Development of a 1D to 3D CFD coupling methodology on an inlet manifold

Andreas Johansson

June 20, 2018
Abstract

We investigate the coupling between 3D simulations of the flow in an engine manifold with simplified 1D models of the engine performance. The inlet manifold in an existing 1D engine model is replaced with a more sophisticated CFD manifold simulation. The rest of the 1D system is left untouched. The CFD model can better capture mixing effects and the geometry of the manifold. The lorry developer Scania provides the 1D engine model together with the corresponding manifold geometry. Inexperienced readers are introduced to related work and underlying theory in the area. The CFD manifold simulation is developed and validated in terms of sensitivity to the grid, choice of turbulence model and surface roughness. Comparisons with the original system are made for a selection of relevant engine predictions along with manifold flow quantities. Minor effects in the predictions are observed at a hefty price in computational time and in implementation effort.
Sammanfattning

## Contents

1 Introduction 7
  1.1 Background ........................................... 7
  1.2 Intended Readers ..................................... 7
  1.3 Related Work .......................................... 8
    1.3.1 Catalyst simulations based on coupling of 3D CFD tool with 1D .......... 8
    1.3.2 A finite element solver and energy stable coupling for 3D and 1D fluid models ......... 8
    1.3.3 1D finite difference and 3D CFD for flow coupling .................. 9
  1.4 Thesis Objective ..................................... 9

2 Computational Fluid Dynamics 11
  2.1 Governing Equations .................................. 11
  2.2 Discretization ......................................... 12
    2.2.1 Finite Difference Method ......................... 12
    2.2.2 Finite Volume Method ............................... 14
  2.3 3D CFD ................................................. 15
    2.3.1 Direct Numerical Simulation ...................... 15
    2.3.2 Reynolds Average Navier Stokes .................. 16
    2.3.3 Large Eddy Simulation .............................. 17
  2.4 1D Flow Models ....................................... 18
  2.5 Coupling Conditions .................................. 18
    2.5.1 Gas species communication ........................ 20

3 Methodology 21
  3.1 GT-Power Engine Model ................................. 21
  3.2 3D Manifold .......................................... 23
    3.2.1 Uncoupled boundary conditions ..................... 26
  3.3 Mesh .................................................. 26
    3.3.1 Mesh independence study ........................... 28

4 Results 30
  4.1 Turbulence model ..................................... 31
  4.2 Surface roughness ..................................... 31
  4.3 co-simulation comparisons ............................. 32

5 Analysis 36
  5.1 Conclusions .......................................... 38
Glossary

**base size** parameter in the star-ccm+ automated meshing technique. it refers to the length for all relative size controls. 26, 27

**Boussinesq hypothesis** A approximation that momentum transferred by eddies in turbulence can be described with a particular viscosity and the mean strain rate tensor S. Equation (13) describes this model. 16

**Central Difference** Differentiation scheme found in equation (9). 14

**CFD** Computational Fluid Dynamics. 7

**co-simulation** Simulations of a coupled problem are modeled and simulated in separate subsystems. 8

**DNS** Direct Numerical Simulation. 15

**EGR** Exhaust Gas Recirculation. A Pipe system returning a part of the exhaust gases back into the engine manifold. 20

**FD** Finite Difference Method. 12

**FEM** Finite Element Method. 12

**fundamental laws of physics** Here refereed to the momentum, energy and mass conservation in classical physics. 11

**FVM** Finite Volume Method. 12

**GUI** Graphical User Interface. 18

**heat transfer coefficient** λ A heat transfer property between a solid surface and a fluid. The property is dependent on the flow such as BL thickness or surface roughness. 22

**ICE** Internal Combustion Engine. 7

**inferior vena cava** A large vein that transports blood back into the hearth from the lower and middle parts of the body. Formed by the merging left and right common iliac veins. 9

**injection pressure** Pressure in the fuel outlet right before injection into the cylinder. High pressure results in very high fuel velocities, necessary to maintain optimal combustion at limited revolution times. 18

**LES** Large Eddy Simulation. 15

**MoC** Method of Characteristic. 9

**PDEs** Partial Differential Equations. 9
Periodic Steady State: The long term, referred to as the steady system response at a particular frequency. Here it will be given as a mean value with a corresponding amplitude. 18

