Calculation of Fluid Dynamic Loads on a Projectile During Firing

- Development of a CFD-modelling Approach

Rikard Fredriksson
Viktor Hellberg
Calculation of Fluid Dynamic Loads on a Projectile During Firing

- Development of a CFD-modelling Approach

Rikard Fredriksson
Viktor Hellberg

Academic supervisor: Jonas Lantz
Industrial supervisor: Harald Svensson
Examiner: Roland Gårdhagen
Abstract

The transition from inner to outer ballistics is a crucial part of the launch of a projectile from a recoilless rifle. Since a launch of the rifle is a rapid process and due to the extreme conditions in terms of accelerations and temperature, physical measurements are hard to achieve.

To gain knowledge about the fluid dynamic loads that act on the projectile during a launch CFD can be a useful technique. In this work a CFD model of the launch process has been developed. Different methods to implement the most important parts of the launch process have been evaluated and compared. An unsteady RANS-model have been utilised in combination with a dynamic mesh to handle the motion of the projectile.

In this work, a spin-stable type of projectile has been analysed. To force the projectile to spin, helical grooves are used inside the launch tube. If the projectile does not fill out and seal the grooves completely, propellant gas can leak through these grooves. In the model it has been evaluated if the leak flow has an impact on the flow field around the projectile and its stability. To simplify the model the grooves were approximated as a gap with constant thickness between the tube and the projectile.

Two different methods to implement the propellant burning have been tested. In the first case a pressure curve known from measurements are implemented. In the second, the mass flow from the combustion is modelled.

This work shows that it is possible to predict the behaviour of the flow during a launch with a CFD model. The leak flow was found to have a significant impact on the flow field in front of the projectile. However, it has also been found that the leakage only have a limited effect on the fluid dynamic forces that works on the projectile during the transition phase.

From this work it has been concluded that CFD can be a useful complement to physical tests and it gives a deeper understanding about the flow when the projectile leaves the launch tube. It has also been concluded that the launch process is an extensive topic and contains many different disciplines; therefore more work is needed to refine the model.

**Keywords:** Fluid Mechanics, CFD, Transient, Ballistics, Recoilless Rifle, Method Development, Dynamic Mesh, CFX, Source Curve, Rigid Body, Leakage
Acknowledgements

This report is a master thesis in computational fluid dynamics and can be seen as the final step on our five year long journey to a master degree in mechanical engineering. The master thesis has been conducted at Saab Dynamics AB in Karlskoga during the spring of 2016 under the supervision of Linköping University.

We would first of all like to express our gratitude to our supervisor Harald Svensson at Saab who have guided us during this whole project and has contributed with much appreciated inputs and knowledge. We are also grateful for the time Torbjörn Green invested in reading our report and for his extensive feedback. Also a large thank to all the people at the analysis department at Saab who have helped us during the project and showed a large interest in our work.

We would also like to thank our opponent Rizad Avdic for his input and thoughts on our work. Last but not least, thanks to our academic supervisor at Linköping University, Jonas Lantz and our examiner, Roland Gårdhagen.

Rikard Fredriksson  
Viktor Hellberg

June 2016
# Nomenclature

## Latin

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$v_o$</td>
<td>Projectile exit velocity</td>
<td>m/s</td>
</tr>
<tr>
<td>$P_r$</td>
<td>Projectile release pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>$U$</td>
<td>Velocity vector</td>
<td>m/s</td>
</tr>
<tr>
<td>$S_m$</td>
<td>Source term</td>
<td>kg/m³ s</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>$I$</td>
<td>Unity matrix</td>
<td>–</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature</td>
<td>K</td>
</tr>
<tr>
<td>$k$</td>
<td>Thermal conductivity</td>
<td>W/mK</td>
</tr>
<tr>
<td>$h_{tot}$</td>
<td>Total enthalpy</td>
<td>J</td>
</tr>
<tr>
<td>$T_{stat}$</td>
<td>Static temperature</td>
<td>K</td>
</tr>
<tr>
<td>$T_{tot}$</td>
<td>Total temperature</td>
<td>K</td>
</tr>
<tr>
<td>$c_p$</td>
<td>Specific heat capacity</td>
<td>J/K</td>
</tr>
<tr>
<td>$p_{stat}$</td>
<td>Static pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>$p_{tot}$</td>
<td>Total pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>$R$</td>
<td>Ideal gas constant</td>
<td>J/kgK</td>
</tr>
<tr>
<td>$N_p$</td>
<td>Number of phases</td>
<td>–</td>
</tr>
<tr>
<td>$r$</td>
<td>Volume fraction</td>
<td>–</td>
</tr>
<tr>
<td>$\forall$</td>
<td>Control volume</td>
<td>m³</td>
</tr>
<tr>
<td>$P$</td>
<td>Linear momentum</td>
<td>kgm/s</td>
</tr>
<tr>
<td>$L$</td>
<td>Angular momentum</td>
<td>kgm²/s</td>
</tr>
<tr>
<td>$F$</td>
<td>Force</td>
<td>N</td>
</tr>
<tr>
<td>$M$</td>
<td>Torque</td>
<td>Nm</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass of projectile</td>
<td>kg</td>
</tr>
<tr>
<td>$C$</td>
<td>Constant</td>
<td>s/m²</td>
</tr>
<tr>
<td>$b$</td>
<td>Constant</td>
<td>–</td>
</tr>
<tr>
<td>$N$</td>
<td>Constant</td>
<td>–</td>
</tr>
<tr>
<td>$A_{eff}$</td>
<td>Effective burning area</td>
<td>m²</td>
</tr>
<tr>
<td>$F_D$</td>
<td>Drag force</td>
<td>N</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Drag coefficient</td>
<td>–</td>
</tr>
<tr>
<td>$A$</td>
<td>Cross section area</td>
<td>m²</td>
</tr>
</tbody>
</table>
Greek

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>ρ</td>
<td>Density</td>
<td>kg/m^3</td>
</tr>
<tr>
<td>μ</td>
<td>Dynamic viscosity</td>
<td>Ns/m^2</td>
</tr>
<tr>
<td>Γ_{disp}</td>
<td>Mesh stiffness coefficient</td>
<td>–</td>
</tr>
<tr>
<td>δ</td>
<td>Displacement</td>
<td>m</td>
</tr>
</tbody>
</table>

Abbreviations and Acronyms

<table>
<thead>
<tr>
<th>Letter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulations</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Solution</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Averaged Navier Stoke</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport</td>
</tr>
<tr>
<td>DOF</td>
<td>Degrees Of Freedom</td>
</tr>
<tr>
<td>GGI</td>
<td>General Grid Interface</td>
</tr>
<tr>
<td>PID</td>
<td>Proportional Integral Derivative</td>
</tr>
</tbody>
</table>
# Contents

Abstract...............................................................................................................................................i
Acknowledgements............................................................................................................................ii
Nomenclature ................................................................................................................................... iii
Contents............................................................................................................................................. v

1. Introduction ...................................................................................................................................1
  1.1 Problem Description .................................................................................................................2
  1.2 Objectives .................................................................................................................................2
  1.3 Limitations ...............................................................................................................................3
  1.4 Outlines ....................................................................................................................................3

2. Background ....................................................................................................................................5
  2.1 Working Principle of the Rifle .................................................................................................5
    2.1.1 Projectile Motion ..................................................................................................................7
    2.1.2 Projectile Release ..................................................................................................................7
    2.1.3 Base Plane Breakage .............................................................................................................7
    2.1.4 Leakage around the Projectile ............................................................................................7
    2.1.5 Friction between the Components ......................................................................................8
    2.1.6 Rotation of the Projectile ................................................................................................. 9
  2.2 Previous Work .........................................................................................................................9

3. Theory ........................................................................................................................................ 11
  3.1 Ansys CFX Solver .....................................................................................................................11
  3.2 Governing Equations .................................................................................................................11
  3.3 Ideal Gas ...................................................................................................................................12
  3.4 Multiphase Flow .......................................................................................................................12
  3.5 Turbulence ...............................................................................................................................13
  3.6 Mesh Displacement ...................................................................................................................14
  3.7 Rigid Body Motion ...................................................................................................................14

4. Method ........................................................................................................................................ 16
  4.1 Setup .......................................................................................................................................16
    4.1.1 Computational Domain ......................................................................................................16
    4.1.2 Boundary Conditions .......................................................................................................17
    4.1.3 Material Properties ............................................................................................................18
    4.1.4 Solver Settings ...................................................................................................................18
  4.2 Mesh .......................................................................................................................................19
  4.3 Inner Ballistics .........................................................................................................................21
    4.3.1 Motion ...............................................................................................................................21
1. Introduction

In this master thesis the launch of a projectile from a handheld recoilless rifle will be studied. The rifle consists of a launch tube that, similar to a small calibre gun, is loaded in the rear end with a cartridge containing the projectile and propellant. When the propellant is ignited the pressure increases which results in a force on the projectile that will accelerate it. The main difference from a small calibre gun is that the rear end of the launch tube is open and at a certain pressure the propellant gas is allowed to flow out in the backward direction. The result is a counteracting force which gives an almost recoilless firing of the rifle. Another difference from a small calibre gun is the projectile velocity. For the rifle studied in this work, the projectile velocity is in the range 200-300 m/s. In comparison, for a small calibre gun it can be up to 1200 m/s. An example of the type of weapon that was studied can be seen in Fig. 1.1.

![An example of the type of rifle subject for the study](image)

**Fig. 1.1: An example of the type of rifle subject for the study [1].**

The aim with this work is to develop a CFD model that can be used to analyse the transient flow around the projectile when it leaves the launch tube. The moment immediately after the projectile leaves the tube will be of particular interest in this study. This part of the launch process is crucial since instabilities in the projectile induced here will greatly affect its continuing path and the precision.

It is known that outflowing propellant gases can affect a projectile after it has left the launch tube [2]. Although, no studies on this particular rifle has been conducted. It is therefore interesting to analyse what fluid dynamic forces that acts on the projectile for this type of weapon and how far the projectile needs to travel in order to reach a steady free stream condition.

Previous studies of projectile launches have mostly concerned firing of small calibre guns from barrels with a closed rear end. The focus has been on shock wave pattern formed around the bullet. In general the exit velocity is much higher for small calibre guns, therefore the conclusions from those results cannot be directly applied on the type of rifle studied in this work and another modelling approach may be needed.
1.1 Problem Description

The launch of a projectile is normally divided into two main phases, the inner and outer ballistics.

- The inner ballistic part is mainly a one-dimensional problem where the projectile is directed by the launch tube and accelerated by the pressure from the propellant gas burning.
- The outer ballistics refers to when the projectile has left the launch tube and is flying in the free air. In this phase the projectile path is decided by aerodynamic loads and gravity.

Both the inner and outer ballistic parts of the launch have been studied previously and models that can be used to predict the behaviour of the projectile exist. However, in reality there is no distinct border between the inner and outer ballistics. A short distance after the projectile has left the tube it is still affected by the outflowing propellant gases. This part of the launch can be referred to as a third intermediate ballistic phase and is crucial for the rest of the projectiles path. Since the projectile in this phase is free to move in all directions, uneven loads can redirect the projectile or cause it to wobble and give unpredictable behaviours.

The launch of a projectile is a rapid process. The time from ignition until the projectile leaves the muzzle is a few milliseconds. Because of high temperatures and the extreme accelerations, physical measurements of the forces on the projectile are almost impossible to achieve. In order to analyse the projectile during firing, one is therefore limited to visual observations, mostly by use of high speed cameras. A problem with this method is that smoke will limit the visibility. To gain knowledge about why certain behaviours occur, it is desirable to develop a CFD-model of this transition period. This can also make it possible to combine inner and outer ballistic models to make more precise simulations of the complete launch process.

1.2 Objectives

The objective of this work is to develop a CFD-model that can be used to analyse the flow around the projectile during the transition from inner to outer ballistics. The main question is what fluid dynamic loads working on the projectile during this transition period and how these forces affect the stability of the projectile. In addition, the distance from the muzzle to where the projectile can be considered to have reached steady free stream conditions are of interest.

