Dynamic CFD Modelling of Deploying Fins During Transitional Ballistic

Anton Jybrink

Engineering Physics and Electrical Engineering, master's level
2018

Luleå University of Technology
Department of Computer Science, Electrical and Space Engineering
The transition from inner to outer ballistics is a crucial part for the stability of the projectile. A projectile is mainly stabilized in two ways, with fins or by rotation. This work is limited to analyze a fin stabilized projectile.

The launch of the projectile and the deployment of the fins are a quick process, therefore high forces and high temperatures will affect the stability of the projectile. Due to these factors, it is hard to quantify experiments to analyze the stability of the projectile.

To gain knowledge about how the forces will affect the path of the projectile during the launch and the deployment of its fins Computational Fluid Dynamics (CFD) can be a useful technique.

In this work, a 2D methodology have been developed in Ansys® Fluent to analyze the launch of a projectile and the deployment of the fins. A RANS-model have been used in combination of dynamic mesh in order to handle the movement of the projectile. The projectile accelerates due to a pressure rise which have been initialized by a mass flow and energy curve as a source term.

This work indicates that it is possible to predict the flow behavior and the forces influencing the projectile and the deploying fins. This work used a 2D model throughout the simulations and a 3D model is therefore needed to further compare and validate the simulation methodology.
First of all, I would like to express my gratitude to my supervisor Torbjörn Green at Saab Dynamics who have guided me during this project. I would also like to express my gratitude to my supervisor at Luleå University of Technology Gunnar Hellström and to all who have helped me during this project. Additionally I would like to express my gratitude the friendly people at the department, for creating such a nice working environment. Finally I want to thank my family and girlfriend for always being supportive.

Anton Jybrink

Karlskoga, June 2018
# NOMENCLATURE

## 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>DOF</td>
<td>Degrees Of Freedom</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Method</td>
</tr>
<tr>
<td>FVM</td>
<td>Finite Volume Method</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
</tbody>
</table>

## Greek Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>δ</td>
<td>Displacement</td>
<td>m</td>
</tr>
<tr>
<td>(\vec{\epsilon})</td>
<td>Strain tensor</td>
<td>1</td>
</tr>
<tr>
<td>γ</td>
<td>Diffusion coefficient</td>
<td>(m^2s^{-1})</td>
</tr>
<tr>
<td>(\lambda)</td>
<td>Lame’s first parameter</td>
<td>Pa</td>
</tr>
<tr>
<td>(\mu)</td>
<td>Shear modulus</td>
<td>(Nm^{-2})</td>
</tr>
<tr>
<td>(\mu_v)</td>
<td>Molecular viscosity</td>
<td>(kgm^{-1}s^{-1})</td>
</tr>
<tr>
<td>(\nu)</td>
<td>Kinematic viscosity</td>
<td>(m^2s^{-1})</td>
</tr>
<tr>
<td>(\phi)</td>
<td>General scalar</td>
<td>1</td>
</tr>
<tr>
<td>(\rho)</td>
<td>Density</td>
<td>(kgm^{-3})</td>
</tr>
<tr>
<td>(\vec{\sigma})</td>
<td>Stress tensor</td>
<td>(Nm^{-2})</td>
</tr>
<tr>
<td>(\vec{\alpha})</td>
<td>Angular acceleration</td>
<td>(rads^{-2})</td>
</tr>
<tr>
<td>(\vec{\omega})</td>
<td>Angular velocity</td>
<td>(rads^{-1})</td>
</tr>
</tbody>
</table>
# Roman Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\vec{v}$</td>
<td>Velocity</td>
<td>$ms^{-1}$</td>
</tr>
<tr>
<td>$\vec{a}$</td>
<td>Acceleration</td>
<td>$ms^{-2}$</td>
</tr>
<tr>
<td>$d$</td>
<td>Boundary distance</td>
<td>$m$</td>
</tr>
<tr>
<td>$E$</td>
<td>Energy</td>
<td>$J$</td>
</tr>
<tr>
<td>$\vec{F}$</td>
<td>Force</td>
<td>$N$</td>
</tr>
<tr>
<td>$R$</td>
<td>Universal gas constant</td>
<td>$Jmol^{-1}k^{-1}$</td>
</tr>
<tr>
<td>$\vec{g}$</td>
<td>Gravitation vector</td>
<td>$N$</td>
</tr>
<tr>
<td>$h_j$</td>
<td>Entalphy</td>
<td>$Jmol^{-1}$</td>
</tr>
<tr>
<td>$I$</td>
<td>Unit tensor</td>
<td>1</td>
</tr>
<tr>
<td>$L$</td>
<td>Inertia</td>
<td>$m^4$</td>
</tr>
<tr>
<td>$J_j$</td>
<td>Diffusion flux</td>
<td>$molm^{-2}s^{-1}$</td>
</tr>
<tr>
<td>$K_{ij}$</td>
<td>Spring constant</td>
<td>$Nm^{-1}$</td>
</tr>
<tr>
<td>$k$</td>
<td>Molecular conductivity</td>
<td>$Sm^2mol^{-1}$</td>
</tr>
<tr>
<td>$k_t$</td>
<td>Turbulent thermal conductivity</td>
<td>$Wm^{-1}k^{-1}$</td>
</tr>
<tr>
<td>$l$</td>
<td>Length</td>
<td>$m$</td>
</tr>
<tr>
<td>$D_m$</td>
<td>Mass diffusion coefficient</td>
<td>$m^2s^{-1}$</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass</td>
<td>$kg$</td>
</tr>
<tr>
<td>$v_g$</td>
<td>Mesh velocity</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$M$</td>
<td>Moment</td>
<td>$Nm$</td>
</tr>
<tr>
<td>$M_w$</td>
<td>Molecular weight</td>
<td>$kgmol^{-1}$</td>
</tr>
<tr>
<td>$p_g$</td>
<td>Gauge pressure</td>
<td>$Pa$</td>
</tr>
<tr>
<td>$p_s$</td>
<td>Static pressure</td>
<td>$Pa$</td>
</tr>
<tr>
<td>$S$</td>
<td>Source term</td>
<td>1</td>
</tr>
<tr>
<td>$S_h$</td>
<td>User defined volumetric heat source</td>
<td>1</td>
</tr>
<tr>
<td>$D_T$</td>
<td>Thermal diffusion coefficient</td>
<td>$m^2s^{-1}$</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature</td>
<td>$k$</td>
</tr>
<tr>
<td>$D_t$</td>
<td>Turbulent diffusivity</td>
<td>$m^2s^{-1}$</td>
</tr>
<tr>
<td>$\vec{u}_m$</td>
<td>Mesh displacement velocity</td>
<td>$ms^{-1}$</td>
</tr>
<tr>
<td>$u$</td>
<td>Velocity</td>
<td>$ms^{-1}$</td>
</tr>
<tr>
<td>$\tau_{eff} \cdot \vec{v}$</td>
<td>Viscous dissipation</td>
<td>$Wkg^{-1}$</td>
</tr>
<tr>
<td>$V$</td>
<td>Volume</td>
<td>$m^3$</td>
</tr>
<tr>
<td>$Y_i$</td>
<td>Mass fraction</td>
<td>1</td>
</tr>
</tbody>
</table>
## CONTENTS

