figure 4.7 : Static pressure field : contours of $((p_p-p_w)/(1/2\rho U_1^2))$, from $x/h=-1$ to $x/h=6$ ...... 19
Figure 4.8 : Static pressure field contours near the origin ........................................ 20
Figure 4.9 : Static pressure field profiles at different stations ....................................... 22
Figure 4.10 : Schematic diagram of the flow pattern in the duct, from $x = -h$ to several $h$ ......... 22
Figure 4.11 : Mass average stagnation pressure along the duct .................................. 24
Figure 4.12 : Mass average stagnation pressure along the duct (log scale) ...................... 24
Figure 4.13 : Mass average stagnation pressure along the duct (region of early mixing) .......... 25
Figure 4.14 : Area average static pressure along the duct .......................................... 26
Figure 4.15 : Mass average kinetic energy along the duct (log scale) ............................ 26
Figure 4.16 : Dissipation function profiles along the duct (before $x/h=0$) ...................... 27
Figure 4.17 : Dissipation function profiles along the duct (early mixing after $x/h=0$) ........... 28
Figure 4.18 : Dissipation function profiles along the duct (early mixing – zoom in) .......... 28
Figure 4.19 : Dissipation function components at $x/h=0.1$ ................................... 30
Figure 4.20 : Dissipation function profiles along the duct (late mixing) ......................... 30
Figure 4.21 : Schematic representation of the results for the case of parallel mixing flows ......... 31
Figure 5.1 : Sketch of centrifugal compressor blades .................................................. 33
Figure 5.2 : Model of centrifugal compressor, with a blockage screen ......................... 35
Figure 6.1 : Contours of radial velocity component around a blade, $w_r/w_r_{in}$ .................. 37
Figure 6.2(a) to (h) : Radial velocity at different $r/R$ stations ................................... 38
Figure 6.3 : Relative velocity at all stations ................................................................ 38
Figure 6.4(a) to (f) : Plots of circumferential relative velocity $w_{th}$ at different $r/R$ stations ... 39
Figure 6.5 : Plots of circumferential relative velocity at all stations ............................... 40
Figure 6.6 : Reduced static pressure contours ($p_{red}/(0.5\rho w_{r_{in}}^2)$) ........................ 41
Figure 6.7 : Reduced stagnation pressure contours (representative scale) .................... 42
Figure 6.8 : Reduced stagnation pressure plots at different stations .............................. 43
Figure 6.9 : Mass average reduced stagnation pressure for $r=R$ (log scale) .................... 43
Figure 6.10 : Dissipation function profiles at different stations (from $r=R$ to $r=2R$) ........ 44
Figure a1 : Velocity profile from the similarity solution (Schlichting) ............................ 49
Figure b1 : $y$-velocity profile from the similarity solution ........................................... 51
Figure b2 : Dissipation function profiles from the similarity solution ......................... 55
Figure c1 : Control Volume definition for integral equations ....................................... 61
Figure c2 : Simple case of 2 uniform parallel flows (2D) ............................................ 62
List of Tables

Table 4.1: Entropy generation for different Reynolds numbers........................................32
Table 7.1: Summary of two mixing problems examined.......................................................46
Table c1: Some direct results of a control volume analysis (simple 2D case of mixing parallel flows, \(\lambda=0.5\)).................................................................63
Conventions and Nomenclature

Conventions:
The directions in the 2-dimensional coordinate system is referred to as a “longitudinal” coordinate (along the mean direction of the flow) and a “transverse” coordinate (perpendicular to the mean direction). The Cartesian coordinate system is defined by x as the main direction of the flow, and y as the transverse coordinate. The Polar system is defined for the centrifugal compressor case, with r the main direction of the flow, and θ the transverse coordinate. The rotation vector $\tilde{\Omega}$ defines the z-axis of the corresponding 3-dimensional cylindrical coordinate system.

Nomenclature:

Letters:

- A: cross-sectional area
- $c_p$: specific heat at constant pressure
- CV: control volume – reference volume for calculations
- d: differential quantity
- E: internal energy
- e: internal energy per unit mass
- Ek: Ekman number
- $f$: non-dimensional stream function (of η)
- g: non-dimensional stream function (of x,y)
- H: enthalpy
- h: enthalpy per unit mass
- h: duct height – reference length in the y direction
- k: thermal conductivity
- L: reference length
- M: Mach number
- m: mass
- n: number of moles
- p: static pressure
- $p_t$: stagnation pressure
- Q: heat transfer
- q: heat flux vector
- $\mathcal{R}$: universal gas constant
- R: gas constant
- R: radius at the trailing edges of the blades (centrifugal compressor case)
- Re: Reynolds number
Symbols:

- \( \nabla \) nabla operator (gradient, divergence)
- \( \partial \) partial derivative symbol
- \( \gamma \) specific heat ratio
- \( \delta \) shear layer half-thickness
- \( \delta \) infinitesimal variation (replaces derivatives for non state functions like \( s_{\text{gen}} \))
- \( \Delta \) difference, e.g. \( A\nabla \)
- \( \eta \) similarity dimensionless transverse coordinate
- \( \theta \) angle (centrifugal compressor case)
- \( \lambda \) ratio of the inlet velocities (case of two parallel mixing flows)
- \( \mu \) dynamic viscosity
- \( \nu \) kinematic viscosity
- \( \rho \) fluid density
- \( \psi \) stream function
- \( \tau \) viscous stress tensor
- \( \phi \) dissipation function (subscripts are used for different velocity fields)
- \( \Omega \) irreversibility distribution ratio
- \( \infty \) infinity
- \( \propto \) proportional to
- \( o(\ ) \) negligible compared to
- \( \approx, \sim \) approximately equal to, equivalent to

Subscripts:

- \( 1 \) higher velocity (upper) inlet component
- \( 2 \) lower velocity inlet component
- \( A \) point where the shear layer reaches the wall boundaries
- \( \text{down} \) value on the lower side of the splitting wall (case of parallel flows)
- \( e \) exit conditions (mixed out state)
- \( \text{gen} \) generated quantity
- \( i \) fixed position of a coordinate system (\( x_i \) stands for a defined position)
- \( i,\text{in} \) inlet conditions
- \( i,j \) vector components in a sum (\( x_i \) and \( x_j \) represent the coordinate system)
- \( \text{loss} \) value of the losses in a non-conserved quantity (here, stagnation pressure)
- \( \text{max} \) maximum value
min  minimum value
mixed value corresponding to the mixed out conditions
R value at reference radius R
rel derivative in the relative frame
red reduced quantity in the rotating relative frame
t stagnation quantities
up value on the upper side of the splitting wall (case of parallel flows)
x,y,z components in x,y,z directions (Cartesian coordinates)
r,θ, th components in r and θ directions (Polar coordinates)
w values at the walls (angle θ)

Overbar symbols :

→ (e.g. $\vec{x}$) vector
. (e.g. $\dot{m}$) rate of change of a quantity in a defined volume (time derivative)
$\overline{M}$ (e.g. $\overline{p}_T$) mass-flow average quantity
$\overline{A}$ (e.g. $\overline{p}_T$) area average quantity
$\sim$ (e.g. $\overline{w}_r$) mass-flow average quantity
~ non-dimensional quantities
1. Introduction

Entropy generation processes have two main origins, in thermal diffusion and in viscous fluid mixing. Entropy generation through fluid mixing occurs when flows of different properties are put in contact. Although a control volume analysis can often be used to compute the losses of various elements of turbomachines, the method does not show the internal processes of entropy generation. We would like here to examine items within the control volume, and illustrate how the detailed, or local, approach relates to the global approach.

The study will focus on two specific problems: a simple case of parallel mixing flows in a duct of constant area and a model of the wake downstream of the blades of a centrifugal compressor.
2. Previous work on mixing flows and shear layers

As background to the present study we describe briefly relevant information on mixing, entropy generation and control volume analysis.

2.1. Control volume and conservation equations

We start with a control volume analysis for steady-state incompressible flow as in Figure 2.1, which represents a control volume for a 2-dimensional situation. As defined here the control volume is a straight channel. At the inlet a non-uniform flow enters the channel, and at the outlet the flow is fully mixed out (i.e. uniform). There are no sources of mass, work, or heat in the channel.

The conservation laws can be written as follows, with \( u_e \), \( p_e \) and \( p_{\text{st}} \) the velocity, static pressure and stagnation pressure at the exit (mixed out conditions). Equation (2.1.1) is conservation of mass for an incompressible flow, and defines the area average.

\[
\bar{u}_e^A (x_i) = \frac{1}{h} \int_h^y u_e \, dy = u_e .
\]  

Equations (2.1.2) and (2.1.3) are conservation of momentum in the longitudinal x-axis with viscous stresses neglected at the boundaries.

\[
\bar{p}^A (x_i) + \rho \bar{u}_e^2 (x_i) = p_e + \rho u_e^2 ,
\]  

or in terms of stagnation pressure

\[
\bar{p}_e^A (x_i) + \frac{1}{2} \rho u_e^2 (x_i) = p_e + \frac{1}{2} \rho u_e^2 .
\]
The equations for conservation of energy are not needed to determine the velocity and pressure, because the mechanical problem is decoupled from the thermal problem for an incompressible flow.