It is to be investigated how the rifle can be modelled in order to capture the flow behaviour known from real testing and what simplifications that can be made. For instance, how much of the inner ballistic phase needs to be included and how leakage of propellant gas around the projectile can affect the flow.
In order to answer these main questions the problem needs to be divided into smaller problems which mainly can be defined as:

- Develop a modelling technique, including computational domain, mesh and setup, which is suitable for the problem.
- Investigate what physics that need to be included in the model and how this can be implemented.
- Compare models with different degrees of complexity to determine what is most important to focus on in the modelling in order to capture the flow physics that are known from real tests.

1.3 Limitations

This work is to be conducted within the framework of a master thesis, corresponding to approximately 20 weeks full time work. Since the work is done by two persons the total time for the project is around 1600 hours. This means that there is a limit in how extensive the study can be.

The launch of a projectile is a complex process involving rigid body dynamics, aerodynamics, combustion, thermodynamics and several other areas. Due to this, a number of delimitations were necessary in order to perform the study with the available resources.

Since the transition phase from inner to outer ballistic conditions was the focus in this study, the inner ballistic phenomena in terms of combustion and thermodynamics will not be covered. The focus in this work is CFD-modelling and only the parts of the inner ballistic phase that is relevant in this perspective is to be considered.

For the simulations computers with six cores, 3.5 GHz processor and 64 GB RAM was used. This means that there was a limit in computational power which limits the domain dimensions and mesh sizes in order to get reasonable simulation times.

The simulations were performed with Ansys CFX 16.0. The reason for using CFX was that a model of a similar problem was available and was used as a starting point.

1.4 Outlines

This report is divided into different sections that will describe the work in detail. In the introduction the problem is described in sense of what the goal is and what the limitations are. In the background section a deeper description of the problem is given. In this section previous work is also introduced and what has been concluded in those studies.

The theory section is used to describe the most important equations and theory behind the solver. However, since there are a lot of equations behind the simulations not all will be described and therefore if the reader wish to read more about the solver theory the Ansys Theory Guide is recommended [3]. In the method section the numerical setup and different methods to implement parts of the launch in the model is described. Also the mesh convergence and domain size is investigated in this section.
In the result and discussion section the results will be displayed and discussed. The method and modelling approach will also be covered and discussed in this section. In the conclusion the most important results and lessons is covered. Some future work is also proposed to get a picture of what the next steps in this work might be. As a final section, some perspectives are discussed in regard to today’s society.
2. Background

Today simulation models of the inner ballistic part of a launch as well as the outer ballistic phase including effects at target impact exist. Inner-ballistic models can be used to predict the projectile muzzle exit velocity, $v_0$. These $v_0$ estimates are then used in outer-ballistic models to simulate the path of the projectile. In reality, there is a certain deviation in the projectile’s trajectory, for instance due to that it never leaves the tube completely straight. There are also small variations in $v_0$, geometry and projectile mass due to tolerances in the production. This can be accounted for by doing several simulations and letting the projectiles direction and $v_0$ vary slightly at the muzzle exit.

The reason for this direction deviation in reality is not completely known. One reason can be that the launch tube is not a perfect rigid body but oscillates slightly during a launch. Another possible theory is that forces from outflowing propellant gases disturb the projectile just after that it has left the muzzle. It has been shown in previous studies [2] that this is the case for various types of launching systems with different calibres. However, [2] only covers launch tubes with closed rear ends. This means that the pressure in the tube, when the projectile exits the muzzle, is much higher than for the type of rifle studied in this work.

The scope of this work is to develop a model to analyse the flow around the projectile in the intermediate-ballistic phase to be able to calculate the fluid dynamic forces that acts on the projectile and possibly can affect its stability. If the instabilities of the projectile induced at the muzzle exit can be predicted, it might be possible to simulate the whole launch cycle from ignition to target impact. This can lead to a large cost reduction in the development of new products since real trials are very expensive and require a lot of validation and testing to secure statistically significant results. Therefore, a CFD model of the transition ballistic phase has potential to contribute to a more efficient development work and more optimised designs of new projectiles.

2.1 Working Principle of the Rifle

As described earlier the rifle consists of a tube that is loaded with a cartridge containing the projectile and the propellant substance. From the beginning the propellant is enclosed by the projectile and the rear end of the cartridge inside the tube. When the propellant is ignited a pressure is built up and at a certain pressure the projectile is released from the cartridge and starts to accelerate forward. At this moment a counteracting force makes the rifle accelerate in the opposite direction, i.e. recoils backward.

The rear end of the cartridge (referred to as the base plane, see Fig. 2.1) is designed to burst at a given pressure. This allows the propellant gas to flow out in the rearward direction. At the rear end of the tube there is a funnel (referred to as the venturi in this work, see Fig. 2.2) and the high pressure from the outflowing propellant gases now gives a force on the venturi walls acting in the forward direction. This force counteracts the rearward acceleration of the rifle that is up to 1000 G [4]. The result is that during a launch the rifle only moves a couple of millimetres and the user experience the firing as recoilless. The principle of the initial part of the launch is shown in Fig. 2.3.
Fig. 2.1: The base plane is indicated by the red marker.

Fig. 2.2: The venturi is indicated by the red area [1].

Fig. 2.3: The principle of the inner ballistic part of the launch. a) When the propellant is ignited the pressure rises in the cartridge. b) At a certain release pressure the projectile starts to accelerate. At this moment a counteracting force accelerate the rifle in the opposite direction. c) The pressure continues to rise and at a certain level the base plane breaks. The outflowing gas now gives a force on the venturi wall that counteracts the movement of the rifle.

On the inside wall of the launch tube helical grooves are present, see Fig. 2.4. These grooves force the projectile to spin along its longitudinal axis and help stabilise it in the air. There are also rifles with smooth tubes that instead have projectiles equipped with stabilising fins. This type of projectiles will however not be covered in this study.

Fig. 2.4: The helical grooves that are used to force the projectile to spin around its longitudinal axis.

The complete launch process is complex, which means simplifications are necessary. From a CFD perspective of the intermediate ballistic phase, it is important to capture the phenomenon that will affect the projectile. The flow outside the launch tube in the vicinity of the muzzle will be affected before the projectile reach the end of the tube. Therefore, some parts of the inner ballistic phase of the launch needs to be considered.

A number of parts of the launch cycle have been identified to have significant impact on the wanted results and must be handled in the model in some way. In the following sections these parts will be further explained.
2.1.1 Projectile Motion

The motion of the projectile inside the tube will affect the flow around the muzzle at a quite early stage. When the projectile starts to accelerate a pressure wave is created in front of it and propagates out from the tube [5–7]. The pressure changes in the vicinity of the muzzle must be captured accurately in order to calculate the right forces on the projectile, therefore the inner ballistic motion of the projectile needs to be considered in some way.

The projectile velocity during the inner ballistic phase can either be seen as a known entity, or the pressure acting on the projectile can be used to drive the motion and calculating the velocity by solving the equations of motion for the projectile.

If the equations of motion are to be solved during the simulation the pressure increase, due to the burning of propellant, needs to be implemented in the model.

2.1.2 Projectile Release

The projectile is clamped into the cartridge, therefore a certain force is needed before it is released and starts to accelerate. The release force is time dependent and its characteristics are known. To simplify the model the release force was approximated as a specific value which means that the release pressure \( P_r \) can be calculated.

If the projectile velocity is to be calculated during the simulation, this delay between the propellant ignition and the start of the projectile acceleration needs to be considered in the model in order to get correct values and the correct behaviour of the flow.

2.1.3 Base Plane Breakage

The base plane disk is made of a brittle material which breaks when a certain pressure is reached inside the tube. Depending on how the pressure increase is implemented in the model the rear outflow and the breakage of the base plane may be crucial. The problem here is that the breakage is a rapid structural mechanics problem that needs to be represented in some way; the time it takes for the base plane to go from intact to broken is in the order of 0.1 ms.

Trials have been conducted where the pressure have been increased slowly by use of compressed air to see when the disk break. However, these tests do not capture the transient behaviour in a real firing. The propellant burning is so rapid that the pressure continues to rise during the time the disc bursts, and for a precise model it may not be enough to treat the base plane as just intact or broken depending on the pressure.

2.1.4 Leakage around the Projectile

As mentioned earlier, helical grooves inside the launch tube forces the projectile to spin in order to stabilise it during the trajectory to the target. It is not completely known how these grooves affect the flow in the tube. Around the projectile there is a ring called slipping ring of a deformable material that is forced into the grooves in order to achieve the rotation. The material in this ring depends on the type of projectile and can be either a soft metal, for instance copper, or a plastic material.
It is assumed that the slipping ring fills out the grooves completely. However, when slow motion movies from test firings are studied, it can be seen that propellant gas flow out from the tube before the projectile leaves the muzzle. Two possible reasons for this have been proposed.

- One possibility is that the slipping ring is deformed in such way that the grooves are not completely sealed which allows for a continuous flow around the projectile, see the yellow area in Fig. 2.5, during a launch.
- The other possible reason for the leak flow is that after the projectile is released, it travels a short distance before it enters the grooved section of the tube. During this short distance, propellant gas may flow around the projectile before the slipping ring seals the grooves, see Fig. 2.6.

In some studied slow motion movies it appears like there is a short puff of propellant gas leaving the tube before the projectile. This should indicate that it only leaks around the projectile the short period before it enters the grooves. But as this behaviour was not obvious in all studied movies both possible reasons for the leak flow was considered.

![Fig. 2.5: The projectile inside the tube where the yellow indicates the leakage due to the grooves.](image)

![Fig. 2.6: a) The leakage created due to that the first part of the sequence is not completely sealed. b) The slipping ring has sealed the grooves.](image)

### 2.1.5 Friction between the Components

Since there is contact between the projectile and the tube inner wall a friction force will act on the projectile. The friction force will be highly dependent on the material in the slipping ring and can have a significant impact on the projectile velocity.

This force has been measured experimentally by pushing a projectile through the tube with a hydraulic piston. These kinds of experiments results in a rather constant friction force through the whole tube. However, it is uncertain how well these measured forces correspond to a real firing case. In reality with high velocity and temperature for instance a copper material may act like a lubricant and the friction force can be highly dependent on the velocity [8]. However, the measurements give an indication of what magnitude the force should have.
2.1.6 Rotation of the Projectile

In order to get a stable flight, the projectile is rotating due to the groves in the launch tube. The rotation speed of the projectile when it leaves the tube is known. It is also known that the projectile maintain most of the rotational speed during the flight until it reach the target. Therefore, the rotational speed can be seen as constant during the short transition phase that will be considered in this work.

The rotation will result in a rotational velocity component in the boundary layer. The rotation might be important to take into account for an increased accuracy in the near wall representation.

2.2 Previous Work

The problem to simulate a projectile leaving a tube has been addressed in some previous works and therefore these references were a starting point in the creation of the model in this project. Most of the previous works has its focus on bullets leaving a rifle and therefore the model differs a bit in comparison to this work. Since the velocity for a bullet can be between 180-1220 m/s [9] (for example [6] uses a velocity of 600 m/s and in [7] the velocity is as high as 1224 m/s) there is quite a big difference in comparison to the studied recoilless rifle where the velocity of the projectile is around 250 m/s. There are also some differences in the boundary conditions. Previous works have been about rifles with closed rear ends. For recoilless rifles though, the base plane breakage leads to some other properties and boundary conditions.

In previous studies [6,10] the movement of the projectile is modelled with help of a dynamic mesh that updates how the mesh looks as the projectile moves through the domain. This gives a very computational heavy model but with a good mesh all the time. While [7,11,12] uses a moving mesh where the elements is compressed or stretched but it is still the same mesh and this leads to a not as heavy model as the re-generating mesh. In [11] they also uses the re-generating mesh option if the elements gets too stretched, then the mesh is generated again and then the moving mesh is restarted with the new mesh. It can be seen that these two methods dominates how the motion of the projectile is created and both methods gives good results in the studied reports. But there is also discussed in [6] that the method of compressing the elements would be better since the computational power is decreased a lot and therefore in their future work it was suggested that a study with this setup should be done.

There is one more method that is used to simulate the motion of the projectile and that is to use a immersed solid approach [13,14]. The idea of the immersed solid method is to have a stationary mesh and letting the solid object move through the grid. By adding a source of momentum to the nodes that are inside the immersed solid the flow is forced to follow the solid object [3]. This method makes the model less complex since no dynamic mesh is used but due to limitations in CFX this method is not available for this work. This is due to that in CFX the immersed solid approach only works for single-phase and incompressible fluids [15].