**Chapter 1 – Introduction**

1.1 Background ................................................. 1
  1.1.1 Ballistics ............................................. 2

1.2 Problem description ........................................... 3

1.3 Limitations .................................................. 4

1.4 Previous work ............................................... 5

**Chapter 2 – Theory**

2.1 Governing equations ............................................. 7
  2.1.1 Mass conservation equation ............................. 7
  2.1.2 Momentum conservation equations .................... 7
  2.1.3 Energy conservation equations ......................... 8

2.2 Flow characteristic ........................................... 8

2.3 Turbulence model ............................................. 8

2.4 Gas law ....................................................... 9
  2.4.1 Ideal gas law ........................................... 10
  2.4.2 Ideal gas mixing law .................................... 10

2.5 Species transport ............................................. 10

2.6 Rigid body motion ............................................. 11

2.7 Dynamic mesh ................................................. 11

2.8 Update methods for the dynamic mesh ....................... 12
  2.8.1 Smoothing method ...................................... 12
  2.8.2 Dynamic layering method .............................. 15
  2.8.3 Remeshing method ...................................... 15

**Chapter 3 – Method**

3.1 Geometry ...................................................... 17
  3.1.1 Computation domain ................................... 18

3.2 Mesh ......................................................... 20

3.3 Setup ........................................................ 21
  3.3.1 Non-conformal zones ................................... 21
  3.3.2 Materials ................................................. 22
  3.3.3 Models .................................................. 22
3.3.4 Boundary conditions ............................................. 24
3.3.5 Dynamic mesh .................................................. 25
3.3.6 Rigid body motion .............................................. 25

Chapter 4 – Results and Discussion ............................. 27
4.1 Combustion model .................................................. 27
4.2 Velocity .............................................................. 29
4.3 Flow behaviour ...................................................... 30
4.4 Force .................................................................. 33
  4.4.1 Force on projectile .............................................. 33
  4.4.2 Force on fin ...................................................... 35
4.5 Boundary conditions .............................................. 37

Chapter 5 – Conclusion ................................................. 39

Chapter 6 – Future Work ............................................... 40
Chapter 1

Introduction

This master thesis is the last project in the degree of engineering physics and electrical engineering specializing in computational methods at Luleå University of Technology. It has been carried out in collaboration with Saab Dynamics AB in Karlskoga a subsidiary of Saab AB.

The goal with this thesis is to find a suitable CFD methodology to calculate the dynamical forces that appear on deploying fins on a projectile after exiting the gun barrel.

1.1 Background

In the area of unguided weapon system, the launch of a projectile with caliber 84 mm is made from a man-portable gun barrel. An example of such system can be seen in Fig. [1]. The moment when the projectile exits the gun barrel is a crucial part since instabilities induced on the projectile in this area will affect the projectile path and precision. There are two main principles to stabilize an unguided projectile, with fins or by rotation. This thesis work focus on a fin stabilized projectile.

The launch of a projectile starts with the ignition of the propellant. While the propellant is burning it generates an expansion of propellant gas which in turn builds up a pressure. As the gas expands the pressure rises inside the gun barrel, generating a force that drives the projectile forward. While the projectile is moving forward the propellant gas mass flow will fill up the new space behind the projectile and drive it forward until it exits the gun barrel. If there is a gap between the projectile and the gun barrel propellant gas may slip past. The leaked propellant gas, that is in front of the projectile, can affect
the stability of the projectile.

During the launch, while the projectile is inside the gun barrel the fins are folded towards the muzzle. Directly when the projectile exits the muzzle the fins starts to deploy. The stability of the projectile is critical during the time while the fins are deploying, due to the high forces acting on it such as pressure shocks and high temperature gases.

![An example of the unguided weapon system Carl-Gustaf M4, manufactured by Saab Dynamics AB. Photo: Hans Berggren ©Saab AB [2014].](image)

### 1.1.1 Ballistics

According to [Oxford dictionaries](#) ballistics is *the science of projectiles and firearms*. More precisely, ballistics describes the flight path of a projectile from the gun barrel to the target. The field of science is divided into four parts, internal, transitional, external and terminal ballistics.

Internal ballistics treat the process inside the gun barrel from when the propellant is ignited until the projectile exits the gun barrel. Including how the propellant is burning and how the expansion of gases will generate a pressure rise that drives the projectile forward [Ballistics](#).

The transitional ballistics treat the process when the projectile leaves the muzzle until the only forces acting on the projectile are due to external ballistics [Introduction to ballistics](#). The transitional ballistics is a crucial part of the launch since the projectile is exposed to pressure chocks and high mass flows from the propellant gas.

The external ballistics treat the process in the air and how aerodynamic loads (e.g. drag force and wind) and the gravity will affect the path of the projectile
1.2. Problem description

The terminal ballistics treat the process when the projectile hits the target. The internal, transitional, external and terminal ballistics are visualized in Fig. 1.2.

The time when the fins are deploying are part of the transitional ballistics. In this area it is very hard to quantify experiments since:

- Deployment of fins is a quick process (a few milliseconds).
- High forces due to high accelerations.
- High temperature ($>2000 \, k$).
- Soot

One way to analyze the flow around the projectile while the fins are deploying is to film with a high-speed camera. But during the launch of a projectile smoke will limit the visibility.

1.2 Problem description

In this work a generic projectile of caliber 84 mm is analyzed for method development. The projectile is fin stabilized. While the projectile is inside the gun barrel the fins are folded towards the muzzle. After the projectile exits the gun barrel the folded fins deploy, as illustrated in Fig. 1.3.
Due to limitations to quantify experiments it is desirable to have an effective CFD methodology to analyze the aerodynamic loads on the fins during the deploying process.

![Illustration of fin deployment.](image)

### 1.3 Limitations

This work is limited to approximately 20 weeks full time work. This limits how extensive the study can be and therefore some simplifications have been done. These are:

- A simplified generic projectile and gun barrel model.
- The friction force between the projectile and the gun barrel is neglected.
- In reality the projectile is clamped to a cartridge until a certain force/pressure is reached. This is neglected.
- The breech of the gun barrel is treated as a wall.