2.2. Background work on Entropy Generation (Bejan, 1996)

Bejan [2] presents an in-depth discussion of the origins of losses in engineering. Irreversible processes, characterized by dissipation, give rise to losses in available energy, so generation of entropy is directly related to mechanical energy losses. Heat flow is one mechanism for entropy generation, and viscous dissipation is the other. The dissipation function $\Phi$, defined in (2.2.1), can be used to describe the viscous dissipation of mechanical energy.

$$\Phi = \sum_{ij} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} = 2 \left( \frac{\partial u_x}{\partial y} \right)^2 + 2 \left( \frac{\partial u_y}{\partial x} \right)^2 + \left( \frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} \right)^2. \quad (2.2.1)$$

The relation between the rate of entropy generation per unit width in the control volume and the dissipation function is given by (2.2.2) for a 2-dimensional situation (see Appendix C):

$$\frac{dS_{gen}(x_i)}{dt} = \int_{C_{(a)}} \frac{\rho}{2 \Phi} \Phi dV. \quad (2.2.2)$$

Entropy generation can be expressed with a control volume analysis. It is useful to introduce the concept of mass average stagnation pressure, defined in (2.2.3) (Greitzer et al., 2004).
For incompressible flow, and uniform stagnation temperature, the relation between the rate of entropy generation, and the variation of the mass average stagnation pressure between the inflow and the outflow, is given by (2.2.4) (see Appendix C):

$$\frac{\partial S_{\text{gen}}(x)}{\partial t} = -\frac{u_e A_e}{T_t} \left( p^{M}_t(x) - p_{se} \right) = \frac{\dot{m}}{\rho T_t} \left( p^{M}_t(x) - p_{se} \right),$$  \hspace{1cm} (2.2.4)

where $p_{se}$ is the stagnation pressure at the exit, $T_t$ the stagnation temperature of the flow, and $\dot{m}$ the mass flow per unit width.

It is also useful to turn the entropy production rate into a dimensionless dissipation coefficient $C_d$ defined by (2.2.5):

$$C_d = \frac{\frac{T_t}{\dot{m} U_1^2} \frac{\partial S_{\text{gen}}(x)}{\partial t}}{\frac{1}{2} \rho U_1^2} = \frac{\Delta p_{\text{loss}}}{\Delta p_{\text{loss}}}.$$  \hspace{1cm} (2.2.5)

For the control volume the relation between the global mechanical energy loss, and the local viscous dissipation is:

$$\frac{\Delta p_{\text{loss}}}{\frac{1}{2} \rho U_1^2} = \frac{T_t}{\dot{m} h U_1^2} \int_{C_{\text{vol}}} \frac{\nu}{T} \frac{\partial \phi}{\partial x} \, dx dy,$$  \hspace{1cm} (2.2.6)

where $\nu = \frac{\mu}{\rho}$ is the kinematic viscosity.

The key to understanding the entropy generation thus resides in a local knowledge of the mechanisms of fluid mixing.

2.3. Background work on Shear Layer mixing (Schlichting, 1968)

To introduce the mechanism of viscous dissipation in fluid mixing, we start with the analysis of a shear layer between two parallel streams in a laminar flow. The analytical solution is detailed in Appendix A. With $U_1$ the velocity of the higher stream, the velocity profile in the longitudinal direction is given by (2.3.1):

$$u_e = U_1 \frac{\partial f}{\partial \eta} (\eta),$$  \hspace{1cm} (2.3.1)
where $f$ is a function of the similarity “dimensionless transverse coordinate”, $\eta = y \sqrt{\frac{U_1}{v x}}$, obeying the shear layer equation (2.3.2):

$$f f'' + 2 f'' = 0 .$$  \hspace{1cm}  (2.3.2)

To solve for $u_x$, we use the boundary conditions:

$$u_x \rightarrow \infty \rightarrow U_1 ,$$

$$u_x \rightarrow \infty \rightarrow U_2 .$$

Figure 2.2 gives the numerical solution for the dimensionless longitudinal velocity, as a function of $\eta$, for an inlet velocity ratio $\lambda = \frac{U_2}{U_1}$ of 0.5.

![Figure 2.2](image)

**Figure 2.2**: Similarity solution velocity profile for a shear layer between two streams (Schlichting, 1963)

From (2.3.2), we can develop a first estimate for the length to mix the flow. We define boundaries for the shear layer at a lateral distance where the velocity reaches 99% of its value at infinity, namely $U_1$ at the top (2.3.3), and $U_2$ at the bottom.

$$u_x = U_1 - 0.01 (U_1 - U_2) .$$ \hspace{1cm}  (2.3.3)
The flow is symmetric about the central plane of shear mixing, so it is convenient to define the shear layer thickness as $2\delta$. The value of the dimensionless coordinate, $\eta$, is 3.48 at the 1% boundary of the shear layer.

Using this value of $\eta$, we estimate the length where the shear layer would reach 10% of a reference height $h$, as:

$$ y_{10\%} = 0.1 \times \left( \frac{h}{2} \right). $$

Thus,

$$ x_{10\%} = \frac{y_{10\%}^2 U1}{\nu \eta^2} = \frac{0.01 \times U1 h^2}{4 \times 3.48^2 \times \nu} = \frac{1}{4844} \frac{Re}{1-\lambda} h. $$

In (2.3.4), $Re = \frac{\Delta U h}{\nu} = \frac{(U1-U2) h}{\nu} = \frac{(1-\lambda) U1 h}{\nu}$ is defined as the Reynolds number of the flow.

The length where the shear layer would reach the edges of a duct of height $h$ is given by (2.3.5):

$$ y_A = \left( \frac{h}{2} \right), $$

$$ x_A = \frac{y_A^2 U1}{\nu \eta^2} = \frac{1}{4.844} \frac{Re}{1-\lambda} h. $$

For example with $Re=100$ (the case examined in Chapter 2), we find:

$$ x_{10\%} = 0.0413 \ h, \text{ and } x_A = 4.13 h. $$
Figure 2.4: Spatial evolution of the shear layer (theoretical model)

To evaluate the length for mixing, we need to define a criterion of uniformity. At the center of the shear layer, the velocity is the mean of $U_1$ and $U_2$ and is equal to the mixed out velocity.

$$u_e = \frac{U_1 + U_2}{2} = \frac{1 + \lambda}{2} U_1$$  \hspace{1cm} (2.3.6)

We can define the variations of the velocity field to a percentage of $\Delta U$. From this criterion we can estimate the corresponding length using the similarity solution for a free shear layer. For a situation where the velocity difference across the duct is no more than 5% of the velocity difference $U_1 - U_2$:

$$u_x = u_e + \frac{0.05}{2} (U_1 - U_2) \quad \text{at} \quad y = \frac{h}{2} , \quad \text{we find} \quad \eta_{15\%} = 0.0986 \text{ for } \lambda = 0.5. \quad \text{This yields}$$

$$x_{15\%} = \frac{y^2 U_1}{\nu \eta_{15\%}^2} = \frac{U_1 h^2}{4 \times 0.0986^2 \times \nu} = \frac{1}{0.0389} \frac{\text{Re}}{1-\lambda} \ h . \quad (2.3.7)$$

For the velocity to deviate by no more than 2% of the difference $U_1 - U_2$, we find $\eta_{2\%} = 0.0422$. This yields

$$x_{2\%} = \frac{y^2 U_1}{\nu \eta_{2\%}^2} = \frac{U_1 h^2}{4 \times 0.0422^2 \times \nu} = \frac{1}{0.00712} \frac{\text{Re}}{1-\lambda} \ h . \quad (2.3.8)$$

For $\text{Re}=100$,

$$x_{15\%} \approx 5140 \ h , \quad x_{2\%} \approx 28000 \ h .$$

These results overestimate the actual mixing length, but as will be seen the shear layer analysis does give a useful description of the early stages of the mixing.
3. Definition of the model for the case of parallel flows mixing

3.1. Definition of the case of study: physical model and restrictions

We examine the mixing of two laminar flows in a two-dimensional straight duct of constant cross-sectional area. The two flows come from two parallel inlets, and have equal cross-sectional area. The mixing is laminar. There are essentially mixed out conditions at the exit.

The inlet conditions are represented by inviscid walls of length $|x_i| = h$ as shown in Figure 3.1.

![Figure 3.1: Sketch of the flow domain](image)

The fluid is incompressible, with uniform density and viscosity, and no body forces. There is no heat transfer or work done on the fluid.

An appropriate Reynolds number is defined as $Re \approx \frac{\Delta U}{\nu h}$ where $\Delta U$ is the velocity difference at the inlet: $\Delta U = U_1 - U_2$:

$$Re = \frac{U_1 - U_2}{\nu h} = \frac{U_1 (1 - \lambda)}{\nu h}.$$
Ideally we would like to have a large value of Reynolds number to illustrate internal flows in turbomachinery. However the larger the Reynolds number, the longer the mixing region (and the longer the computation). In addition, there are no qualitative changes expected once the Reynolds number is large enough such that inertial forces are important. The value used here for the detailed results was thus 100, although calculations were also carried out at other values of Re.