In previous works different turbulence models have been used. There does not seem to be a model that is overrepresented in some way. Most of the common models can be found, for example the standard k-ε [5] but also more computational heavy schemes such as Large Eddy Simulations (LES) [6]. The choice of turbulence model does not seem to affect the previous results since most of the works gets equivalent results with some small difference and they capture the same behaviour in the flow.
When it comes to the computational domain, most of the previous works has used the same setup. The tube and projectile are modelled in the middle of a large cylinder representing the surrounding atmosphere. This cylinder has had some different sizes but it can be seen that the dimensions for the cylinder is much larger than the pipe. In [7] for instance, the diameter of the cylinder is 40 times the projectile diameter and the length of the domain is 80 times the projectile diameter.

The interaction between the slipping ring and the grooves is a fairly unknown area since there is no way to monitor how the slipping ring deforms during the complete test. It is only possible to see a before and after result [8]. There have been some studies in this field in order to investigate how the slipping ring affects the friction on the projectile. It has been concluded that there is a lot of properties that affect the friction force e.g. material, dimensions, temperature and velocity. The friction force can be divided into different stages with different properties for the force. At the first stage when the projectile is accelerated fast and the slipping ring undergoes plastic deformation at high strain rates, there is a large friction force. As a second stage the friction force decreases due to the temperature increase in the material which leads to surface melting. The melted material will act as a lubrication between the slipping ring and the grooves [8,16].
3. Theory

In this section some of the relevant theory behind the method used in this work will be presented. Since this is an extensive work involving several different disciplines not all theory will be described in detail. It will be assumed that the reader has basic knowledge in solid and fluid dynamics and is familiar with CFD.

3.1 Ansys CFX Solver

CFD is a technique to numerically solve the governing equations for a fluid dynamic problem. In this project the CFD software Ansys CFX was used. CFX is a node based implicit finite volume solver that divides the domain into small control volumes and solves the equations iteratively for each control volume. This gives an approximation of the result for each control volume and can then be translated into the complete domain [3,17].

In order to run a CFD simulation in Ansys CFX, three different types of software are needed [3,15]

- Geometry Generation Software – This software is used to create the domain that is to be analysed and contains all geometrical entities. In order to work with CFX the geometry needs to be three-dimensional. In this project Ansys Design Modeler was used.
- Mesh Generation Software – In order to translate the geometry to a number of control volumes a mesh generation software is needed. For this Ansys Meshing was used.
- Solver software – CFX is the solver used for the calculations. It contains two parts. The first part is the Physics Pre-processor where all boundary conditions, material properties and definition of the run are specified. As a second step the Solver and CFD Job Manager calculates the solution with the finite volume method.

In addition a post-processing software which allows the user to graphically display the results from the simulation is needed. For this Ansys CFD-Post was used.

3.2 Governing Equations

The set of equations that govern a compressible fluid flow are the unsteady Navier-Stokes equations and the continuity equation. These are solved by CFX in their conservation form [3].

The conservation of mass is represented by the continuity equation and can be written as

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = S_m
\]  

(1)

Where \( \rho \) is the fluid density, \( \mathbf{U} \) is the velocity vector and \( S_m \) is a source term.
The conservation of momentum is presented by the Navier-Stokes equations and can be expressed as
\[
\frac{\partial (\rho U)}{\partial t} + \nabla \cdot (\rho U \otimes U) = -\nabla p + \nabla \cdot \tau
\] (2)

Here \(p\) is pressure and \(\tau\) is described by
\[
\tau = \mu \left( \nabla U + (\nabla U)^T - \frac{2}{3} I \nabla \cdot U \right)
\] (3)

Where \(\mu\) is dynamic viscosity and \(I\) is a unity matrix.

The total energy in the flow is described by the energy equation as
\[
\frac{\partial (\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho U h_{tot}) = \nabla \cdot (k \nabla T) + \nabla \cdot (U \cdot \tau)
\] (4)

Where \(T\) is temperature, \(k\) is thermal conductivity and \(h_{tot}\) is the total enthalpy which can be expressed in term of the static enthalpy \(h(T, p)\) by
\[
h_{tot} = h + \frac{1}{2} U^2
\] (5)

### 3.3 Ideal Gas

The assumption of ideal gas and the second law of thermodynamics give that
\[
\rho h_{tot} = \rho_{stat} \exp \left( \frac{1}{R} \int_{T_{stat}}^{T_{tot}} \frac{c_p(T)}{T} dT \right)
\] (6)

where \(T_{stat}\) and \(T_{tot}\) is the static and total temperature, \(c_p\) is the specific heat capacity, \(\rho_{stat}\) and \(\rho_{tot}\) is the static and total pressure and \(R\) is the ideal gas constant [3].

### 3.4 Multiphase Flow

To handle the different properties of the propellant gas and the surrounding air, the simulation was set up as a multiphase model. The different phases will be indicated by an index \(\alpha\) where \(\alpha = 1..N_p\), \(N_p\) is the number of phases (two in this case). In this work a homogeneous multiphase model was used. This is a somewhat simplified multi fluid model [3] that assumes that all components share a common velocity field and that all transported quantities, except for volume fraction, are the same for all phases.

For the multiphase model the continuity equation (1) is rewritten into
\[
\frac{\partial}{\partial t} (\rho_{\alpha} U_{\alpha}) + \nabla \cdot (\rho_{\alpha} U_{\alpha} U) = S_{ma}
\] (7)
Where \( r_\alpha \) is volume fraction and \( S_{ma} \) is the mass source of fluid \( \alpha \). The production of propellant gas from the combustion will be implemented as a mass source of the propellant gas phase. In the momentum and energy equations (2)-(5), \( \rho \) and \( \mu \) is given by

\[
\rho = \sum_{\alpha=1}^{N_p} r_\alpha \rho_\alpha
\]

and

\[
\mu = \sum_{\alpha=1}^{N_p} r_\alpha \mu_\alpha
\]

The conservation of volume demands that the volume fractions sum up to unity

\[
\sum_{\alpha=1}^{N_p} r_\alpha = 1
\]

### 3.5 Turbulence

Almost all flows that appears in industrial applications becomes turbulent at high Reynolds numbers [17]. If properties for air at room temperature and the diameter and maximum velocity of the projectile are used the Reynolds number for the studied case becomes \(1.5 \times 10^6\). This will result in turbulent areas. It is therefore necessary to in some way represent the effects that the turbulence has on the flow. Different ways of handling the turbulence exists.

The most advanced method is to fully resolve the turbulent fluctuations, known as DNS (Direct Numerical Solution). This means that the unsteady Navier-Stokes equations are solved on a mesh small enough to capture the smallest scales and with a time step size shorter than the fastest fluctuations in the flow. This method is however so computational heavy that it is impractical to use for engineering purposes with today’s computer capacity.

An alternative is to resolve the largest scales of the turbulence while the smallest fluctuations are modelled in some way. This is often referred to as LES (Large Eddy Simulations). This can be an alternative if the turbulent effects are of great importance, in most applications though; it is not motivated since it still is very computational demanding.

The by far most common way to handle turbulence is to model all its effects on the flow. All fluid quantities are decomposed into a time averaged part and a fluctuating part, \( \bar{\Phi} \) and \( \phi' \) giving

\[
\phi(t) = \bar{\Phi} + \phi'(t)
\]

This is called Reynolds decomposition and if it is applied on velocities and inserted in the Navier-Stokes equations the RANS (Reynolds Averaged Navier Stokes) equations are achieved. This give rise to some additional terms called Reynold stresses in the RANS equations which needs to be handled in order to close the equation system. The most well documented method to come around this problem is to use a two-equation eddy viscosity
model, often k-\(\varepsilon\) or k-\(\omega\) models [17]. For more information about turbulence modelling literature about basic CFD-modelling for instance [3,15,17] are recommended.

For the simulations performed in this work the SST k-\(\omega\) model was used. This model is well documented and is known to perform well in aerodynamic applications [18]. The k-\(\omega\) model is in general more accurate in near wall regions, which is important when forces on a body are to be calculated. The k-\(\varepsilon\) model is more accurate and preferable in the bulk flow. The idea of the SST k-\(\omega\) is to blend these two models to receive accurate results both in near wall regions and in the far field flow.

### 3.6 Mesh Displacement

To handle the movement of the boundaries of the projectile, a deformable mesh was used. In CFX the motion of nodes in a sub-domain of the mesh can be specified. In this case the motion of the domain surrounding the projectile was determined by solving the equations of motion for the projectile.

For the remaining nodes where the motion was not specified CFX uses a *Displacement Diffusion* method [15]. This means that the displacements applied on the specified subdomain are diffused to the rest of the mesh by solving

\[
\nabla \cdot (\Gamma_{disp} \nabla \delta) = 0
\]

(12)

Where \(\Gamma_{disp}\) is a mesh stiffness coefficient and \(\delta\) is the displacement relative to the previous mesh. Equation (12) is solved iteratively at the start of each time step loop.

In order to preserve the relative volume of the mesh elements, the stiffness was increased in areas with small elements, meaning large elements take a larger part of the deformation. The mesh stiffness is varied in the domain according to

\[
\Gamma_{disp} = \left(\frac{\nabla_{ref}}{\nabla}\right)^{c_{stiff}}
\]

(13)

Where \(\nabla\) is the control volume size and \(\nabla_{ref}\) is the mean control volume size in the domain. The exponent \(c_{stiff}\) decides how quickly the stiffness is changed. The value of \(c_{stiff}\) was put to 2.0 which is the default value in CFX, no further investigation how a change in this value affect the result was done.

### 3.7 Rigid Body Motion

The calculated pressure in the tube was used to drive the motion of the projectile. CFX has the possibility to solve the equations of motion for a rigid body with six degrees of freedom, where the body is defined by a number of surfaces, in this case the wall surfaces of the projectile.

The equation of motion for a rigid body can be written according to (14) and (15), which state that the rate of change of linear and angular momentum \(P\) and \(L\) are equal to the applied force and torque \(F\) and \(M\) respectively [3].
\[
\frac{dP}{dt} = F \tag{14}
\]
\[
\frac{dL}{dt} = M \tag{15}
\]

In this case, the projectile was only allowed to move in the longitudinal direction of the pipe, i.e. 1 DOF, meaning that the angular momentum equation could be neglected. The equation of linear momentum could be simplified to the following one-dimensional problem

\[
m\ddot{x} = F = F_{\text{Aero}} + F_{\text{Ext}} \tag{16}
\]

Where \( x \) is position in the direction of the translation, \( m \) is the mass of the projectile, \( F_{\text{Aero}} \) is the resulting aerodynamic force and \( F_{\text{Ext}} \) is an applied external force, for instance friction between the tube and the projectile. Equation (16) is solved iteratively in CFX by use of the Newmark integration scheme; for more information, see [3].

If the angular momentum equations are to be solved, also the moments of inertia for the body need to be calculated. Since the rotational velocity of the projectile is rather constant during the short part of the launch considered in this work, the rotation of the projectile can be handled in other ways, see section 4.8.
4. Method

In order to get the desired results a well-posed model needs to be proposed and within this section all parts will be thoroughly described. This part contains the numerical setup for the domain and mesh, and a mesh and domain analysis. In the background section the working principle of a rifle was described and the methods to implement the different parts will be described.

4.1 Setup

In this section the computational domain, geometries, boundary conditions, material properties and general solver settings used in the models will be described.

4.1.1 Computational Domain

The computational domain consists of a cylinder placed uniaxial with the launch tube. The launch tube is modelled with an inner diameter D=84 mm, length L=850 mm and wall thickness of 10 mm. At the rear end of the tube the venturi was located. Fig. 4.1 shows a cross section of the launch tube geometry and the computational domain.

The total radius of the computational domain is 800 mm and extends 1650 mm in front of the launch tube muzzle. In order to evaluate if the domain was made large enough a domain size analysis was done and it can be seen in section 4.9.2 Domain Size.