Also it is assumed that the air will be treated as perfect air (thermally perfect) and has a condition of 1 atmospheric (atm) pressure and has a temperature of 300 k.

It is desirable to have a 3D cylindrical sector of 30° model. Due to limitations in the software Ansys® Fluent this is not possible. The limitation is with the dynamic mesh. The working methodology for the dynamic mesh is that it
will mark mesh cells in a domain that exceed user defined skewness and size criteria when a body/wall is moving. These marked cells will then be remeshed. But when a model of a 3D cylindrical sector is used, Ansys\textsuperscript{®} Fluent can not split the tetrahedral (triangular) elements. For a full 3D model the dynamic mesh works as it should but with a bigger model comes more computational time. Therefore this methodology is developed for a 2D model in order to save computational time.

### 1.4 Previous work

Previous work has been done in the area of simulating the launch of a projectile from a gun barrel. These works done by Fredriksson R, Hellberg V [2016], Xavier S. [2011] and Christer and Carl [2013] focused on spin stabilized projectiles. The most common thing in these works was the use of a dynamic mesh, i.e. the mesh will be updated as the projectile moves forward.

In the work done by Fredriksson R, Hellberg V [2016], Ansys\textsuperscript{®} CFX was used to find a CFD methodology for a projectile launch. In this work, the simulation starts with a combustion model behind the projectile in the gun barrel. The combustion model is tested with 2 different methods. The first case a measured pressure curve was implemented and for the second case a mass flow was modelled and implemented. When the projectile starts to move/is moving the mesh is updated. The area around the projectile has a constant mesh that will move with the projectile. This will lead to a less computational heavy model in comparison to if the mesh should be updated around the projectile for every time step. The elements behind and in front of the area with constant mesh will be stretched and compressed respectively. In this case, the projectile will have a maximum velocity of about 280 ms\textsuperscript{−1}. Their work shows that it is possible to predict the flow behaviour from a projectile launch with a CFD model.

Xavier S. [2011] used Ansys\textsuperscript{®} Fluent to simulate the pressure blast of a projectile which is launched from a gun barrel. The projectile will accelerate from a high pressure and high temperature that is initialized behind the projectile. When the projectile is moving the mesh will be updated in the same manner as for Fredriksson R, Hellberg V [2016]. Xavier S. [2011] varied the mass of the projectile, the initialized pressure and temperature behind the projectile to investigate the effects on the flow field. This leads to, the projectile gets different maximum velocities which is about 700 and 900 ms\textsuperscript{−1} respectively. Xavier S. [2011] showed that it is possible to capture the flow behaviour.
Christer and Carl [2013] used OpenFOAM to make a computational study of unsteady gun tube flows. At the start of the simulation, it was assumed that the projectile had a velocity of 600 m s$^{-1}$. All propellant combusted and the volume behind the projectile was filled with high pressure and high temperature gases. In this case, the mesh will be remeshed when the projectile had traveled a certain distance. For every remeshing, the number of elements will increase. This will lead to a more computational heavy method compare to Fredriksson R, Hellberg V [2016] and Xavier S. [2011]. For the dynamic mesh setup, in this case, Christer and Carl [2013] discuss that the results should be compared against sliding mesh or over set grid technique.
In this section, the theory behind the solution is described. It is preferable if the reader has some background in CFD and/or mathematics, because the most rudimentary details are omitted.

2.1 Governing equations

The governing equations are the Navier-Stokes equations for transport of mass, momentum and energy. The definition of these equations in ANSYS Fluent can be seen below.

2.1.1 Mass conservation equation

The conservation of mass also known as the continuity equation is defined as

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S, \]  

(2.1)

where \( \rho \) is the density, \( \vec{v} \) is the velocity vector and \( S \) is a source term, for example, a mass flow curve.

2.1.2 Momentum conservation equations

The conservation of momentum is defined as

\[ \frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p_s + \nabla \cdot (\vec{\sigma}) + \rho \vec{g} + \vec{F}, \]  

(2.2)

where \( p_s \) is the static pressure, \( \vec{\sigma} \) is the stress tensor, \( \rho \vec{g} \) and \( \vec{F} \) are the gravitational force and external force respectively. The stress is described by

\[ \vec{\sigma} = \mu_v[(\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \cdot \vec{v} I], \]  

(2.3)
where $\mu_v$ is the molecular viscosity and $I$ is the unit tensor.

### 2.1.3 Energy conservation equations

For heat transfer problems the energy equation is used and it is defined as

$$
\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v} (\rho E + p_s)) = \nabla \cdot \left( (k + k_t) \nabla T - \sum_j h_j \vec{J}_j + (\vec{\tau}_{eff} \cdot \vec{v}) \right) + S_h, \quad (2.4)
$$

where $E$ is the energy, $k$ is the molecular conductivity, $k_t$ is the turbulent thermal conductivity, $T$ is the temperature, $h_j$ and $\vec{J}_j$ is the enthalpy and diffusion flux of species $j$ respectively. The term $\vec{\tau}_{eff} \cdot \vec{v}$ is the viscous dissipation and $S_h$ is an user-defined energy source, for example, as an energy curve.

### 2.2 Flow characteristic

When the projectile is moving through the air, the air molecules are disturbed and moves around the projectile. How these molecules will move i.e. with a smooth or chaotic behaviour. The behaviour are dependent on the shape and speed of the projectile (moving object), the mass, viscosity and the compressibility of the air (surrounding fluid).

The flow characteristic is often determined from the dimensionless variable Reynolds number, that express the ratio between the inertial forces and the viscous forces according to,

$$
Re = \frac{ul}{\nu}, \quad (2.5)
$$

Where $u$ is the velocity magnitude of the fluid with respect to the object, $l$ is the length of the object that faces the fluid and $\nu$ is the kinematic viscosity of the fluid.

At low Reynolds numbers laminar (smooth) flow occurs while at high numbers turbulent (chaotic) flow occurs. In this work it is assumed that the flow is turbulent because the flow around the projectile will have a velocity greater than 0.3 Mach ($\approx 100 \text{ ms}^{-1}$).

### 2.3 Turbulence model

In order to catch the behavior of a turbulent flow, a good turbulence model is needed. There are several different turbulence models that can be used.
2.4. Gas law

The most advanced model is the Direct Numerical Solution (DNS). This model resolves the whole spectrum of turbulent fluctuations. For a simulation using this model the mesh must be fine enough to capture the smallest scales. This method is so computational heavy and with today’s computers it would be impractical to use.