Prismatic cell layer: Additional prismatic cells placed along the flow boundaries to accurately capture the boundary layer. Standard technique in CFD. 26

RANS: Reynolds Average Navier-Stokes Equations. 15

RPM: Revolutions Per Minute. 18

RSM: Reynolds stress equation model. 17

Spectral Method: Method for solving PDEs. Fast Fourier transforms with a sum of basis functions are utilized. 12

SST: Menter’s Shear Stress Transport model. 31

Thermal conductivity $k$: A heat transfer property within a material. Describes how heat is conducted according to Fourier’s heat law. 22
1 Introduction

1.1 Background

In the last decades Computational Fluid Dynamics (CFD) have been used extensively as a part of the development of various parts in Internal Combustion Engines (ICEs). Many of the improvements in combustion and gas exchange processes have been achieved thanks to CFD. The manifold is a crucial part for an effective combustion because it distributes air into the cylinders. Detailed flow behavior and accurate flow predictions of such as mass flow rate and pressure drop are some of the characteristics that makes CFD a valuable tool in the ICE industry[10]. The main limitation of CFD compared with the traditional 1D models is the severe increase in computational costs. Full scale 3D simulations of large flow systems such as an entire engine or a cooling systems are still too demanding[6]. The speed of a 1D simulation makes it suitable for testing a numerous different configurations during development but lacks the details of a 3D simulation. While the 1D simulation tools fail to capture all details and provide accurate results for complex flow situations, it has proven sufficient for some simple, yet common, parts in flow systems. Pipes are considered to be well modeled with 1D tools[10]. By strategically combining the two methods, one can use the benefits of each method while mitigating the disadvantages. In particular, the computational domain can be split into two parts. One part where the 1D code is sufficient and one part where 1D codes have proven insufficient. CFD is used only on those parts where 1D models are considered insufficient. The main advantage is that this enables good metric resolution in the 3D domain, although with a lower computational cost. The resulting accuracy and computation time compared to full 3D simulation varies a lot between different geometries and flow characteristics. Commercial software manufacturers have started to implement the option to automatically share flow quantities and fluid characteristics at each time step. This simplifies co-simulations with other software. Flow coupling can be implemented single way, meaning from an upstream domain to a downstream domain. It can also be a two way coupling where information is shared in both direction during each time step. The problem in the latter often originates at the intersection surface of the 1D domain and the 3D domain. The 1D average data can be inadequate as boundary condition for inhomogeneous flow areas in a 3D model which relies on point wise data[9].

1.2 Intended Readers

This thesis exemplifies an application where 1D-3D fluid coupling proved unnecessary. It also presents the coupling methodology of 1D flow models and finite volume CFD in detail along with the uncoupled algorithms. A reader, experienced in this topic will find the manifold description combined with Results chapter interesting. The readers, unfamiliar in the topic will be introduced to the algorithms and the underlying physics in the background chapter. Knowledge in basic fluid mechanics, calculus,
algebra and vector/tensor operations will make the content of this thesis easier to approach. Knowledge in combustion and engines makes interpretation of the results clearer.

1.3 Related Work

In recent years, the potential benefits of coupled 1D-3D flow algorithms have been brought into the light and is used in academic work and in industrial development. So far both excellent and mediocre results have been reported from coupling 1D and 3D codes for flow prediction. For some applications coupling has proven beneficial compared to the corresponding uncoupled simulations. In other problems either of the uncoupled models are preferable over the coupled simulation. Different approaches for efficient coupling have been proposed. Some previous works, especially early studies are academical. Today it is also found as a possible tool in flow system simulation. I will briefly summarize and present a selection of works conducted within this topic.

