FACULTY OF ENGINEERING AND COMPUTING

School of Mechanical, Aerospace and Automotive Engineering

PROJECT

Interview Review

CFD assessment of a steady and pulsed jet in a catalytic converter with a 2D mesh and a 3D mesh.

Author

Mara Salut Escartí Guillem

Supervisor

Dr Humberto Medina

May 13, 2017

Declaration of Originality

This project is all my own work and has not been copied in part or in whole from any other source except where duly acknowledged. As such, all use of previously published work (from books, journals, magazines, internet, etc) has been acknowledged within the main report to an item in the References or Bibliography lists.

I also agree that an electronic copy of this project may be stored and used for the purposes of plagiarism prevention and detection. I understand that cheating and plagiarism constitute a breach of University Regulations and will be dealt with accordingly.

Copyright Acknowledgement

I acknowledge that the copyright of this project and report belongs to Coventry University.

Signed: Mara Salut Escarti Guillem Date: 09.05.2017

Office Stamp

Acknowledgements

I want to express my gratitude to Coventry University and all the people who are part of it for giving me this opportunity. Especially to my supervisor Dr. Humberto Medina, for his encouragement and passion. I would have never end my project without our casual meetings in front of the library.

I am grateful for all the people I have known during this time and that today I can consider my friends. I am taking home a lot of things. Although the suitcase is big, too many things does not fit, but the heart is always plenty of space.

'A constant element of enjoyment must be mingled with our studies, so that we think of learning as a game rather than a form of drudgery, for no activity can be continued for long if it does not to some extent afford pleasure to the participant.'
Desiderius Erasmus of Rotterdam

'Scientists discover the world that exists, engineers create the world that never was.'
Theodore Von Karman

'Perqué hi haurà un dia que no podrem més i llavors ho podrem tot.'
Vicent Andrés Estellés

'All that I am or hope to be I owe to my mother.'
Abraham Lincoln

Abstract

Due to the increase of the cars pollution, the automotive industry has included catalytic converters to its exhaust gases in the vehicles. But the flow behaviour inside of the catalytic monolith and in the diffuser affects highly to the performance and efficiency of this systems. In this study a CFD study through the software OpenFOAM has been applied to this configuration. The $v2f$ turbulence model have been used for the assessent of the steady and transient flow conditions. Moreover, each flow conditions have been analysed in a 2D mesh and in a 3D mesh. The results for the steady flow have been validated against a measured data by the HWA technique. It has been found that the configuration with the 3D mesh capture better the flow behaviour in the first part of the lateral distance while the 2D mesh capture better the second peak of velocity. Finally, regarding the pulsating flow results the velocity contours for both cases have been compared realising that they are quite similar.

Contents

List of Figures

List of tables

Notation

ϵ	Turbulent dissipation rate
μ	Dynamic viscosity
ν	Kinematic viscosity
ν_R	Viscosity ratio
ν_T	Kinetic eddy viscosity
$\overline{U}$	Time-averaged velocity
ρ	Density
τ_{ij}	Reynold-stress tensor
ΔP	Pressure drop
d_h	Hydraulic diametre
d_m	Viscous term for the porous media model
f	Damping Function
f_m	Inertial term for the porous media model
g	Gravitational acceleration
k	Kinetic energy
L	Length scale
p	Pressure
P_i	Porous intertial resistance tensor
P_u	Porous viscous resistance tensor
Re	Reynolds number
S	Source term
s_{ij}	Strain rate tensor
T	Tempreture scale
t	Time
t_{ij}	Viscous stress tensor
U	Mean velocity
u	Vector velocity
u'	Fluctuating velocity
U_i	Mean velocity at the inlet
$v2$	Normal stress function
x	Vector position
y_+	Non dimensional distance from surface
A	Wetter surface
P	Perimetre

1 Introduction

1.1 Background

Due to the increase in the number of cars in circulation, the emissions regulations have become more and more stringent. In order to reduce the exhaust gases of the cars the automotive industry has included catalytic systems in its vehicles. Catalytic converters, by catalysing a redox reaction, converts the pollutants of the exhaust gases into less harmful emissions.

The catalytic converter is located after the tailpipe of the motor. The catalyst is a monolith structure composed of small diameter channels. Having this channels the total reaction area is improved, producing a more effective result. Inside of the channels an energy loss is produced resulting in a decrease of the efficiency. If a smaller length monolith is used, less energy is lost when the flow goes through it. As a result, if the reaction volume wants to be maintained the transversal section of the catalyst needs to be bigger than the one from the pipe.

Due to space limitations a short wide-angled diffuser is used to connect the inlet pipe and the monolith. Although, the propagation of the flow across this diffusers increase the flow poor distribution (Clarkson, 1995 [7]). In figure 1 it is shown the typical catalyst assembly.

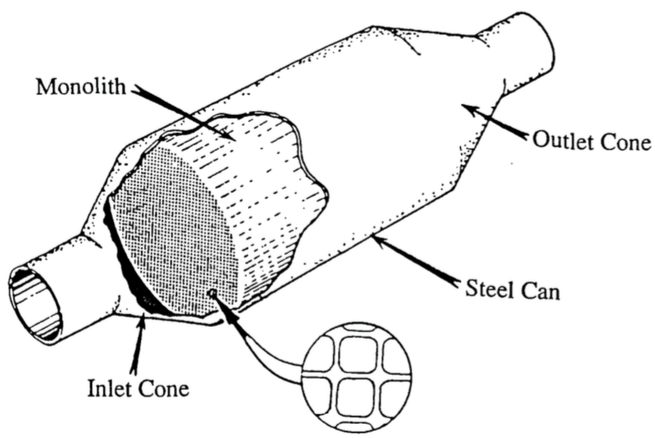


Figure 1: Typical monolith type catalyst assembly (Clarkson 1995:4).

Previous studies on catalytic systems agree on the fact that the poor distribution of the flow arriving to the catalyst affects to its performance and efficiency (Porter et al., 2016 [16]). Due to this, it is highly important to understand the flow distribution in order to develop the best possible design.

Computational Fluid Dynamics, better known as CFD, have been widely used to analyse the performance of the flow on this configuration. CFD is an numerical tool which use modelling procedures to investigate the behaviour of the flow. It is a good alternative to empirical studies because it has a lower operational cost regarding time and money. This is why it has been used in a lot in different engineering areas with excellent outcomes.

The three principals approach of modelling the turbulence are: Direct Numerical Simulation (DNS), Large Eddy Simulation (LES) or Reynolds-averaged Navier-Stokes (RANS). DNS has been applied to this configuration by Cokljat et al. (2003 [8]) and Ozhan et al. (2014 [14]) with good results. However, the implementation of DNS requires a high operational cost leading it as a researching tool. Another approach is the use of LES as Catalano et al. (2003 [6]) did. They compared the outcome against results from RANS and Unsteady-RANS simulations. LES simulations provided the best results while URANS had a good agreement with the experimental data and RANS predictions were not as good as the others. Respect DNS, the simulations performed in LES does not require such dense mesh but they still are associated with an important operational cost. As the RANS approach is the combination of good results and a reasonable computational cost, it will be along with the URANS approach the turbulence model used.

1.2 Aims and objectives

This assessment aims to analyse, through the open software *OpenFOAM*, the flow behaviour of the system consisting of a wide-angle diffuser and a catalytic converter.

As stated by Porter (2016) [15], the study of steady flows add important knowledge about the performance of a catalytic system. Nevertheless, the operational conditions of the flow across the exhaust gases are those of a pulsed one. A recent study, reveals that the results of the simulation of the steady flow in 3D showed good agreement with the ones perfomed in 2D (Borisov 2016, [5]). According to this, the 2D mesh can approximate satisfactorily the real conditions. Along with this, the operational cost will be improved.

In order to cover all this aspects, the study is going to be done in increasing order of complexity. It will cover all the aspects following the consecutive path:

- Steady jet in 2D modelled with RANS (Reynolds-Averaged Navier-Stokes) equations.
- Steady jet in 3D modelled with RANS (Reynolds-Averaged Navier-Stokes) equations.
- Pulsed jet in 2D modelled with URANS (Unsteady RANS) equations.
- Pulsed jet in 3D modelled with URANS (Unsteady RANS) equations.

The objectives of this study are to:

- Carry out a literature investigation to gain an understanding of the problem identifying the flow distribution linked to the diffusor and the monolith in each configuration.
- Define the geometries and the meshes with the open source software Salome.
- Perform a series of RANS and URANS simulations for a two-dimensional and a three-dimensional domain and obtain the velocity and pressure distribution in each configuration.
- Explain the results obtained with the CFD simulations and discuss the accuracy by comparing them with the experimental data that is counted.

1.3 Structure of the project

The following sections will describe more deeply the theoretical approach that has been carried out and its results.

In Chapter 2 the pertinent literature for the understanding of the project is given. Moreover, in this chapter are also mentioned the previous studies. Then, the methodology is shown in the Chapter 3. In the mentioned chapter, the approach followed and the simulation procedure can be found.

Chapter 4 details the results achieved through the different simulations and compare them. Lastly, in Chapter 5 the conclusions of this assessment study will be exposed.

2 Literature Review

As it has been mentioned, this section will highlight the theory that has been used to develop the mathematical model. It reviews the studies that had been done about catalytic converters and the knowledge that it has been gained. There are also introduced the governing equations of the domain and the turbulent model that will be applied.

2.1 Previous studies

The geometry of the catalytic converter has been widely studied during the last years. It is namely formed by an inlet pipe, a diffuser, a porous monolith and an outlet sleeve. In figure 2 it is displayed the flow behaviour in a catalytic converter. This pictures reflects what has been found in previous studies. In the first picture can be noted the flow field in the diffuser and the typical velocity profile inside monolith. In the second one it is displayed the recirculation bubbles formed at channel walls that file inside of the monolith.

The inlet pipe connect the exhaust system with the diffuser. The velocity inlet in its operational conditions is unsteady. During years catalytic converters were studied with steady flow due to the high computational time that assess an unsteady flow takes. Moreover, it was believed that concentrating in simple configuration would lead to a better understanding of the relationship of the variables involved. This way the design of more complex configuration would be improved. Benjamin et al. (1996 [3]) stated that the velocity profile arriving to the diffuser through the inlet has a big impact over the development of the flow.

As it has been stated previously, short wide-angled diffusers were added to catalytic converters because of space constraints. Due to the geometry of this kind of diffusers, the expansion of the flow produces flow separation phenomena. A big improvement for the research was the introduction of the non-intrusive measurements techniques. Quadri et al. (2013) conducted PIV measurament in a planar diffuser. This study showed that a free shear layer causes the separation of the core jet from recirculating vortices which remain in both parts of the diffuser. As it can be seen in Figure 2, when the jet reaches the centre of the entrance of the monolith it spreads radially. This leads to that a part of the incoming jet enters the monolith but the rest of the flow moves away from the monolith entering in recirculating regions placed between the monolith and the diffuser. The blue line inside the monolith in figure 2 represents the velocity profile inside of it. The flow that enters into the central part of the channels goes axially, rising higher velocities, while the flow approaching the most distant channels enters obliquely. The flow approximating the channels situated close to the wall come to a standstill. As a result, there is a pressure rise that force some flow to enter into the closest channels to the wall, producing a velocity peak. In consequence, the flow is said to enter into the monolith poorly distributed.

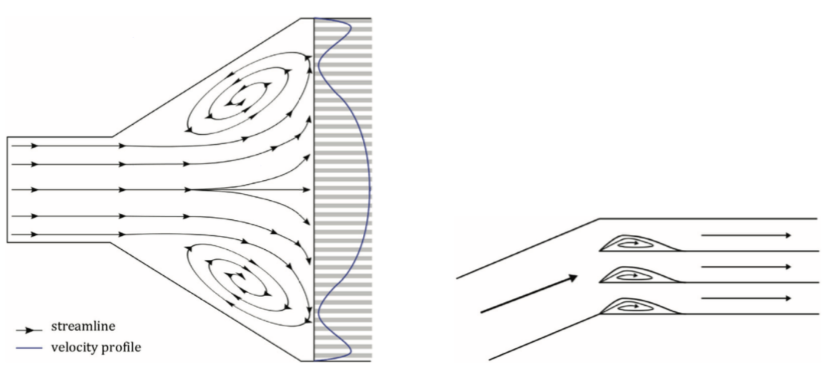


Figure 2: Flow behaviour in a catalytic converter (Porter et al. 2016:8436).

Consequently, it can be said that the geometry of the diffuser is a key factor in the evolution of the velocity profile that arrives to the monolith. Wollin (2002 ,[24]) studied the influence of different diffusers with different expansion angles. Bigger diffuser angles result to produce a less uniform profile than the smaller one in pulsating conditions.

As it has been said, the monolith is a structure formed by small diameter channels. The flow poor distribution that is achieved produce important effects to the pressure drop as well as to the conversion efficiency and the degradation rate of catalyst. Due to the importance of the flow uniformity, several studies have been done in order to assess the influence of different variables. Reaching the conclusion that the flow physics are more uniform when a smaller Re is set and with higher monolith resistance (Clarkson, 1995 [7]).

Along with the PIV technique, HWA measurements have been done in an attempt to characterise the flow downstream of the monolith. Obtaining experimental data from before and behind the diffuser the understanding achieved of the flow phenomena is improved. Mat Yamin (2013, [25]) measured different pulsating flow regimes, finding that with a smaller value of Re or with higher frequency the flow reattached downstream of the separation bubbles.

In the further investigation of flow behaviour in the monolith, Arias-Garcia et al. (2001, [2]) assessed a close coupled catalyst in steady and pulsating flow conditions. It was observed that the maldistribution for the pulsating conditions resulted in a improvement of the flow uniformity. To analysed the influence of the pulsating flow, Benjamin et al. (2002, [4]) contrast different flow regimes. It was highlight that with bigger frequency the flow maldistribution was reduced producing a smaller pressure drop.

In an attempt of validate different turbulence models, Borisov (2016 [5]) applied the viscosity ratio approach to five different models for a steady flow in a 2D domain of a catalytic converter. The SST-k-w result to be the one with best agreement with the experimental results along with more stability. Due to this, it was implemented to a 3D domain. As the results did not showed a big improvement it was concluded that in order to spend less operational cost the 2D domain was good enough. Furthermore, this study analysed an unsteady flow where all the turbulence models provided similar results.

Porter (2016, [15]), also assessed the unsteady flow using the URANS approach and validate this results against PIV measurements. This study highlights that the CFD approach overpredicted the vorticity magnitudes in the phased-averaged condition while showed good agreement for the instantaneous values. This suggests that other turbulence models could be used in order to obtain more detailed knowledge. This study also did an important contribution developing an hybrid model for modelling the monolith. This hybrid model combine the porous media approach and the individuals channels one. This models represents the equilibrium between a lower operational cost and catching adequately the flow behaviour. Despite of this improvement, in this assessment the porous media will be implemented because it does not require such a dense mesh.