An alternative to DNS is the Large Eddy Simulation (LES). This model resolves only largest turbulent fluctuations while the smallest are modelled in some way. This model is recommended to use if the turbulent effects have a great impact on the solution. In most applications it is not motivated to use due to it is very computational heavy.

The most commonly used and not so computational heavy model are the Reynolds Average Navier-Stokes (RANS) Turbulence Models. For these models the Navier-Stokes solution is decomposed into a mean and a fluctuating component. The velocity, pressure and other scalar quantities can be written as

\[ u_i = \bar{u}_i + u'_i \quad \text{and} \quad \phi = \bar{\phi} + \phi' \]  \hspace{1cm} (2.6)

respectively. Where \( u_i \) denotes the velocity (\( i = 1, 2, 3 \)), \( \phi \) denotes a scalar such as energy or pressure, the vector and prim symbols denote the mean and the fluctuating components respectively. By inserting these two equations in eq. (2.1) and (2.2) gives the RANS equations.

In this project, a turbulence model named Shear Stress Transport (SST) \( k-\omega \) is used, which is a type of RANS turbulence model. This model is used because it is in general more accurate in near-wall regions, i.e when forces on a body should be calculated and less computational heavy compared to DNS and LES.

2.4 Gas law

In this project two different fluids (chemical species) is used, air and propellant gas. In order to calculate the right properties of the mixing fluid the ideal gas mixing law is used. Further, the ideal gas law is used to calculate the density of the fluid.
2.4.1 Ideal gas law

The ideal gas law is defined by

$$\rho = \frac{p_{op} + p_g}{RT \frac{1}{M_w}},$$

(2.7)

where $p_{op}$ is the operating pressure, $p_g$ is the gauge pressure, $R$ is the universal gas constant, $T$ is the temperature, $M_w$ is the molecular weight. If the fluids velocity is greater than 0.1 mach ($\approx 34 \text{ ms}^{-1}$) the operating pressure should be set to zero. This is because for higher mach number flows the operating pressure is less significant. The pressure changes in these flows are much larger than in low mach number flows. Thus, eq. (2.7) is rewritten to

$$\rho = \frac{p_g}{RT \frac{1}{M_w}}.$$  

(2.8)

2.4.2 Ideal gas mixing law

The equation for the ideal gas mixing law is almost the same as the ideal gas law. It is defined by

$$\rho = \frac{p_{op} + p_g}{RT \sum_i \frac{Y_i}{M_{w,i}}} \Rightarrow \rho = \frac{p_g}{RT \sum_i \frac{Y_i}{M_{w,i}}}.$$  

(2.9)

Where $Y_i$ and $M_{w,i}$ is the mass fraction and the molecular weight of species $i$.

2.5 Species transport

When two or more different fluids are used the mass fraction of each species has to be calculated. The general form of the conservation equation for chemical species is defined as

$$\frac{\partial}{\partial t} (\rho Y_i) + \nabla \cdot (\rho \vec{v} Y_i) = -\nabla \cdot \vec{J}_i + R_i + S_i,$$

(2.10)

where $R_i$ is the rate of change due to reaction sources and $S_i$ is the rate of change due to other sources of species $i$ respectively. This general equation is solved for $N - 1$ species, where $N$ is the total number of species.

The diffusion flux is calculated according to,

$$\vec{J}_i = -(\rho D_{m,i} + \rho D_t) \nabla Y_i - D_{T,i} \frac{\nabla T}{T},$$

(2.11)

where $D_{m,i}$ is the mass diffusion coefficient for species $i$, $D_t$ is the turbulent diffusivity, $D_{T,i}$ is the thermal diffusion coefficient.
2.6 Rigid body motion

In order to get a body to move due to the forces acting on it, the Six Degree Of Freedom (6DOF) solver is used. This solver uses the forces and the moment on the body to calculate the translational and the angular motion of the body. The translational motion (acceleration) of the body is calculated from Newton’s second law according to

\[ \vec{a} = \frac{1}{m} \sum \vec{F}, \]  

(2.12)

where \( m \) is the mass and \( \vec{F} \) is the force vector. While the angular motion (acceleration) is calculated by

\[ \vec{\alpha} = \frac{1}{L} \left( \sum \vec{M} - \vec{\omega} \times L \vec{\omega} \right), \]  

(2.13)

where \( L \) is the inertia tensor, \( \vec{M} \) is the moment vector and \( \vec{\omega} \) is the angular velocity vector.

2.7 Dynamic mesh

The dynamic mesh model can be used to model flows where the domain is changing with time due to a motion on a boundary or on a body inside the domain. In this work the projectile is allowed to move inside the domain and on a symmetry boundary condition, therefore the dynamic mesh method is needed.

The motion of a boundary can either be a prescribed motion or an unprescribed motion. A prescribed motion can be, for example, a user input value of translational and/or angular acceleration/velocity of a body. While an unprescribed motion is based on the solution for the current time step. Which means that the translational and/or angular acceleration/velocity is calculated according to eqs. (2.12) and (2.13).

In order to conserve the solution when the dynamic mesh is used the integral form of the conservation equation can be written as

\[ \frac{d}{dt} \int_V \rho \phi dV + \int_{\partial V} \rho \phi (\vec{v} - \vec{v}_g) \cdot d\vec{A} = \int_{\partial V} \gamma \nabla \phi \cdot d\vec{A} + \int_V S_\phi dV, \]  

(2.14)

for a general scalar, \( \phi \), on an arbitrary control volume, \( V \), whose boundary is moving. In eq. (2.14) \( \vec{v}_g \) is the mesh velocity of the moving mesh, \( S_\phi \) is the source term of \( \phi \) and \( \partial V \) is used to represent the boundary of the control volume ANSYS\textsuperscript{\textregistered} Fluent.
2.8 Update methods for the dynamic mesh

There are three different dynamic mesh update methods in *Ansys® Fluent*. In this section a brief introduction to these methods are described, if the reader wants to know more details about these methods, the [ANSYS® Fluent](https://www.ansys.com/products/simulation/ansys-fluent) Ansys theory guide is recommended.

In order to adjust the mesh in a zone with a moving and/or deforming boundary, there are three main methods to use. These methods are:

- Smoothing method,
- Dynamic layering method,
- Remeshing method.

2.8.1 Smoothing method

The smoothing method can be used in zones that has triangular, tetrahedral, quadrilateral, hexahedral, prism, poly and cutcell mesh. This method absorbs the motion of a moving or deforming boundary by moving the interior and the boundary nodes without changing the number of nodes. There are three different smoothing methods available. These are:

- Spring-based smoothing,
- Diffusion-based smoothing,
- Linearly elastic solid.