1.3.1 Catalyst simulations based on coupling of 3D CFD tool with 1D

J.Štěpánek et al. implemented a similar, modular approach although on a CO/NOx chemical reaction inside a catalyst system[27]. The authors argue that a 1D model fails to represent the flow through the individual channels inside a monolith converter. They also mentions how the computational effort becomes severe with a full size 3D domain. In their approach, the 3D software works as a host and calculates the flow field inside the converter shell. The 3D software calls the 1D channel model for the individual monolith channels. For each time step, the CFD calculates flow rate, pressure and temperature in the converter which is used as input for the 1D chemical reaction models. The 1D models solves for the enthalpy flux inside each monolith channel based on the information received from the 3D CFD host. The 1D models sends quantities such as temperatures and pressures back to the 3D tool which updates the flow field in the converter. Evaluation of the co-simulation is made on a single channel monolith. The 3D + 1D channel simulations shows similar results as the pure 1D model for this simple case. An entire catalyst is then dynamically co-simulated. The simulation successfully confirms that the initialization time for the CO/NOx reaction depends on the channel positions. A conclusion unable to be made with only a 1D model.

1.3.2 A finite element solver and energy stable coupling for 3D and 1D fluid models

An alternative 1D-3D model coupling condition is presented by Tatiana et al [7]. The one dimensional flow model can be derived from the Navier-Stokes equations. It is based on a long axisymmetric and elastic pipe. A source function is added as it is required in the later hemodynamic simulations. Also a rigid fluid 3D model was used. The model is suitable
in cylindrical domains in blood flow simulations. A 1D/3D coupling is developed, based on continuity of stress and the linear combination of energy flux and mass flux. The continuity is applied at the intersection area, separating the 1D and the 3D domain. In most works, mass flux is made continuous. The authors use the linear combination of mass and energy flux instead, as an attempt to reduce common mismatches in energy flux at the intersection. They mention that the linear combination continuity does not secure a perfect match in either the mass flux nor the energy flux. Relevancy of the method is presented with a coupled inferior vena cava filter simulation. Results are compared with a trusted 1D model.

1.3.3 1D finite difference and 3D CFD for flow coupling

In this paper[10], a 1D-3D methodology is developed to illustrate the potential of co-simulations for acoustics in engine exhaust systems. Particularly, a dynamic 1D model is used in the duct systems, where the flow is associated with no viscosity. Acoustic response in terms of a pressure pulse through the system is simulated. Both a test rig and the analytical solution for Sod’s problem is used for verification[10]. Sod’s problem has been used extensively for code verification and has a well known and verified analytical solution. The problem describes a pressure pulse generated by a breaking membrane, separating a high pressure area from a low pressure area within a duct [26]. The pressure pulse breaks down at the outlet where non-linearities are considerable. A Method of Characteristic (MoC) is applied to the section, separating the 1D flow domain and the 3D code at the non-linear outlet region. The MoC is in short a solver for hyperbolic PDEs which can be applied on the 1D inviscid Navier-Stokes equation referred to as the Euler equation. Riemann invariants, pressure and temperature is transferred through the interface surface to establish the coupling to the downstream 3D geometry. Results with co-simulation are closely matching both the experimental and analytical values. Time saved with co-simulation were stated to a factor $10^3$-$10^2$ compared to a pure 3D simulation while still maintaining the same metric resolution in the interesting outlet region[10]. The authors presses on how the technique may be an effective approach on describing sound waves in exhaust systems. The idea is to use 1D models in the upstream channel system with coupling to a 3D domain in the outlet area, where non-linear effects are considerable.

1.4 Thesis Objective

This master degree thesis simulates the flow inside a manifold and it constructs a 1D-3D coupling methodology for a truck combustion flow system. It presents the foundations of 1D-3D fluid coupling along with its advantages. Additional truthfulness from a strategically placed 3D domain can improve the 1D engine model. Co-simulations are compared with the original 1D model to evaluate the increase of accuracy. Unfortunately, no experimental data is available. Accuracy gain evaluations are hence restricted to expected quantities, model sensitivity and to global system
predictions. A concise description of the methodology can be summarized like this.

1. Develop and evaluate a CFD manifold environment.
2. Replace the 1D manifold in the engine system with the CFD manifold.
3. Compare coupled system behavior to the original, uncoupled 1D system.

Focus is placed at the potential of full system co-simulations for combustion development. The 1D simulation software GT-POWER is used for the engine flow simulations. GT-POWER is a 1D simulation software, specialized in full scale engine simulations. GT explains in the following coupling manual [2], how their 1D models often fails to capture mixing effects and cylinder variations in manifolds. This further justifies co-simulations in this system. A schematic figure for the coupled flow system is presented.