2.2 Governing equations

The basic equations which described the flow dynamics are the conservation of mass, conservation of momentum and the energy equation. Applying this equation to our control volume, the governing equations are the ones that follow:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0 \tag{1}$$

$$\rho \frac{\partial}{\partial t}(u_j) + \rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial t_{ij}}{\partial x_j} + \rho g_i \tag{2}$$

Equation 1 is the conservation of mass equation, where the vectors u_i and x_i are velocity and position, ρ is the density and g_i is the gravitational acceleration. Equation 2 is the conservation of momentum equation or Newton's second Law. Where t_{ij} is the viscous stress tensor defined by Wilcox (2010, p.39 [23]) as

$$t_{ij} = 2\mu s_{ij} \tag{3}$$

where μ is molecular viscosity and s_{ij} is the strain-rate tensor

$$s_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{4}$$

To simplify equations 1 and 2 we can consider the flow as incompressible and we will despise the gravitational forces, thus:

$$\frac{\partial}{\partial x_j} u_j = 0 \tag{5}$$

$$\frac{\partial}{\partial t}(u_j) + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial t_{ij}}{\partial x_j} \tag{6}$$

Considering equation 5, we can rewrite the convective term of equation 6 as:

$$u_j \frac{\partial u_i}{\partial x_j} = \frac{\partial}{\partial x_j}(u_j u_i) - u_i \frac{\partial u_j}{\partial x_j} = \frac{\partial}{\partial x_j}(u_j u_i) \tag{7}$$

leading to the Navier-Stokes equation in conservation form:

$$\frac{\partial}{\partial t} u_j + \frac{\partial}{\partial x_j}(u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(2\mu s_{ij}) \tag{8}$$

At this point, we need to introduce the Reynolds Averaging concepts because the turbulence is composed of random fluctuations of the flow. Reynolds averagin, introduced by Reynolds in 1895, express all quantities as the sum of the mean and fluctuation part (Wilcox, 2016 [23]). Thus, we can express the instantaneous velocity, $u_i(x,t)$ as the sum of a mean, $U_i(x)$, and a fluctuating part, $u_i'(x,t)$, so that

$$u_i(x,t) = U_i(x) + u_i'(x,t) \tag{9}$$

The mean velocity, $U_i(x)$ is the time-averaged velocity that can be defined as in equation pi. Its mean velocity is again the same time-averaged value, i.e. $\overline{U_i} = U_i$. Meanwhile, the time average of the fluctuating part of the velocity is zero.

$$U_i = \lim_{T\to\infty} \frac{1}{T} \int_t^{t+T} u_i(x,t)\, dt \tag{10}$$

If we apply the time averaging to the Navier-Stokes equation, we obtain the Reynolds-averaged Navier-Stokes equation (RANS) for an incompressible flow:

$$\frac{\partial U_j}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}(2\mu s_{ij} - \overline{u_j' u_i'}) \tag{11}$$

Du to the time averaging some momentum fluxes appear acting as apparent stresses across the flow (Wilcox, 2010:33). This tensor is the term $\overline{u_j' u_i'}$, known as the Reynolds-stress tensor and it is denoted as

$$\tau_{ij} = \overline{u_j' u_i'} \tag{12}$$

This tensor is composed by six components which along with the mean flow properties are the unknown values that create the closure issue. At this point, there are four equations, the mass conservation and the three components of equation 11. Moreover there are ten unknowns, the 6 six components of the Reynolds-stress tensor along with the three component of the velocity and the pressure. Hence, the closure issues can be resolved modelling the tensor τ_{ij} with a turbulent model. A good turbulence model is the key to model the flow behaviour.

Regarding the URANS equations, they are the same as in equation 11. They are called Unsteady-RANS because during the computation the transient term $\partial U_j/\partial t$ is retained. For URANS, a time step is implemented for the unsteady solution. Meanwhile in RANS the flow properties are disintegrated into their mean and fluctuating components and integration over time is performed (Salim, 2011 [19]).

2.3 $v2 - f$ turbulence model

In the setting of a free flow, the kinetic energy of the Reynolds stress tensor components is redistribute by the pressure fluctuations, this effect is known as 'return to isotropy'. The existence of walls produces the 'wall-blocking effects'. The Reynolds stress becomes highly anisotropic because the transfer of energy between the stream-wise and the wall-normal velocity fluctuations is diminished by remote interaction of pressure fluctuations with the solid wall (SolKeun, 2012) [21]. Due to this, the wall normal component, v^2, and shear stress are strongly reduced.

To model the turbulence of the flow the linear eddy viscosity that has been chosen is the $v2 - f$ model, where $v2$ is the wall normal stress function and f is the elliptical function. . This model has been chosen due to its good results in previous studies. Although, there are models with slightly better results at some points, this model can be used for all the conditions studied here. Moreover, as in the last studies this turbulence model has been one of the most used, follow this tendency has been attempted.

The $v2 - f$ model was firstly suggested by Durbin in 1991 [9], as an alternative to the $k - \epsilon$ model. The $k - \epsilon$ is a two-equation model that resolves the closure problem for the kinetic energy, k, and the dissipation rate ϵ. In addition to this, the $v2 - f$ model also solves the v^2 and the f functions. It has been proven that this model works effectively, but achieving the convergence is complicated due to the walls boundary conditions that require a coupling of variables at walls (Laurence et al., 2004 [11]). For this reason, an improved model developed by Lien and Kalitzin (2001 [12]) is used.

It consist of four equations, three equations of transport for k, ϵ, and $v2$ along with an elliptic relaxation equation for the f term.

Turbulent Kinetic Energy

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \epsilon + \frac{\partial}{\partial x_j} \left[(\nu + \nu_t/\sigma_k) \frac{\partial k}{\partial x_j} \right] \tag{13}$$

Dissipation Rate

$$\frac{\partial \epsilon}{\partial t} + U_j \frac{\partial \epsilon}{\partial x_j} = C_{\epsilon 1} \frac{\epsilon}{k} \tau_{ij} \frac{\partial U_i}{\partial x_j} - C_{\epsilon 2} \frac{\epsilon^2}{k} + \frac{\partial}{\partial x_j} \left[(\nu + \nu_t/\sigma_k) \frac{\partial \epsilon}{\partial x_j} \right] \tag{14}$$

$\overline{v}^2$ transport equation

$$\frac{\partial \overline{v}^2}{\partial t} + U_j \frac{\partial \overline{v}^2}{\partial x_j} = kf - 6\overline{v}^2 \frac{\epsilon}{k} + \frac{\partial}{\partial x_j} \left[(\nu + \nu_t/\sigma_k) \frac{\partial \overline{v}^2}{\partial x_j} \right] \tag{15}$$

Elliptic relaxation equation

$$L^2 \frac{\partial^2 f}{\partial^2 t} - f = \frac{1}{T} \left[(C_1 - 6) \frac{\overline{v}^2}{k} - \frac{2}{3}(C_1 - 1) \right] - C_2 \frac{P_K}{k} \tag{16}$$

Closure Coefficients

$$C_\mu = 0.22, \qquad \sigma_k = 1, \qquad \sigma_\epsilon = 1.3 \tag{17}$$

$$C_{\epsilon 1} = 1.4 \left[1 = 0.05 \sqrt{\frac{k}{\overline{v}^2}} \right] \qquad C_{\epsilon 2} = 1.9 \tag{18}$$

$$C_1 = 1.4, \qquad C_2 = 0.3, \qquad C_L = 0.23, \qquad C_\eta = 70 \tag{19}$$

The length scale of the model, as defined in equation 20, is based on flow properties, specifically the kinetic energy, k, the dissipation rate, ϵ, and the kinematic viscosity , ν, (SolKeun, 2012) [21].

$$T = max \left[\frac{k}{\epsilon}, 6\sqrt{\frac{\nu}{\epsilon}} \right] \qquad\qquad L = C_L max \left[\frac{k^{\frac{3}{2}}}{\epsilon}, C_\eta \frac{\nu^{\frac{3}{4}}}{\epsilon^{\frac{1}{4}}} \right] \tag{20}$$

2.4 Summary

RANS equations will be applied to the steady flow while URANS equations will be used to solve the unsteady flow. The main difference is that a time step is implemented for the unsteady solution while for the RANS the flow properties are disintegrated into their mean and fluctuating components and integration over time. From RANS equations arise the closure problem due to the fact that with the presence of the Reynolds-stress tensor there are ten unknowns variables but only for equations. In order to solve the closure issue the turbulence model applied is the $v2f$ developed by Lien and Kalitzin (2001 [12]). This model consists of four equations, three of transport for the kinetic energy, k, the dissipation rate, ϵ and the wall normal stress, $v2$, and an elliptic relaxation equation for the f term.

3 Methodology

According to all the stated on the previous chapter, the open software *OpenFOAM* script has been employed in order to produce a mathematical model for the configuration of study.

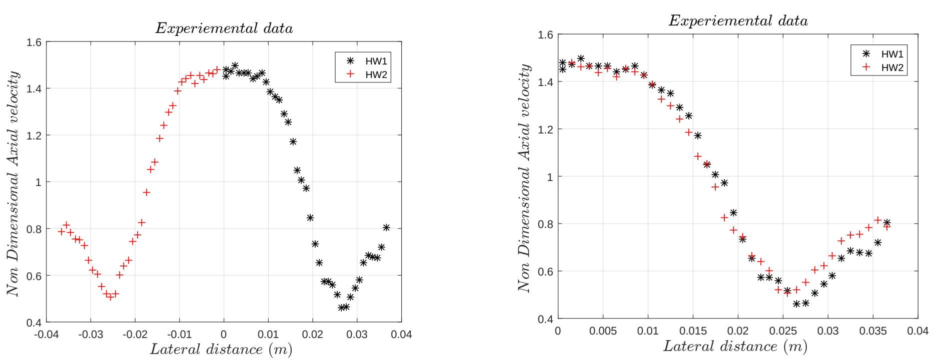
3.1 Research approach

Here is presented an overview of the research approach followed. The research process followed is an inductive method, because a theory is pretend to be obtained from the data and the study has began with a research question. Regarding the research philosophy, a quantitative research has been done, where the aim is to find a relationship between the independent variables and the outcome variable.

The principal research methodology are numerical experiments. Simulation with OpenFOAM in the operating system of OS X and Linux had been carried. The model of approach is RANS and URANS with the turbulence model v2-f. Finally, concerning the data collection, the data used through this assessment can be considered as secondary data: because it has been collected in a indirect way. Examples of secondary data used are papers or thesis. Moreover, the results from simulations by CFD can also be covered in secondary data. The data to validate the results is experimental. It was provided by the supervisor so it can also be considered as secondary data too.

3.2 Experimental data

The experimental data used for the validation of the model was the used previously by Mat Yamin (2012) [25] and Sophie Porter (2016) [15]. This measurements were obtained using a Hot Wire Anemometry, HWA. With this process the axial velocity profile is captured downstream of the monolith. More precisely, the data used was the one that belongs to a Reynold number of $2.2x10^4$. In figure 3 are presented the measured results. In figure (a) is plotted the data for both sides of the symmetry plane. As for the CFD simulations the domain is axisymmetric only one half of the domain will be solved. In order to validate this results, both series of data are represented together because as it can be seen there are slight differences between them.



(a) Results for the full length of the geometry (b) Results modified fot the axissymetric domain

Figure 3: Experimental points

3.3 Mesh

3.3.1 2D Mesh

In figure 4 it is displayed the domain used for the study. It is divided in four main parts: the inlet, the diffuser, the monolith and the outlet. This domain is a planar approximation to the real one. this domain is considered because in previous studies it has been demonstrated that the flow can be held as a two dimensional one. Thus, it is also possible to define it as axisymmetric.

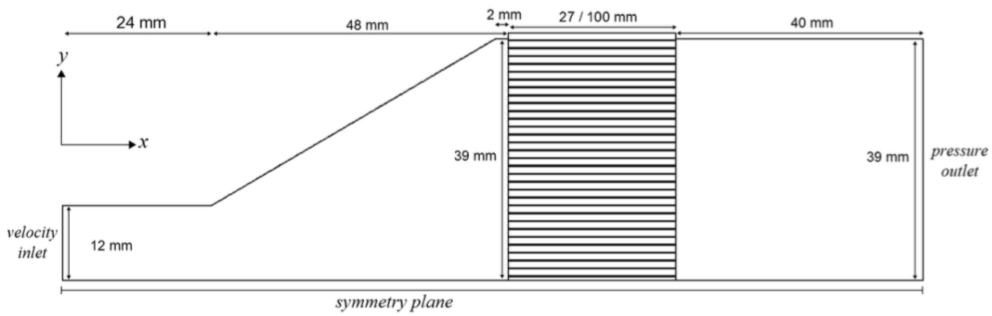


Figure 4: The geometry of the domain (Porter 2016:41).

The mesh used is displayed in the figure 5 and it was provided by Humberto Medina. This mesh has already been used in the studies of Sophie Porter [15] and Dimitar Borisov [5] and so its quality has been proved. The mesh has an overall domain bounding box of 139 mm of height and 39mm of width. It is formed of 28423 cells, which are primarily hexahedral. Despite that it is a 2D domain there is one cell in the Z direction, but the influence of this direction is not taken into account. The width of the cells decrease as you get closer to the wall, resulting in a $y_+ < 1$ that allows the model to capture the flow physics in this region.

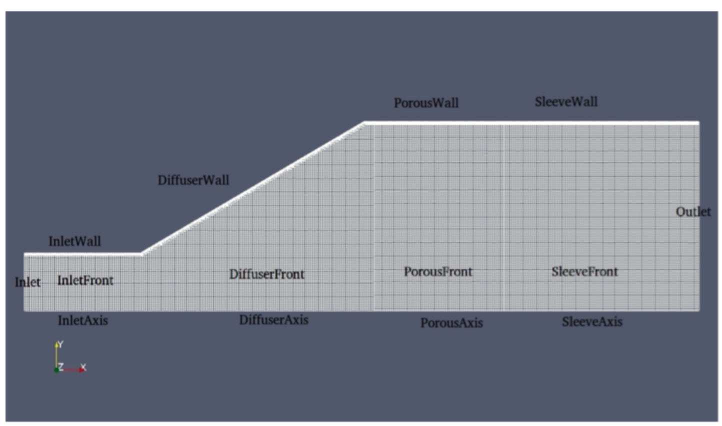
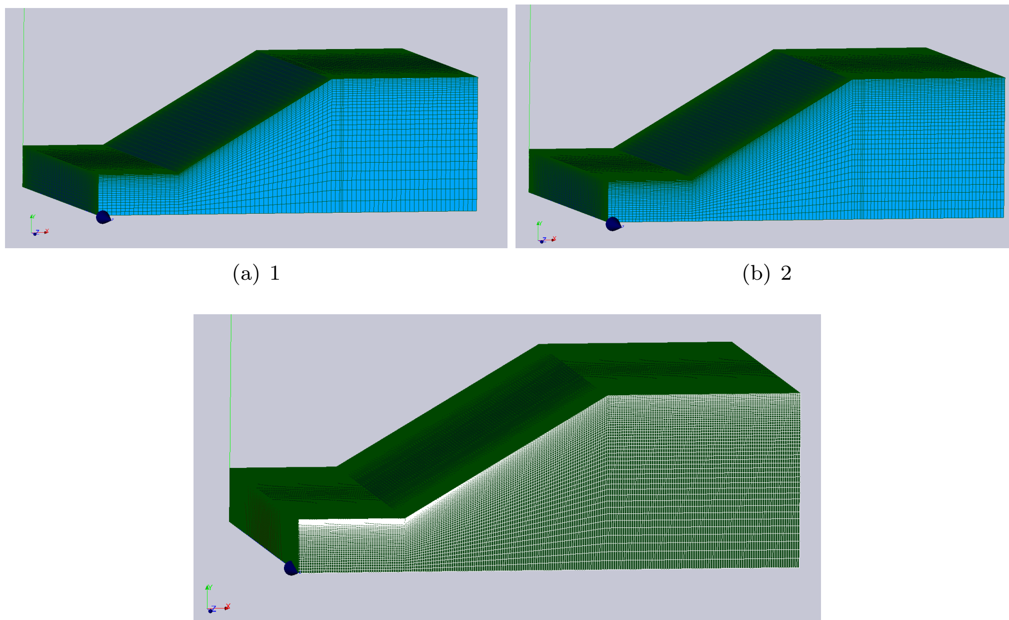


Figure 5: The 2D grid used (Borisov 2016:18).