When the smoothing method is used the mesh quality can deteriorate or cells/faces can become degenerated. This can lead to e.g. negative cell volume which will lead to convergence problem. In order to avoid these problems, it is a good idea to use smoothing and remeshing together.

**Spring-based smoothing**

The spring-based smoothing method is based on the known displacement at the boundary nodes. The edges between two nodes are idealized as a network of interconnected springs. Then, when a displacement occurs at a given node it will generate a force that is proportional to the displacement along all springs that are connected to the node. The force on a node, by using Hook’s law, can be written as
2.8. Update methods for the dynamic mesh

\[ F_i = \sum_{j} n_i k_{ij} (\delta_j - \delta_i) \]  \hspace{1cm} (2.15)

where \( n_i \) is the number of neighbouring nodes connected to node \( i \), \( k_{ij} \) is the spring/stiffness constant between node \( i \) and \( j \), \( \delta_i \) and \( \delta_j \) are the displacement of node \( i \) and its neighbour \( j \). The spring/stiffness constant is defined by

\[ k_{ij} = \frac{k_{fac}}{\sqrt{||x_i - x_j||}}, \]  \hspace{1cm} (2.16)

where \( k_{fac} \) is the user input spring constant factor, which should have a value between 0 and 1. A value of 0 indicates no damping while a value of 1 indicates full damping on the springs.

An iterative equation for the displacement of a node can be obtained when the net force from all springs that is connected to the node is zero, i.e. \( F_i = 0 \) in eq. (2.15). This leads to the following equation

\[ \delta_{i}^{m+1} = \sum_{j} n_i k_{ij} \delta_j^m \sum_{j} n_i k_{ij}, \]  \hspace{1cm} (2.17)

where \( m \) is the iteration number. When the solution has converged the position is updated by

\[ x_i^{n+1} = x_i^n + \delta_i^{\text{converged}}, \]  \hspace{1cm} (2.18)

where \( n + 1 \) and \( n \) denotes the position of the next and the current time step respectively.

**Diffusion-based smoothing**

For the diffusion-based smoothing method, the mesh motion is governed by the diffusion equation

\[ \nabla \cdot (\gamma \nabla \bar{u}_m) = 0 \]  \hspace{1cm} (2.19)

where \( \gamma \) is the diffusion coefficient and \( \bar{u}_m \) is the mesh displacement velocity. In order to solve eq. (2.19) the boundary condition is defined by a user prescribed motion or computed boundary motion (6 DOF). The boundary conditions, on deforming boundaries, are that the mesh motion is tangent to the boundary, which means that the normal velocity component vanishes.

The diffusion equation describes how the boundary motion propagates into the interior of the deforming mesh. The diffusion constant has two different formulations, where the first is based on the boundary distance, \( d \), according to
\[ \gamma = \frac{1}{d^\alpha}, \]  
and the second is based on the cell volume, \( V \), according to
\[ \gamma = \frac{1}{V^\alpha}, \]  
where \( \alpha \geq 0 \) is a user input parameter.

The diffusion equation, eq. (2.19), is solved with two different numerical methods. Either it is solved using the finite element method (FEM) or the finite volume method (FVM). For the FEM solution the displacement velocity is obtained directly at each node while for the FVM it is obtained for the cell center and interpolated to the nodes. Which method that should be used depends on the element types in the mesh, the FEM is used for all meshes except meshes containing polyhedras or hanging nodes (cutcell or adapted meshes) where the FVM is applied. Then the node positions are updated according to
\[ \vec{x}_{\text{new}} = \vec{x}_{\text{old}} + \vec{u}_m \Delta t, \]  
where \( \vec{x}_{\text{new}} \) is the new position vector, \( \vec{x}_{\text{old}} \) is the old position vector and \( \Delta t \) is the time step.

**Linearly Elastic Solid**

For the linearly elastic solid based smoothing method the mesh motion is governed by the following equations
\[ \nabla \cdot \vec{\sigma}(\vec{\delta}) = 0, \]
\[ \vec{\sigma}(\vec{\delta}) = \lambda (Tr\vec{\epsilon}(\vec{\delta})) I + 2\mu \vec{\epsilon}(\vec{\delta}), \]  
\[ \vec{\epsilon}(\vec{\delta}) = \frac{1}{2}(\nabla \vec{\delta} + (\nabla \vec{\delta})^T), \]
where \( \lambda \) is Lame’s first parameter, \( Tr \) is the trace, \( \vec{\epsilon} \) is the strain tensor and \( \mu \) is the shear modulus.

The boundary conditions for this method are obtained either from a user prescribed motion or computed boundary motion (6 DOF). The imposed deformation on the boundary, from the motion of the body, are then transferred into the interior of the mesh, this will cause the mesh to behave as if it is a linearly elastic solid.
The governing equations, eq. (2.23), is solved by using the FEM approach. The displacement of the mesh for the interior and the deforming boundary nodes are obtained directly at the nodes.

### 2.8.2 Dynamic layering method

The dynamic layering method can be used in hexahedral and wedge mesh zones, it is used to add or remove cells nearby a moving boundary. The user specifies an ideal height on each moving boundary then the layer of cells nearby the moving boundary is split or merged with the cells next to it. If the cells in a layer are expanding, the height of the cell is allowed to increase until

\[ h_{\text{min}} > (1 + a_s)h_{\text{ideal}}, \]  

(2.24)

where \( h_{\text{min}} \) is the minimum cell height of the cell layer, \( h_{\text{ideal}} \) is the ideal cell height and \( a_s \) is the layer split factor. Instead, if the cells in a layer are being compressed, they can be compressed until

\[ h_{\text{min}} < a_c h_{\text{ideal}}, \]  

(2.25)

where \( a_c \) is the layer collapse factor. If the condition in eq. (2.24) or (2.25) are met, the cells are split or the compressed cells are merged into new cells.

### 2.8.3 Remeshing method

The remeshing method can either be used alone or with the smoothing method. The remeshing method checks if the size of a cell/face is within the size criteria or if the skewness of a cell/face is within the skewness criteria. If any of these are outside the criterion the corresponding cell/face are marked and remeshed locally. If the new cell/face does not satisfy the criterion they are discarded and the old cell/face are retained. There are several remeshing methods available, they are presented with a short description about how they work below.

- Local cell

Marks interior mesh cells and remeshes these cells if they exceed the user defined skewness or size criteria.

- Zone remeshing

Remeshing a complete cell zone. It is enabled by default when local cell is activated. It will be used if local cell fails to create an acceptable mesh and will be invoked if the skewness > 0.98.
• Local face

This method is only for 3D cases. Marks faces and adjacent cells on deforming boundaries based on skewness and allows local remeshing at the deforming boundaries. This method can not remesh across multiple face zones.