Along with improved accuracy from CFD, a co-simulation automatically provides a 3D visualization framework to the manifold. The competing technique is to use 1D models or experimental data as boundary conditions for a separate CFD manifold analysis. Effort and simulation time comparisons are made. Various turbulence models, LES, different mesh sizes and surface roughnesses are included in creation of the methodology. Star-CCM+ chosen for the 3D CFD simulations. It is valuable that GT-Suite is compatible with Star-CCM+ and tutorials for practical implementations are available on the GT web page.

Figure 1: Flow chart of engine flow system. Only major parts are included. The green box represents the 1D manifold model and the red box represents the CFD manifold replacement.
2 Computational Fluid Dynamics

2.1 Governing Equations

The foundation for all CFD is the governing equations in fluid dynamics. CFD codes predicts the motion described by these equations. Those equations cannot be solved analytically except for a few very special cases[8]. One approach is to approximately solve them using discretization and a computer to solve the resulting large set of equations. These are essentially the classical fundamental laws of physics. Written in text they appear in the following for a fluid[1]:

Mass of the fluid is conserved.

Rate of change of fluid momentum equals the forces acting on the fluid.

Rate of change of fluid energy equals the sum of work and heat on the fluid.

The laws can be written in integral form on a fixed volume. Starting with the mass conservation equation.

\[ \frac{d}{dt} \int_V \rho dV = - \int_S \rho v \cdot n dS \] (1)

The expression can be simplified utilizing the divergence theorem on the RHS and by noting that the relation holds for arbitrary fix volume. The integral relation consequently holds for the integrand as well. The simplified form is given as.

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho v) = 0 \] (2)

This is the first important partial differential equation in fluid dynamics and CFD and is referred to as the continuity equation. Further simplifications can be made in spacial cases such as steady flow where all partial derivatives in time are zero. Another case is incompressible flow where \( \frac{\partial \rho}{\partial t} \) and \( \frac{\partial \rho}{\partial \vec{x}} \) are negligible. For incompressible flow, relation (2) can simply be expressed \( \nabla \cdot \vec{v} = 0 \). The same approach use for deriving the mass conservation can be repeated on the two other integral equations to obtain the partial differential equations for momentum and energy. The momentum equation reads

\[ \frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \times \vec{v}) = \rho \vec{f_v} + \nabla \cdot \vec{T} \] (3)

\( \vec{f_v} \) represents external volumetric forces such as gravity acting on the fluid. \( \vec{T} \) is the Cauchy stress tensor. Sometimes one can find the stress tensor divided in two parts, with pressure separated from the viscous stresses. This equation also have various possible approximations in order to make it numerically approachable. The energy equation can as well be written in vector notation[17].
\[
\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho E \mathbf{v}) = \rho \mathbf{v} \cdot \mathbf{f} - \nabla q + \nabla \cdot (\mathbf{v} \cdot \mathbf{T})
\] (4)

\( q \) represents the externally added heat to the system. Fluids in many application can be regarded as incompressible. In this case, one can consider internal and kinetic energy separately when no expansion or compression is present[17]. The energy equation simplifies to a separate temperature equation.

The average velocity is set to zero at the wall boundaries. This is called the no-slip condition. The velocity profile close to the walls is proportional to a logarithmic distance to the wall. The velocity profile is presented in the expressions below.

\[
u^+ = \frac{1}{\kappa} \ln y^+ + C^+
\] (5)

\[
y^+ = \frac{y u_r}{\kappa}, \quad u_r = \sqrt{\frac{\tau_w}{\rho}}, \quad u^+ = \frac{u}{u_r}
\] (6)

\( y^+ \) is the normalized wall distance and \( u^+ \) is the normalized velocity. \( u_r \) is called the friction velocity or the shear velocity. The relations are used to describe the boundary layer and are valid for \( y^+ > 20 \).

### 2.2 Discretization

Discretization is a multi step process of breaking a partial equations in a continuous domain into discrete algebraic equations[15]. Instead of a continuous solution one receives an approximated solution and the flow quantities \( u_i, P \) at discrete locations. The use of interpolation then gives an approximate continuous solution. A detailed discretization methodology for each specific algorithm will not be included. The interested reader can find out more about the specific discretization in basic computational fluid dynamics literature or see the reference list in the end. Instead, the important steps of discretizing a set equations will be presented. More specifically, two particularly ways of discretization within CFD. Those are Finite Difference Methods (FDs) or Finite Volume Methods (FVMs). Note that there are others in CFD, such as Spectral Methods and Finite Element Methods (FEMs).