The value chosen for $\lambda$ was 0.5 to avoid regions of backflow. To examine the effects of shear layer mixing only, there is no shear stress at the upper and lower walls.

3.2. Definition of the numerical model: boundary conditions, accuracy and convergence

The Reynolds number and velocity ratio define the inlet conditions. The results are in terms of non-dimensional quantities and thus reflect general results for our set of assumptions. The solver used is Fluent. The grid is drawn using the software Gambit. From the discussion in Section 2.3, development of the shear layer corresponding to 10% of $h$ is to be expected around $x=0.04h$. The mesh size at the trailing edge of the separating wall is set to $4.10^{-4}h$, corresponding to the distance where the shear layer has a height of 1% $h$. To enable calculations in a reasonable time, a progressive mesh with a small successive cell-to-cell ratio, 1.01, was used. With 146,000 cells, the calculation converged from a uniform state in less than one day. Convergence is defined when non-dimensional residuals of the flow equations (continuity and momentum) sum up to less than $1.10^{-13}$.

The numerical results were extracted from Fluent and post-processed in Matlab.
4. Results for the duct mixing simulation

4.1. Velocity profiles, and length for mixing to be accomplished

We examine the x-component of velocity first. Figure 4.1 shows contours of $\frac{u_s}{U_1}$ in the initial mixing region for $-1 < \frac{x}{h} < 7$.

*Figure 4.1 : Contours of $u_s/U_1$ from $x/h=-1$ to $x/h=7$ (shear layer early mixing)*

Figure 4.1 is complemented by Figures 4.2(a) to 4.2(n), which show the x-component of velocity at different locations along the duct. There are two curves in each figures corresponding to $x \geq 0$ : one for the numerical simulation and one for the similarity solution. In the latter the “origin” of mixing has been set at the end of the separating wall ($x=0$) ; this provided the best correlation with the numerical calculations. The profile computed shows a good correlation with that of the similarity solution within the shear layer until approximately $x=10$. 
Some features can be commented on in Figures 4.2(a) to 4.2(n).

Figures 4.2(a) to 4.2(d) show that the velocity profile starts to change before the trailing edge \( \frac{x}{h} = 0 \) to reach a mean value at the edge. This is necessary to satisfy continuity of the velocity field at the trailing edge; there is no discontinuity at the trailing edge (Figure 4.2(e)).

The velocity profile follows the evolution of the similarity profile for only a short distance (Figures 4.2(e) to 4.2(j)). From the location where the velocity at the walls starts to be reached by the
shear layer (around $\frac{X}{h} = 4$ at the top, $\frac{X}{h} = 6$ at the bottom), the actual profile shows a different evolution.

For $\frac{X}{h} = 0$ to $\frac{X}{h} = 4$, the velocity is not uniform outside the shear layer near the boundaries of the duct. Figures 4.2(e) to 4.2(j) show that the velocity in the upper region (higher velocity) increases to a maximum then decreases, while the velocity in the lower region (lower velocity) decreases to a minimum then increases. The explanation of this observation will be given in Section 4.2, where the pressure field is analyzed. Figures 4.2(i) and 4.2(j) show that the velocity profile is not symmetric. At $\frac{X}{h} = 1$ the velocity at the center is slightly higher than the mean velocity. At $\frac{X}{h} = 5$ we see that the velocity in the upper region has decreased faster than the velocity in the lower region has increased. The value of $\frac{X}{h}$ where the shear layer reaches the wall (at height $\frac{h}{2}$) is about 4.1 at the top, and 5.8 at the bottom. The shear layer estimate was $\frac{X}{h} = 4.1$ on both sides.

From $\frac{X}{h} = 5$ to $\frac{X}{h} = 200$, Figures 4.2(j) to 4.2(n) show that the flow mixes out more rapidly than in the similarity solution; for example, there is a 2% velocity difference between the center and the boundaries of the duct at $\frac{X}{h} = 45$, as illustrated in Figure 4.3. This is 3 orders of magnitude less than the 28 000 calculated in Section 2.3! The shear layer similarity analysis is thus not representative after $\frac{X}{h} \approx 5$. 
Figure 4.3: Contours of $u_x/U_1$ from $x/h=-1$ to $x=50$, with a 2% step
The y-component of velocity is shown in Figures 4.4 and 4.5, non-dimensionalized by $U_1$. Figure 4.4 presents contours of $\frac{u_y}{U_1}$ near the inlet ($\frac{x}{h} = -1$ to $\frac{x}{h} = 7$) and Figures 4.5 and 4.6 shows the y-component velocity profiles at different stations, at two different scales. The similarity solution is again shown for comparison.

![Figure 4.4: Contours of $u_y/U_1$ from $x/h=-1$ to $x/h=7$ (shear layer early mixing)]
Figure 4.5(a) to (l): Transversal velocity component, $u/yU_1$, for $x/h < 1$
There are several features of the y-component of velocity that should be noted. First, the value of $u_y$ is everywhere small compared to U1. The highest values of $\frac{u_y}{U_1}$ are at the trailing edge of the separating wall and have magnitude about 0.05. The maximum value of $\frac{u_y}{U_1}$ does not occur at the center of the duct. The mixing cannot start with a discontinuity (as implied by the shear layer analysis), but rather a wake profile. We see two maxima at $\frac{x}{h} = 0$, the highest being $\frac{u_y}{U_1} = 0.0538$ at
\( \frac{y}{h} = -0.0184 \). Figures 4.5(c) and 4.5(d) show that the flow profile starts to evolve before the trailing edge of the separating wall. From Figures 4.5(e) to 4.5(j), we see that until \( \frac{x}{h} = 0.2 \) the direction of the transverse flow is from the (lower) region of lower velocity, to the (upper) region of higher velocity. This will be explained in Section 4.2 when the pressure field is analyzed. Downstream of \( \frac{x}{h} = 0.2 \) the direction of \( u_y \) becomes negative (Figures 4.5(k) and 4.5(l)), and roughly resembles the similarity solution. However, from Figures 4.6(a) to 4.6(f), the profile of \( \frac{u_y}{U_l} \) evolves more rapidly than the similarity solution after approximately \( \frac{x}{h} = 5 \). Again, the flow reaches a quasi mixed out state at \( \frac{x}{h} = 45 \) (see Figures 4.6(d), 4.6(e) and 4.6(f)).

4.2. Static pressure field

The above description of the velocity field allows us to interpret the dissipation distribution. Before this, however, it is useful to describe the pressure field. Contours of static pressure are given in Figure 4.7 and 4.8. Figure 4.9 is a plot of the static pressure profile at different locations.

\[ \frac{y}{h} = -0.0184. \]  

In Figures 4.7, 4.8 and 4.9, we see that except near \( \frac{x}{h} = 0 \), the pressure differences are everywhere small compared \( \frac{1}{2} \rho U_l^2 \). After \( \frac{x}{h} = 1 \), the effect of the pressure differences on the flow
can be neglected. Between $\frac{X}{h} = 0$ and $\frac{X}{h} = 1$, the pressure differences are concentrated in an area of radius roughly 0.01h around the origin. After 0.01h (Figure 4.9) static pressure differences do not exceed 10% of $\frac{1}{2} \rho U^2$. Close to $\frac{X}{h} = 0$, Figure 4.8 shows that the pressure differences cannot be neglected. Figure 4.9 shows the static pressure difference in this region. The high flow and low flow streams have a common value at the trailing edge of the separating wall, where the velocity field cannot be discontinuous. The higher velocity stream has to decrease rapidly, and cannot decrease through friction, and the low flow has to increase rapidly. At the origin, the transverse pressure gradient is the highest. Between $\frac{X}{h} = 0$ and $\frac{X}{h} = 5$, the pressure differences observed near the origin lead to a contraction of the high flow stream, and an expansion of the low flow stream.

Figure 4.8: Static pressure field contours near the origin
We can also check roughly that viscous effects do not play a major part in the region before $x/h = 0$.

If the viscous dissipation is negligible, we apply Bernoulli flow between $x/h = -1$ and $x/h = 0$ and the extrema of static pressure $p_{\text{max(up)}}$ and $p_{\text{min(down)}}$ at the trailing edge are:

$$p_{\text{max(up)}} - p_1 \approx \frac{1}{2} \rho u_1^2 - \frac{1}{2} \rho u_2^2 = \frac{1}{2} \rho u_1^2 \left( \frac{3+\lambda}{4} \right) (1-\lambda),$$

and

$$p_2 - p_{\text{min(down)}} \approx \frac{1}{2} \rho u_2^2 - \frac{1}{2} \rho u_1^2 = \frac{1}{2} \rho u_1^2 \left( \frac{3\lambda+1}{4} \right) (1-\lambda).$$

The numerical results at the inlet, for $Re=100$, give $\frac{p_1 - p_e}{\rho u_1^2} = -0.0776$ and $\frac{p_2 - p_e}{\rho u_1^2} = -0.1724$.

This leads to $\frac{p_{\text{max(up)}} - p_e}{\rho u_1^2} \approx 0.36$ and $\frac{p_{\text{min(down)}} - p_e}{\rho u_1^2} \approx -0.48$.