• Region face

Marks faces and adjacent cells on deforming boundaries based on the size criteria. If the size criteria are reached the cells splits or deletes immediately adjacent to the moving boundary.

• Cutcell zone

This method is only for 3D cases and remeshes a complete cell zone. The remeshing occurs either at a predefined intervals or when the mesh quality is poor.

• 2.5D

This method applies only for extruded 3D geometries (2D triangular mesh that is revolved/extruded). It marks faces on a deforming boundary based on user defined skewness and size criteria. Cells can also be marked by using a size function. When the criteria are reached the triangular surface are remeshed and smoothed on one side and the changes are then extruded to the opposite side.
CHAPTER 3

Method

This section contains the geometry of the model, the numerical setup for the computational domain and the setup in Ansys® Fluent. In section 1.1 it was described how the projectile accelerates due to a pressure rise. The methodology on how this was implemented is described in this chapter. Also, this methodology focuses on a symmetric 2D model but can be applied to a 3D model with some minor changes.

3.1 Geometry

As mentioned in section 1.2 a generic projectile with a fin was used in the simulation. The shape of the projectile is simplified compared to a real projectile. The shape is simplified because this was a methodology development and there exist many different types of projectiles.

The geometry of the projectile is symmetric and has a maximum height of 42 mm, a minimum height of 20 mm and a length of 200 mm. While the wing has a length of 85 mm and a height of 15 mm; see Fig. 3.1.
3.1.1 Computation domain

The computational domain was divided into several smaller domains:

1. Gas domain.
2. Projectile domain.
3. Air domain.
4. Far field domain.

These domains are visualized in Fig. 3.2.

The gun barrel is placed along the x-axis and is 600 mm long with a height of 48 mm and a wall thickness of 16 mm. The projectile is placed in the middle of the gun barrel. The distance from the rear end of the projectile to the breech of the gun barrel is 200 mm and the distance between the projectile front to the muzzle is 200 mm. The total height of the domain is 3130 mm and the total length is 6320 mm (side to side of domain 4). Domain 4 is used to overcome reversed flow problems in the simulation, i.e. flow bunches back at the boundary. The geometry of the computational domain is visualized in Fig. 3.3.
3.1. Geometry

Figure 3.2: The computational domain divided into several smaller domains.

Figure 3.3: Geometry of the computational domain with the coordinate system placed at the projectile front.
A gap between the projectile and the gun barrel wall was needed to obtain a stable simulation using the dynamic mesh. In reality the gap between the gun barrel and the projectile is negligible. Although, some propellant gas may slip past.

### 3.2 Mesh

Since this project was a methodology study a mesh study on how different mesh sizes affect the solution was omitted. It were assumed that the initial mesh was acceptable. When the dynamic mesh is used there is no way to affect the mesh quality disregard to the size and the skewness criterions in the dynamic mesh setup. Thus the mesh quality will vary with time.

As said in section 3.1.1 the computational domain was divided into several smaller domains. This because different mesh methods and sizes have been applied in each domain.

In domain 2, the triangle mesh method have been applied. This because the dynamic mesh methods, diffusion based and remeshing, should work. Also in this domain the mesh size is the smallest, it was initialized with a size of 1.8 mm. All others domains have a mix of triangle and quadrilateral mesh elements but with different sizes. Domain 4 have the largest mesh size in order to smear out the solution which would lower the chance of reversed flow at the boundaries.

Edge sizing have been applied to the outer edges where two domains meet in order to get a smooth transition between the domains. The mesh size of the edge corresponds to the mesh size in the finer domain.

The mesh is visualized in Fig. 3.4.
3.3 Setup

In this part the setup for the simulation is described. The general settings for these simulations have been:

- Pressure based method,
- Solution of transient type,
- Turbulence model SST $k - \omega$,
- Energy equation is enabled,
- Operating pressure is set to 0 atm,
- Reference pressure is set to 1 atm.

3.3.1 Non-conformal zones

In order to calculate the solution between different domains, that has different mesh methods and sizes. The edge of a domain must be matched with the corresponding edge of the other domain. This can be done in the mesh interface.

Figure 3.4: The applied mesh in the computational domain.


settings where the two edges are selected and then the matching option was chosen.

### 3.3.2 Materials

In order to use a combustion model that mimics the burning propellant, the corresponding properties for the propellant gas must be inserted. The properties for the propellant gas are defined by its density, specific heat, thermal conductivity, viscosity and the molecular weight. These properties were defined using a user defined material database.

This method is used to define the propellant gas and the perfect air properties. The propellant gas was then inserted into domain 2, while perfect air was inserted into domain 1, 3 and 4.

### 3.3.3 Models

**Combustion model**

To get the projectile to accelerate due to the pressure rise in the gun barrel a generic mass flow and energy curve was used. These curves should mimic the pressure and energy rise in the gun barrel. The curves were inserted as source terms in domain 1 and are visualized in Figs. 3.5(a) and 3.5(b) respectively. Note that the units for the mass flow and the energy is per unit volume, hence the original mass flow and the energy curve had to be scaled with the volume of the gun barrel.
3.3. Setup

Figure 3.5: Generic mass flow and energy curve that were inserted as source terms to mimic the pressure and energy rise.

The mass flow curve is calculated from a pressure curve, this pressure curve can either be measured experimentally or be calculated. In this case, a calculated pressure curve have been used. The pressure curve is then used to calculate the mass flow according to

\[
\dot{m}(t) = b P(t)^N A_{\text{eff}}(t) \rho_p.
\]  

(3.1)

Where \( b \) and \( N \) are propellant dependent constants, \( P \) is the pressure, \( A_{\text{eff}} \) is the effective burning area of the propellant and \( \rho_p \) is the propellant density.

The energy curve is calculated by

\[
E(t) = \dot{m}(t) \cdot C,
\]  

(3.2)

where \( C \) is the propellant caloric value.

As the mass flow and energy curves are generic they have been modified to give the projectile a velocity that resembles real Carl-Gustaf munition.

**Transport equation**

In order to calculate how the propellant gas will move and be mixed with the perfect air, the species transport equation was enabled. The following fluid properties is calculated with the corresponding equation:

- **Density** ideal gas law.
- **Specific heat** mixing law.
- **Thermal conductivity** ideal gas mixing law.
- **Viscosity** ideal gas mixing law.
- **Mass diffusivity** set as a constant.

### 3.3.4 Boundary conditions

Four different boundary condition were used. These conditions are explained below.

1. **Wall**
   This condition was applied to the projectile, the fin and the walls of the gun barrel. These walls were set to the no-slip condition which sets the velocity to zero on the boundary.