Since the geometry is axisymmetric, the computational cost was reduced by using a wedge piece and applying symmetry boundary conditions. In the future when more computational power is available it is desirable to run the simulations in full 3D. This will also be necessary in order to calculate radial forces properly or if a non-axisymmetric projectile geometry is used. The model was developed with this in mind and constructed in such way that it easily can be scaled up to a full 360 degrees 3D model.

![Fig. 4.1: The computational domain and the different boundary conditions. Where 1 is wall-, 2 is opening- and 3 is symmetry-condition.](image)

The projectile used in the simulations can be seen in Fig. 4.2. This is a generic model of a projectile where the shape has been somewhat simplified. This geometry was selected since many different types of projectiles exist. Since this project is more of a method development this general projectile geometry was considered a good choice to simplify the work. The real
Projectiles often have this typical shape with some small differences and therefore this simplified geometry should give a good picture of the general structures of the flow around the projectile.

In order to avoid that the distance between the projectile and the tube inner wall goes to zero when there is no gap between the projectile and the wall, a notch was added to the geometry. This area is highlighted in Fig. 4.2 and it makes it possible to create a good quality mesh and avoid highly skewed elements.

**Fig. 4.2:** The projectile that was used during the simulations. In the picture the little edge that was created can be seen and it has a size of 1 mm.

### 4.1.2 Boundary Conditions

In this model three types of boundary conditions were used, wall, opening and symmetry. These conditions were used with some different settings for the different parts in the domain. Below follows the different conditions and in Fig. 4.1 the placement of the different conditions can be seen.

1. **Wall** – This condition was used for all sides of the projectile and launch tube. Since the properties of the walls were unknown the standard values in CFX were used. The wall was set to “No Slip Wall” i.e. the velocity at the wall is equal to zero.
2. **Opening** – For the outer sides of the domain, opening boundary conditions were used. These boundaries can be seen as far away from the areas of interest and should be undisturbed; therefore the pressure was set to zero relative to the reference pressure. This type of boundary condition was also used for the rear end boundary of the venturi. That this was valid was tested by compare with a model where a larger domain around the venturi was included. It was found that this did not have an impact on the studied results. The turbulence intensity was set to 5 percent (in CFX this is “Medium” turbulent).
3. **Symmetry** – Since this case was modelled as a wedge piece of the domain, symmetry conditions was used on the sides. This means that the normal velocity component at the symmetry plane is zero and there is no flux across the boundary.
4.1.3 Material Properties

Compressible flow was used for both air and the propellant substance. This requires that the different fluids had the correct defined properties for compressible flow. The properties varies with temperature [3]. Therefore, the specific heat capacity needs to be defined for the different fluids as seen in (6). This was done by creating a user function with the correct properties for both fluids.

4.1.4 Solver Settings

The general settings that were used for all cases can be seen in Tab. 4.1 and the more specific settings for the different cases can be seen in the specific sections since these varies.

Tab. 4.1: The settings that was used for the simulations. The table is divided in the same way as the headings in CFX.

<table>
<thead>
<tr>
<th>Domain</th>
<th>Increase Near Small Volumes, Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh Stiffness</td>
<td>Homogeneous Model</td>
</tr>
<tr>
<td>Multiphase</td>
<td>Total Energy</td>
</tr>
<tr>
<td>Heat Transfer</td>
<td>Shear Stress Transport</td>
</tr>
<tr>
<td>Turbulence</td>
<td>1 [atm]</td>
</tr>
<tr>
<td>Reference Pressure</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Rigid Body</th>
<th>X axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Degrees Of Freedom</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Opening Boundary Condition</th>
<th>0 [Pa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Opening Relative Pressure</td>
<td></td>
</tr>
<tr>
<td>Turbulence</td>
<td>Medium (Intensity = 5%)</td>
</tr>
<tr>
<td>Opening Temperature</td>
<td>293[K]</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Wall Boundary Condition</th>
<th>No Slip Wall</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass &amp; Momentum</td>
<td>Smooth Wall</td>
</tr>
<tr>
<td>Wall Roughness</td>
<td></td>
</tr>
<tr>
<td>Heat Transfer</td>
<td>Adiabatic</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Solver Control</th>
<th>High Resolution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Advection Scheme</td>
<td></td>
</tr>
<tr>
<td>Transient Scheme</td>
<td>Second Order Backward Euler</td>
</tr>
<tr>
<td>Turbulence Numeric</td>
<td>First Order</td>
</tr>
<tr>
<td>Numbers of Iterations (Min-Max)</td>
<td>5-30</td>
</tr>
<tr>
<td>Convergence Criteria</td>
<td>$10^{-4}$</td>
</tr>
</tbody>
</table>

The initial conditions that were used for the different parts of the domain can be seen in Tab. 4.2. The numbering refers to the same numbering as in Fig. 4.3. The part of domain 1 that is behind and in front of the projectile can be seen to be part of domain 2 and 3 in terms of initial conditions.
**Tab. 4.2:** The initial conditions that was used on the different domains. The domain numbers refers to Fig. 4.3 numbering.

<table>
<thead>
<tr>
<th>Domain 2 &amp; 5</th>
<th>Velocity</th>
<th>0 [m/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Relative Pressure</td>
<td>Projectile Release Pressure or 0 [Pa]</td>
</tr>
<tr>
<td></td>
<td>Temperature</td>
<td>Propellant Burn Temperature</td>
</tr>
<tr>
<td></td>
<td>Volume Fraction</td>
<td>100% Propellant Gas</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Domain 3 &amp; 4</th>
<th>Velocity</th>
<th>0 [m/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Relative Pressure</td>
<td>0 [Pa]</td>
</tr>
<tr>
<td></td>
<td>Temperature</td>
<td>293 [K]</td>
</tr>
<tr>
<td></td>
<td>Volume Fraction</td>
<td>100% Air Ideal Gas</td>
</tr>
</tbody>
</table>

### 4.2 Mesh

The computational grid was divided into two parts, one dynamic part and one stationary. A schematic picture of the mesh principle is shown in Fig. 4.3. The dynamic part of the mesh is a cylinder with the same diameter as the inner diameter of the launch tube and starts at the rear end of the tube and extends forward through the whole domain, regions 1, 2 and 3 in Fig. 4.3.

As the projectile moves forward the elements in region 2 and 3 are stretched and compressed respectively while the elements around the projectile in region 1 are constant and moves with the projectile throughout the simulation. This approach makes it possible to maintain a good mesh around the projectile and means that no re-meshing is needed as the projectile moves. In the outer part of the domain, region 4 and 5, the mesh is stationary all the time.

On the interface between the stationary and dynamic part of the domain, a General Grid Interface (GGI) was used, dotted lines in Fig. 4.3. In CFX the GGI-connection makes it possible to have non-matching grid on each side of a contact surface and maintain strict conservation for all fluxes of all equations across the interface [3]. The GGI-interface does not require the connected surfaces to be of equal size and different conditions can be applied on the non-overlapping areas. In this case the non-overlapping areas correspond to the inner wall of the launch tube and the envelope surface of the projectile, see Fig. 4.4. On these areas non-slip wall boundary conditions were used.

![Fig. 4.3: The schematic principle of the mesh that was used for the model.](image-url)
The mesh was created by use of Ansys Meshing. In region 2, 3 and 4 in Fig. 4.3 a structured mesh with hexahedron elements was used. In region 5 and around the projectile in region 1, an unstructured tetrahedron mesh was used. The reason for this was to make it possible to easy change the geometry of the projectile to another one without a lot of time-consuming mesh work.

To improve the resolution of the boundary layer around the projectile a number of inflation layers were used. The near-wall flow was handled by use of wall functions and the near-wall mesh was created to receive a \( y^+ \)-value in the range of 30-100. This means that the first near wall cell is located in the log-law region of the boundary layer, which is recommended when wall functions are used to model the near-wall flow [18].

![Fig. 4.4:](image)

**Fig. 4.4:** The non-overlapping areas where wall boundary conditions were applied.

The domain was divided in areas of different mesh refinements. In the rear domain in the venturi the mesh was rather coarse since details in the flow in this area were not of interest. Most of the elements were concentrated around the projectile and the area around the tube muzzle, approximately 40% of the total number of elements. The mesh with the different refinement zones is seen in Fig. 4.5 and in Fig. 4.6 a magnification of the area around the projectile is shown.
In order to evaluate that the mesh was sufficiently fine to not affect the result a mesh sensitivity analysis was performed, see section 4.9.1 Mesh Sensitivity.

4.3 Inner Ballistics

Even though the inner ballistic phase of the launch was not the main focus in this work it needs to be considered. In previous studies of small calibre guns it has been shown that the flow around the muzzle is heavily disturbed before the bullet leaves the barrel [7,10]. It is reasonable to assume that this is the case also for the studied recoilless rifle. The question is therefore where to start the simulation. Since it was unknown how far the projectile can move inside the tube before the surrounding flow field is so affected that it will have an impact on the intermediate ballistic results, it was decided to include the whole inner ballistic trajectory of the projectile.

4.3.1 Motion

The motion of the projectile can be handled in two different ways. If the pressure inside the tube during a launch is known from experiments the force on the projectile can be calculated. By integration of Newton’s second law the velocity can be calculated externally and supplied to the solver as a pre described motion.

The other alternative is to treat the projectile as a rigid body and use the calculated pressure to solve the equations of motion during the simulation. This is the approach that was chosen to focus on. This method means the pressure rise in the tube needs to be implemented in the model in some way, as will be described in the following sections.
4.3.2 Implementation of the Pressure Rise

The projectile is accelerated inside the launch tube by a high pressure created by burning of a propellant. This pressure rise was implemented as a mass source term of the propellant gas phase \( S_{mp} \) in the continuity equation (7). The source term can in CFX be specified either per volume with unit kg/m\(^3\)s or as a total mass source on a domain with unit kg/s. The source term was added to the fluid domain inside the tube behind the projectile. To determine the value of the source term some different approaches were used.

4.3.3 Pressure Curve

For the simplest model of the pressure rise it was assumed that the pressure inside the launch tube was known. The pressure can be measured by drilling small holes and place pressure sensors inside the tube behind the projectile during test firing. A typical pressure curve is shown in Fig. 4.7.

To get the pressure in the model to follow a given curve, the source term \( S_{mp} \) was calculated by use of a simple proportional controller (P only-controller in control theory [19]) according to

\[
S_{mp} = C(p_a - p_d) \tag{17}
\]

Where \( C = 10^5 \, s/m^2 \) is a constant, \( p_a \) is the average pressure in the source domain and \( p_d \) is the desired pressure from the measured curve.

![Measured Pressure Curve](image)

**Fig. 4.7:** A typical pressure curve during launch. The pressure is given from experimental measurements.

The pressure curve was controlled according to equation (17) in two different ways in order to decide which approach gives the best result.

First, the source term was added until the pressure reached the maximum value and after that the source term was set to zero. In this way the pressure drop was not controlled but driven of the rear outflow and the volume increase due to that the projectile moves forward.
It was found that this approach did not correspond well to the given pressure curve since the pressure decreased too fast when the addition of mass was stopped, resulting in too low projectile velocities.

The second approach was to let the source term follow the pressure curve after the maximum value has been reached. In this way the source is used to control the pressure drop when the projectile leaves the tube. This method gave velocities that better correspond to reality which indicates that the propellant continue to burn after the maximum pressure is reached.

When the source term was calculated according to (17) a rather noisy signal of $S_{mp}$ was achieved, see Fig. 4.8. When the calculated pressure inside the tube was studied, pressure oscillations behaving like standing waves were seen. This is a phenomenon that has not been observed in measurements.

To ensure that these oscillations were not a result of the fluctuations in the source term, the source term from (17) was filtered by use of a 4th order polynomial curve fitting. When this filtered source curve was used, no oscillations in the pressure were observed. How the pressure varies along a longitudinal line inside the tube just before the projectile leaves the muzzle for the two types of source curves can be seen in Fig. 4.9.

To avoid that these unphysical fluctuations had an impact on the results, it was decided to use this filtered source curve as a reference model for the following simulations. To test the models for different pressure and velocities, this reference source curve was scaled by multiplying with a factor.

**Fig. 4.8:** The red line show the source term calculated according to (17). The blue line shows the filtered source term, which later was used as a reference.