The results observed in Figure 4.9 are $\frac{p_{\text{max(up)}} - p_e}{\rho u_1^2} \approx 0.4$ and $\frac{p_{\text{min(down)}} - p_e}{\rho u_1^2} \approx -0.65$.

The inviscid approximation for thus gives a rough estimate of the pressure extrema and show that viscous effects coming there do not play a major role. Figure 4.9 shows that the pressure differences are important only in the region close to the end of the splitting wall at $x=0$, $y=0$. After $x=0.1$ the pressure variations in the flow are negligible.

Figure 4.10 is a schematic diagram that represents the main evolution of the flow near the inlet of the duct. After $x=0$, the flow is divided into three areas. Viscous mixing occurs in the shear layer, and outside the shear layer the evolution of the flow is determined by pressure differences.
Figure 4.9: Static pressure field profiles at different stations

Pressure peak (repelling the flow)

Region of contraction (acceleration of the flow)

Pressure hollow (attracting the flow)

Region of expansion (deceleration of the flow)

Figure 4.10: Schematic diagram of the flow pattern in the duct, from $x = -h$ to several $h$
4.3. Dissipation function and mass average stagnation pressure

Figure 4.11 shows the stagnation pressure along the duct. We see that the dissipation occurs between \( \frac{x}{h} = 0 \) and \( \frac{x}{h} = 45 \), and the mass average stagnation pressure does not vary noticeably after \( \frac{x}{h} = 45 \). Figure 4.12 shows an inflexion point of the dissipation (in log scale) at \( \frac{x}{h} = 5 \), corresponding to the location where the shear layer reaches the wall. Between \( \frac{x}{h} = 5 \) and \( \frac{x}{h} = 20 \) the shear layer does not expand anymore, and the velocity profile becomes uniform, so that at \( \frac{x}{h} = 20 \) more than 90% of the mixing is achieved. Figure 4.13 shows the region near the origin.

The viscous losses for \( \frac{x}{h} < 0 \) are negligible compared to the mixing losses. Figure 4.13 also shows the maximum dissipation rate at \( \frac{x}{h} = 0 \), where the two incoming flows meet. The overall stagnation pressure loss is:

\[
\frac{\Delta p_{\text{loss}}}{\frac{1}{2} \rho u_i^2} = 0.0783.
\]

As expected, this result corresponds to a control volume analysis, provided that we take into account the static pressure difference between the two inlet streams at this low Reynolds number (Appendix C):

\[
\frac{\Delta p_{\text{loss}}}{\frac{1}{2} \rho u_i^2} = \frac{(p_1 - p_e) + \lambda(p_2 - p_e)}{(1 + \lambda) \frac{1}{2} \rho u_i^2} + \frac{3}{4} \left(1 - \lambda^2\right). \tag{4.1}
\]
Figure 4.11: Mass average stagnation pressure along the duct

Figure 4.12: Mass average stagnation pressure along the duct (log scale)
Pressure rise in the duct can be seen in Figure 4.14. The mass average kinetic energy, plotted in Figure 4.15, is a measure of the non-uniformity of the flow across the duct. The mass average kinetic energy from the similarity solution (where pressure variations are neglected) and the numerical results are shown for $\frac{x}{h} < 100$. In the numerical results, we observe that the variations in the velocity field outside the shear layer lead to a greater non-uniformity, and a stronger mixing. The two curves cross each other at $\frac{x}{h} = 5$. 

Figure 4.13: Mass average stagnation pressure along the duct (region of early mixing)
Figure 4.14: Area average static pressure along the duct

Figure 4.15: Mass average kinetic energy along the duct (log scale)
Figures 4.16, 4.17 and 4.18 represent the dissipation function at different locations and give a description of the viscous dissipation per unit area. The integral of $\phi$ gives the local rate of dissipation with axial distance, the rate of change of the mass average stagnation pressure.

Figure 4.16 represents the dissipation upstream of the trailing edge of the separating wall. At $\frac{x}{h} = -1$, the dissipation is zero. There are several orders of magnitude between the dissipation rate in the region $\frac{x}{h} < -0.1$ and the region close to the origin ($\frac{|x|}{h} < 0.01$). From $\frac{x}{h} = -0.1$ to $\frac{x}{h} = 0$, the dissipation rate grows up to $10^4$ times, with a maximum at the center of the duct.

![Figure 4.16: Dissipation function profiles along the duct (before x/h=0)](image-url)
Figure 4.17: Dissipation function profiles along the duct (early mixing after $x/h=0$)

Figure 4.18: Dissipation function profiles along the duct (early mixing – zoom in)
Figures 4.17 and 4.18 show the dissipation function between $\frac{X}{h} = 0$ and $\frac{X}{h} = 0.1$. The maximum of dissipation is at the center of the duct. The maximum decreases from $\frac{X}{h} = 0$ to $\frac{X}{h} = 0.1$ by approximately 2 orders of magnitude. Figure 4.20 shows the decrease along the duct for larger values of $\frac{X}{h}$.

The region of dissipation expands laterally from the center to the boundaries of the duct, as the shear layer develops. After the shear layer has reached the walls of the duct, the dissipation function profile tends to a uniform distribution across the duct. At small distances from the origin, there are “bumps” of dissipation on the sides of the main dissipation peak, as in Figure 4.18. These bumps are of magnitude $10^3$ to $10^4$ times less than the peaks at the center, and provide negligible contribution to the overall dissipation.

Figure 4.19 shows the components of the dissipation function at $\frac{X}{h} = 0.1$. The major terms that lead to the bumps are $\left(\frac{\partial u_x}{\partial x}\right)^2$ and $\left(\frac{\partial u_y}{\partial y}\right)^2$ (these are equal). The bumps are caused by the non-uniformity of the flow outside the shear layer. More precisely, the velocity gets higher than $U_1$ above the shear layer, and lower than $U_2$ below.

In Figures 4.16 to 4.20, we observe that the dissipation profile outside the shear layer has a noticeable asymmetry. In Figure 4.20, we see that the dissipation is larger in the lower region, where both pressure rise and shear forces contribute to decrease the flow velocity.
Figure 4.19: Dissipation function components at \( x/h = 0.1 \)

Figure 4.20: Dissipation function profiles along the duct (late mixing)
Figure 4.20 shows that the shape of the dissipation function changes noticeably after the shear layer has reached the walls. After approximately $\frac{x}{h} = 5$, shape changes from bell-like to “dome-like”, and becomes more uniform. The dissipation function then decreases until the exit.

4.4. Summary

Figure 4.21 recaps the major features of the mixing process for this case. The duct can be divided into 4 different parts.

![Diagram](image)

No mixing
- velocity changes at the separating wall due to pressure gradients
- small dissipation, non-symmetrical flow evolution

Shear layer expansion
- shear layer mixing, velocity outside the shear layer not constant
- dissipation mainly in the shear layer, centered on $y = 0$, decreases along the duct, small dissipation outside the shear layer, due to non-uniform flow (“bumps”)

Late mixing
- shear layer all across the duct, uniform pressure field
- dome-like dissipation profile through the duct, magnitude decreases with $x$

Mixed out state
- $\frac{\Delta h}{U1-U2} < 0.02$
- negligible dissipation

Figure 4.21: Schematic representation of the results for the case of parallel mixing flows ($\lambda=0.5, Re=100$)
The overall mechanical energy loss due to fluid mixing is \( \frac{\Delta p_{\text{loss}}}{\frac{1}{2}\rho U^2} \approx 0.0783 \).

<table>
<thead>
<tr>
<th>Re</th>
<th>( \frac{\Delta p_{\text{loss}}}{\frac{1}{2}\rho U^2} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>0.1150</td>
</tr>
<tr>
<td>100</td>
<td>0.0783</td>
</tr>
<tr>
<td>1000</td>
<td>0.0751(^1)</td>
</tr>
<tr>
<td>CV analysis with uniform inlet pressure</td>
<td>0.0625</td>
</tr>
</tbody>
</table>

\(^1\): the duct was not long enough, so the total loss was estimated.

Table 4.1: Entropy generation for different Reynolds numbers

Table 4.1 shows the decrease of mass-average stagnation pressure in the duct for different Reynolds number, and the result derived from a control volume analysis, using the assumption of equal inlet pressure in the two streams. The inlet velocities are the same for all cases, and the inlet stagnation pressures are different (only the viscosity of the fluid was changed). As Re increases, the mechanical energy loss gets closer to the result derived from a control volume analysis. The pressure difference observed at the end of the splitting wall decreases, the flow is more unidirectional, and the mixing situation there is closer to a free shear layer situation.
Another generic mixing situation is a wake in a centrifugal compressor. This has different features from the duct mixing situation, including Coriolis and centrifugal forces in the rotating frame.