#### 2.2.1 Finite Difference Method

FD is an old technique that have been used for approximations when calculations were done by hand. Much emphasis is invested at introducing the inexperienced readers to the FD approach, leaving less to the FVM. The reason is the simplicity and insight the method provides for solving equations in discretized domains. I will start to illustrate with help of a picture done by Tu Jiyan along with others[15]. For instructive reasons, a two dimensional grid will be of study. A 3D discretization can be made analogous with 3 indices, \((i,j,k)\).
For the structural grid in figure 1, let \( T \) be a scalar field of choice. Then indices \((i,j)\) are related as following:

\[
T(x+\Delta x,y)_{i,j} = T(x,y)_{i+1,j} = T_{i+1,j} \quad i \in \{0,1,\ldots,N-1\} \\
T(x,y+\Delta y)_{i,j} = T(x,y)_{i,j+1} = T_{i,j+1} \quad j \in \{0,1,\ldots,N-1\}
\]

which is obvious when observing Figure 1. By utilizing relations (5), one can approximate partial derivatives in a rather satisfying fashion.

Consider a Taylor expansion of the scalar field \( T(x+\Delta x,y)_{i,j} \).

\[
T(x+\Delta x,y)_{i,j} = T(x,y)_{i,j} + \left( \frac{\partial T(x,y)}{\partial x} \right)_{i,j} \Delta x + \frac{1}{2} \left( \frac{\partial^2 T(x,y)}{\partial x^2} \right)_{i,j} \Delta x^2 + O(\Delta x^3) 
\]

And also expanding \( T(x-\Delta x,y)_{i,j} \) into a Taylor series.

\[
T(x-\Delta x,y)_{i,j} = T(x,y)_{i,j} - \left( \frac{\partial T(x,y)}{\partial x} \right)_{i,j} \Delta x + \frac{1}{2} \left( \frac{\partial^2 T(x,y)}{\partial x^2} \right)_{i,j} \Delta x^2 + O(\Delta x^3) 
\]

Now, subtract equation (7) from equation (6) and rewrite the left sides with notation presented in expression (5).

\[
T(x,y)_{i+1,j} - T(x,y)_{i-1,j} = 2 \left( \frac{\partial T(x,y)}{\partial x} \right)_{i,j} \Delta x + O(\Delta x^3) 
\]

Finally rewriting the partial derivative in eq.(8) into a algebraic equation with a second order approximation error \( \Delta x^2 \).

\[
\left( \frac{\partial T(x,y)}{\partial x} \right)_{i,j} = \frac{T_{i+1,j} - T_{i-1,j}}{2\Delta x} + O(\Delta x^2) 
\]
This is a *Central Difference* discretization for a partial derivative. Longer or shorter Taylor series expansions allows for other algebraic approximations with different order of the truncation error. Examples are forward an backward differences with a linear truncation error.

\[
\left( \frac{\partial T(x, y)}{\partial x} \right)_{i,j} = \frac{T_{i+1,j} - T_{i,j}}{\Delta x} + O(\Delta x) \quad \text{Forward difference}
\]

\[
\left( \frac{\partial T(x, y)}{\partial x} \right)_{i,j} = \frac{T_{i,j} - T_{i-1,j}}{\Delta x} + O(\Delta x) \quad \text{Backward difference}
\]

The corresponding derivation can be done for the partial derivatives in \( y \) and by utilizing the indice \( j \). Further, a steady scalar field with no time dependence was chosen. Time discretization is done similar to space with an additional index for time. Since the approach is the same for time I will refer the interested reader to basic literature in numerical calculations.