**Fig. 4.9:** Normalized pressures for the different source curves, where the normalization was done against the minimum value of the pressure. The pressure is monitored on a longitudinal line inside the tube. It can be seen that the unfiltered curve varies, while the filtered curve only increases.

### 4.3.4 Combustion Model

The goal is to be able to make simulations also when pressure curves from real tests are not available. This means that a model of the propellant burning must be implemented.

The mass flux of gas due to the propellant burning can be described by the following empirically developed expression
\[ \dot{m}(t) = bP(t)^N A_{\text{eff}}(t) \rho \] (18)

Where \( b, N \) are constants depending on type of propellant, \( P \) is the pressure, \( A_{\text{eff}} \) is the effective burning area of the propellant and \( \rho \) is the density of the propellant substance.

\( A_{\text{eff}} \) is dependent on how much of the initial amount of propellant has been burnt and decreases with time. The expression for \( A_{\text{eff}} \) was implemented by help of a loop that updates the value in every time step. The total amount of mass added, depends on the initial amount of propellant substance. For the simulations where this combustion model was used the mass was added as a total mass source term.

### 4.4 Projectile Release

The projectile is clamped in the beginning of the firing process; this needed to be considered when the combustion model was used to calculate correct accelerations and velocities. In order to get the projectile clamped some different methods were proposed:

- **Fixed Rigid Body** – This method was done by not allowing the rigid body motion of the projectile until the release pressure was reached. When the release pressure was reached the simulation was stopped and the settings were changed to solve the equations of motion of the projectile in the axial direction. The simulation was then restarted.

- **Big Projectile Mass** – Another method that was evaluated was to increase the mass of the projectile until it reached the specific pressure. The thought behind this method was that if the mass of the projectile were large the pressure would not be able to move the projectile. When the specific pressure is reached the mass is decreased to the real value again. This method was implemented by making the projectile mass used in the equation of motion (16) dependent on the pressure with an if-statement that makes it possible to change the mass without stopping the simulation.

- **Counteracting Force** – As a final method a counteracting force was added to the rigid body solution. This force would hold the projectile in place until it reached the specific pressure. The force was calculated by taking the pressure behind the projectile and multiply it with the area of the projectile, i.e. \( 0.042^2 \pi \, \text{m}^2 \)

### 4.5 Base Plane Breakage

In order to model the base plane two quite different approaches was investigated. The first approach was to use a wall as the plane and remove it at a later stage. The second approach was to use a porous domain.

With the wall approach a wall boundary condition is used as a starting condition at the base plane location, see Fig. 4.10. This means that at the beginning there is no motion in the venturi and no flow across the base plane. When the base plane breakage pressure is reached the simulation is stopped by a stop criterion. The wall boundary condition is removed and the fluid is allowed to flow out through the venturi. When the base plane boundary is removed and the simulation is restarted high velocity and pressure gradients occur. Before the solution has stabilised small time steps are required, \( 10^{-8} \text{s} \) was used. When the flow had stabilised the time step size was ramped up to \( 10^{-5} \text{s} \) within 100 time steps.
The second approach was to change the fluid inside the venturi to a porous domain. This means that the whole venturi is used to represent the base plane. In CFX a type of porous elements can be used to limit the flow.

The idea with this method was to, at the start, set the porous value to zero which would represent that the base plane is intact and there is no flow out through the venturi. Then the value is increased to unity, which would represent that the domain behaves as a fluid domain i.e. the domain is completely open as when the base plane is broken. With this approach, it should be possible to simulate the breaking process of the base plane disk.

### 4.6 Leakage around the Projectile

In order to evaluate how the leak flow through the grooves affects the projectile two different setups were used. First, the leak flow was neglected. Second, the leakage was modelled as a small gap between the projectile and the wall.

#### 4.6.1 Without Leakage

In this model there was no gap between the projectile and the launch tube which can be seen in Fig. 4.11. This model represents that the projectile would seal tight during the launch. In order to get this model to work the sidewall of the projectile was modelled with help of the non-overlapping condition that was set to a no-slip wall. Due to that there is no “real” wall in the geometry the rigid body motion was calculated with the forces acting on the rear and front end of the projectile.

![Fig. 4.11: The computational domain for the geometry where there is no gap between the projectile and tube. Note that there is no connection between the area behind and in front of the projectile when it is inside the tube.](image)
4.6.2 With Leakage

To account for the leak flow through the grooves in the tube a second model with a small gap between the projectile and the tube was created, see Fig. 4.12. The gap between the tube inner surface and the projectile was 0.5 mm, half the depth of the groves. The choice of using half the groove depth will give approximately the same leakage area as in reality. This has been used in previous studies of other types of launching systems and has shown to be a good approximation of the groves [4]. The gap was created by increasing the inner diameter of the tube.

![Fig. 4.12: The projectile inside the launch tube with a small gap around the projectile. The grey part indicates the fluid domain. In this setup the fluid is allowed to pass the projectile inside the tube.](image)

4.7 Friction between the Components

The friction force between the projectile and the tube can be implemented by adding a counteracting force in the equation of motion of the projectile. This friction force can either be chosen as a constant value in order to get correct exit velocity, or a varying force can be implemented. In reality the friction force is a function of temperature and velocity [8] and if the exact force is known from measurements this can be implemented. To simplify the model in this work, the friction force was neglected. Instead the pressure curve was adapted to give reasonable exit velocities. This was considered acceptable since this was not an exact model of a specific case.

4.8 Rotation of the Projectile

The rotation of the projectile can be implemented in two different ways, with help of the rigid body or by specifying a velocity on the projectile wall. The method of using the rigid body will be hard to implement since this requires that the projectile could rotate in the mesh and this has not been implemented in the model.

By using the projectile wall and specify a rotational velocity in the boundary condition settings the rotation can be easier implemented in the model. With this method the rotational velocity will be fixed during the whole simulation. Though it is known that the projectile maintain around 90 percent of its rotational velocity until target impact. Therefore the change in angular momentum in the vicinity of the muzzle that is in focus in this work will be negligible and the simplification is acceptable.

4.9 Mesh Sensitivity & Domain Size Analysis

Since the main purpose with this work was to develop a method to analyse the fluid dynamics around the projectile in the intermediate ballistic phase, and only a generic projectile geometry was studied, a meticulous mesh independency study was not done.
However, even if the exact numbers were not important, it needed to be ensured that the mesh and domain size were sufficient so conclusions about the principal method could be drawn. Therefore a somewhat simplified mesh sensitivity analysis was performed. Since the study was done at an early stage of the project the geometry of the computational domain was slightly modified to the final model.

### 4.9.1 Mesh Sensitivity

To evaluate how fine the mesh needed to be a 30-degree wedge piece of the geometry was studied. The mesh independency study was done early in the project. At this stage some functions that later on was implemented in the model were not present. At the rear part the venturi was excluded and only a single phase model with air as fluid was used.

Results from an initial mesh consisting of approximately 1.3 million elements were compared against results from meshes with 1.6 million and 2 million elements. The refinements were done rather uniformly over the domain with a slightly focus on the areas of interest, i.e. around the projectile and the muzzle, especially for the 2 million mesh most focus was put on the area around the projectile.

To evaluate the results, pressure and velocity profiles along a number of lines were compared at two different times, 5 and 7 ms from start, corresponding to just before the projectile leaves the tube and when it have travelled a short distance from the muzzle, see Fig. 4.13. Also the resulting force working on the projectile during the launch was studied and can be seen in Fig. 4.17.

![Fig. 4.13: The placement of the lines that were studied in order to evaluate mesh independence and the projectiles position at the different times that was studied.](image)

On line 1 and 2 no difference in the results between the meshes could be observed. In the area in front of the muzzle, small differences in the velocity field could be observed between the 1.3 million mesh and the other two. The largest deviation was found behind the projectile after it had travelled some distance from the muzzle, see Fig. 4.15. In front of the projectile only a small deviation in velocity could be observed as seen in Fig. 4.14.

If the driving pressure inside the tube is studied no differences between the meshes can be noticed, Fig. 4.16 shows the average pressure in the domain behind the projectile during the launch process. This means that the area behind the launch tube is sufficiently resolved to model the rear outflow.

Also, if the force acting on the projectile was considered all meshes gave similar results as seen in Fig. 4.17. Worth to notice is that the force does not decrease as smooth as the pressure drops, which means that the pressure are not constant in the whole tube after the maximum
pressure has been reached. However, all meshes capture the same variations. This variation in the force is probably a result of the source term implementation, as described in section 4.3.3.

![Fig. 4.14: The velocity profile for line 4 at the time 5 ms for the three different meshes.](image1)

![Fig. 4.15: The velocity profile for line 3 at the time 7 ms for the three different meshes.](image2)

![Fig. 4.16: The average pressure in the volume behind the projectile during the launch.](image3)

![Fig. 4.17: The force on the projectile for three different meshes.](image4)

**4.9.2 Domain Size**

In order to get accurate results the computational domain needs to be sufficiently large. If the computational domain is too small the boundary conditions will affect the results and if the domain is too big the computational power needed will be unnecessarily large. Both these problems needed to be investigated so that the computational domain has the right size. This was done by studying the flow in the domain and the main concern for the flow is the diameter of the domain. The length of the domain was analysed by studying the flow on a cross sectional plane. The velocity at the end of the domain is equal to zero when the projectile reaches the maximum distance it will travel, see Fig. 4.18. It has also been shown in previous studies that the most important parts happens behind the projectile and not in front of it [10] therefore this domain size was seen as sufficient.
In order to evaluate how big the diameter of the domain needs to be velocity and pressure were studied along the lines in Fig. 4.13. The results are seen in Fig. 4.19 and Fig. 4.20. The values were normalized against the maximum absolute value of each line in order to display all lines in the same graph. It can be seen that most of the lines have reached the zero value before the end of the domain. It can be seen that the velocity on line 1 and 2 does not reach zero but are forced down by the boundary condition.

The reason for the higher velocities on these lines is the rear outflow that was included in the model used for the domain size study. In the geometry used later on the rear outflow were not allowed to interfere with the flow in the area of interest. Since the velocities on the other lines does not seem to be forced down to zero in the same way the domain diameter was considered sufficient.

With help of this study the domain size was specified to, radius of 800 mm and a length after the pipe of 1650 mm.

### 4.10 Evaluated Models

In the previous sections methods to simulate the different parts of the launch has been described. The different methods that have been evaluated can be seen in Fig. 4.21. The colour represents how the different methods will be further investigated. Green represents parts that will be investigated further in the result section and have known input parameters.
Yellow represents models that works in principle but have some uncertainties in the input parameters and are not deeper analysed. Red represents models that did not give a satisfying outcome or was not considered as good choices and will therefore not be further analysed.

![Diagram showing the different methods analysed](image)

**Fig. 4.21**: The different methods that were analysed in order to get a working model that gives a good and realistic representation of the weapon. The green represents methods that will be further analysed in the result section. Yellow is methods that works but needs to be improved in regards to the given input. Red is parts that were not any further analysed.

In order to evaluate the different methods a number of test cases were defined. The models did not have any friction or rotation on the projectile. The filtered source curve that was used can be seen in Fig. 4.8 and is used as a reference curve.

In order to decide how leakage affects the flow the following simulations were performed.

- No leakage where the analysis was initialized from the release pressure with the reference source curve \(v_0=280 \text{ m/s}\).
- Leakage where the analysis was initialized from the release pressure with the reference source curve \(v_0=280 \text{ m/s}\).

To see how the model responds to different pressures two additional simulations on the model without leakage were made. For these two simulations the reference source curve was scaled to 80 and 90 percent.

- No leakage where the analysis was initialized from the release pressure with the 0.9* reference source curve \(v_0=250 \text{ m/s}\).
- No leakage where the analysis was initialized from the release pressure with the 0.8* reference source curve \(v_0=225 \text{ m/s}\).

And in order to see how well the combustion model works the following simulation was done.

- No leakage where the combustion model was used with the base plane and projectile release implemented by stopping the simulations and change settings.

In total, five different models were studied in order to give an understanding of the flow and how well the models work.
4.11 Studied Entities

To evaluate how the model corresponds to reality some different characteristics of the flow were studied and evaluated. This was done by comparing some different entities for example velocity, volume fraction, pressure and forces with observations from test firings.