3.3.2 3D Mesh

In order to assess a deeper analysis a 3D mesh is also used. This mesh was generated in the open-source software Salome by Boristov (2016, [5]) and used for his analysis. One of the main difference with the 2D mesh is that the width is of 96 mm. The second important difference is that the porous media is modelled as a 2mm region while the 2D mesh has a 27 mm region. It is formed of 4162328 cells with a maximum $y_+ = 5.84$ and a minimum $y_+ = 0.023$.



(a) 1

(b) 2

(c) 3

Figure 6: 3D Mesh

Two mesh independence study has been carried out with the steady flow condition. On one hand, it has been analysed the dependence of the results with the global mesh. In figure 6, the three studied grids can be seen. It has been changed the number of segments in which the side mesh is discretised. The first mesh shown in figure 6 is the less refined and the third one is the most refined. This is the one that has been used through this study.

The second mesh dependence study revolve around the porous media mesh. In the original mesh this region was divided in two cells in the X-axis direction. The number of cells has been improved to 4 and 6 in two new grids. This has been done in order to assess if the refinement of this part of the mesh ables to capture a better flow behaviour.

3.4 Monolith model

A high operational cost can be achieved if the monolith is resolved through performance its complicated geometry. But the pressure drop that this surface does over the flow can be reproduced by modelling an analogous reduced surface.

Hafsteinsson [10] states that the pressure drop can be modelled by attenuating the derivate of the time and adding a sink term to the momentum equation

$$\frac{\partial U_j}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + S_i \tag{21}$$

The source term, S_i, is composed by a viscous loss term and a inertial loss term. It is represented by the

Darcy-Forcheimer equation

$$S_i = -\left(\mu D + \frac{1}{2}\rho|u_{jj}|F\right) \tag{22}$$

Equation 22 can be written as a second degree polynomial with respect to velocity as

$$S_i = -(P_i|u| \cdot u + P_u \cdot u) \tag{23}$$

where P_i is the inertial resistance tensor and P_u is the viscous resistance tensor.

Porter (2016 [15]) characterised the axial resistance tensors of a 27 mm and a 100 mm monolith. For this study only the 27 mm monolith values will be used. She measured the values of the velocity and the pressure drop produced per unit of length. In figure 7, are represented the points of both monoliths and a fitted second order line for each.

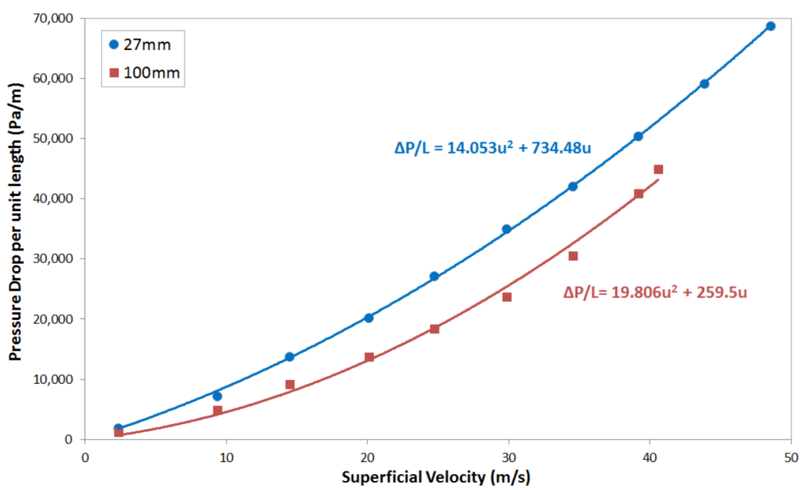


Figure 7: Measured pressure drop per unit length as a function of superficial velocity (Porter, 2016:35).

From figure 7 can be obtained a second degree function for the 27 mm monolith as

$$\frac{\Delta P}{L} = 14.053 \cdot u^2 + 734.47 \cdot u \tag{24}$$

Hence, the inertial resistance tensor is $P_{ix} = 14.053 kg/m^4$ and the viscous resistance tensor is $P_{ux} = 734.48 kg/m^3$. This values are the ones that will correspond to the 2D mesh because the porous monolith region in this mesh, measures 27 mm.

In the 3D mesh shown in figure 6, the region corresponding to the porous monolith has a 2 mm of length in order to reduce the operational cost. As the pressure drop is applied per unit of length, to reproduce the effect of the 27 mm monolith the resistance tensors must be multiplied by the factor $27/l$. Where l is the length of the reduced monolith. Due to this the values of the axial resistance tensors must be changed in concordance. It is easy to see that the new values for a 2 mm monolith will be

$$\frac{\Delta P}{L} = \frac{27}{2}14.053 \cdot u^2 + \frac{27}{2}734.47 \cdot u = 189.716 \cdot u^2 + 9915.345 \cdot u \tag{25}$$

To implement the effect of the porous monolith in the simulations it has been followed the approached presented by Hafsteinsso (2009, [10]). According to this, the code shown in the listing 1 was developed. The porous media is added to a region of the mesh, in our configuration this cell zone is called porosity for the 2D mesh and PM for the 3D mesh. In *DarcyForchheimerCoeffs* the values of the inertial and viscous tensors are defined. The two vectors d and f are added to the diagonal of the matrix D_{ij} and F_{ij}. In *coordinateSystem* a local coordinate system for the cell zone is defined.

Listing 1: Porous media model

```
explicitPorositySourceCoeffs
{
  type            DarcyForchheimer;
  selectionMode   cellZone;
  cellZone        porosity;

  DarcyForchheimerCoeffs
  {
    d  d [0 -2 0 0 0 0 0] (3.9593e7 -1e2 -1e2);// d = alpha/1.85508E-5 (alpha = 734.48 kg/m^3-s)
    f  f [0 -1 0 0 0 0 0] (23.735  -1e2 -1e2) ;// f = 2*beta/1.18415   (beta = 14.053 kg/m^4)

    coordinateSystem
    {
      type    cartesian;
      origin  (0 0 0);
      coordinateRotation
      {
        type    axesRotation;
        e1      (1 0 0);    //(0.70710678 0.70710678 0);
        e2      (0 0 1);
      }
     }
   }
}
```

The values that must be included in the *DarcyForchheimerCoeffs* are the ones deducted from put together equation 22 and equation 23 as

$$P_i = \frac{1}{2}\rho f_m \qquad\qquad P_u = \mu d_m \tag{26}$$

$$f_m = 2\frac{P_i}{\rho} \qquad\qquad d_m = \frac{P_u}{\mu} \tag{27}$$

Taking into account that the density is $\rho = 1.184 kg/m^3$ and the dynamic viscosity is $\mu = 1.855e{-5} Ns/m^2$, in table are presented the values that must be included in the code for each mesh

Table 1: Porous media values for each mesh.

	2D mesh	3D mesh
Inertial term	$f_m = 3.9593 \cdot 10^7$	$f_m = 53.54 \cdot 10^{77}$
Viscous term	$d_m = 23.735$	$d_m = 320.3583$

3.5 Steady flow

The solver that has been used to solve the steady flow is the SIMPLE algorithm. The acronym stands for Semi-Implicit Method for Pressure-Linked Equations. It is basically a guess-and-correct procedure for the calculation of pressure and velocities. Hence, the method is iterative and when other scalars are coupled to the momentum equations, the calculation is done sequentially (Versteeg, 1995 [22]). This iterative process can be outlined as:

- Guess a pressure field $p*$ (or use p^n from last iteration).
- Solve the momentum equation for the velocity field $u*$, based on the guesses pressure $p*$, which will not be divergence-free.
- Solve the pressure correction equation for Δp.
- If Δp is 'small enogh' then the equations are solved, exit, else
- Correct the pressure $p*$ using $\Delta p = (\delta p_* + \delta p)$
- Correct the velocities u_* with the velocity correction formula
- Go to the second step

The discretization schemes are set in the fvScheme dictionary as follow: ddtScheme should be steadyState. The gradSchemes is set to Gauss linear scheme, that specifies a finite volume discretisation that requires a linear interpolation of the cell centers to face centres. The divSchemes are set to bounded Gauss linearUpwind gradSchemelimited, where the *bounded* option include the discretisation of the third-term with the advection term. Finally the laplacianSchemes are selected as Gauss linear corrected. The selected RAS turbulence model is v2f model with the coefficients as expressed in section 2.

The incoming velocity in this configuration is constant and uniform throughout the diffuser entry. The simulations have been done for a inlet Re number of 22 000, in concordance with the experimental data. This is a value for the inlet velocity of $U_i = 8.79675 m/s$. The Reynolds number has been calculated with the hydraulic diameter of square pipes, as

$$Re = \frac{d_h \cdot \nu}{U_i} \qquad\qquad d_h = \frac{4 \cdot A}{P} \tag{28}$$

where ν is the kinematic viscosity and d_h is the hydraulic diameter of a square pipe calculated as a function of the wetted area, A, and the perimeter, P.

The density of the fluid is $\rho = 1.18415 kg/m^3$ and the dynamic viscosity is $\mu = 1.85505 \cdot 10^{-5} Ns/m^2$. Hence, the kinematic viscosity is

$$\nu = \frac{\mu}{\rho} = 1.5666 \cdot 10^{-5} m^2/s \tag{29}$$

Regarding the implementation of the $v2f$ model as explained in the section 2. The turbulent kinetic energy has been calculated as stated by Wilcox (2010, [23]), as the half of the turbulent trial where making the assumption that the Reynolds stresses are equal

$$k = \frac{1}{2}(\overline{u}^2 + \overline{v}^2 + \overline{w}^2) = \frac{1}{2}(3 \cdot \overline{v}^2) = \frac{3}{2}(T_u \cdot U_i)^2 \tag{30}$$

In equation 30, T_u is the turbulence intensity that is set to a 1 %, due to the possible uncertainty. Thus, the value of the turbulent kinetic energy will be $k = 0.011607 m^2/s^2$. Assuming the isotropy of the turbulence, the normal stress function leads to be calculated as

$$\overline{v}^2 = \frac{3}{2} \cdot k = 0.0077383 m^2/s^2 \tag{31}$$

This values will be used in the implementation of the v2f turbulence model through all the study cases.

3.5.1 2D domain

The first configuration of study is the steady flow in a 2D domain. For this conditions, it has been applied the viscosity ratio approach to calculate the initial boundary conditions. The viscosity ratio can be calculated as the ratio of the turbulent viscosity and the viscosity of the fluid.

$$\nu_R = \frac{\nu_T}{\nu} \tag{32}$$

The kinematic eddy viscosity for the $v2f$ model can be calculated with equation 34

$$\nu_T^{\overline{V}^2} = C_\mu \cdot \overline{v}^2 \cdot T \tag{33}$$

Regarding the length scale of the $v2f$ model defined in equation 20, it has been demonstrated that only one produce suitable results. The kinematic eddy viscosity can be written as

$$\nu_T^{\overline{V}^2} = C_\mu \cdot \frac{3}{2} \cdot k \cdot \frac{k}{\epsilon} \tag{34}$$

The turbulent dissipation rate can be clear from the previous expression and the definition of the viscosity ratio

$$\epsilon = C_\mu \frac{3}{2} \frac{k^2}{\nu_T} = C_\mu \frac{3}{2} \frac{k^2}{\nu_R \cdot \nu} \tag{35}$$

The viscosity ratio approach revolve around calculating the dissipation ratio from a viscosity ratio value. This way the different results can be organize in accordance with the different viscosity ratios. Boristov (2016, [5]) emphasizes that the use of this approach gives an intuitive way of defining the viscosity ratio according to the turbulence intensity.

Consequently, for this conditions, the turbulent dissipation rates used in function of the viscosity ratio are presented in table 2.

Table 2: Viscosity ratio and dissipation rate values for the steady flow in a 2D domain.

ν_R	ϵ
1	1.25865
5	0.25173
10	0.12586
20	0.063

In the table 3 are presented the corresponding boundary conditions. The boundary conditions applied in the walls for the fields k, $\overline{v}^2$, ϵ and f are mostly those pertaining to the OpenFOAM v2f model. For the inlet and the outlet regions has been mainly used the Dirichlet and Neuman boundary conditions. The Dirichlet condition determine and set the value of a variable at a certain boundary of the domain. This condition in OpenFOAM is set as fixedValue. The Neuman boundary condition specifies the values of the normal derivative of a solution at a certain boundary of the domain. In OpenFOAM the zeroGradient condition establish that the gradient of the respective quantity is zero in relation to the orthogonal direction of the surface applied .

Table 3: Boundary conditions.

	dInlet	dOutlet	Walls
U	fixedValue	InletOutlet	fixedValue 0
p	zeroGradient	fixedValue 0	zeroGradient
k	fixedValue	zeroGradient	KLowReWallFunction
$\overline{v}^2$	fixedValue	zeroGradient	v2WallFunction
ϵ	fixedValue	zeroGradient	epsilonLowReWallFunction
f	zeroGradient	zeroGradient	fixedValue 0
ν_T	calculated	calculated	nutUWallFunction

Finally, for this case it has to be said that in the controlDict dictionary, the time step deltaT is set to 100 s as the endTime is set to 40000 s.

3.5.2 3D domain

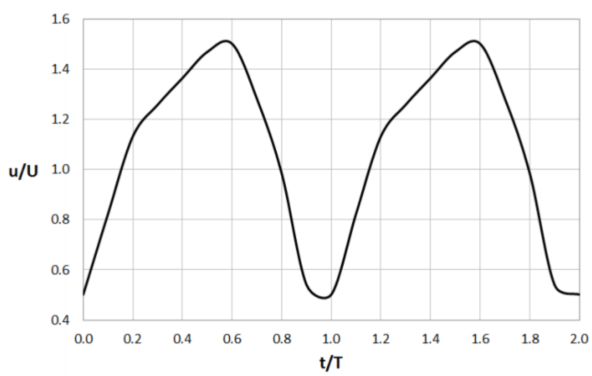
The assessment of the steady flow in the 3D domain it has been the next step of the study. The boundary conditions applied are the same that the ones showed in the table 3. However, only the value of a viscosity ratio of 10 has been analysed because it was the best result for the 2D domain study. The solver and the discretization used is the same as the one showed in section 3.5.1. Despite, in the controlDict the endTime is set to 200000s and the writeInterval is 800s.