Figure 5.1 is a sketch of a wake in a 2-dimensional centrifugal compressor. A 20-blade centrifugal compressor is used, having straight and infinitely thin radial blades. Calculations show that the distance of mixing after the end of the blades did not exceed 5 times the width of the channel \( W_{R} \) at the trailing edges; at this location the flow can be considered mixed out.

\[
W_{R} \approx \frac{2 \pi R}{20} \approx 0.314 R
\]

**Figure 5.1 : Sketch of centrifugal compressor blades**

At the inlet a representation of rotating channel flow based on a quasi 1-dimensional approximation is useful. For irrotational inviscid flow, the analytical result is given by Greitzer et al. [5, Section 7.8]:

\[
w_{r} \approx 2 \Omega r + \frac{w_{r} W}{2 \Omega r}, \tag{5.1}
\]
where $Q$ is the angular velocity of the rotating machine, $\theta_w$ is the semi-angle of the passage, $W$ is the channel arc length at radial station $r$ (taken equal to the width for small values of $\theta_w$), and $w_r$ and $w_\theta$ are respectively the radial and angular relative velocity components.

The Reynolds number of the relative flow is defined as $Re = \frac{\bar{w}_r W}{v}$, where $v$ is the kinematic viscosity. It is here set to $Re = 2.5 \times 10^5$. The calculation also uses a $k-\varepsilon$ turbulence model.

The Navier-Stokes equation in the relative frame has two body forces, centrifugal force and Coriolis force. The centrifugal force can be incorporated through the definition of a reduced pressure. For a steady flow we thus have:

$$\nabla \cdot \bar{\omega} = -\frac{1}{\rho} \bar{\omega} \bar{p}_{red} - 2 \bar{\Omega} \times \bar{w} + \bar{f}_{visc}, \quad (5.3)$$

where $p_{red} = p - \frac{\rho \Omega^2 r^2}{2}$ is the reduced static pressure and $\bar{\Omega}$ is the rotation vector $(\Omega \bar{e}_z)$.

To compare the different terms of (5.3), we can use non-dimensional numbers representing the relative influences of the physical quantities. The Rossby number compares the Coriolis force to the inertial force and is defined as $Ro = \frac{\bar{w}_r}{\Omega W}$.

Its value was set as 2.6 in our study, a representative magnitude encountered in turbomachinery. The Ekman number compares viscosity to the Coriolis force. Its value $Ek = \frac{\nu}{\Omega W^2} = \frac{Ro}{Re} = 1.04 \times 10^{-5}$ shows that the flow profile is essentially determined by the pressure field and Coriolis forces.

The exit static pressure is set to zero. The blades have no slip boundary conditions. The velocity profiles given previously, together with the Rossby number and the Reynolds number, define the boundary conditions at the inlet. $R \approx \frac{10}{\pi} W \approx 3.2 W$ is the radius at the trailing edges of the blades.

To achieve high resolution, the mesh (generated with Gambit) was focused on the mixing areas at the trailing edges of the blades. We used 18,900 cells to allow rapid calculations. Convergence was taken to be when the residuals reduced to $10^{-11}$ for all the governing equations (continuity, momentum).

An appreciable energy loss was desired to illustrate the mixing processes in a rotating environment. The wake created by the boundary layers on the blades was not thick enough, and we
thus created a larger wake by adding a screen on the suction side of the compressor. The velocity in
the wake is about half the velocity of the free stream at \( r=R \), and the width of the wake is 20% of the
passage width. These were created using a flow blockage (a screen-like configuration), placed on the
suction side of the compressor, at \( r=0.9R \). Figure 5.2 is a sketch of the model.

Figure 5.2: Model of centrifugal compressor, with a blockage screen
6. Wake in a centrifugal compressor: results

This chapter shows the results of the wake mixing calculations. As with the parallel mixing problem, we describe first the velocity fields and pressure. We then discuss the evolution of the reduced stagnation pressure and the dissipation function. Comparisons are also made between the rotating blade row and the parallel mixing situation.

6.1. Relative velocity profiles, area of mixing

Figure 6.1 gives the contours of the radial relative velocity around a blade, referenced to the inlet average radial velocity at the center of the passage, \( W_{rin} \). The mixing region spans from approximately \( r=0.9R \) to \( r=1.1R \). Figures 6.2(a) to 6.2(h) give the radial velocity at different radial stations. Figure 6.3 is a composite plot of these profiles, with the velocity \( \frac{W_r}{W_{rin}} \) replaced by the volume flow \( \frac{W_r}{W_{rin}} \times \frac{r}{R} \).
Figure 6.1: Contours of radial velocity component around a blade, \( w_r / w_{r,\text{in}} \)
Figure 6.2(a) to (h): Radial velocity at different r/R stations

Figure 6.3: Relative velocity at all stations
At $r=R$ the velocity in the wake is roughly half the velocity in the free channel ($0.9 \times w_{\text{in}}$). The wake represents roughly 20% of the passage. After the trailing edge, the area average radial relative velocity decreases as $r$ increases because of conservation of mass. On Figure 6.3 we see that the flow profile is convected by the circumferential velocity.

Figure 6.4(a) to (f): Plots of circumferential relative velocity $w_{\psi}$ at different $r/R$ stations
Figures 6.4(a) to 6.4(f) show the features of the circumferential component of relative velocity, and Figure 6.5 compares them at different radial stations. The angular velocity has two local minima and two local maxima. Near $r=R$, the wake ($3.6^\circ < \theta < 9^\circ$) has less circumferential velocity than the free stream. The flow at the end of the blades has zero circumferential velocity. The maxima and minima are convected with the flow, i.e. move to lower angles, and their difference decreases.
6.2. Reduced static pressure field

![Figure 6.6: Reduced static pressure contours](image_url)

Figure 6.6 shows contours of the reduced static pressure, for the range of values after $r=R$ (the values at the screen are out of scale). After $r=R$, we observe that in the wake (suction side), the reduced static pressure is lower than in the free stream. This suggests that like the parallel mixing case, pressure forces are important in the early mixing region where they tend to drive the flow from the free stream to the wake region and diminish the velocity difference.

6.3. Losses: dissipation function, and reduced stagnation pressure

Reduced stagnation pressure is defined as follows:
\[ p_{\text{red}} = p_{\text{red}} + \frac{1}{2} \rho w^2 = p - \frac{1}{2} \rho \Omega^2 r^2 + \frac{1}{2} \rho w^2. \]

The non-dimensional value of stagnation pressure represented in Figure 6.10 is:

\[ \tilde{p}_{\text{red}} = \frac{p_{\text{red}} - p_{\text{red}} \text{ (mixed out state)}}{\frac{1}{2} \rho \Omega^2 R^2}. \]

Figure 6.7: Reduced stagnation pressure contours

At \( r=1.5R \), the reduced stagnation pressure is essentially uniform. Figure 6.8 shows the reduced stagnation pressure profile at different radial locations from \( r=R \) to \( r=1.5R \). The profile at \( r=R \) has a peak on the suction side, that decreases with radius. At \( r=1.2R \), the peak has moved from \( \theta=9^\circ \) to \( \theta=2^\circ \) because of convection by the circumferential velocity.
Figure 6.8: Reduced stagnation pressure plots at different stations

\[ \frac{p_{\text{red}} - p_{\text{red (mixed)}}}{\frac{1}{2} \rho \Omega^2 R^2} \]

Figure 6.9: Mass average reduced stagnation pressure for \( r=R \) (log scale)
Figure 6.9 shows the mass average reduced stagnation pressure along the passage. The flow mixes out relatively quickly, with the losses concentrated near the edge of the blades. The overall loss is 0.071 times the reference rotational kinetic energy at $r=R$. We can also calculate the loss in percentage of the reference inlet kinetic energy (at $r=0.75R$, based on the mean velocity):

$$0.071 \times \frac{Q^2 R^2}{W_{in}^2} = 0.071 \times \frac{1}{R^2} \times \frac{R^2}{W_{in}^2} = 0.071 \times \frac{1}{R^2} \times \frac{R^2}{W^2} \times \frac{1}{0.75^2} \approx 0.19,$$

which is also 0.11 times the kinetic energy at $r=R$.

We now look at the dissipation function, to define the spatial location of the losses. The dissipation function is:

$$\phi = \sum_{i,j} \left( \frac{\partial w_i}{\partial x_j} + \frac{\partial w_j}{\partial x_i} \right) \frac{\partial w_i}{\partial x_j}.$$

Since the difference between the velocity in the absolute frame and the relative frame corresponds to a solid-body rotation, the dissipation function is equal in both frames of reference; $\phi_u = \phi_w = \phi$, and $\rho \frac{\partial S_{gen}}{\partial t} = \frac{\mu}{T} \phi$. The dissipation is not frame-dependent and is an appropriate measure of the local entropy generation due to viscous stresses.

![Figure 6.10: Dissipation function profiles at different stations (from $r=R$ to $r=2R$)](image_url)
Figure 6.10 gives the dissipation function at different radial stations. Two passages are represented to emphasize the peak at the trailing edge of the blades (at $\theta = 9^\circ$). At the end of the blades the peak of high local dissipation is convected from $r=R$ to $r=1.05R$. A second local peak is observed on the suction side of the channel around $\theta = 5^\circ$, due to the mixing of the wake with the free stream after the blockage at $r=0.9R$. The two shear layers eventually merge to a single peak between $r=1.05R$ and $r=1.1R$. At $r=1.2R$, the maximum of the dissipation function has decreased more than 2 orders of magnitude from its value at $r=R, \theta = 9^\circ$. The dissipation is negligible after $r=1.5R$. 
7. Comparison of the results, and concluding remarks

Table 7.1 recaps the main features of the two cases examined.