4.11.1 Velocity

The velocity of the flow leaving the barrel is hard to analyze in the experimental setup since there is extreme conditions and therefore experiment values should be taken with some doubt. But the velocity of the flow can be approximated by calculating the distance the smoke travels between two frames in a slow motion film from a test firing; this gives a rough estimation of the velocity. The projectile velocity $v_0$ when it leaves the tube is better known and can be precisely measured.

4.11.2 Volume Fraction

The volume fraction was compared against experimental data of when the projectile leaves the barrel. This was done by comparing how the cloud of smoke looks at different times for the experimental and numerical solution. The experimental values were given from a test firing where the area around the muzzle was recorded with a high-speed camera.

4.11.3 Pressure

The pressure was monitored in order to see that the pressure curve corresponded well to the experimental curve in order to get a good velocity. The pressure outside the launch tube was also monitored by studying contour plots on planes in the domain, and compared between different models.

4.11.4 Force

In order to evaluate if the projectile has reached a free stream condition the aerodynamic force acting on the projectile was monitored. When the force level out and stabilises around a constant value the projectile can be considered to be in free stream condition and the length of the intermediate ballistic phase can be estimated.

To validate the model the calculated free stream force was compared against a theoretical calculated drag force, according to

$$F_D = \frac{1}{2} \rho v^2 C_D A$$  \hspace{1cm} (19)

$F_D$ is the drag force, $\rho$ is the density of the air, $v$ is the speed of the projectile relative the air, $C_D$ is the drag coefficient and $A$ is the cross section area of the projectile. A typical projectile has a drag coefficient $C_D = 0.3$ [4,20,21].
5. Results & Discussion

In the following sections the results from the studied cases, described in section 4.10, will be presented and discussed. Also the methods to implement the different parts of the launch process, seen in Fig. 4.21, will be discussed in sense of how they work or why they did not give satisfying outcome.

5.1 Reference Position

In order to have a reference point the position \( x \) of the projectile at the instant when it leaves the tube was defined as the zero position \( x=0 \) m, see Fig. 5.1. A positive distance will correspond to a position outside the tube and a negative distance is inside the tube. Also when time is used this projectile position will be referred to as time \( t=0 \) s.

![Fig. 5.1: The predefined zero position for the projectile.][1]

5.2 Projectile Velocity

When the projectile velocity is displayed as function of the distance the projectile has traveled, it can be seen that the velocity has almost reached the maximum value when the projectile is exiting the tube, see Fig. 5.2. When the projectile is exiting the tube the velocity is 99.5 percent of the maximum value. The maximum velocity is reached approximately 100 mm from the muzzle as seen in Fig. 5.3.

An increase in projectile velocity after the muzzle exit is also found by [2]. They study a number of different launch systems with different calibres and found the velocity increase in general to be less than one percent, which is in agreement with the result in this work. They also conclude that the velocity increase in most cases can be neglected.

It can also be seen that there is a slight difference in the maximum velocity for the model with and without leakage. The maximum velocity for the leakage model is 279.8 m/s and 277.9 m/s for the non-leakage model.
5.3 Leak Effects on the Flow

To evaluate the differences between the model with and without leakage the flow field around the muzzle was analysed and compared to pictures from a test firing. The area around the muzzle at six different instants of the launch, both from the simulations and snap shots from an experimental trial recorded with a high-speed camera can be seen in Fig. 5.4-Fig 5.9. In the experimental pictures red lines are drawn to highlight the behaviour of the outflowing smoke that is observed when the slow motion film is studied.

The upper part of the simulation pictures show a contour of the velocity magnitude in the range 0-600 m/s, the lower part shows volume fraction of propellant gas from 0-100% where red indicate propellant gas and blue is air.

The figures are divided into a, b and c where a is the experimental result, b is the simulation without leak and c is the simulation with leak. The projectile position that is given for the experimental pictures are estimated by assuming that the projectile has a constant velocity equal to $v_0$ and counting backward from the reference position i.e. this is a rough estimation of the position. The exit velocity $v_0$ in the test firing was estimated to be 300 m/s.

The experimental picture in Fig. 5.4a is the first frame from the high speed camera where any propellant gas can be observed. The time and distance is approximated to be -2 ms and -600 mm. It can be seen that the flow change direction right at the muzzle to an almost 45 degree outward direction. By measuring the distance the smoke travels in one picture frame it can be concluded that the velocity of the muzzle outflow is at least 400 m/s.

The results from the model with leakage were found qualitatively agreeing with the experimental pictures. The picture from the model with leakage in Fig. 5.4c shows a similar behaviour as in the experiment at time t=-2.49 ms and distance x=-593 mm. However, for the model without leakage in Fig. 5.4b there is no velocity at the muzzle at the same time. Obviously no volume fraction of propellant gas will be seen in front of the projectile before it has left the tube in the model without leakage.
For the experimental flow at time $t=-1.25\text{ ms}$ the projectile position is approximately $x=-375\text{ mm}$. The flow still change direction at the muzzle and a vortex is created, see Fig. 5.5a. The same behaviour is found in the leakage model at $t=-1.94\text{ ms}$ and $x=-505\text{ mm}$, as can be seen in Fig. 5.5c. For the model without leak a small velocity increase is seen inside the tube but the area outside the tube is still unaffected, see Fig. 5.5b.
Fig. 5.4: a) Experimental picture at t=-2 ms, x=-600 mm b) Model without leakage at t=-2.49 ms, x=-593 mm c) Model with leakage at t=-2.49 ms, x=-593 mm. For the simulated results, the upper part displays velocity and the lower part shows volume fraction of propellant gas.

Fig. 5.5: a) Experimental picture at t=-1.25 ms, x=-375 mm b) Model without leakage at t=-1.94 ms, x=-505 mm c) Model with leakage at t=-1.94 ms, x=-505 mm. For the simulated results, the upper part displays velocity and the lower part shows volume fraction of propellant gas.
For the experimental flow at $t=-0.75$ ms and $x=-225$ mm, the outflow stream has narrowed and the flow follows the direction of the tube before it turns around into a vortex as seen in Fig. 5.6a. The same behaviour is found in the leakage model at $t=-1.44$ ms and $x=-389$ mm, see Fig. 5.6c. In the model without leakage the velocity in the tube has only reached 200 m/s and the flow outside the tube is only lightly disturbed as seen in Fig. 5.6b.

Fig. 5.7a show the instant immediately before the projectile leaves the tube. Also here the flow follows the direction of the tube before it turns into a vortex. The leakage model shows a similar behaviour but the vortex core seems to have travelled slightly further away from the muzzle than in the experimental picture, see Fig. 5.7c. The model without leakage show a velocity field that have a similar shape as the cloud of smoke in the experiment but the vortex is much smaller as seen in Fig. 5.7b.
**Fig. 5.6:** a) Experimental picture at $t=-0.75$ ms, $x=-225$ mm  
   b) Model without leakage at $t=-1.44$ ms, $x=-389$ mm  
   c) Model with leakage at $t=-1.44$ ms, $x=-389$ mm. For the simulated results, the upper part displays velocity and the lower part shows volume fraction of propellant gas.

**Fig. 5.7:** a) Experimental picture at $t=-0.5$ ms, $x=-150$ mm  
   b) Model without leakage at $t=-0.24$ ms, $x=-69$ mm  
   c) Model with leakage at $t=-0.24$ ms, $x=-69$ mm. For the simulated results, the upper part displays velocity and the lower part shows volume fraction of propellant gas.
Fig. 5.8a show the projectile immediately after it has left the tube. Hot propellant gas flow straight out from the muzzle at an angle of approximately 45 degrees. This is also the case for the leakage model as seen in Fig. 5.8c. Also the model without leakage show a similar behaviour but the outflowing gas has formed a small vortex, see Fig. 5.8b.

In Fig. 5.9a the projectile has travelled a short distance (approximately 75 mm), the outflowing propellant gas has now turned into a vortex behind the projectile. This is also the case in the simulations, see Fig. 5.9b and Fig. 5.9c. At this instant both the model with leakage and without gives similar results of the flow field around the muzzle and projectile.
Fig. 5.8: a) Experimental picture at t=0 ms, x=0 mm b) Model without leakage at t=0.16 ms, x=42 mm c) Model with leakage at t=0.16 ms, x=42 mm. For the simulated results, the upper part displays velocity and the lower part shows volume fraction of propellant gas.

Fig. 5.9: a) Experimental picture at t=0.25 ms, x=75 mm b) Model without leakage at t=0.91 ms, x=252 mm c) Model with leakage at t=0.91 ms, x=252 mm. For the simulated results, the upper part displays velocity and the lower part shows volume fraction of propellant gas.

As shown above, there are large differences in the flow field in front of the projectile between the model with and without leakage. This can also be seen if the pressure is studied. Fig. 5.10 and Fig. 5.11 show the pressure and velocity field when the projectile is still inside the tube. It
can be seen that the velocity inside the tube is much higher when the leakage is included in the model. This high velocity in front of the projectile results in a pressure wave around the muzzle. When this pressure wave leaves the muzzle a low pressure is created in front of the projectile inside the tube. Since the fluid velocity is supersonic inside the tube in the leak model a shockwave is created at the muzzle seen as a sharp limit between high pressure and low pressure in Fig. 5.11. As seen in Fig. 5.10 the fluid velocity in front of the projectile is much lower for the model without leakage (subsonic) and no low-pressure zone is created inside the tube. The pressure wave created around the muzzle is also much weaker.

The low pressure in front of the projectile in the leak model can explain why the exit velocity of the projectile is slightly higher for this model than for the model without leakage as shown in section 5.2.

![Fig. 5.10: Pressure and velocity for the model without leakage. Time t=-1.2 ms, distance x=-325 mm.](image1)

![Fig. 5.11: Pressure and velocity for the model with leakage. Time t=-1.2 ms, distance x=-325 mm.](image2)

In Fig. 5.12 and Fig. 5.13 the pressure and velocity inside the tube are more closely seen (note the different velocity scales). In Fig. 5.12 it can be seen that in the model without leakage, the fluid velocity in front of the projectile does not reach higher values than the velocity of the projectile and the pressure wave created in front of the projectile has a uniform front.

The leak flow results in much higher fluid velocity inside the tube, up to 3000 m/s and a shockwave pattern is created that result in the low pressure in front of the projectile, see Fig. 5.13.
Fig. 5.12. The upper part of the figure shows the pressure inside the tube for the model without leakage. The pressure scale is the same as in Fig. 5.10. The lower part shows velocity magnitude. Time t=-1.2 ms, distance x=-325 mm.

Fig. 5.13. The upper part of the figure shows the pressure inside the tube for the model with leakage. The pressure scale is the same as in Fig. 5.10. The lower part shows velocity magnitude. Note the shock waves inside the tube which can be seen as sudden pressure differences. Time t=-1.2 ms, distance x=-325 mm.

In general it has been shown in the comparison between the model with and without leakage that there are large differences in the flow behavior in front of the projectile. The model with leakage gives results that correspond qualitatively with physical tests. However, in the simulations the flow phenomenon that was observed around the muzzle occurs when the projectile is further inside the tube compared to the experimental case. The reason for this can be differences in projectile velocity. Another reason can be that the modelling approach used for the leak flow gives too high gas velocity in the tube. This will result in that the flow reach the end of the tube faster and the observed phenomenon outside the muzzle occur at an earlier stage.

From the results in Fig. 5.4-Fig. 5.9 it can be concluded that the leakage has a large effect on the flow in front of the projectile and if a precise model is to be created the leak flow probably must be included. However, the possibility that the leak only occur during the short distance before the projectile enters the grooved section of the tube has not been modelled. The fact that the flow phenomenon observed around the muzzle occurs at an earlier stage of the launch in the model than in reality indicates that the leak flow is too large in the model compared to physical experiment. It is possible that a model where the leakage is limited to only a short part of the tube would give results closer to reality.