3.6 Pulsed flow

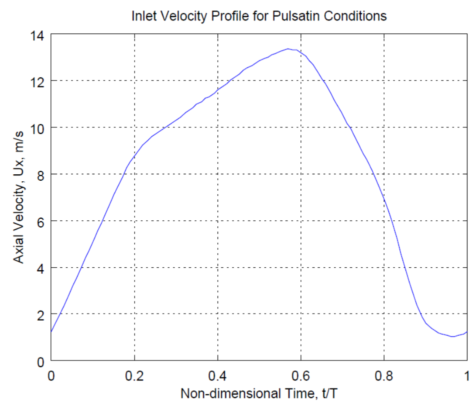
The next step in the study is the assessment of the pulsing flow. The solver used for the transient studies is PIMPLE. This solver is the merger of the SIMPLE solver and the PISO (Pressure Implicit with Split Operator) one. The PISO solver is suitable for transient simulations where it is necessary to fully solve the velocity-pressure coupling for each time step (Aguerre et al., 2013 [1]). The non-linear effects of the velocity are reduced setting small time steps characterized by Courant numbers below one. However, these small time steps can be problematic to achieve convergence and you are limited to use a $Co < 1$. Hence, PIMPLE uses bigger time steps respectively with bigger Courant numbers.

Regarding the discretization schemes, in this configuration the ddtScheme is set as Euler, as this case is transient this option set the time discretisation scheme . The gradSchemes are set to Gauss linear scheme as in the previous case. The divSchemes are set to Gauss linearUpwind gradSchemelimited, where the *bounded* option has been eliminated. Finally the laplacianSchemes are selected as Gauss linear corrected, where the *corrected* scheme adds a correction to mantain the second-order accuracy. The selected RAS turbulence model is v2f model with the coefficients as expressed in section 2. In the controlDict dictionary, the time step deltaT is set to 0.0001s, the writing interval is 0.001s and the endTime is set to 1s.

One of the pricnipal differences of this configuration is that the inlet velocity is periodic and uniform. It has been set in the inlet boundary condition as a signal. This signal as stated by Porter and Borisov have a frecuency of 50 Hz, which means that the pulse period is $T = 0.02s$. The signal is the same used by Borisov (2016, [5]). It is based on the experimental data measured by Yamin et al. (2013 [25]) and there is a 6 of discrepancy between them. In figure 8 is presented the cycle-averaged inlet velocity. The picture (a) is the pulsed shape used by Porter et al. (2016, [17]). The picture (b) is the pulsed shape that has been used for this study. Ar it can be seen only till the non-dimensional time of 1 has been used.



(a) Phased-average mean velocity (Porter 2016:3 [17])



(b) Inlet pulse for the velocity BCs (Borisov 2016:25 [5])

Figure 8: Inlet velocity signal

3.6.1 2D domain

The first configuration of the pulsating flow has been studied in the 2D mesh presented in 3.3.1. As a result, the porous model is the respective to the 27 mm one. In order to solve the $v2f$ turbulence model the viscosity ratio chosen is $\nu_R = 10$. Regarding the boundary conditions, they are the same as the ones established in table 3. Excepting the boundary condition for the inlet velocity. It has been used *uniformFixedValue*, where the *uniformValue* is *tableFile* because the values are introduced with an external *.txt* file.

3.6.2 3D domain

The 3D domain has been used to assess the last configuration of the pulsed flow. Thus, in this case the porous model is the one regarding the 2 mm region. The value of 10 for the viscosity ratio has been set to implement the turbulence model. The boundary conditions are the same that for the previous case.

3.7 Summary

Two grids have been used, one regarding a 2D domain and the other a 3D domain. Each of them is related to a different characterisation of the porous media model. This model reproduce the effect of the catalytic monolith adding a source term to the momentum equation. this source term is divided in a viscous and in a inertial viscous terms. The geometry has been studied for a steady flow and for a pulsating flow in each of the domains. Furthermore, a study of the influence of the mesh density in the region of the porous media has been done. Finally, it has to be highlight that the implementation of the $v2f$ model has been done through the viscosity ratio approach. For the steady flow in the 2D domain different values of viscosity ratio has been assessed. A viscosity ratio of 10 has been set for the rest of configurations because it was the best result.

4 Discussion of results

In this section are presented the results of the performed simulations with the $v2f$ model. It is divided in two main parts: steady flow and pulsating flow. The results performed in *OpenFOAM* are now port-processed with the software *paraView*.

4.1 2D Steady Flow

Results from the steady conditions are presented firts. Figure 9 illustrates the velocity contour displayed in *ParaView*. It can be noted the separation of the flow in the expansion through the diffuser. The flow spreads radially when reaches the monolith and a recirculating zone can be seen in the diffuser wall. Furthermore, across the monolith and the outlet sleeve the velocity maldistribution can be appreciated along with the stagnated and second peak region. Finally, in the outlet sleeve a boundary layer is developed.

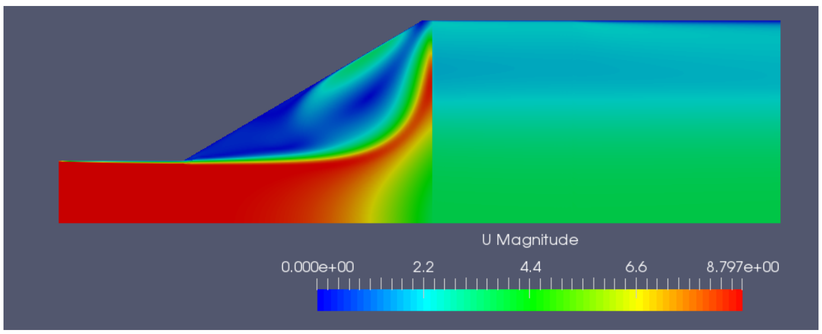


Figure 9: Velocity profile contour for steady flow in a 2D domain.

Figure 15 displays a graph that contrast the experimental data with the simulated one. It presents the non dimensional axial velocity against the lateral distance. There is one line with dots for each of the simulated values of viscosity ratio and the black dots represent the measured data that has been presented in 3.2.

The experimental data measured in Porter et al. (2016, [16]) capture the flow behaviour as it has been explained. The flow starts with a peak at the lateral distance of 0, this is at the core of the monolith, due to the axial entrance of the flow in this region. As the distance from the center of the monolith increase the velocity of the flow decrease as a consequence of the inclination of the velocity that reach the monolith and the presence of this one. Reaching the minimum at a lateral distance of 0.027 m approximately. Finally, there is a second maximum peak produced by the pressure rise that force the flow to enter into the channels.

The results obtained from the simulations differ slightly with the experimental ones. At the center of the monolith it can be seen that the velocity is underestimate while the minimum value is overpredicted. However, the second peak is predicted quite accurate. It can be said that the CFD approach provides an acceptable approximation. Regarding the different viscosity ratio values, the value of $\nu_R = 20$ is the less accurate that underestimate a bit more the first peak and overpredicts the last part. Between the other 3 values the ratio of $\nu_R = 10$ is the most precise. So it has been used for the rest of the simulations.

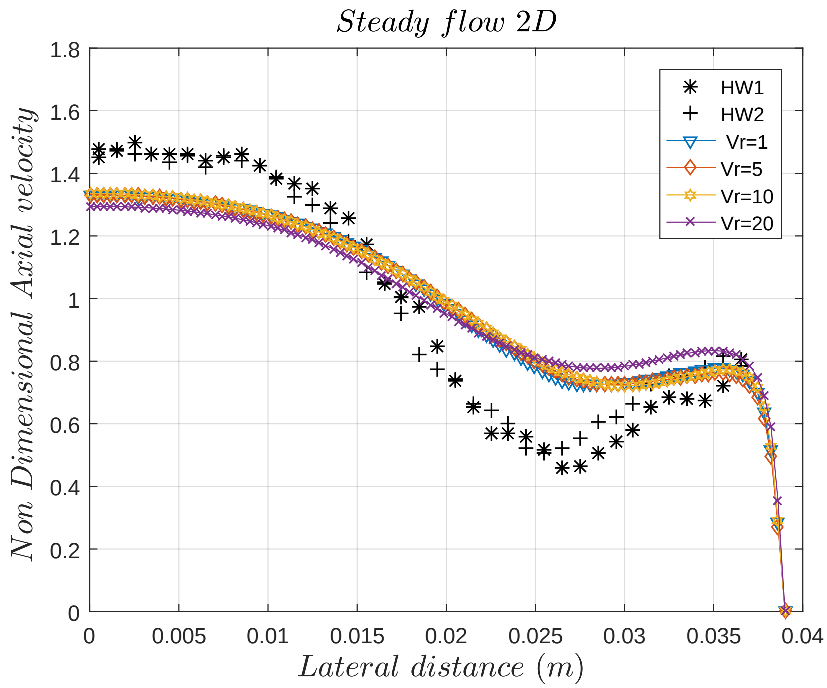


Figure 10: Results from the steady flow in a 2D mesh.

4.2 3D Steady Flow

The second step in this analysis is steady flow in the 3D domain. In figure 11 is displayed the velocity contour. It can be appreciate how the jet spreads radially along with the recirculating region in the wall of the diffuser. In the monolith the maldistribution is present. The minimum of the velocity and the secondary peak can be noted.

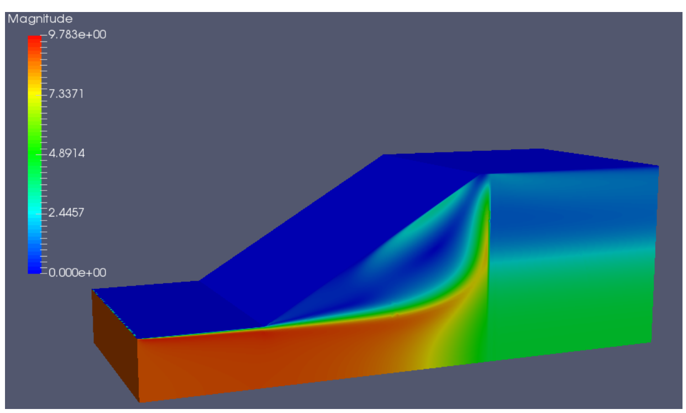


Figure 11: Velocity profile contour for steady flow in a 3D domain.

Representing the non dimensional axial velocity against the lateral distance for this case it is obtained the figure 12. Along with the experimental data it is presented the in the blue line the results for the 3D mesh with 2 cells in the porous media region. According to the results, the maximum peak is overestimated with respect to the experimental data. As the flow spreads radially it can be seen how the simulated results follow the tendency of the measured results. The minimum is place in 0.0275m. which is quite approximate to the experimental one. Finally, the closest part to the wall i underestimated, this means that the model have not captured the effect of the flow absorption due to the pressure in this region.

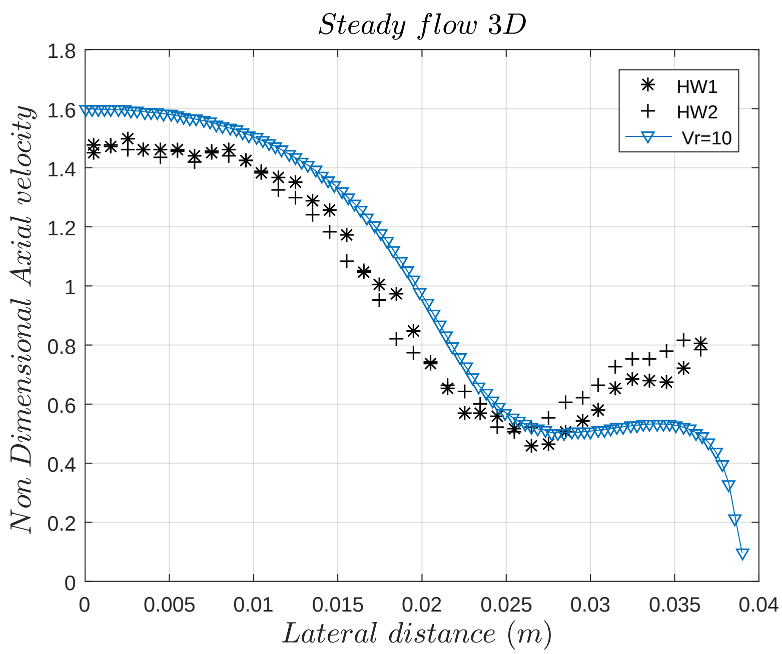


Figure 12: Results from the steady flow in a 3D mesh.

4.2.1 Mesh dependence study

In figure 13 is presented the dependence mesh study of the model. There are displayed the three simulations for each of the meshes presented in section 3.5.2. It can be seen that with the different meshes very similar results have been obtained. Due to this, the original mesh provided by Borisov.

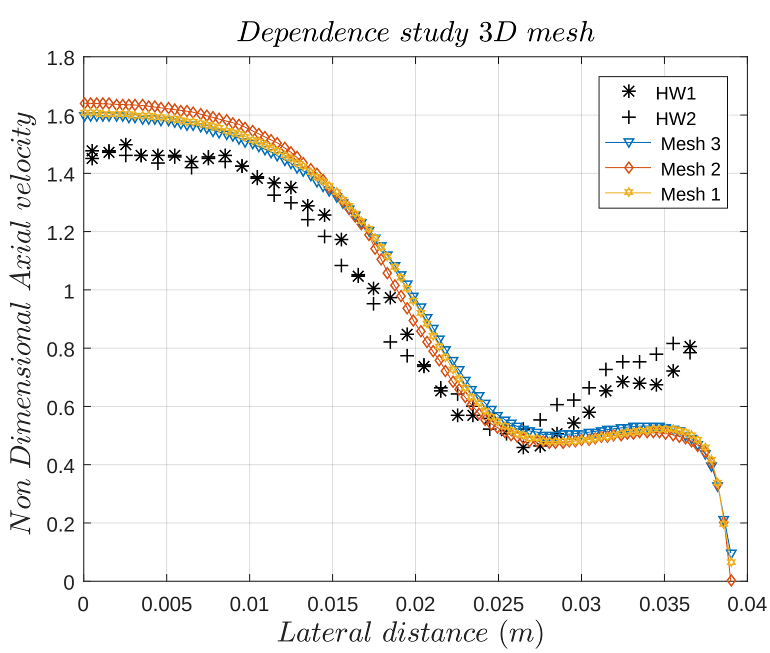


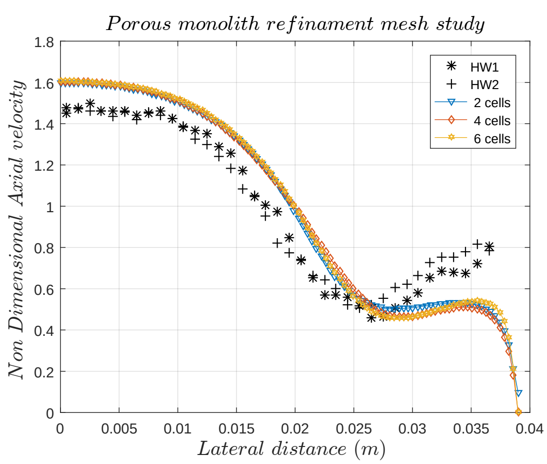
Figure 13: Independence study mesh

4.2.2 Porous media mesh dependence study

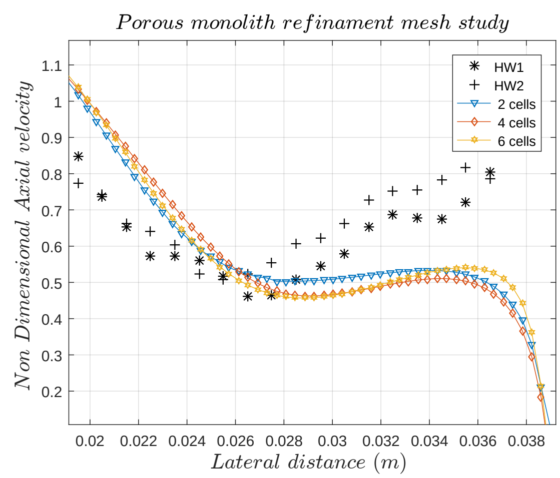
In this section are presented the results of the study that compare the effect of different mesh density in the region of the porous monolith. This study was assess in order to analyse if with an improvement of the mesh density the model will be able to capture better the flow behaviour.