2. **Pressure inlet**
   This condition was applied on one side of domain 4. For this inlet, the total pressure and temperature was set to 1 atm and 300 $k$ respectively.

3. **Pressure outlet**
   This condition was applied on two sides of domain 4. For these outlets the total pressure and temperature are set to 1 atm and 300 $k$ respectively.

4. **Symmetry**
   As the model was a half 2D model, the symmetry boundary condition was applied on the boundary along the x-axis.

The boundaries of the geometry with corresponding condition are visualized in Fig. 3.6, where green corresponds to pressure inlet, blue to pressure outlet, black to wall and yellow to symmetry.
3.3. Setup

3.3.5 Dynamic mesh

In order to update the mesh in domain 2 when the projectile is moving forward, the dynamic mesh method was used. While the projectile moves forward the mesh elements in front and behind the projectile gets compressed and stretched respectively. To handle the compressed and stretched elements so they do not get too compressed or to stretched the remeshing method was used.

The settings for the remeshing method was; it should update the mesh if any cell/face has a lower or larger size than 1 or 2 mm respectively. But also if any cell/face has a higher skewness than 0.4. Only using the remeshing method can be computational heavy. Therefore the smoothing method with the diffusion based method with a user input value of 0.25 was also used.

The projectile and the fin were defined as rigid bodies while the symmetry line in domain 2 was allowed to deform when the projectile moves.

3.3.6 Rigid body motion

In order to get the projectile to accelerate from its initial position the 6 DOF solver was used. The projectile will then accelerate due to the pressure that
acts on it, which is generated from the precalculated mass flow and energy curve. Due to the pressure rise, a User Defined Function (UDF) was written to describe the mass, moment of inertia and how the projectile is allowed to move. In this case, the projectile was allowed to move in x-axis and rotational about z-axis, hence 2 DOF.

The fin was set to move with the projectile. Saab Dynamics AB fin stabilized projectiles has a mechanism to control the fin deployment. In the simulations, the mechanism was simplified to have a prescribed motion of the fin. The prescribed motion of the fin was also implemented using a UDF.
Results and discussion

In this section, the results from the simulation are presented and discussed. For this methodology a generic projectile was used. To get the projectile to accelerate from its initial position due to the pressure rise a mass flow and an energy curve was inserted as source terms. To verify the used methodology some of the results are compared to results in Fredriksson R, Hellberg V [2016]. It is important to remember that the study in Fredriksson R, Hellberg V [2016] is for a spin stabilized projectile. The geometry for the projectile and gun barrel differs, therefore the result may vary between the cases. But it gives a hint of the methodology’s quality and trust.

4.1 Combustion model

The pressure curve that appears in domain 1 due to the burning propellant is visualized in Fig. 4.1(a). Where the black cross indicates when the projectile front is at the muzzle. The black circle indicates where the fin starts to deploy, at this point the projectile has about 1 mm left in the gun barrel.

The simulated pressure curve is compared to a measured and simulated pressure curve from Fredriksson R, Hellberg V [2016] shown in Figs. 4.1(a) and 4.1(b) respectively. Comparing Figs. 4.1(a) and 4.1(b) indicates that the behaviour of the pressure curve seems realistic. With an almost flat line at the beginning, then a rapid increasing, with a steep angle and then a slower decreasing pressure. However, the magnitude of the pressure curve is not realistic for a real weapon, this is due to the simulated model is in 2D.

The simulated temperature and mass flow in domain 1 is visualized in Figs. 4.1(c) and 4.1(d) respectively. From these figures it is seen that the maximum
temperature is about 2900 k and the maximum out flowing mass flow is about 60 kgs$^{-1}$. At this time the pressure also reach the maximum value about 2.42 MPa according to Fig. 4.1(a). This is a realistic behaviour, that these curves reaches their maximum value at the same time.

Comparing Figs. 4.1(a) 4.1(c) and 4.1(d) at the time around 7 ms it is seen that the pressure, temperature and mass flow starts to increase. This is mostly due to an underpressure is created in the gun barrel and a mixture of air and propellant gas will flow inward instead.

Figure 4.1: Resulting pressure, temperature and mass flow in the gas domain, compared with a measured and resulting pressure curve from Fredriksson R, Hellberg V [2016].

(a) The resulting total pressure in the gas domain.

(b) Comparison between a measured and simulated gas model pressure curve. Courtesy of Fredriksson R, Hellberg V [2016].

(c) The resulting temperature in the gas domain.

(d) The resulting mass flow in the gas domain.
4.2 Velocity

It takes about 3.5 \( ms \) for the projectile to reach the muzzle, a travelling distance of 0.2 \( m \) at this point the projectile velocity was 186.6 \( m/s \). The corresponding values for the projectile to exit the gun barrel was 4.4 \( m/s \), 0.39 \( m \) and 227.6 \( m/s \) according to Figs. 4.2(a) and 4.2(c). This can be compared with the maximum velocity of 238.9 \( m/s \) which occurs at 8 \( ms \), corresponding to a travelling distance of 1.26 \( m \) according to Figs. 4.2(b) and 4.2(d).

The difference between the two lower velocities and the maximum velocity is about 28.0 % and 4.96 % respectively. Why the difference was so large is most likely due to the geometry of the gun barrel, which work as a cannon. Due to the geometry the mass flow can only exit through the muzzle. This will lead to a continued increase in velocity of the projectile after it has exit the gun barrel since the mass flow still will affect the projectile.
(a) Projectile velocity as function of time during the motion.

(b) Projectile velocity as function of time when it has left the gun barrel.

(c) Projectile velocity as function of distance during the motion.

(d) Projectile velocity as function of distance when it has left the gun barrel.

Figure 4.2: Projectile velocity as function of time and distance during the whole motion and when it has left the gun barrel.

### 4.3 Flow behaviour

In order to verify the behaviour of the gas flow in the simulations, the velocity field and propellant mass fraction around the muzzle is analyzed and compared to experimental figures from Fredriksson R, Hellberg V [2016].

In Fig. 4.3 two snap shots from two different times are visualized. For these two snap shots, the behaviour of the out flowing gas from an experiment is visualized. The velocity field and the propellant mass fraction of the gas from the simulation are visualized respectively.

The left and right column in Fig. 4.3 corresponds to 2 m/s and 1.25 m/s before
the projectile exits the gun barrel. Two of the figures from Fredriksson R, Hellberg V [2016] can be seen in Figs. 4.3(a) and 4.3(b), which are snap shots from a high speed camera during an experiment. In these figures red lines are drawn to visualize the behaviour of the out flowing gas. The velocity fields are visualized in Figs. 4.3(c) and 4.3(d) while the propellant mass fraction are visualized in Figs. 4.3(e) and 4.3(f).