It can also be argued that the gap of half the grooves depth used in the model is too large. This corresponds to the leak flow area shown in Fig. 2.5. In reality though, at least parts of the grooves are filled with the slip ring. Therefore a narrower gap would probably be closer to reality and result in a lower leakage then in the current model. However, a problem with reducing the gap is that it will be hard to maintain a good quality mesh in the leakage area.
The leak flow through the grooves is a subject that needs more research to validate in which way the precursor flow in the tube is best modelled. The differences between the models, especially in the pressure field in the tube and around the muzzle are quite large. It should therefore be possible to measure the pressure at some locations during a test firing to decide which model that is closest to reality.

If the pressure waves created at the muzzle could be modelled with high accuracy another possible usage of the model could be to simulate the sound pressure that the operator is exposed to when firing the rifle.

### 5.4 Force on the Projectile

One of the main purposes of the model is to analyse if the outflowing propellant gas can disturb the projectile and make it unstable. The fluid dynamic forces acting on the projectile were therefore studied.

With the exit velocity of $v_0 = 280 \text{ m/s}$ and assuming that $c_d = 0.3$ for the projectile, the free stream drag force can be estimated according to (19). This gives a force of $65 \text{ N}$ and the simulated force on the projectile can be expected to level out around this value.

Fig. 5.14 shows the force in the x-direction during the whole launch. It can be seen that the force increases fast at the beginning when the pressure is increased in the tube and when the pressure is decreased the force is also decreased. This is a correct behaviour since the force is a function of pressure. A sudden decrease in the force can be seen when the projectile leaves the tube. It can also be noticed that there are no difference in behaviour between the model with and without leakage.

In Fig. 5.15 the force in the x-direction when the projectile has left the tube is seen. In order to see the variations the y-axis has been limited to ±500 N. When the projectile leaves the tube the force drops rapidly to approximately -200 N, thereafter it is increased again up to around 100 N before it drops again and level out at a negative value. It can be noted that the force drops below zero when the projectile has travelled a distance of 100 mm from the muzzle. This means that the projectile continues to accelerate until this distance, i.e. as long as a positive force acting in the x-direction. This confirms what was previous seen for the velocity, which reaches its maximum value at $x=100 \text{ mm}$, see section 5.2.

When studying Fig. 5.15 it can be seen that the force oscillates and never level out at a constant value. The leakage model gives a more unstable force than the non-leakage model. The reason for these oscillations is probably that the boundary conditions start to affect the flow in the domain. When the pressure waves formed around the muzzle reach the domain boundaries they are reflected and give rise to pressure variations in the domain that are not physically valid. The boundary reflections in the leakage model can be seen in Fig. 5.16 and Fig. 5.17. In Fig. 5.16 the precursor high pressure wave is seen just before it reaches the right boundary of the domain. When it reaches the boundary it is reflected and turned into a low pressure wave that propagates inward in the opposite direction, see Fig. 5.17.

It can however, be seen that the force in both models have a tendency to level out and oscillates around approximately -100 N when $x=1000 \text{ mm}$, which is in the same order of magnitude as the theoretical calculated drag force of -65 N.

The reasons for the larger oscillations in the model with leakage is most likely that the pressure wave formed in front of the projectile is much stronger and the reflected pressure waves have a larger impact on the flow field in this model. It can thus be concluded that a
much larger domain where the pressure waves are not allowed to reach the boundaries are needed to get accurate predictions of the projectile force.

**Fig. 5.14:** The force acting on the projectile during the projectile motion. The different lines indicate with and without leakage.

**Fig. 5.15:** The force on the projectile when it has left the launch tube and is traveling in the air. The different lines indicate with and without leakage.

**Fig. 5.16:** The pressure field for the leakage model close to the right end boundary. The precursor high pressure wave is seen just before it reaches the right end of the domain. Time t=0.95 ms, distance x=266 mm.

**Fig. 5.17:** The pressure field for the leakage model close to the right end boundary. Note how the high pressure wave seen in Fig. 5.16 is reflected as a low pressure wave propagating in the left direction. Time t=1.75 ms, distance x=490 mm.

From a projectile stability point of view it is the radial forces that are most interesting. A disadvantage of the wedge piece model used in this project is that correct values of the radial forces cannot be achieved. For this axi-symmetric case the radial forces should sum up to zero, this is not the case for the wedge piece. The force in the z-direction however, should sum up to zero since the model was generated symmetrically around the xy-plane. As can be seen in Fig. 5.19 the forces in z-direction are close to zero which validate that the models work correctly.

As seen in Fig. 5.18 the radial force in the y-direction is large at the instant when the projectile leaves the tube but decreases quickly and levels out around zero. It can be seen that the first 200 mm in the positive x-direction are the most crucial and if the outflow of propellant gas is not perfectly symmetric it most likely can have an impact on the projectile stability. At approximately 500 mm the y-force has stabilised for the model with no leakage. In the model with leakage the force oscillates slightly around zero which can be a result of boundary interference. Another reason for the less oscillating force in the model without leakage can be that the envelope surface of the projectile is not included as described in
section 4.6. When the total projectile surface area is smaller the model does not become as sensitive to variations in the pressure.

Fig. 5.18: The force in the y-direction that acts on the projectile when it has left the launch tube with and without leakage.

Fig. 5.19: The force in the z-direction that acts on the projectile during the launch for leak and no leak. The disturbance in the leakage model curve does probably originate from numerical issues or convergence problems.

In order to understand why the forces behaves as shown in Fig. 5.14-Fig. 5.19, the pressure and velocity fields around the muzzle and projectile was studied.

Fig. 5.20 and Fig. 5.21 show the projectile at x=200 mm which is where the largest force in the negative x-direction is found for both the model with and without leakage. Here it can be seen that the pressure wave created when the projectile leave the tube are on its way to pass the projectile.

The pressure wave results in a high pressure area in front of the projectile which gives a contribution to the negative force. At the same time a low pressure wake is created behind the projectile when the propellant gas is accelerated when it leaves the muzzle. Also this low pressure wake gives a negative contribution to the projectile force.

When the high pressure wave has passed the projectile a low pressure wake is formed around the front of the projectile while a high pressure area is formed around the rear end, see Fig. 5.22 and Fig. 5.23. This results in the positive force on the projectile seen in Fig. 5.15 at x=500 mm.

It can also be seen that the pressure and velocity around the projectile when it has left the tube looks rather similar for both the model with and without leakage. Worth to notice is that the pressure wave is sharper and stronger for the model without leakage, compare Fig. 5.20 and Fig. 5.21. The reason for this is probably that the outflow of propellant gas starts suddenly when the projectile leaves the tube for the non-leakage model. In the leakage model there is instead, a continuous outflow that is gradually increased when the projectile exits the muzzle. This results in a not as sharp pressure wave.
The general behaviour of the flow and the pressure wave propagation found in this work is similar to findings in previous studies of small calibre bullets [7,10]. A main difference is that a supersonic bullet will catch up and pass the precursor pressure wave while a subsonic projectile, as in this study, instead get passed by the pressure wave created behind the projectile. This means that the outflow of propellant gas, when the projectile have exit the tube probably have a larger impact on the stability for a recoilless rifle with subsonic projectile velocity than for a small calibre gun.

From the study of the projectile forces it can be concluded that it is the outflow of propellant gas when the projectile has left the tube that has the largest impact on the results. Both the model with and without the leakage included gives similar results and characteristics of the forces. It cannot from this study be excluded that the differences in the calculated forces between the models origins from the model setup instead of physical differences.

Especially in the model with leakage it has been seen that the strong pressure wave created in front of the projectile is reflected on the boundaries and therefore can have an impact on the
results. It is therefore necessary to conduct a more meticulous study of the mesh, time step and domain size to consider the results more trustworthy.

### 5.5 Combustion Model

In order to evaluate the combustion model where the source term was calculated according to (18), the pressure curve from the simulation was compared to a real measured curve. The comparison is seen in Fig. 5.24.

It can be seen that the curve from the combustion model show the same characteristics as the real curve. The reason for the simulated curve does not follow the real curve exactly is because that the constants in (18) do not correspond to the propellant in the test firing.

By choosing parameters that corresponds to the propellant used in reality it should be possible to receive a pressure curve that closely matches the real curve. It also needs to be more closely investigated what base plane breakage pressure to use in the model. Since the aim with this project was not to develop a precise model of the inner ballistic phase though, no further effort was put into the work to refine the combustion model. It can however, be concluded that this method to implement the pressure increase works in principle.

The use of the combustion model would be useful particular in an early stage in the development process when no real measurements are available. It would therefore be interesting to continue the work with refining this model.

![Measured Pressure Curve vs Propellant Gas Model](image)

**Fig. 5.24:** The different pressure curves where the pressure curve that was given from the propellant gas model is compared against the measured curve.

### 5.6 Different Source Curves

In order to see how the model responds to a change in input parameters two additional simulations on the model without leakage was performed where the reference source curve was scaled to 80 and 90 percent.
Fig. 5.25 shows the calculated projectile velocity for the different source curves. A decrease in the source term also gives a decrease in velocity as expected. The results indicate that there is a linear relationship between the input and the output. In these simulations, it was seen that a decrease in the source curve of ten percent results in a decrease in exit velocity of ten percent, see Fig. 5.25.

![Fig. 5.25: The projectile velocity with different source curves. It can be seen that a source decrease of ten percent leads to a ten percent smaller velocity.](image)

Also when the force on the projectile is studied, the model responds to a change in the source curve in a consequent manner as expected. In Fig. 5.26 it is seen that the maximum force on the projectile inside the tube decreases when the source is decreased. In Fig. 5.27 it can be noted that a lower exit velocity results in a larger maximum negative force when the projectile has left the tube, and that the maximum negative force is reached at a position closer to the muzzle. This can be explained by that the lower exit velocity means that the projectile will be overtaken by the pressure wave at an earlier stage.

![Fig. 5.26: The force on the projectile with different source curves. It can be seen that the force decreases with decreased source curve.](image)

![Fig. 5.27: The force on the projectile when the projectile has left the tube with different source curves.](image)

The purpose of the study with different source curves was to see that the model corresponds in a consequent manner to a change in input parameters. The results show that the model works properly and could be used to analyse trends in projectile behaviours depending on different inputs.
5.7 Domain and Mesh

It has been seen in the results that the pressure waves created at the muzzle reach the boundaries of the domain where they are reflected and therefore have an impact on the flow field later on. This means that the domain used for the simulations are too small. Especially in the leakage model where the precursor pressure wave created in front of the projectile is much stronger it can be seen that these boundary reflections interfere with the subsequent flow field in the domain.

The domain used in this study extends 20 times the projectile diameter in front of the muzzle and have a diameter of 20 times the projectile diameter. It was seen when studying the model with leakage that the domain was too small at the end since the flow is reflected on the boundary. There is also a big difference if previous works are studied where for instance [10] uses a domain of 40 times the projectile diameter. However they also have projectile velocities three times higher than used in this work.

The reason that not a larger domain was used was because of limitations in computational power. Also since the aim with this study was to develop a modelling approach rather than calculate exact values, the increased simulation times for a larger domain was not motivated. The domain size study that was performed was also done at an early stage in the project on a model without leakage and indicated that the domain was sufficiently large. The fact that the pressure and velocities were unaffected close to the boundaries may not be sufficient to conclude that the domain is large enough though. It can also be the case that the boundary conditions themselves force the properties of the flow to approach the boundary values.

The model has been found to be quite sensitive to changes in the mesh and also to time step size in order to become stable. Especially the flow just at the muzzle has been found to be troublesome to resolve. In general the model without leakage has shown to be more stable and less time step size sensitive. The leakage results in much higher velocities in front of the projectile. When this leak flow reach the muzzle, high gradients and small vortices are created around the sharp edges of the muzzle. This means that smaller time steps and finer mesh are required for the leakage model.

Fig. 5.28 show the velocity field around the muzzle when the projectile is about to leave the tube. It can be seen that the high velocity from the leak flow results in a number of small vortices. The vortices varies transient in size and position. These small vortices did not appear in the non-leakage model and can be the reason why that model is more robust and easier to converge.

![Image](image.png)

**Fig. 5.28:** The velocity field at the muzzle at the moment when the projectile leaves the tube. The high velocity from the leak flow results in several small vortices seen in the vector plots.
The sharp edges at the muzzle may also contribute to locally very high gradients. From a convergence point of view it would probably be better to round off the corners and make the geometry smoother. This will however make it harder to use a structured grid in this part of the domain.