The results are presented in figure 14. In the first picture is presented the non dimensional velocity for the complete lateral distance of the monolith. Here can be seen that the results at the beginning are very similar, as expected because in this region there is no change. To analyse better the results, the second picture displays a zoom over the final region where the mesh desntiy difference will be noted. It can be seen how the less dens mesh capture worse the velocity profile because the profile is more uniform. On the other hand, the most dens mesh provides produces a better velocity profile, where the minimum and second peak are better represent. The intermediate mesh is similar to the most dens but the 2 peak is no captured accurately.

Hence, it can be said that if the density mesh of this region is improved better results will be obtained. Furthermore, it has to be considered that this approach of modelling the porous media pretends to minimise the mesh density and consequently the time operational cost. Therefore, if the density mesh of the monolith is increased this requirement is not completely accomplished. However, the density of the mesh used is smaller than if the hole intrinsic channels region had been represented.



(a) View of the complet lateral distance



(b) Zoom in the last part of the section

Figure 14: Dependence of the porous media region density mesh

4.3 Comparison of steady condition results

If we analyse together the results for the steady conditions with a viscosity ratio of $\nu_R = 10$ the figure 15 is obtained. It can be seen that the case with the 2D domain overestimates the minimum of the velocity but predict accurately the second peak of the velocity. On the other hand, the configuration with the 3D domain capture the flow behaviour quite good until the minimum, when it reaches the second peak there is a big underestimations of the values.

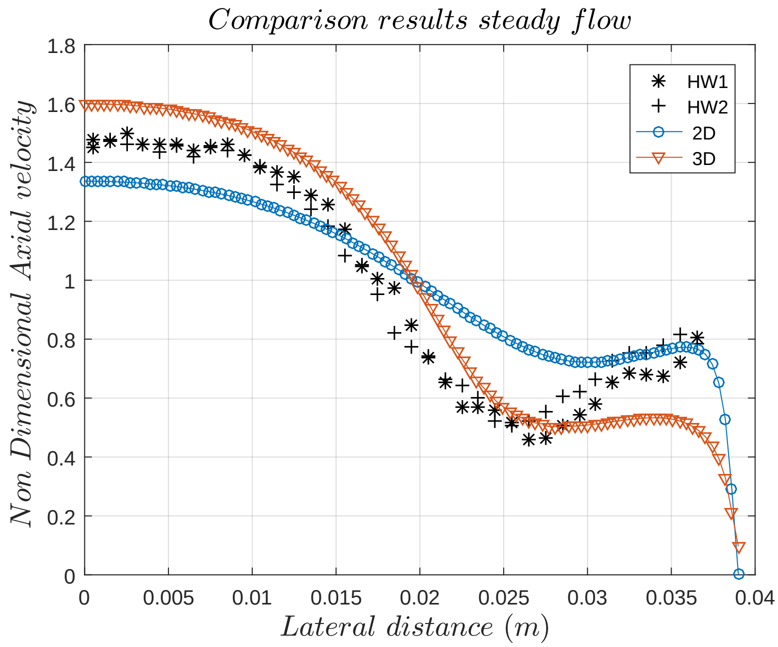


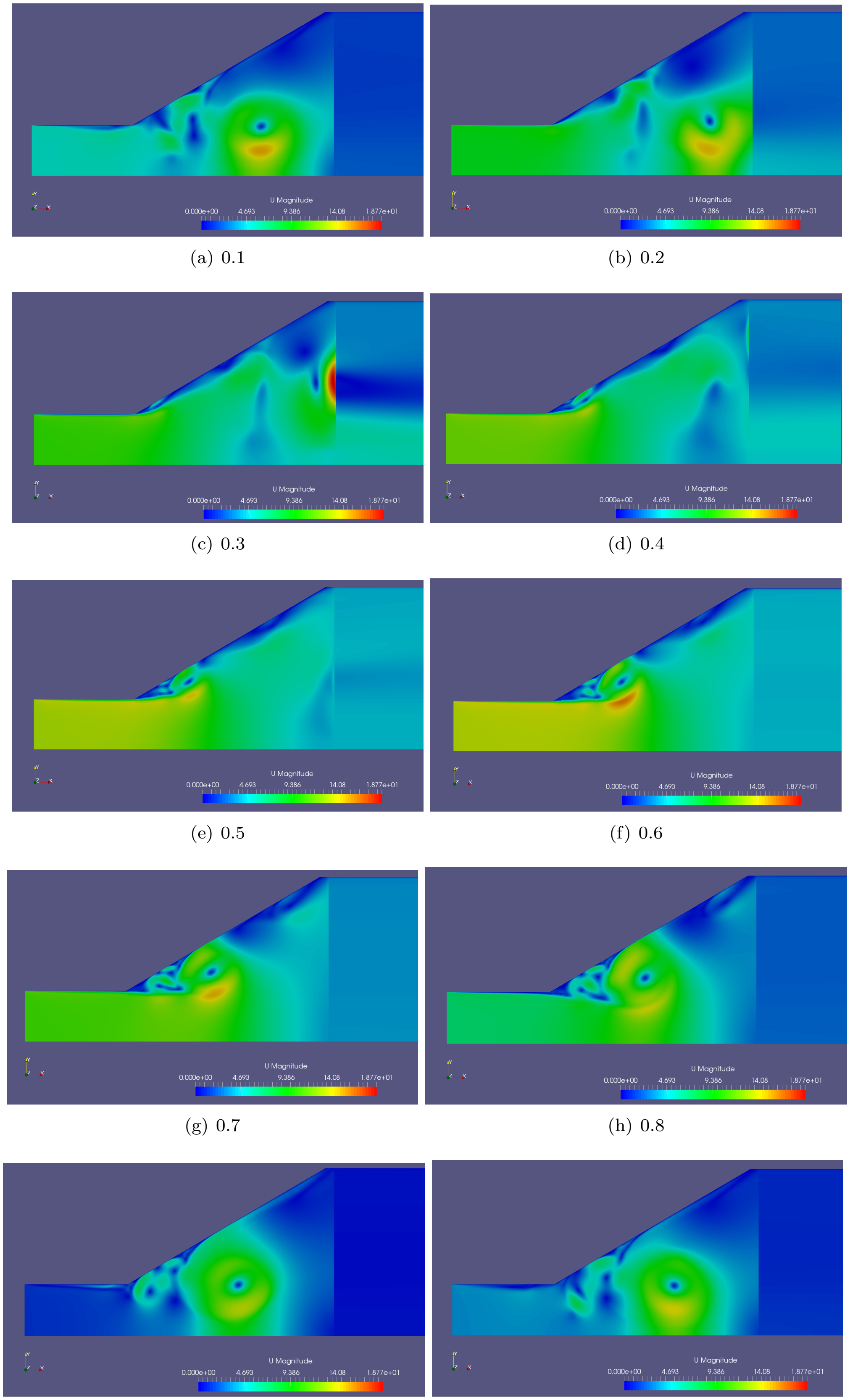
Figure 15: Comparision of results for the steady flow.

The differences can be due to the porous monolith model. When the 2D mesh has been used the porous model had the normal length, but in the 3D mesh the region is characterised in order to produce the same effect. Other reason, can be the turbulence model. In his study Borisov (2016, [?]) , studied the 3D steady flow with the $SST - K - \omega$ because of his stability. In his study, there was a slightly difference between both results and he concluded that there was no need to study the steady case with a 3D domain due to the higher operational cost over the 2D one. Therefore, a more deeply study of the implementation of the $v2f$ in the 3D domain should be done.

4.4 2D Pulsed Flow

The results obtained from the transient case in the 2D mesh are presented in figure 16. This results have been obtained from the 11^{th} cycle of the simulation where the results seem to have converged. In figure 16 is presented the velocity contour for one cycle. Considering that the time is non-dimensional, the cycle is divided in two parts respect to inlet velocity conditions. The maximum inlet velocity is found between a the time of 0.5 and 0.6, so before of this point the flow is accelerating and after the flow is decelerating.

At the beginning, the recirculating zone from the previous cycle is still present at the core of the geometry. As the inlet velocity increases the core jet moves the recirculating region into the monolith. Meanwhile, at the entrance of the diffuser the separation of the flow can be seen. In time 0.3, 0.4 and 0.5 one can see how the recirculating zone is created between the core jet and the diffuser wall. As the inlet velocity reaches its maximum the recirculating bubble becomes bigger and this produces leads to a another smaller recirculating bubble. When the inlet velocity starts to decrease, the main recirculating zone stops being extended due to the incoming velocity that was feeding this region. At the final steps of the cycle the bubble that had remained attached to the wall of the diffuser starts moving into the core of the diffuser. It can be said that this results represent the expect behaviour more or less.



(a) 0.1 (b) 0.2

(c) 0.3 (d) 0.4

(e) 0.5 (f) 0.6

(g) 0.7 (h) 0.8

(i) 0.9 (j) 1

Figure 16: Pulsating conditions velocity contours in the 2D mesh.

4.5 3D Pulsed Flow

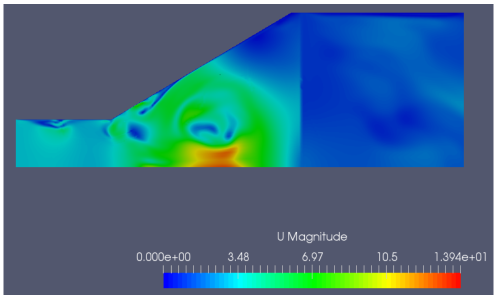
The results obtained from the transient case in the 3D mesh are presented in figure 17. It has to be noted the high operational cost used to resolve this case. While the case with the 2D mesh was fully solved with 3 days, this case with the 3D mesh at the third day was still in 0.12 s (having in mind that the final time was 1 s). Due to this reasons and time limitations the images showed are from the 8_{th} cycle and so are not as good as they could be.

In this occasion, there are also two main parts differentiated by the maximum inlet velocity that for this configuration is placed at the non dimensional time 0.6. In the first time step, along with the remain recirculating region from the previous cycle it is interesting to point the small recirculating bubble that is present in the inlet to the diffuser. As in the previous case, as the flow moves into the diffuser the recirculating region existing goes into the monolith. It is remarkable from this times that the stagnate flow that was initially at the end of the diffuser wall have been expanded approaching the jet core. At the time 0.4 and 0.5 the new recirculating region is being created at the first part of the diffuser wall. When the maximum velocity inlet has been reached, another recirculating bubble develops from the first one. In the time 0.7, it can be noted how these bubble are feed from the jet core. In the time 0.8 both bubbles have been combined in a bigger one that is still being extended. As the inlet velocity decrease the recirculating region stops growing. Finally at the time 1 the big bubble separates from the wall and moves into the core of the diffuser. It is at this point when from the recirculating zone a part of the flow moves downwards into the inlet wall. This is what produces the recirculating zone in the first time step.

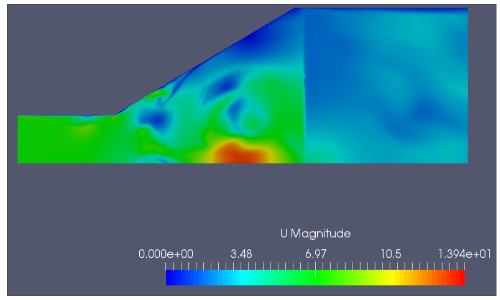
4.6 Comparison of pulsating condition results

In this section the behaviour of the pulsating flow in both domains will be compared. In both cases the maximum velocity inlet is placed at the non dimensional time 0.6. At the start of the fluid inlet acceleration stage in can be noted that the recirculating zone in the 3D mesh is no very well defined. Moreover, the mentioned recirculating region that is develop for the 3D mesh between the time 1 of the previous cycle an the time 0.1 is present in the time 0.9 of the 2D mesh. But here is a thin layer that disappear. For both cases the recirculating region starts to develop between the times 0.3 and 0.4, but it is more pronounced in the 2D mesh. In this stage it has to be highlight that when the bubbles reach the monolith face in the 2D domain the velocity peak is the face of the monolith while in the 3d domain the velocity peak is in placed in the axis wall. Furthermore, it is highly remarkable that in the this times in the 2D domain the velocity spread faster into the diffuser than in the 3D domain. It can be seen how the behaviour of the flow that spreads into the diffuser for the 0.2 time in the 2D mesh is the same (except the recirculating zones) that in the 0.4 of the 3D mesh. This leads to the fact that while in the 2D domain, the velocity is uniform after the started bubble in the times 0.5 and 0.6, in the 3D domain the velocity is poorly distributed.

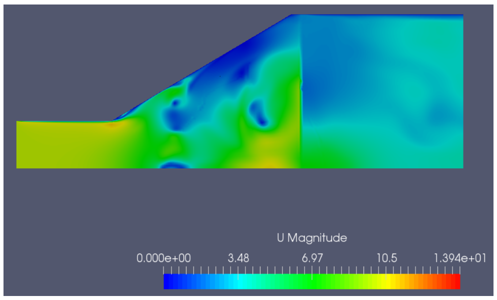
At the deceleration of the inlet velocity it is noteworthy how in the time 0.7 the diverse recirculating bubble perform. While for the 2D mesh they have been segregated in the 3D mesh they join into a big recirculation bubble. Finally, it can be seen how as the inlet velocity cecrease the recirculating region that was attached to the diffuser wall moves into the core of the diffuser for both configurations.