<table>
<thead>
<tr>
<th>Case</th>
<th>Parallel flows in a duct</th>
<th>Wake in a centrifugal compressor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Viscosity and flow regime</td>
<td>laminar flow</td>
<td>turbulent k-ε model</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>100</td>
<td>$2.5 \times 10^5$</td>
</tr>
<tr>
<td>Rossby number</td>
<td>$\infty$</td>
<td>2.6</td>
</tr>
<tr>
<td>Body forces</td>
<td>none</td>
<td>centrifugal + Coriolis</td>
</tr>
<tr>
<td>Inlet boundary conditions</td>
<td>Two uniform parallel streams</td>
<td>1-D velocity field passed through a screen</td>
</tr>
</tbody>
</table>

Table 7.1: Summary of two mixing problems examined

The conclusions of our study of the two conducted problems are:

1) For the case of the straight channel, we have calculated the characteristics of the mixing of the two flows. For Re=100, the length of mixing is approximately 45 times the height of the duct. The local dissipation profiles have been calculated. The plots show small bumps of dissipation outside the shear layer, due to the non-uniformity of the flow and the pressure difference at the inlet. The results give a clear example of connection between local and global features of mixing flows.

2) For the wake after a centrifugal compressor, we have calculated the relative velocity and pressure profiles along the radius. For Re=$2.5 \times 10^5$ and Ro=2.6, the length of mixing has the order of
magnitude of the passage width between two blades. The plots of dissipation show two shear layers that merge.
Appendix A: Similarity solution of the Shear Layer Theory (Schlichting)

Schlichting [8] solves a simple case of two parallel mixing flows. The flow is considered incompressible, steady, of constant viscosity \( \mu \), uniform pressure in the longitudinal direction \( \bar{e}_y \) or \( p = p(y) \), and derivatives in the longitudinal direction \( x \) neglected with respect to those in the transversal direction \( y \). Under these conditions, we first have the continuity equation for incompressible flows:

\[
\text{div} \ (\mathbf{v}) = 0 \quad \iff \quad \frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} = 0. \tag{1}
\]

The equation for linear momentum (Navier-Stokes equation) in the x-direction is:

\[

p \left( u_x \frac{\partial u_x}{\partial x} + u_y \frac{\partial u_x}{\partial y} \right) = -\frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u_x}{\partial x^2} + \frac{\partial^2 u_x}{\partial y^2} \right),
\]

which simplifies to:

\[

p \left( u_x \frac{\partial u_x}{\partial x} + u_y \frac{\partial u_x}{\partial y} \right) = \mu \frac{\partial^2 u_x}{\partial y^2}. \tag{2}
\]

To solve this system of equations, we use the stream function \( \psi \), defined as follows:

\[

\frac{\partial \psi}{\partial y} = u_x, \quad \frac{\partial \psi}{\partial x} = -u_y.
\]

\( \psi \) thus satisfies the continuity equation for the velocity field. (2) transforms to:

\[

\frac{\partial \psi}{\partial y} \frac{\partial^2 \psi}{\partial x \partial y} - \frac{\partial \psi}{\partial x} \frac{\partial^2 \psi}{\partial y^2} = \nu \frac{\partial^3 \psi}{\partial y^3}, \tag{3}
\]

with \( \nu = \frac{\mu}{\rho} \) the kinematic viscosity.

A similarity variable \( \eta \propto \frac{\nu}{\sqrt{x}} \) can be defined as \( \eta = y \sqrt{\frac{U_1}{\nu x}} \).

We define also a non-dimensional function to represent \( \psi \): \( f(\eta) = \frac{\psi(x,y)}{\sqrt{\nu U_1 x}} \). (3) becomes then:

\[

f \ f'' + 2 \ f''' = 0. \tag{4}
\]

\(^1\) It is noticeable that \( \eta \) won't work as a similarity variable for \( \psi \), and therefore won't work either as a similarity variable for \( u_x \). The similarity variable is defined to solve only the main velocity field \( u_x \).
has no analytical solution at this time, but can be solved numerically. The boundary conditions taken by Schlichting are:

\[
\frac{df}{d\eta} \quad \eta \rightarrow +\infty \rightarrow 1 \quad \text{or} \quad u_\infty \quad \eta \rightarrow +\infty \rightarrow U1
\]

\[
\frac{df}{d\eta} \quad \eta \rightarrow -\infty \rightarrow \lambda \quad \text{or} \quad u_\infty \quad \eta \rightarrow -\infty \rightarrow U2
\]

The function \( u_\infty \) is set arbitrarily to be equal to 0 at the origin—this choice has no incidence on the solution for \( u_\infty \) (we will see in the Appendix B why this boundary condition is not correct for an estimation of \( u_\infty \)).

The numerical solution is plotted on Figure a1:

![Figure a1: Velocity profile from the similarity solution (Schlichting)](image)

This solution depends only on the \( \lambda \) parameter (and not for example on the Reynolds number, within the limits of the assumption of laminar flow).
Appendix B: Dissipation associated with the similarity solution

We want to determine the transversal velocity field $u_y$, and the dissipation function of the similarity solution. We use the previous non-dimensional variable $\eta$, and function $f = f(\eta)$, to get

$$u_y = -\frac{\partial \psi}{\partial x}(x,y) = \frac{1}{2} \sqrt{\nu U_1 \frac{h}{x}} \left( \eta \frac{df(\eta)}{d\eta} - f(\eta) \right).$$

With the Reynolds number $Re = \frac{(1-\lambda)U_1 h}{\nu}$, we can put together non-dimensional terms:

$$u_y = \frac{1}{2} U_1 \sqrt{\frac{(1-\lambda)}{\nu} \frac{h}{x}} \left( \eta \frac{df(\eta)}{d\eta} - f(\eta) \right). \quad (5)$$

Unlike $u_x$, $u_y$ depends not only on the lambda factor (through $f$), but also on the Reynolds number. $u_y$ is not a function of $\eta$. We can foresee in the last equation some "non-similar" effects to be observed, i.e. for a same value of $\eta$ a strong increase of $u_y$ close to the point where the mixing starts ($x \to 0$), and a decrease of the relative importance of $u_y$ far from the origin. The behaviour of $u_y$ at the origin is predictable because we know that $f$ is finite there (more precisely we know that the expression within the brackets isn't equal to zero).

The boundary conditions need now to be revised, as $f$ cannot take a random constant value at $\eta = 0$ without changing the solution. A natural boundary condition corresponds to the annihilation of $u_y$ in the upper region (and consequently in the lower region, as the numerical solution is an even function for $f$). This condition can be written:

$$\eta \frac{df}{d\eta} - f \xrightarrow{\eta \to \infty} 0 \quad \text{or} \quad f \xrightarrow{\eta \to \infty} 1 \quad \text{or} \quad f \xrightarrow{\eta \to \infty} \eta$$

The new boundary conditions are:

$$\frac{df}{d\eta} \xrightarrow{\eta \to \infty} 1 \quad \text{or} \quad u_x \xrightarrow{y \to \infty, x \text{ const}} U_1 \quad ; \text{the longitudinal velocity outside the shear layer in the upper region is } U_1,$$

$$\frac{df}{d\eta} \xrightarrow{\eta \to \infty} \lambda \quad \text{or} \quad u_x \xrightarrow{y \to \infty, x \text{ const}} U_2 \quad ; \text{the longitudinal velocity outside the shear layer in the lower region is } U_2,$$
\[
\frac{f}{\eta} \xrightarrow{\eta \to \infty} 1 \quad \text{or} \quad \frac{\psi(x,y)}{y} \xrightarrow{y \to \infty} UI \quad \text{; the function } \psi \text{ is asymptotically linear in } y. \text{ The numerical solution, for } \lambda=0.5, \text{ is :}
\]

![y-velocity profile from the similarity solution](image)

\begin{equation}
\phi = \sum_{i,j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} = 2 \left( \frac{\partial u_x}{\partial x} \right)^2 + 2 \left( \frac{\partial u_y}{\partial y} \right)^2 + \left( \frac{\partial u_y}{\partial x} \right)^2 + 2 \left( \frac{\partial u_x}{\partial y} \right) \left( \frac{\partial u_y}{\partial x} \right). \tag{6}
\end{equation}

Figure b1 : y-velocity profile from the similarity solution

The \( u \), velocity field depends on \( Re \) and on the longitudinal coordinate \( x \). We can characterize the entropy generation associated with the similarity solution for the velocity profile. The dissipation function \( \phi \) defined in the first chapter and in Appendix C is directly related to the velocity field:
From the similarity solution, we get:

\[
\phi = \frac{1}{2} \frac{U_1^2}{x^2} \eta^2 \left( \frac{\partial f}{\partial \eta} \right)^2 + \frac{U_1^3}{x} \left( \frac{\partial^2 f}{\partial \eta^2} \right)^2 + \frac{1}{16} \frac{V U_1}{x^2} \eta^4 \left( \frac{\partial^2 f}{\partial \eta^2} \right)^2 + \frac{1}{4} \sqrt{V U_1} x^2 \eta^2 \left( \frac{\partial^2 f}{\partial \eta^2} \right) u_y + \frac{1}{4x} w_x^2
\]

\[
- U_1 \frac{U_1}{V} x^{-\frac{3}{2}} \frac{\partial^2 f}{\partial \eta^2} u_y.
\]

\[
2 \left( \frac{\partial^2 u}{\partial \eta^2} \right)
\]