Another problem with the use of a structured grid is that the mesh becomes unnecessary fine in areas that are not important to resolve. For a future model, if computational power is not limited, it could be better to use an unstructured tetrahedron mesh in the stationary part of the domain. This will make it easier to refine the mesh only in the areas where it is needed.

Regarding the time step size it was hard to find a proper value that works for the whole simulation. During large parts of the simulation time steps of 0.01 ms were used. However very small time steps results in an extremely small opening at the moment when the projectile leaves the tube. This results in locally very high velocities and Courant numbers that several times caused the simulation to crash. The solution was to use larger time steps just when the projectile leaves the tube. It was also tested to use automatically adapted time steps which is a possibility in CFX. Generally this resulted in even smaller time steps and impractical simulation times. The simulations with manually adapted time steps took in average around 40 hours to complete.

**5.8 Models**

The model discussion will be divided into the different parts of the launch with the same headers as in Fig. 4.21. This section is used to discuss how the models work and why some models are not further analysed.

**5.8.1 Pressure Rise**

One of the most important part of the model is the pressure increase implementation. Two methods with different degrees of complexity and requirements of input values have been evaluated.

The model that uses a measured pressure curve is simplest to implement. Since the projectile release and the base plane breakage are included in the real pressure curve, the complexity of the model is reduced. But it was also found that oscillations in the pressure inside the tube were induced when the source term was calculated according to (17), since the source curve becomes noisy. The reason for this noise is that the magnitude of the source was calculated proportionally to the error between the actual and the desired pressure, known as a proportional or P only-controller. A possibility is to implement the source as a more advanced control system. For instance, a PID-controller where also the previous values and the derivative of the pressure curve are accounted for could give a more accurate response.

It was not further investigated if these oscillations in the pressure have an impact on the flow field during the transition from inner to outer ballistics. But to ensure that the oscillations did not affect the result it was decided to use a filtered source curve. This however, means that two simulations were needed, one with the P only-controller and one with the filtered source curve. This is not an optimal solution and increases the computational cost.

For the future, it may be more preferable to use the combustion model. This method is shown to give correct characteristics of the pressure increase but needs more refining. The main challenge with this model is that it is affected by the breakage of the base plane which needs to be implemented in a proper way. This adds more complexity to the model and gives another source for errors. It should not be that hard to tune this model to work properly but
since the inner ballistic phase was not the main focus in this project no further effort was put into this.

It can be said that both these models work well and give proper results so therefore the model used depends on the situation. If the pressure curve is known, the simpler model might be preferable. But if the pressure curve is unknown the combustion model is the only possibility.

5.8.2 Projectile Release

The projectile release is quite straightforward to implement in the model, since the projectile can be seen as fixed or not fixed. In reality there is a short transition phase, but this phase occurs during such a short time that there is no needed to implement it. The method that is recommended, and used, is to fix the rigid body in all directions at the start and when the release force has been reached the projectile is allowed to move in the x-direction and the equations of motion are solved. If a known pressure curve is used the simulation can be started from the release pressure which is the easiest way.

To model the release by increasing the mass of the projectile was not any further investigated. The method of using a counteracting force on the projectile in order to prevent it to move was unnecessary complicated. Due to numerical errors, the counteracting force did not become the same as the rigid body force and therefore the projectile had a small velocity when it should be clamped.

5.8.3 Base Plane

The modelling of the base plane breakage have not been deeply analysed in this project. The base plane is only of interest if the combustion model is to be used. It was found to have a large impact on the pressure inside the tube. In the combustion model simulation that was performed, the base plane was modelled as a wall boundary that was removed at a certain pressure. What pressure to use as the break pressure needs to be further investigated in order to get a good agreement with reality.

Since the base plane breakage in reality is a transient phenomenon another modelling approach would be preferable. The method of using a porous media can be an alternative, but this method was not investigated in this project due to limitations in time.

If a known pressure curve is used, the base plane can be ignored since this curve already contains information about the base plane and how it breaks. This makes the model with a pressure curve more attractive. Generally, it can be said that the base plane breakage is a large topic in itself and more work is needed in order to model it precisely.

5.8.4 Leakage

When studying the comparison between the model with and without leakage it is obvious that the leakage has a large impact on the flow field in front of the projectile. The uniform gap around the projectile is not a perfect representation of the grooves. However, making a precise geometrical model of the rifling is not justifiable since a huge amount of small elements would be needed in the mesh and the model would become extremely computationally demanding.
When comparing the results from the two models with experimental photos, the leakage model shows better agreement with the experiments. It can therefore be concluded that there is a certain leakage in reality. However, it is impossible to say how well the simulated concentration of propellant gas corresponds to reality, since it is unknown how much propellant gas that is needed to be visually observed. To make a more precise evaluation the concentration of propellant gas in the air around the muzzle must be measured in some way. The choice of using a gap with the thickness of half the depth of the grooves gives an overestimation of the leakage. This gap thickness gives approximately the same flow area as if no parts of the grooves are filled by the slip ring.

The two models can be said to represent the following two extreme cases

- The grooves are completely sealed by the slipping ring
- No part of the grooves are sealed

The reality is most likely somewhere in between, and what thickness of the gap that gives a good representation of reality needs to be further investigated. This may also vary for different types of projectiles and depending on the material used in the slipping ring.

It might also be that the reality is a mix of these two models, where there is a leakage at the beginning before the projectile enters the grooved section and then the gap is sealed.

### 5.8.5 Friction

The friction force was not any further investigated in this work but as described in the method section 4.7, the force can be implemented as an external force on the rigid body. This force needs to be investigated further in order to get a good agreement with reality. For a future model and if the friction force is known it could though be easily implemented.

### 5.8.6 Rotation

In order to include the rotation of the projectile in the model some changes needs to be made which would make the model more computation heavy. If the rotation should be implemented as a specified tangential velocity on the projectile wall, the symmetry condition would need to be changed to a periodic boundary condition. This was found to put larger demands on the solver memory and made the simulation more computationally heavy. Alternatively, the model would need to be created as a 360-degree full 3D model. This is also valid if the rotation should be handled by solving the equations of motion also for the rotation which would lead to a 2-DOF model.
6. Conclusions

It can be concluded that the task of creating a model that can simulate the launch of a projectile is an extensive subject and contains many different disciplines. It has been shown that it is possible to create a model, which corresponds well to reality and gives a result in the right order. The models tested in this work have shown results that correspond well to what could be observed in reality and also to findings in previous studies in open literature. Since only a generic type of projectile has been studied the results have not been validated against real tests in terms of exact numbers. But the same type of behaviour is found in the model as in experiments. By implementing more physics and more precise models of different parts of the launch the simulations could be further improved.

The method of using a dynamic mesh where the elements are stretched and compressed to handle the motion of the projectile works well. From a computational point of view, this is a rather effective method since the numbers of mesh elements are constant throughout the simulation. However, the deformation results in highly skewed elements which can be problematic in the areas where the largest part of the deformation takes place. To improve the results it could be preferable to combine this dynamic mesh with a criterion that automatically stop the simulation and regenerate a new mesh when the deformation becomes too large.

Regarding the leakage effects on the flow, it has been concluded that it is a leakage around the projectile and it has a large impact on the flow field in front of the projectile. The model where the leakage was included gave results closer to what can be observed in reality when the flow in front of the projectile are considered. However, the method to model the leakage used in this work probably gives an overestimation of the leak flow.

For the outflow of propellant gas behind the projectile when it has left the tube both models give similar results. It is also this outflow that has the largest impact on the forces acting on the projectile. If the calculated forces are compared between the models, the leakage seems to have a rather small impact. It is therefore possible that the leakage only have an impact on the visual results. The leakage effects can also differ from different types of projectiles. In this work, only results from one test firing have been used for comparison. Therefore more work is needed to find the best way to model the leak flow for different cases.

Since the leakage makes the model more complex and computational heavy and only have a limited effect on the projectile forces, it may not be the number one thing to focus on when refining the model. Nevertheless, for precise results the leakage needs to be taken into account.

The main purpose of the model was to analyse if the fluid dynamic loads can affect the stability of the projectile during the intermediate ballistic phase. With the present model it is not possible to say exactly how the projectile is affected. For this a full 3D model would be necessary. It has however, been shown that large forces acts on the projectile when it has left the tube. It is therefore possible that the outflow of propellant gas can affect the stability.

One of the goals with the model was to analyse how far from the muzzle the projectile is affected of the outflowing propellant gas. The present model is not precise enough to decide exactly when the projectile is in free stream condition, but the results indicate that it is in the order of one meter from the tube.
In order to create a pressure inside the launch tube both the combustion model and pressure curve gave an increase in pressure that corresponds well to experiments. Both models has its advantages and disadvantages, the combustion model requires more input in comparison to the pressure curve. But the best way to handle the pressure rise is probably to develop the combustion model since this model is not dependent of measurements from test firings.

A big concern when using a CFD model of the launch process is the computational power. The models that were studied were quite downscaled in order to get results in a reasonable period of time. To use the model to get reliable results it will be necessary to enlarge it to a full 3D model. But if the computational power is not a concern this type of model can be a good addition to today’s tools.

As a final conclusion it can be said that it is possible to create a CFD model that simulates the launch of the projectile with accuracy. But there is still much room for improvements of the models presented in this work.
7. Future Work

Since this work mainly was to develop a modelling technique to make it possible to analyse the transition phase with CFD there is room for much improvements and refinements in the model for future investigations. In order to evolve this work it can be divided into four different areas:

- Implement more physics into the model and evaluate how it affects the model. In this work some of the most important parts have been studied, for example leakage and pressure. The model can be implemented with more physics, e.g. rotation, friction etc. This needs to be evaluated to see how it affects the model and the result. It could also be interesting to implement more DOF in the model so that the projectile can move in all direction, but it would require some changes in the model. This would make it possible to further analyse if the projectile becomes unstable when it leaves the tube.
- Make a more meticulous study of the mesh and time step. Since in this work the main focus was to create a well posed model, and to be able to investigate it, the simulation time needed to be decreased. Therefore the mesh and time step used were as large as they could be and still give a valid result. Therefore it would be interesting to make a study and see how the mesh and time step affects the result and the studied entities.
- Since it was seen that the flow is affected by the boundary for the model with leakage, it needs to be investigated how much larger the domain needs to be. This needs to be done so that the free stream force can be investigated further.
- It is also interesting to continue analyse which entities that can be evaluated with this method and how well these results compares to experimental values.

One of the main things that need to be investigated is if the leakage can be modelled by having leakage at the beginning and after some distance close the gap. This would lead to a representation of when the slipping ring enters the grooves. This model would be interesting to compare to the other models and see how well it corresponds to the experimental values.

Since a moving mesh is used where the elements are stretched, it leads to that the elements after some time becomes highly skewed. Therefore it would be helpful if the simulation contained a criterion that updates the mesh automatically when it is needed. As of right now this is done manually when it seems like the mesh needs to be updated. It should not be any problem to implement this since it has been done before in previous works.

Since the experimental values that were evaluated were taken from different experimental setups it makes it hard to compare the results. Therefore, it would be interesting to make some experimental trials that are specified for this work in order to get better experimental values to compare with. Some entities that would be interesting to monitor is the amount of propellant gas leaving the tube and the velocity of the gas. The pressure in and around the launch tube would also be useful to measure.
8. Perspectives

The work and the results presented in this thesis can be put into a larger context. In this section some brief comments about the project from an economic, environmental and ethical point of view will be given.

In an economical perspective these results and the development of precise simulation models in general, have potential to lead to a large cost reduction since the number of expensive experimental trials can be reduced. At the same time the cost for computer power and time for model development are increased, but this cost is often smaller than the manufacturing cost of prototypes. A precise simulation model will also make it possible to test many configurations and small changes in for instance geometry in a quantitative way that would not be reasonable with an experimental method.

Also in an environmental point of view the work can give a good impact since this method of simulating the results can help reducing experimental trials. This will reduce emissions since there are many different materials in the manufacturing process which in some cases are highly toxic. Also the working environment can be improved since there is always a risk when working with and handle explosives for the experimental trials.

The defence industry in general, is a sensitive area with many different opinions in regards to a social and ethical point of view. The authors choose not to address this subject and leave to the readers to have their own opinions.
References