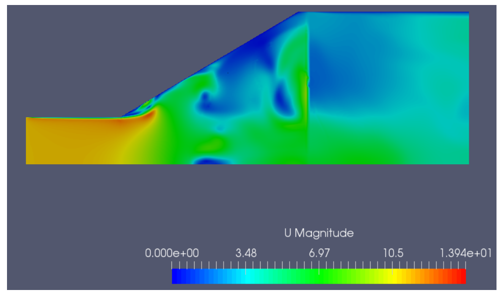
(a) 0.1



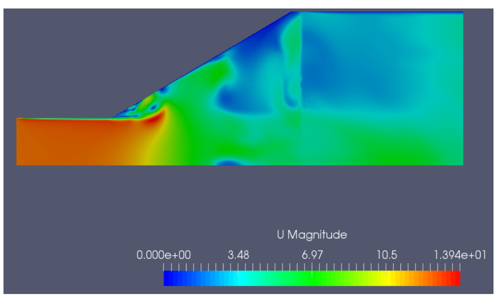
(b) 0.2



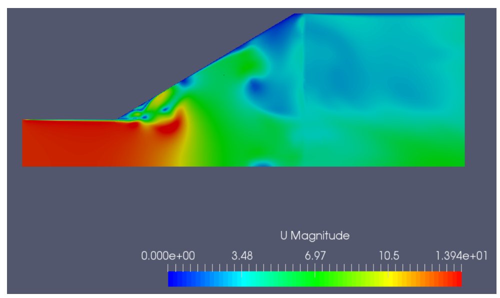
(c) 0.3



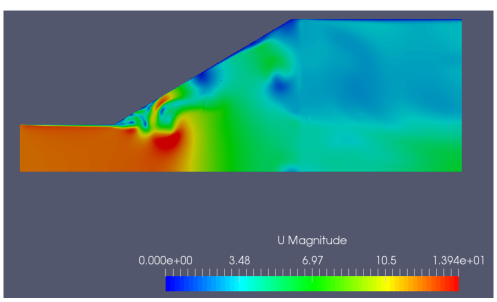
(d) 0.4



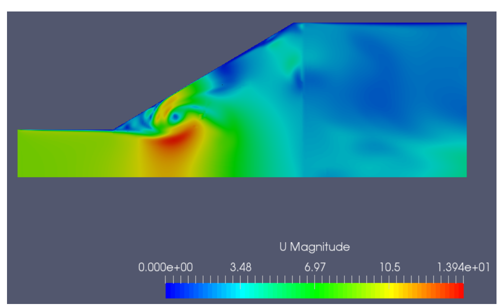
(e) 0.5



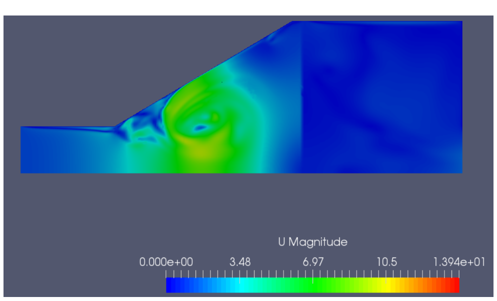
(f) 0.6



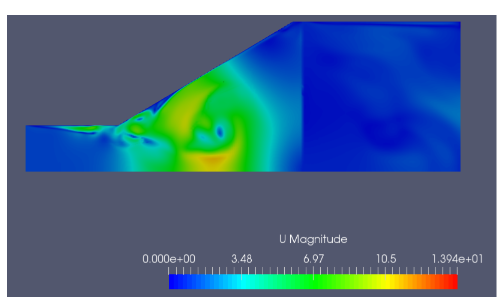
(g) 0.7



(h) 0.8



(i) 0.9



(j) 1

Figure 17: Pulsating conditions velocity contours in the 3D mesh.

5 Conclusions

An assessment of the flow behaviour in a catalytic converter have been done. The main differences have been that the it have been studied in steady and transient conditions along with the use of a 2D mesh and a 3D mesh. Each of the flow conditions has been analysed in both meshes. Regarding the steady flow, the configuration in 3D is able to capture better the flow behaviour until the minimum and there is an underprediction of the second velocity peak. Meanwhile, the 2D domain capture very well the second peak. Furthermore, an investigation about the dependency of the results with the porous media mesh have been done. As expected, a refinement of the mesh lead into a better approximation to the experimental data. However, the improvement is not very remarkable.

Concerning the pulsating flow, it has been analysed the velocity contour in the inlet and the diffuser for a cycle. Comparing both results, it is remarkable to say that between the times 0.6 and 0.7 for the 3D domain, the main recirculating bubble and the secondary one that leads from the first, combine into a bigger one while in the 2D mesh they remain segregated.

Finally, it has to be said that despite the fact that the results obtained for the transient configuration are quite similar, in my opinion the pulsating flow in the 3D domain needs to be studied and compared to experimental data in order to validate the model.

6 References

References

[1] Aguerre, H.J., Marquez, S., Gimenez, J.M. and Nigro, N.M. (2013). 'Modelling of compressible fluid problems with OpenFOAM using dynamic mesh technology' *Mecanica Computacional* Vol XXXXII, pages 995-1011.

[2] Arias-Garcia, A., Benjamin, S.F., Zhao, H. and Farr,S. (2001)'A comparison of steady, pulsating ow measurements and CFD simulations in close coupled catalysts.' *SAE Technical Paper 2001-01-3662*

[3] Benjamin, S.F., Clarkson, R.J., Haimad,N., and Girgis, N.S. (1996). 'An experimental and predictive study of the flow field in axisymmetric automotive exhaust catalyst systems.' SAE Technical Papers 961208.

[4] Benjamin, S.F., Roberts, C.A., and Wollin, J. (2002). 'An experimental and predictive study of the flow field in antisymmetric automotive exhaust catalyst systems.' *SAE Technical Papers 961208.*

[5] Borisov, D. (2016) *Assessment of the predictive capabilities of CFD for a planar diffuser with a porous monolith downstream* Unpublished dissertation. Coventry: Coventry University.

[6] Catalano, P., Wang, M., Iaccarino, G. and Moin, P. (2003) 'Numerical simulation of the flow around a circular cylinder at high Reynolds numbers.' *International Journal of Heat and Fluid Flow* 24, pages 463-469

[7] Clarkson, R.J. (1995). A theoretical and experimental study of automotive catalytic converters. Unpublished PhD thesis. Coventry: Coventry University.

[8] Cokljat, D., Kim, S.E., Iaccarino, G., and Durbin, P.A. (2003) 'Acomparative assessment of the V2F model for recirculating flows.' *41st Aerospace Sciences Meeting and Exhibit* held 6-9 January 2003 at Reno, Nevada.

[9] Durbin, P.A. (1991). *Near-wall turbulence closure modeling without damping functions.* Theoretical and Computational Fluid Dynamics 3.

[10] Hafsteinsson, H. E. (2009) *Porous Media in OpenFOAM.* Department of Thermo and Fluid Dynamics of Chalmers University of Technology.

[11] Laurence, D.R., Uribe, J.C. and Utyuzhnikov.,S.V. (2014) 'A robust formulation of the v2-f model.' *Flow Turbulence and Combustion* 73, pages 169-185.

[12] Lien, F. and Kalitzin, G. (2001) 'Computations of transonic flow with the v2-f turbulence model.' *International Journal of Heat and Fluid Flow* 22, pages 53-61.

[13] Mat Yamin, A.K. (2012). *Pulsating flow studies in a planar wide-angled difuser upstream of automotive catalyst monoliths.* Unpublished PhD thesis. Coventry: Coventry University.

[14] Ozhan, C., Fuster, D., and Da Costa, P. (2014) 'Multi-scale flow simulation of automotive catalytic converters.' Mechanics of the fluids. Universite Pierre et Marie Curie - Paris VI,2014.

[15] Porter, S.J. (2016) *An assessment of CFD applied to a catalytic converter system with planar diffuser.* Unpublished Phd thesis. Coventry: Coventry University.

[16] Porter, S.J., Saul, J.M., Aleksandrova, S., Medina H, H.J., and Benjamin, S.F. (2016) 'Hybrid Flow Modelling Approach Applied to Automotive Catalysts' *Applied Mathematica Modelling* 40, pages 8435-8445.

[17] Porter, S.J., Saul,J.M., Yamin, A.K.M., Aleksandrova, S., Benjamin, S.F. and Medina H, H.J., (2016) 'Pulsating Flow in a Planar Diffuser Upstream of Automotive Catalyst Monoliths: A CFD Study' *Pacific Symposium on Flow Visualization and Image Processing.* Held 15-18 June 2015 at Naples, Italy.

[18] Quadri, S.S., Benjamin, S.F., and Roberts, C.A. (2013). 'Flow measurements across an automotive catalyst monolith situated downstream of a planar wide-angled diffuser'. *Proceedings of the Institution of Mechanical Engineers*, Part C: Journal of Mechanical Engineering Science Volume 224(2):321-328.

[19] Salim, S. M., Ong, K. C. and Cheah, S. C. (2011) 'Comparison of RANS, URANS and LES in the prediction of airflow and pollutant dispersion' *Proceedings of the World Congress on Engineering and computer Science 2011 Vol II* held 19-21 October 2011 at San Francisco, USA.

[20] Saul, J.M, Porter, S.J., Mat Yamin, A.K., Aleksandrova, S., and Benjamin, S.F. (2016) *Influence of cyclic variance on the performance of URANS for pulsating flow upstream of an automotive catalyst monolith.* Coventry University, Departament of Mechanical, Automotive and Manufacturing Engineering, England.

[21] SolKeun, J. and Shariff, k. (2012) 'Detached-Eddy Simulation Based on the $v^2 - f$ Model'. in *Seventh International Conference on Computational Fluid Dynamics (ICCFD7)* held 9-13 May 2012 at Big Island, Hawaii.

[22] Versteeg, H.K. and Malalasekera, W. (1995) *An Introduction to Computational Fluid Dynamics: The Finite Volume Method.* Longman Scientific and Technical. ISBN 0-582-21884-5

[23] Wilcox, D. (2010) *Turbulence Modeling for CFD* 3rd edn. DWC Industries. ISBN 978-1-928729-08-2

[24] Wollin, J. (2002). *A study of pulsating ow in automotive exhaust catalyst systems.* Unpublished Phd thesis. Coventry: Coventry University.

[25] Yamin, A.K., Benjamin, S.F., Roberts, C.A. (2013) 'Pulsating flow in a planar diffuser upstream of automotive catalyst monolith' *International Journal of Heat Fluid Flow* Volume 40, pages 43-53.

Appendices

A Project Proposal

Final Year Undergraduate Project Proposal Form

Student Name	Mara Salut Escartí Guillem
Course	Project 320EKM
Email	escartim@uni.coventry.ac.uk
Project Module	320EKM / 330EKM

Supervisor	Dr Humberto Medina

Project Title:
A proposed title for the project (Should be meaningful, relevant and concise)

3D study using URANS of a steady and pulsed jet in a diffusor without resistance.

Synopsis:
Explain the background to the project, and provide an overview of what you intend to do (approximately 500 words)

Due to the increase in the number of cars in circulation, the emissions regulations have become more and more stringent. In order to reduce the exhaust gases of the cars the automotive industry has included catalytic systems in its vehicles. Catalytic converters facilitate the conversion of pollutants in the pulsating exhaust flow into less harmful emissions by catalysing a redox reaction.

In our case of study, the inlet of the catalytic converter is connected to a planar wide angled diffusor. This produce that the flow which arrives to the catalyst is non-uniformly distributed. The level of flow maldistribution is very important factor in the design of the catalytic converter because affects its conversion efficiency. It is essential to obtain a detailed understanding of the flow and the impact of a certain design over its development and physics.

This work aims to analyse the flow behaviour in 3D of the diffusor and the catalytic converter combined. There will be two configurations: (i) steady jet and (ii) pulsed jet. In previous research the behaviour of a steady jet has been studied but in reality the flow of the car's engine is a pulsed jet. It has been found that the results when there is a pulsed jet are not as accurate as they should be. For this reason, I am going to study both configurations with a different approach.

As stated above, this topic has already been studied using the commercial CFD solver STAR-CCM+. Where the Reynolds averaged Navier-Stokes (RANS) equations were combined with a turbulence model called v^2f (v^2 is the normal stress function and f is the elliptic function).

For this work, the CFD solver will be OPENfoam and instead of using the RANS equations to solve the problem the URANS (unsteady Reynolds averaged Navier-Stokes) equations will be used. The difference between RANS and URANS is that an addition unsteady term is present in the URANS momentum equation. Also the turbulence model that will be used to close the equations will be the v^2f model.

The 3D mesh will be defined in the software SALOME. This mesh will be used in both cases. Regarding the geometries, the regions belonging to the monolith will be modelled as a combination of individual channels of length 13 mm and a porous medium of length 1 mm. This model will be called hybrid model and it will provide a compromise between the computational efficiency of the porous model and geometrical accuracy of individual channels.

Finally flow predictions will be compared with experimental data in the diffuser and downstream of the monolith.

Client:
Provide a description of your client (if any), and contact details.

None

Objectives (provide from 5 to 8): *List the overall objectives of the project. These should be measurable, and will be used to assess the level of achievement of the project.*

- Carry out a literature investigation to gain an understanding of the problem identifying the flow distribution linked to the diffusor and the monolith in each configuration.
- Define the geometries and the meshes with the open source software Salome.
- Perform a series of URANS simulations for a three-dimensional domain and obtain the velocity and pressure distribution in each configuration.
- Explain the results obtained with the CFD simulations and discuss the accuracy by comparing them with the experimental data that is counted.

Project Deliverables (provide from 5 to 8):
Provide a list of key deliverables of the project (which may be one for each of the above objectives). These can be studies, reports, recommendations, etc.

- Brief report about the literature review where to explain the state-of-the-art.
- Report with the details of the methodology followed with the program OPENfoam.
- Report explaining the results obtained and making a comparison between the cases of study.

Why are you interested in the project?
Provide a reason for your interest, and describe what greater general interest it serves. Who else could benefit from it?

First of all, I am interested in this project because I will be able to learn how to performance Computational Fluid Dynamics (CFD) simulations and improve my knowledge about this topic. CFD is one of the most extended technics of research on the aerodynamics field. Furthermore, it will provide me the opportunity to study a real configuration where to apply all I have learn during my degree.

In my opinion this project can help to understand the flow behaviour at the end of the car's engine improving the operational effectiveness of the catalytic converters. As a result this can reduce the amount of harmful exhaust gases that are expelled to the air.

What are the key questions the project attempts to answer (provide from 1-3)?

- How is the pulsed jet behaviour different from the steady jet one?
- How can the reduction of the harmful exhaust gases be improved?

How will you judge whether your project has been a success?

Firstly, one of the most important things is that all the objectives have to be met. As it is a quantitative project I still do not know which results I will obtain. Due to this, I can't judge if the project has been a success looking at the results. In other words, maybe the results are not as expected but that not mean that the project has been a failure. It will be a success if I am able to discuss accurately and understand the results obtained.

What research methods do you intend to use?

As this project is a quantitative project the research approach will be deductive. The first step will be carry out a literature investigation in order to get a basic theory background. Secondly, a series of URANS simulation will be performed in the open source software OPENfoam. Due to the lack of time the experimental data needed will be provided by the supervisor. It is important to highlight that the simulations will be resolved in the cluster provided by Coventry University.

What primary and/or secondary data sources do you intend to use?

The principal data for this project will be primary. It will be obtained from the open source software OPENfoam. This software performance CFD simulations toyou're your system of study.