In non-dimensional values:

\[
\phi \left( \frac{U_1^2}{h^2} \right) = \frac{1}{2} \frac{h^2}{x^2} \eta^2 \left( \frac{\partial f}{\partial \eta} \right)^2 + h \frac{Re}{(1-\lambda)} \left( \frac{\partial^2 f}{\partial \eta^2} \right)^2 + \frac{1}{16} \frac{h^3}{x^3} \frac{(1-\lambda)}{Re} \eta^4 \left( \frac{\partial^2 f}{\partial \eta^2} \right)^2 + \frac{1}{4} \left( \frac{h}{x} \right) \frac{Re}{U_1} \eta^2 \frac{\partial^2 f}{\partial \eta^2} u_y + \frac{1}{4} \left( \frac{h}{x} \right) \frac{Re}{U_1} \eta^2 \frac{\partial^2 f}{\partial \eta^2} w_x
\]

\[
+ \frac{1}{4} \frac{h^2}{x^2} \frac{u_x^2}{U_1^2} - \left( \frac{h}{x} \right) \frac{Re}{(1-\lambda)} \frac{u_y}{U_1} \frac{\partial^2 f}{\partial \eta^2}.
\]

Here we see explicitly that the \( \phi \) function is not similar. To evaluate the importance of the relative terms at different x positions, we use \( \frac{u_y}{U_1} \propto \left( \frac{h}{x} \right) \). When \( \frac{x}{h} \rightarrow 0 \), the third, fourth and fifth terms are dominant. The major component of dissipation is \( \left( \frac{\partial u_x}{\partial x} \right)^2 \). The middle term is negative (because \( u_y \) is negative), and the solution gives an interesting function, that below a certain value of \( \frac{x}{h} \) has zero values at a defined \( \eta \). The function produces "bumps" in the \( u_y \) velocity field. When \( \frac{x}{h} \rightarrow \infty \), the second term is dominant. The major component of dissipation is \( \left( \frac{\partial u_x}{\partial y} \right)^2 \). This function is a quite "regular" bell-like function, that decreases and goes flat to zero with x. Plots of the \( \phi \) function are given on Figure b2 for different values of \( \frac{x}{h} \).
\[ \eta' \]

\[ x/h = 10^{-6} \]

\[ y/h \]

\[ u_1^2 a^2 \]

\[ x/h = 10^{-5} \]

\[ y/h \]

\[ u_1^2 a^2 \]

\[ x/h = 10^{-4} \]

\[ y/h \]

\[ u_1^2 a^2 \]
Figure b2: Dissipation function profiles from the similarity solution
On Figure b2 we can appreciate the relative influences of the different terms varying with $\frac{x}{h}$. The influence of the $\left(\frac{\partial u}{\partial x}\right)^2$ factor leads to "bumps", here for values of $\frac{x}{h}$ much below. However, these bumps are not related to those observed in the study.
Appendix C: Entropy generation in mixing flows

We present here Bejan's work [2] [3] on combining Fluid Mechanics and Thermodynamics to derive relations between entropy generation, mass average stagnation pressure, and dissipation function. Our study is limited to the effects of viscous mixing of fluid flow, so we assume negligible heat transfer. We also assume incompressible flow. We start with the fundamental thermodynamic identity, with the assumption of local thermodynamic equilibrium:

$$TdS = dE + pdV = dH - Vdp = dH - nRT\frac{dp}{p}$$

for a perfect gas, with $R$ the universal gas constant ($R \approx 8.314$).

In quantities per unit mass,

$$Tds = dh - RT\frac{dp}{p},$$

where $R = \frac{R}{\Omega}$ is the individual gas constant with $\Omega$ the molar weight of the gas

$$Tds = c_p dT - RT\frac{dp}{p},$$

$$ds = c_p \frac{dT}{T} - R\frac{dp}{p}.$$ 

The same result applies for stagnation quantities:

$$ds = c_p \frac{dT_i}{T_i} - R\frac{dp_i}{p_i}.$$ 

Without any work or heat addition, the first principle of thermodynamics gives for a perfect gas $dh = c_p dT = 0$. Under these conditions,

$$ds = - R\frac{dp_i}{p_i} = - R d(\ln p_i).$$

And for reference values taken at mixed out conditions:

$$s - s_e = - R \ln \frac{p_i}{p_{i_e}}.$$ 

57
To get a simpler expression for (8), we need to assume that the flow is nearly incompressible, which is a consequence of low Mach number. Under these assumptions, 
\[ \frac{dp_t}{pt} \approx 0 . \]

\[ |pt - ne| < p ne \quad (or \ p_t = p ne + o(ne)) . \]

This last result allows us to rewrite (8) as:

\[ s - se \approx - R \frac{p_t - ne}{p ne} , \text{ or} \]

\[ \rho \ T_t (s - se) \approx -(p_t - ne) . \] (9)

The assumptions made for this result are perfect gas : \( PV = n gT \) (or \( P = \rho RT \)), no external shaft work, no working (non-conservative) forces, no heat addition, low Mach number of the flow:

\( M = \frac{|u|}{\sqrt{\gamma RT}} \ll 1 \) and small pressure variations.

In the case of incompressible flow, (9) is a direct result from \( T_t ds = c_e dT_t - \frac{dp_t}{\rho} . \)

The next step deals with Fluid Mechanics. The Navier-Stokes equation for a viscous flow with no body force is:

\[ \rho \ \frac{d\vec{u}}{dt} = \rho \left( \frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla) \vec{u} \right) = -\vec{\nabla} p + \vec{\nabla} \cdot \tau , \]

where \( \tau \) is the viscous stress tensor, and \( \vec{\nabla} \cdot \tau = \text{Div} (\tau) = \sum \frac{\partial \tau_{ij}}{\partial x_j} \vec{e}_i . \)

The variation of kinetic energy is then:

\[ \rho \ \frac{d(\vec{u}^2)}{dt} = -\vec{\nabla} \cdot \vec{p} + \vec{\nabla} \cdot (\pi(\vec{u})) + \vec{\nabla} \cdot \vec{q} . \] (10)

In the absence of any work added to the fluid, the energy equation is:

\[ \rho \ \frac{d}{dt} \left( e + \frac{\vec{u}^2}{2} \right) = -\vec{\nabla} \cdot (p \vec{v}) + \vec{\nabla} \cdot (\pi(\vec{u})) - \vec{\nabla} \cdot \vec{q} + P , \]

where \( e \) is the internal energy per unit mass, \( \pi(\vec{u}) \) represents the viscous stress of the flow, \( \vec{q} \) is the heat flux vector per unit volume, and \( P \) is the heat power dissipated per unit volume. (10) and the identity \( h = e + \frac{P}{\rho} \) yields:

\[ \rho \ \frac{dh}{dt} = \frac{dp}{dt} + (\tau \cdot \nabla \vec{u}) - \vec{\nabla} \cdot \vec{q} , \]
where \( (\tau \nabla \vec{u}) = \sum_{ij} \alpha_i \frac{\partial u_i}{\partial x_j} \) is the tensor product between the viscous stress matrix and the Jacobian matrix of the velocity field.

We now introduce the entropy per unit mass:

\[
Tds = dh - \frac{dp}{\rho},
\]

which leads to:

\[
\rho \frac{ds}{dt} = (\tau \nabla \vec{u}) - \nabla \cdot \vec{q}.
\]  

For a Newtonian fluid, \( \eta = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \left( \nabla \cdot \vec{u} \right) \delta_{ij} \).

And with Fourier's law, \( \vec{q} = -k \nabla T \), we get

\[
\rho \frac{ds}{dt} = \frac{\mu}{T} \sum_{ij} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \frac{2}{3} \mu \left( \nabla \cdot \vec{u} \right)^2 + \frac{1}{T} \nabla \left( k \nabla T \right).
\]

We introduce at this point the dissipation function \( \phi = \sum_{ij} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \), and use the result:

\[
\frac{1}{T} \nabla \left( k \nabla T \right) = \nabla \left( \frac{k}{T} \nabla T \right) + \frac{k}{T^2} \left( \nabla T \right)^2.
\]

(this is exactly the divergence theorem for the integral equation).

Now

\[
\rho \frac{ds}{dt} = \frac{\mu}{T} \phi - \frac{2}{3} \mu \left( \nabla \cdot \vec{u} \right)^2 + \frac{k}{T} \left( \nabla T \right)^2 + \frac{k}{T^2} \left( \nabla T \right)^2. \tag{12}
\]

To extract from this last result the entropy generation, we need to write:

\[
\rho \frac{ds}{dt} = \rho \frac{\delta s_{gen}}{dt} + \frac{\dot{Q}}{T},
\]

where \( \delta s_{gen} \) is the entropy created and \( \dot{Q} \) is the rate of heat received per unit volume. It is easy to show that \( \frac{\dot{Q}}{T} = \nabla \left( \frac{k}{T} \nabla T \right) \).