It can be seen from the experimental figure, Fig. 4.3(a), that the out flowing gas changes direction and gets a sharp angle relative to the gun barrel. The same behaviour was also captured in the simulations, Figs. 4.3(c) and 4.3(e).

It can still be seen from the experimental figure, Fig. 4.3(b), that the out flowing gas gets a sharp angle relative to the gun barrel. But now it seems like the angle of the out flowing gas was lower than in 4.3(a). In Fig. 4.3(b) but a bit in front of the muzzle it seems like the gas has started to rotate and created a vortex. The same behavior with a lowered angle of the out flowing gas and the creation of a vortex was captured in the simulations, Figs. 4.3(b) and 4.3(f).
Result and discussion

(a) Experimental figure about 2 ms before projectile exits the gun barrel. Courtesy of Fredriksson R, Hellberg V [2016].

(b) Experimental figure about 1.25 ms before projectile exits the gun barrel. Courtesy of Fredriksson R, Hellberg V [2016].

(c) Velocity field about 2 ms before projectile exits the gun barrel.

(d) Velocity field about 1.25 ms before projectile exits the gun barrel.

(e) Propellant mass fraction field about 2 ms before projectile exits the gun barrel.

(f) Propellant mass fraction field about 2 ms before projectile exits the gun barrel.

Figure 4.3: Comparison between experimental and simulated results.
4.4 Force

The main purpose with this project was to find a methodology in order to analyze the dynamical loads on the fin under the imposing process. Therefore the forces acting on both the projectile and fin are scrutinized.

4.4.1 Force on projectile

The forces acting in the x-direction follows the behaviour of the pressure curve, Fig. 4.4(a). This behaviour was expected since the force is pressure dependent. The increasing force, that can be seen after 5 ms, Fig. 4.4(c) mostly depends on the outflow of gases behind the projectile after it exits the muzzle. Also, the increasing force that occurs after 10 ms was due to the fin angle was such the mixture of propellant gas and air will be pushed on the projectile i.e. the flow had a higher velocity than the projectile. The increased force is visualized in Fig. 4.5 where the contour represents the pressure and the arrows represents the velocity.

The forces acting on the projectile in the y-direction was larger than the forces in the x-direction. Also, in this direction the force had the same behaviour as the pressure curve, Figs. 4.4(b) and 4.4(d). The reason why the forces in this direction was so large, is mostly due to the geometry which is a half projectile with a symmetry condition. If the geometry would have been a full 2D projectile, about same magnitude of forces should act on the other side. This would result in a total force with a smaller magnitude in the y-direction.
(a) Force in x-direction acting on the projectile.

(b) Force in y-direction acting on the projectile.

(c) Force in x-direction acting on the projectile when it has exit the gun barrel.

(d) Force in y-direction acting on the projectile when it has exit the gun barrel.

Figure 4.4: Forces that are acting on the projectile in x- and y-direction.
4.4. Force

4.4.2 Force on fin

The oscillating behaviour that is seen in Figs. 4.6(a) and 4.6(b) up to about 4 ms was due to when the propellant gas progress forward in the gun barrel. While the propellant gas progress forward it will collide with the edges of the projectile and fin. When that happens, the flow starts to rotate and will affect the projectile and fin with different magnitude of force at different locations. The rotating motion (vorticity) of the flow is visualized in Fig. 4.7(a).

The rapid increasing force that is seen at about 4 ms in Figs. 4.6(a) and 4.6(b) was due to when the first part of the projectile has exit the gun barrel and the part in front of the fin was outside the gun barrel. This will lead to an over-pressure in the gun barrel which will influence the fin, visualized in Fig. 4.7(b).

When the projectile is in the air and the fin deploys the forces is decreasing.
in x-direction and y-direction. There is one exception at 6 ms which is mostly dependent on the mass flow. The minimum force on the fin in x-direction is about $-4000$ N and occurs when the angle of the fin is almost 90°. The corresponding point but in y-direction it is seen that the force have increased. After this point the force in x-direction increases until the fin stops in its final position and the forces start to decrease (about 14 ms). While for the force in y-direction starts to decrease. This is shown in Figs. 4.6(c) and 4.6(d).

Figure 4.6: Forces that are acting on the fin in x- and y-direction.
4.5. Boundary conditions

Analyzing the force influencing the projectile at the same time as the fin is exposed for the maximum force. It seems like the force influencing the fin do not influence the projectile. A way to adjust this is to write a UDF where the forces influencing the fin is set as external forces on the projectile.

4.5 Boundary conditions

In this case the pressure inlet and outlet boundary condition have been used, this will generate a high velocity in the point where they meet. This high velocity can influence the solution if the boundary is near the projectile. In this case it seems like the affect on the projectile is negligible sine the velocity of the propellant gas is greater than the initialized velocity from this point. Otherwise, to avoid this problem, the boundary condition pressure far field can be used. The velocity field with the flow direction is visualized in Fig. 4.8.

Also, when a symmetry condition is used, like in this case, the information about how the von Karman vortex street will affect the projectile is lost.
Figure 4.8: Visualization of the velocity and the flow direction due to the applied boundaries condition.
It can be concluded that simulating the launch of a projectile with fin deployment dynamically is a daunting task that includes many disciplines. It has been shown that it is possible to create a model that mimic a realistic behaviour. But in order to validate the results a 3D model must be created.

The dynamic mesh methods, remeshing and smoothing, is an expensive computational method. But using these methodology’s results in a good mesh quality throughout the simulation.

The main purpose with this project was to obtain a methodology that can calculate the dynamical forces acting on the fin deployment. The results indicates a realistic behaviour, but a 3D model must be created to get a more realistic behaviour and result.
CHAPTER 6

Future work

Since the aim of this project was to develop a methodology there is room for improvements and refinements in future investigations. The improvements can be split up into different areas such as:

- A more realistic geometry for this case such as the breech should not be treated as a wall.
- Include more physics, like:
  - Friction force exerted on the projectile from the gun barrel.
  - Let the projectile be clamped until a certain pressure is acting on it.
  - In a 3D model include all 6 DOF in order to analyze if the projectile becomes unstable when exiting the gun barrel.
  - Set the forces from the fin as external forces on the projectile.
- Make a study of the mesh and time step in order to see how these will affect the solution. Because this work was to develop a methodology the mesh and time step was set as big as possible to reduce the computational cost, but still give reasonable results.
- To reduce the computational cost of the dynamic mesh, the overset mesh technique should be tested and be compared with the used technique in this setup.

Since this methodology is created with a generic projectile and source terms it is hard to validate the results. It would be interesting to use real values of these and then compare the results with an experiment.
REFERENCES