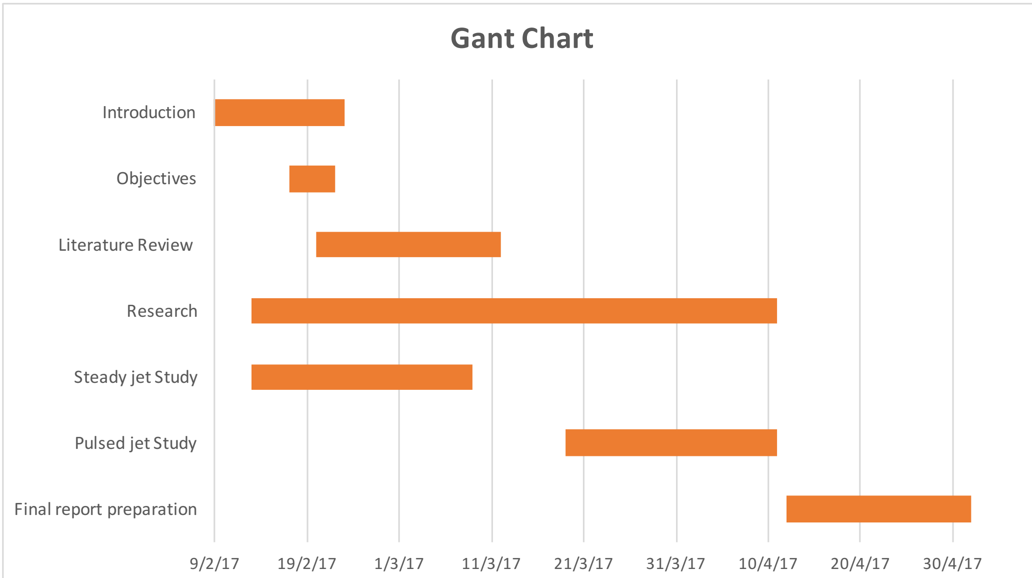
This principal data will be compared with experimental data. As the experimental one will be provided by the supervisor it can be considered as secondary data. Also it is possible to include some other secondary data as for example data from books and journals.

Estimate the number of hours you expect to spend on each of the major project tasks:
(The tasks below are only examples. You will need to edit the table to suit your own project).

The approximate time that every task will take is:

TASK	*START DATE*	*DURATION*	*END DATE*
Introduction	8- Feb	10	17-Feb
Objectives	17-Feb	5	19-Feb
Literature Review	20-Feb	20	4-Mar
Research	13-Feb	110	11-Abr
Steady jet Study	13-Feb	25	10-Mar
Pulsed jet Study	19-Mar	85	11-Abr
Final report preparation	12-Apr	60	1-May

Based on a pattern of 20 hours/week distributed in 5 days (Mon-Fri), starting on 9th of February 2017, it is expected to finish the project by no earlier than 11th of April 2017, about two weeks before the deadline (2nd May) giving some flexibility about possible delays.



Signature:

Mara Salut Escartí Guillem

Date:

13th February, 2017

B LogBook

B.1 Introduction

This report aims to complement the first draft of the project report. Pretends to present the project management that has been carried out. This documents is divided in three main sections:

- A logbook, provides weekly information on progress in relation with the project development. In this part the important deadlines and problems can also be found. The week will start on Friday because is the day when I have my meetings with my supervisor and from this point is when I change or start some points.
- Gantt chart. Actually, there will be two Gantt charts one is the original one and the second one is an approximation to reality.
- An annex that includes the revisions of the meetings with the supervisor. In them it is included the points brought to the meetings and the key action points draw from the meeting.

The initial project management was based on a pattern of 20 hours/week distributed in 5 days (Mon-Fri), starting on 9th of February 2017, it is expected to finish the project by no earlier than 11th of April 2017, about two weeks before the deadline (2nd May) giving some flexibility about possible delays.

Due to the fact that as I am an ERASMUS student I started the classes later. Furthermore, my first choice in order to develop my project was the module Aerospace Individual Project but as the students of that module had already started their projects on the previous semester me and my two classmates from Spain, were changed to this module. As a result, we got an extension from registry. The final deadline is now 16th of May. However, it has been submitted the 12 of May.

B.2 Logbook

Week 1 2017/02/07 - 2017/02/10

This week start on the 7th of February because due to not having a project supervisor, I decided not to leave the university that day without one. I spoke with Dr Humberto Medina and he results to be an expert in aerodynamics. He offered me three projects one about boundary layers, the second more related to mass and energy transfer and lastly one about the flow behaviour on catalytic converters.

He sent me an article of each one in order to read them and understand them better. Also, he asked me to write a small project proposal. Thus, I spent that days reading the articles. Finally, I decided to study the one about catalytic converters and I wrote a proposal about this. One important thing about the project is that this week I decided the name of the project.

One of the main problems of this week, was to understand properly what the articles were saying. As English is not my mother tongue, I am not very used to read very technical texts. So I took a while for me to read and understand them all.

Week 2 2017/02/10 - 2017/02/17

This week was the deadline for the project proposal of the module and also for complete the ethics. As I have already wrote the one for my supervisor it was easier.

On this week's meeting my supervisor i also met a Phd student who also developed a project about catalytic converters. He was very helpful because he taught me the basics of the software OpenFOAM. Also, he talk with the computers technician and I got an user on some good computers. Furthermore, he recommended me some books that were helpful for him and sent me more articles with some cases to analyse in OpenFOAM.

After the meeting I focused on the software that I was going to use: OpenFOAM. OpenFOAM is an open software that performance on operating systems as Linux or Ubuntu. I have an apple computer so my operating system is OS X. At the beginning I decided to work with the university computers but at the end I was able to install it on my computer.

The installation of the program also took a while of time. And I had some problems with ParaView that it is a post-processor software that helps to visualise the results obtained with the simulations.

Week 3 2017/02/17 - 2017/02/24

During this week with everything working I began to get a deeper knowledge of OpenFOAM by doing the tutorials of the user guide. I also started to analyse the 2D configuration that the Phd student had sent me. I had some problems here because my OpenFOAM is not the same as the one he was using. So I had problems when running the case that I was not able to understand.

This week I also read an article that my supervisor sent me about the last conference they did regarding my topic of study.

Week 4 2017/02/24 - 2017/03/3

On this week meeting my supervisor explained to me all the problems I was having with the software and we managed to fix them. I learnt that I am able to run a case in parallel with the 4 processors of my computer. On the meeting we also talk about his opinion about asking for an extension and he agreed. Finally, we stated that the main difference between previous studies and mine was the 3D study and that the hybrid model to model the monolith is too difficult.

During this week I started to think about how to write my interview review and how to structure it. Furthermore, I was also thinking about the procedure with the 3D configuration.

Weeks 5 and 6 2017/03/3 - 2017/03/17

During this weeks I suffered and operating standstill because on the first week I went home (in Spain). Just after that I left with one of my modules in Coventry University to Dubai for a whole week. In my opinion this caused a big consequence. Even though that while being in Dubai I read some articles being that much time without being in more contact with the project made me lose the thread.

Furthermore, I missed two meetings with my supervisor due to this trips.

Week 7 2017/03/17 - 2017/03/24

During the trip and with no apparent reason my computer began to get very hot while being suspended. When I was using it normally I could feel that it was working way slower. But I decided to wait until the interview review deadline had passed.

Since we arrived on Saturday I began going to the library in order to read, wrote and get some results. But as I have mentioned before due to the time I spent out of the city I was not as much prepared to write as I thought I was. A hard moment started here because writing at the beginning was very hard because I did not have the proper knowledge. So I began to read about the topic but writing was very hard anyway because I do not have all the English vocabulary as I would like to.

Week 8 2017/03/24 - 2017/03/31

After my weekly meeting with my supervisor I was more focused because he has solved all the doubts that appeared during the previous week.

Anyway I think it was too late because for me what I have written is not as good as I would like it to be. However, I have done my best and now I know how I have to do it for he final report.

Week 9 2017/03/31 - 2017/04/7

This week I started with he 3D configuration. As decided with my supervisor I was working in the university computers that had Linux. The main problem this week was related to the connexion with the cluster. To launch the case I was using the Coventry University cluster, but every time I tried to launch it I failed. It was on the queue and there was an error. So I investigated about the cluster and how they work. With the help of my supervisor I solved the problem adding a code line to one file.

I launched the steady flow in a 3D mesh and the pulsed flow in a 2D mesh. With the 3D mesh I obtained good results, but it was not fully converged. The pulsed flow was not simulated because there was one file remaining.

Finally, I spend the end of the week preparing the interview review that was taking place that Friday.

Week 10 2017/04/7 - 2017/04/14

The feedback after the interview review was good. The teacher highlighted that the name was not adequate and in order to express the boundary conditions is better to do reference them (eg. Dirichlet condition).

The file remaining from the 2D pulsed flow was the one that sets the value of the inlet velocity. I asked for this file to Dimitar Borisov, who kindly shared it with me. Hence, I launched the 2D pulsed case.

As the Easter holidays were coming, on the 10th of April there was another meeting. During this meeting we realised that the porous monolith was not well characterised in the 3D mesh. Moreover, this gave us the idea of study the performance of the flow when improving the resolution of the mesh in the porous monolith area.

Humberto recommended me to investigate how to set up a case with multiple regions, so I did. Moreover, I did a research about how to characterise the porous monolith area. The conclusion was that as the length of this area is smaller in the 3D mesh, the pressure drop parameters must be changed according to its new length.

This week I also introduce myself to the open-source software *SALOME*, that was used to developed the 3D mesh. I changed the resolution of the mesh, varying the global discretization and also extending the porous monolith are, that was originally formed by 2 cells, to 4 and 6 cells.

Finally, as the computers of the University were going to be closed until the 18th of April and I was working in parallel with all the cases, I launched all of them in order to be resolved in the cluster. It can be said that this week, I did a big progress in the project

Week 11 2017/04/14 - 2017/04/21

During the beginning of the week, as the University was closed I worked on the writing of the project. At the end of the week, I was able to analyse the results obtained and share them with my supervisor on the meeting of the 20th of April.

It has to be said that during this week I had problems working in parallel due to the cluster. The cluster allows you to launch different cases and solve them faster, but there is a time limit and some cases stopped at that time without being solved. Thus, this cases need to be launched again.

Regarding, the results obtained the 2D steady cases were found to be good. But they differ noticeably with the 3D steady results. It was found out that one of the values that characterised the porous monolith was incorrect so it was changed. Finally, the results of the 2D pulsed flow seemed to be good.

Week 12 2017/04/21 - 2017/04/28

During this week the cases with the different meshes were launched obtaining very bad results. The 24th of May there was a meeting planned with my supervisor and my mate Pablo. Before this meeting Pablo and I were talking about the problems we were having. We discovered that my problem was that the mesh was in metres and not in millimetres. This was because while setting the new mesh there was a command missing that will set the mesh in mm. Once this problem was solved the results started to be better, but not as good as they were supposed to be.

The most important thing achieved this week was the implementation of the pulsed case in the 3D mesh. I worked with the 2D case and the 3D case in order to obtain this new one.

Week 13 2017/04/28 - 2017/05/5

At this point of the semester, I was trying to end the simulations and focus on writing. But in the meeting with my supervisor, we realised that the boundary conditions of the 3D case were not adequate for the v2f turbulence model. Hence, I changed the boundary conditions and launched the new simulations. The results obtained were good. And the only results that were missing at this point were the ones regarding the 3D pulsed flow.

As stated previously, this week I was more focus on writing. The only day I worked less was on Friday because when I wake up I wasn't able to move my head properly due to a whiplash in my neck.

Week 14 2017/05/5 - 2017/05/12

During the last days the results of the 3D pulsed were suppose to be ready by Monday. But when launching them in an attempt of saving more space disk the wiring step was defined too big and consequently the changes on each cycle were not captured properly. Hence, the simulation was launched again delaying the final delivery. But finally, with all the results the project wa ended and submitted the 12 of May.

B.3 Gant chart

The original Gantt chart is presented in figure 18. As can be noted orginally the deadline was the 2nd of May. As shown in figure 19 we can find the current Gantt chart which includes the real progress of the job.

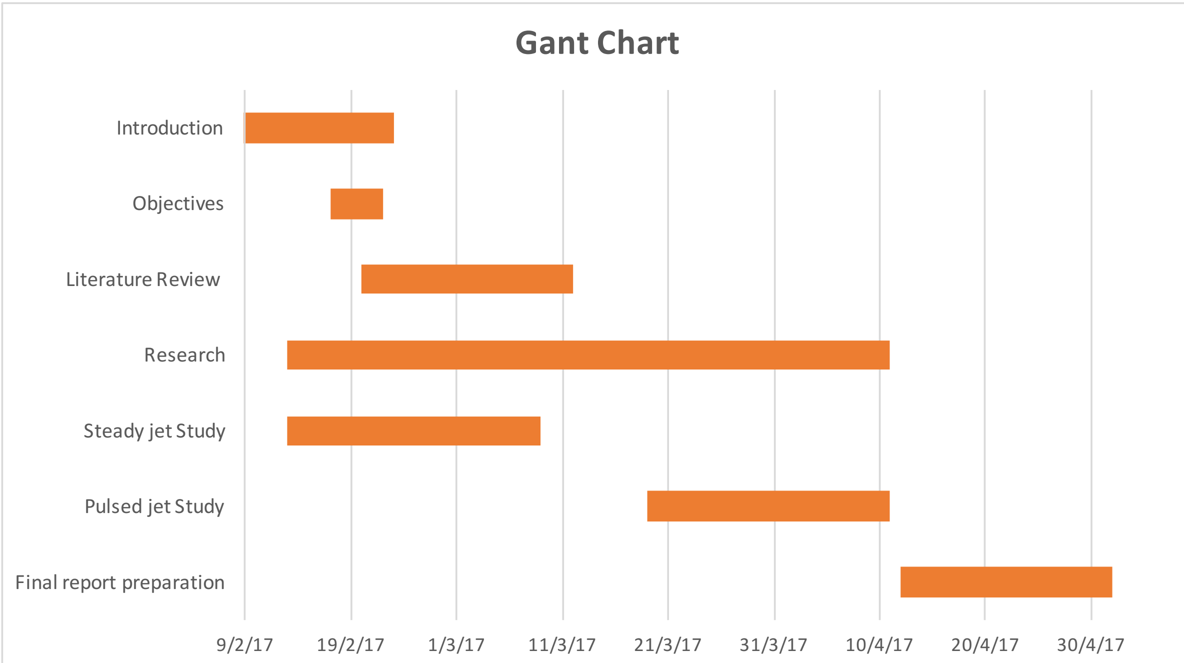


Figure 18: Original gant chart.