Subsequently, (12) becomes

\[
\rho \frac{\delta s_{gen}}{dt} = \frac{\mu}{T} \phi - \frac{2}{3} \mu \left( \nabla \cdot \vec{u} \right)^2 + \frac{k}{T^2} \left( \nabla T \right)^2. \tag{12}
\]

It is here noticeable that this result shows that the entropy generated is always positive (the first and second terms on the right side add up to a sum of squares).
For our study, we assume no heat transfer within the flow, i.e. negligible temperature gradient and low Mach number. Bejan in *Entropy Generation Through Heat and Fluid Flow* defines the irreversibility distribution ratio \( \varphi = \left( \frac{\text{fluid friction}}{\text{heat transfer}} \right) = \frac{\mu \phi}{k T \sqrt{\nabla T}} \) to compare the magnitudes of entropy generation from heat transfer and fluid friction.

Put all together, the last assumptions (12) lead to the relation between the entropy generation and the dissipation function:

\[
\rho \frac{\delta s_{\text{gen}}}{dt} = \frac{\mu \phi}{T} .
\]  

The assumptions made for this result are Newtonian fluid (isotropic viscous fluid, with mechanical elasticity and symmetry properties, Stokes’s assumption, and constant viscosity coefficient –see 1.13 in *Internal Flow*), no body force, no heat transfer or heat source, no external work added, and incompressible flow.

To combine the two main results (9) and (13), we do a control volume analysis. With all the assumptions made, we define a control volume by a stream tube closed by two sections. Because the flow, without external work, tends towards a maximal entropy state, we consider mixed out conditions (uniform flow) at the exit.

Figure c1 shows the control volume for our study, and defines a longitudinal coordinate \( x \).

We recap the integral conservation equations for incompressible steady flow:

\[
\overline{u_x} A(x) A(x) = \int_{A(x)} \nu_x dA = u_e A(x_e) .
\]  

(conservation of mass)

\[
\left[ \overline{p u_x^2} (x_e) + \frac{1}{2} \rho u_x^2 A(x) \right] = \left[ \rho u_e + \frac{1}{2} \rho u_e^2 \right] A(x_e) .
\]  

(conservation of impulsion without external force – viscous forces are neglected at the boundaries)

For steady flow, the entropy inside the control volume is constant:

\[
\frac{D S}{dt} (x) = \int_{A(x)} \rho u_x s dA - \rho u_e s_e A(x_e) + \int_{\partial A(x)} \dot{Q} dV + \frac{\delta s_{\text{gen}}}{dt} = 0 .
\]

The terms on the right side are respectively the entropy inflow and outflow, the heat transfer integral (that equals the heat flux integral over the contour), to be neglected, and the entropy generated.
The definition of the mass average pressure, together with (9) and (14), transforms (16) to:

\[
- \rho u_x A(x_e) \left( \frac{\overline{\rho} M (x_e) - p_e}{T_i} \right) + \frac{\partial \overline{\rho} M_{gen}}{\partial t} = 0,
\]

\[
\frac{\partial \overline{\rho} M_{gen}}{\partial t} (x_i) = \frac{\rho u_x A(x_e)}{T_i} \left( \overline{\rho} M (x_e) - p_e \right) = \frac{\dot{m}}{\rho T_i} \left( \overline{\rho} M (x_e) - p_e \right). \quad (17)
\]

The integral form of (16) is

\[
\frac{\partial \overline{\rho} M_{gen}}{\partial t} (x_i) = \iiint_{CV(x_i)} T \phi dV.
\]

Now we have the direct relation between the dissipation function \( \phi \) at the local scale, and the global mass average stagnation pressure decrease, or mechanical energy loss:

\[
\Delta p_{loss} (x_i) = \overline{\rho} M (x_i) - p_e = \left( \frac{T_i}{\rho u_x A(x_e)} \right) \iiint_{CV(x_i)} T \phi dV,
\]

\[
\overline{\rho} M (x_i) = \Delta p_{loss} (x_i) + p_e. \quad (18)
\]

(conservation of mechanical energy)

To conclude with the control volume analysis of mixing flows, we write the main integral results for the simple case of 2-dimensional flow. We assume a constant area channel, and set the inlet
conditions to two uniform parallel flows. These two different flows come out of two channels of same area.

The control volume analysis gives from (14), (15) and (19):

\[
\frac{\Delta p_{\text{loss}}}{\frac{1}{2} \rho U_1^2} = \frac{(p_1 - p_e) + \lambda (p_2 - p_e)}{(1 + \lambda) \frac{1}{2} \rho U_1^2} + \frac{3}{4} (1 - \lambda)^2,
\]

where \( \lambda = \frac{U_2}{U_1} \). We can solve this set of equations with a convenient number of known quantities.

Except for \( \Delta p_{\text{loss}} \) that has to be known from experimental results, the boundary conditions must provide at least 3 quantities with at least one reference pressure or stagnation pressure. Because the pressure field in incompressible flow is defined to within a constant, we can set for example the exit pressure to be the reference pressure, i.e. \( p_e = 0 \). If we decide to set for example velocity inlet conditions, we can simply express any quantity as a function of \( U_1, \lambda, \) and \( \Delta p_{\text{loss}} \):

\[
p_\text{i} = \left(1 + \frac{\lambda}{1 - \lambda}\right) \Delta p_{\text{loss}} - \frac{1}{2} \rho U_1^2 \frac{(1 + \lambda) (3 - \lambda)}{4},
\]

\[
p_\text{a} = - \left(1 + \frac{\lambda}{1 - \lambda}\right) \Delta p_{\text{loss}} + \frac{1}{2} \rho U_1^2 \frac{(1 + \lambda) (3 - \lambda)}{4},
\]

\[
p_1 = \left(1 + \frac{\lambda}{1 - \lambda}\right) \Delta p_{\text{loss}} - \frac{1}{2} \rho U_1^2 \frac{(1 - \lambda) (1 - 3\lambda)}{4},
\]

\[
p_2 = - \left(1 + \frac{\lambda}{1 - \lambda}\right) \Delta p_{\text{loss}} - \frac{1}{2} \rho U_1^2 \frac{(1 - \lambda) (1 - 3\lambda)}{4},
\]
\[ p_1 - p_2 = 2 \left( \frac{1 + \lambda}{1 - \lambda} \right) \Delta p_{\text{loss}} + \frac{1}{2} \rho U_1^2 \frac{(1 + \lambda)(1 - \lambda)}{2} . \]  

From (23), we observe that \( p_1 - p_2 \) is of the same sign as \( U_1 - U_2 \). The flow of higher velocity has to be the flow of higher stagnation pressure and vice versa. Moreover, there is a minimal stagnation pressure difference to be applied for a defined velocity ratio at the inlet (that corresponds to the isentropic case \( \Delta p_{\text{loss}} = 0 \)).

<table>
<thead>
<tr>
<th>Case</th>
<th>( \Delta p_{\text{loss}} ) ( \frac{1}{2} \rho U_1^2 )</th>
<th>( \frac{p_1}{\frac{1}{2} \rho U_1^2} )</th>
<th>( \frac{p_2}{\frac{1}{2} \rho U_1^2} )</th>
<th>( \frac{p_1}{\frac{1}{2} \rho U_1^2} )</th>
<th>( \frac{p_2}{\frac{1}{2} \rho U_1^2} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Equal inlet pressure</td>
<td>0.0625</td>
<td>0.875</td>
<td>0.125</td>
<td>-0.125</td>
<td>-0.125</td>
</tr>
<tr>
<td>Numerical Simulation</td>
<td>0.0751*</td>
<td>0.8965*</td>
<td>0.1035*</td>
<td>-0.1035*</td>
<td>-0.1314*</td>
</tr>
<tr>
<td>Re=1000</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Numerical Simulation</td>
<td>0.0783</td>
<td>0.9224</td>
<td>0.0776</td>
<td>-0.0776</td>
<td>-0.1724</td>
</tr>
<tr>
<td>Re=100</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Numerical Simulation</td>
<td>0.1150</td>
<td>0.931</td>
<td>0.069</td>
<td>0.069</td>
<td>-0.3554</td>
</tr>
<tr>
<td>Re=10</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*Table c1: Some direct results of a control volume analysis (simple 2D case of mixing parallel flows, \( \lambda=0.5 \))

Table c1 presents some values computed at different Reynolds numbers, together with the solution from a control volume analysis (assuming uniform inlet pressure). We observe that the static pressure of the high velocity channel can be either higher or lower than the static pressure of the low velocity channel. Unlike what we could expect, equal pressures at the inlet is not a good boundary condition at low Reynolds numbers. The mixing cannot be considered at a constant pressure near the inlet. This is the main reason why we used a splitting wall at the inlet, rather than a velocity profile with a discontinuity.
Bibliography


http://www.fluent.com/about/news/newsletters/04v13i1/pdfs/n34.pdf


http://web.mit.edu/fluids/www/Shapiro/ncfmf.html