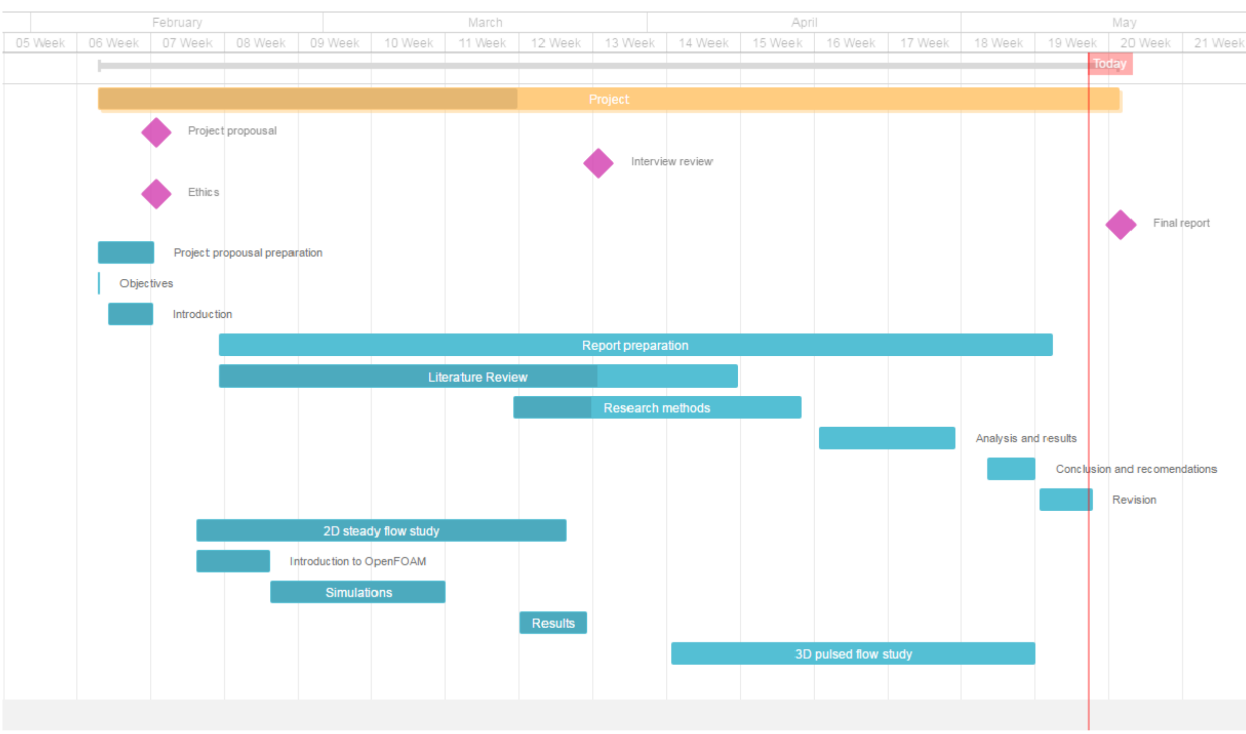


Figure 19: Final gant chart.

C Supervisor Meetings

Supervisor Meeting Form

Please update the below form and use as a template for each of your Director of Studies/Supervisory meetings as evidence.

Failure to evidence contact and engagement with your Director of Studies and/or Supervisors can result in you being withdrawn from the University for non engagement.

Student Name	Mara Salut Escarti Guillem	**Present**
Director of Studies	Humberto Medina	YES/~~NO~~
2nd Supervisor		~~YES~~/NO
3rd Supervisor		~~YES~~/NO3
Date	7/02/2017	
Time	11:00	

Key Points brought to the meeting:

- Discussion of the election of the topic of study between the three possibilities.
- Which kind of study should I do?
- Comparison between steady and pulsed jet doing a RANS study?

Key Action Points

- Decision to make an study of the hybrid flow in a catalytic converter. The study will be done in 3D with URANS and will compare two configurations: steady and pulsed jet.
- Write a proposal project which include: Introduction, aims, objectives, deliverables and a Gant chart.

Date and Time of next meeting: 10/02/2017 10:00

Signature: (Director Of Studies)

(2nd Supervisor)

(3rd Supervisor)

(Student)

Supervisor Meeting Form

Please update the below form and use as a template for each of your Director of Studies/Supervisory meetings as evidence.

Failure to evidence contact and engagement with your Director of Studies and/or Supervisors can result in you being withdrawn from the University for non engagement.

Student Name	Mara Salut Escarti Guillem	**Present**
Director of Studies	Humberto Medina	YES/~~NO~~
2nd Supervisor		~~YES~~/NO
3rd Supervisor		~~YES~~/NO3

Date	10/02/2017
Time	10:00

Key Points brought to the meeting:

- Read the project proposal and discuss its points.
- Revision of the title.

Key Action Points

- Change the tittle to: 3D study using URANS of a steady and pulsed jet in a diffusor without resistance.
- Rewrite the introduction trying to go from more general to more specific (Background, motivation, aim).
- Two deliverables of the result one for the steady jet and other for the pulsed jet.

Date and Time of next meeting: 17/02/2017 10:00

Signature: (Director Of Studies)

(2nd Supervisor)

(3rd Supervisor)

(Student)

Supervisor Meeting Form

Please update the below form and use as a template for each of your Director of Studies/Supervisory meetings as evidence.

Failure to evidence contact and engagement with your Director of Studies and/or Supervisors can result in you being withdrawn from the University for non engagement.

Student Name	Mara Salut Escarti Guillem	**Present**
Director of Studies	Humberto Medina	YES/~~NO~~
2nd Supervisor		~~YES~~/NO
3rd Supervisor		~~YES~~/NO3

Date	17/02/2017
Time	10:00

Key Points brought to the meeting:

- Problems with the visualization of the results of the simulations on *Paraview.* The program doesn't open from the terminal.
- Discussion about which solver adapts better to my study.

Key Action Points

- The best option is download the Paraview program and open it separetly of the terminal. But the file needs to be created first with the commands :
  paraFoam –builtin -touch
- The solver that better adapts to the cases of study is *pimpleFoam* from the incompressible case.
- For the final simulation will be better to use the university cluster. For this get in contact with the person responsible.

Date and Time of next meeting: 24/02/2017 10:00

Signature: (Director Of Studies)

(2nd Supervisor)

(3rd Supervisor)

(Student)

Supervisor Meeting Form

Please update the below form and use as a template for each of your Director of Studies/Supervisory meetings as evidence.

Failure to evidence contact and engagement with your Director of Studies and/or Supervisors can result in you being withdrawn from the University for non engagement.

Student Name	Mara Salut Escarti Guillem	**Present**
Director of Studies	Humberto Medina	YES/~~NO~~
2nd Supervisor		~~YES~~/NO
3rd Supervisor		~~YES~~/NO
Date	24/02/2017	
Time	10:00	

Key Points brought to the meeting:

- Revision of the first configuration of study in 2D. Problems with the mesh due to the turbulence model script.
- Doubts on how to run the case because it needs 4 processors. In the *decomposeParDict* script.
- Revision and discussion of the key dates of the project. (Extension of the interim review)
- Which is the diference between my study and the previous studies? Should I study the hybrid model?

Key Action Points

- Extension of the turbulence model script, now it includes the v2f turbulence model.
- In order to run the case with the 4 processor I have to write in the terminal
  Mpirun –np 4 simpleFoam –parallel >log.simpleFoam &
  Writing this in the terminal the data will be saved in a text file.
- Agreement on the extension of the key dates of the project.
- The difference is the study in 3D and the hybrid model shoudn't be used due to its difficulty.
- Try to get some results from this case and start the next one.

Date and Time of next meeting: 3/03/2017 10:00

Signature: (Director Of Studies)

(2nd Supervisor)

(3rd Supervisor)

(Student)

Supervisor Meeting Form

Please update the below form and use as a template for each of your Director of Studies/Supervisory meetings as evidence.

Failure to evidence contact and engagement with your Director of Studies and/or Supervisors can result in you being withdrawn from the University for non engagement.

Student Name	Mara Salut Escarti Guillem	**Present**
Director of Studies	Humberto Medina	YES/~~NO~~
2nd Supervisor		~~YES~~/NO
3rd Supervisor		~~YES~~/NO

Date	3/03/2017
Time	10:00

Key Points brought to the meeting:

- Revision of the list of contents (with some ideas) to include in the first draft of the report of the project
- Discussion about the templet to follow and how to put the nomenclature in *LATEX*.
- How should I start with the 3D mesh?

Key Action Points

- Agreement on the list of contents and some extra advice received.
- First, I should ask Dimitar for his 3D mesh and some information about how to performance the pulsed jet. Then, the case needs to be changed to my OpenFoam version. Finally, convert the case to a URANs one implementing the pimpleFoam solver. When having some results for the steady case, the pulsed jet will be started.

Date and Time of next meeting: 24/03/2017 10:00

Signature: (Director Of Studies)

(2nd Supervisor)

(3rd Supervisor)

(Student)

Supervisor Meeting Form

Please update the below form and use as a template for each of your Director of Studies/Supervisory meetings as evidence.

Failure to evidence contact and engagement with your Director of Studies and/or Supervisors can result in you being withdrawn from the University for non engagement.

Student Name	Mara Salut Escarti Guillem	**Present**
Director of Studies	Humberto Medina	YES/~~NO~~
2nd Supervisor		~~YES~~/NO
3rd Supervisor		~~YES~~/NO
Date	24/03/2017	
Time	10:00	

Key Points brought to the meeting:

- Which is the best way to see the convergence? How can I plot the residuals?
- Discussion about the software to plot the results of the simulations.
- Which experimental results should I use to verify the data obtained?

Key Action Points

- To see the convergence look at the residuals but also look at a variable (eg. Velocity). This can be done with Octave or with MATLAB. Some scripts to plot this in Matlab or Octave were shared. With this script a maximum of 6 variables can be shown at the same time.
- To plot the results is the same. Either MATLAB or Octave can be used. But I will be using MATLAB.
- The experimental data to be used is the one regarding the Hot Wire Anemometry Tool. This were obtained by Sophie Porter. I can obtain them from the images of the reports or from the EXCEL of the case.
- To outline the results in a report one should follow: Description, Interpretation, Implication, Critique and Contribution.
- Inkscape can be used to draw diagrams.

Date and Time of next meeting: 31/03/2017 10:00

Signature: (Director Of Studies)

(2nd Supervisor)

(3rd Supervisor)

(Student)

Supervisor Meeting Form

Please update the below form and use as a template for each of your Director of Studies/Supervisory meetings as evidence.

Failure to evidence contact and engagement with your Director of Studies and/or Supervisors can result in you being withdrawn from the University for non engagement.

Student Name	Mara Salut Escarti Guillem	**Present**
Director of Studies	Humberto Medina	YES/~~NO~~
2nd Supervisor		~~YES~~/NO
3rd Supervisor		~~YES~~/NO
Date	31/03/2017	
Time	10:00	

Key Points brought to the meeting:

- The OF_PlotResiduals script to plot the residuals is not able to run in MATLAB.
- Debate over whether follow using my personal computer to run the cases or whether use the university ones due to operational problems with my computer.
- How to launch the case to cluster?

Key Action Points

- Agreement on the list of contents and some extra advice received.
- Decission to follow the study with the university computers using OpenFOAM.
- To connect to the university cluster ask for a user. Connect the computer and write in the terminal ' ssh –Y user@zeus.coventry.ac.uk'.
- The case must be launched with the help of the file job_foam.slurm. Write 'qsub foam_job.slurm' to run it.
- To see the case on the cluster queue write 'qstat –u user'

Date and Time of next meeting: 7/04/2017 10:00

Signature: (Director Of Studies)

(2nd Supervisor)

(3rd Supervisor)

(Student)

Supervisor Meeting Form

Please update the below form and use as a template for each of your Director of Studies/Supervisory meetings as evidence.

Failure to evidence contact and engagement with your Director of Studies and/or Supervisors can result in you being withdrawn from the University for non engagement.

Student Name	Mara Salut Escarti Guillem	**Present**
Director of Studies	Humberto Medina	YES/~~NO~~
2nd Supervisor		~~YES~~/NO
3rd Supervisor		~~YES~~/NO
Date	7/04/2017	
Time	10:00	

Key Points brought to the meeting:

- Interview review.
- The pulsed case can't be launched. Because there is an error, it seems like there is one file missing.

Key Action Points

- From the meeting can be highlighted that the 3D results were not converged, that the name was not adequate and that to express the boundary conditions is better to reference the conditions (eg. Dirichlet).
- The file missing is the one that states the pulsed conditions of the velocity.
- The report needs to add more knowledge. To compare the 3D results the vortex images can be compared, represent the mean velocity with the experimental data or analyse the frecuency.

Date and Time of next meeting: 10/04/2017 10:00

Signature: (Director Of Studies)

(2nd Supervisor)

(3rd Supervisor)

(Student)

Supervisor Meeting Form

Please update the below form and use as a template for each of your Director of Studies/Supervisory meetings as evidence.

Failure to evidence contact and engagement with your Director of Studies and/or Supervisors can result in you being withdrawn from the University for non engagement.

Student Name	Mara Salut Escarti Guillem	**Present**
Director of Studies	Humberto Medina	YES/~~NO~~
2nd Supervisor		~~YES~~/NO
3rd Supervisor		~~YES~~/NO
Date	10/04/2017	
Time	10:00	

Key Points brought to the meeting:

- Problems with the convergence of the 3D case. There is something that is not well set and the results are not good.

Key Action Points

- The porous monolith is not well characterised. This is because in the 3D mesh the PM area is smaller. As a result, the values that represent the pressure drop must be changed in order to have ones that agree with the new mesh.
- Furthermore, a mesh study of the porous monolith can be done. Increase the resolution of the mesh in this area and see how it affect to the final solution.
- Start to work in parallel with all the cases. Optimise the time while one case is running prepare the next one.

Date and Time of next meeting: 20/04/2017 10:00

Signature: (Director Of Studies)

(2nd Supervisor)

(3rd Supervisor)

(Student)

Supervisor Meeting Form

Please update the below form and use as a template for each of your Director of Studies/Supervisory meetings as evidence.

Failure to evidence contact and engagement with your Director of Studies and/or Supervisors can result in you being withdrawn from the University for non engagement.

Student Name	Mara Salut Escarti Guillem	**Present**
Director of Studies	Humberto Medina	YES/~~NO~~
2nd Supervisor		~~YES~~/NO
3rd Supervisor		~~YES~~/NO
Date	20/04/2017	
Time	10:00	

Key Points brought to the meeting:

- Presentation of the results of the 2D steady case. Big difference when changing the viscosity ratio.
- The 3D steady case do not adapt to the 2D steady case as much as it is supposed to be.
- Show the results of the 2D pulsed flow case.

Key Action Points

- For the 2D steady case, only use the viscosity ratio recommended in the previous case.
- The residuals of the 3D case seem to be correct and also the velocity profile seems to be fine. No clue of why the results differ.
- Regarding the 2D pulsed case, the results also seems to be right.

Date and Time of next meeting: 28/04/2017 10:00

Signature: (Director Of Studies)

(2nd Supervisor)

(3rd Supervisor)

(Student)

Supervisor Meeting Form

Please update the below form and use as a template for each of your Director of Studies/Supervisory meetings as evidence.

Failure to evidence contact and engagement with your Director of Studies and/or Supervisors can result in you being withdrawn from the University for non engagement.

Student Name	Mara Salut Escarti Guillem	**Present**
Director of Studies	Humberto Medina	YES/~~NO~~
2nd Supervisor		~~YES~~/NO
3rd Supervisor		~~YES~~/NO
Date	28/04/2017	
Time	10:00	

Key Points brought to the meeting:

- The 24th of April there was a meeting planned but in the previous moments talking with my mate Pablo about the problems I was having we discovered that the meshes that had been changed were in meters and no in mm. So I changed the measurements and launch the cases again. Due to this, this day there was no meeting.
- Despite of changing the values that characterise the PM, the results have improved a little bit but not that much.

Key Action Points

- During the meeting we checked the settings of the 3D cases looking for any problems. Finally, it was found that the boundary conditions were not the correct ones. Change them to the corresponding to the v2f turbulence model
  1. V2f → v2WallFunction
  2. K → kLowReWallFunction
  3. Epsilon → epsilonWallFunction
  4. F → fWallFunction

Date and Time of next meeting: 5/05/2017 10:00

Signature: (Director Of Studies)

(2nd Supervisor)

(3rd Supervisor)

(Student)