A limited illustration of how the central difference results into an algebraic system is made. It can result in point wise quantities for a scalar field. A simple one dimensional, made up differential equation will be considered.

\[
\frac{\partial T(x)}{\partial x} + kT(x) = 0, \quad 0 < x \leq a, \quad T_0 = b \quad (12)
\]

With the central difference discretization in a one dimensional domain, equation (10) can be written as.

\[
\frac{T_{i+1} - T_{i-1}}{2\Delta x} + kT_i = 0, \quad i \in 1, \ldots, N \quad T_0 = b \quad (13)
\]

Equation 11 can be set up into a system of equation expressed in matrix notation.

\[
\begin{pmatrix}
  k & \frac{1}{2} & 0 & \cdots & 0 \\
  -\frac{1}{2} & k & \frac{1}{2} & 0 & \cdots \\
  0 & -\frac{1}{2} & k & \frac{1}{2} & 0 & \cdots \\
  \vdots & \ddots & \ddots & \ddots & \ddots & \vdots \\
  0 & \cdots & \cdots & \cdots & \cdots & 0 \\
\end{pmatrix}
\begin{pmatrix}
  T_1 \\
  T_2 \\
  \vdots \\
  \vdots \\
  T_N \\
\end{pmatrix}
= 
\begin{pmatrix}
  b \\
  0 \\
  \vdots \\
  \vdots \\
  0 \\
\end{pmatrix}
\]

The vector \( T \) can now be solved for and a discrete solution is obtainable.

### 2.2.2 Finite Volume Method

The finite volume method is particularly advantageous within CFD. Most other areas of physic are dominated by Finite Element Method which is overall solid. As discussed in the Governing Equations section, All the governing equations within fluid dynamics are conservation equations. Mass, Momentum and Energy are conserved. The FVM is especially successful in solving conservatory differential equations.

The domain is divided into small volumetric cells with a corresponding control volume for each nodal points. Integrating the equations for each
cell is the unique step in the FVM. This generates a set of discretized equations for each cell[29]. An appreciated property of these equations is the transparent physical meaning for each equation since it corresponds to conservation in a specific cell. The grid or mesh does not have to be structured, as long as the control volumes fills the whole domain. The divergence part in the conservation equations can be converted into surface integrals through the divergence theorem. Due to continuity, influx into a specific cell has to be equal to the combined out flux to neighboring cells. This is a valuable aspect that can be utilized for stability. Techniques for discretizing the conservation equations are not covered in here. However, most FVM discretization techniques are related to the FD example in the previous section.

2.3 3D CFD

As computational resources has grown exponentially over the years[25], availability of 3D CFD simulations has grow as well. Discretizing a domain in all direction increases the number of unknowns. whilst it results in additional computational effort, it also allows for more accurate results, more details for optimization and less required modeling. More details are captured compared to the average spatial values of 1D simulations. The accuracy of CFD simulations varies between algorithms and also depends on the problem. To what degree the simulation resolves the partial flow equations varies between types of CFD simulations. In general, simulations are typically divided into 3 categories, DNS, LES and RANS.

2.3.1 Direct Numerical Simulation

The method solves the Navier-Stokes equations. Results from these simulations are known to be extremely accurate. Another advantage with DNS is that limited calibration or knowledge about the flow is necessary. This is an appreciated consequence of no turbulence modeling or simplifications of the flow equations[22]. DNS provides detailed solutions with a high metric resolution for post-processing. This makes DNS great tool for studying flow. In fact, DNS is used as numerical experiments due to the reliability and visualization possibilities. The downside of this method is the requirement computational effort. There are still today no computer that can perform DNS on high enough Reynolds numbers for typical flows in engineering. The estimated computational effort for a DNS scales with \( Re^{11/4} \). This originates from the ratio of the smallest and largest length scales in turbulent fluctuations. A minimum grid resolution in a spatial directions is approximately the ratio of domain size and the kolmogorov length scale \( \frac{L}{\eta} = Re^{3/4}_L \). For all three spatial direction and an estimated time scaling as \( Re^{1/4}_L \), it leads to the above Reynolds number dependence. As of today, Kaneday et al. still has one of the highest \( Re \) used in a DNS. At \( Re \approx 1200 \) a total of 4096\(^4\) grid points were used[18]. Although impressive, practical flow problems within industry requires larger Reynolds numbers.
2.3.2 Reynolds Average Navier Stokes

Most CFD simulations are run with codes that fall into this category. The unexperienced reader is introduced to 3D CFD solver strategies, similar to the one used in this thesis. The approach starts with Reynolds decomposition, a common strategy for approaching turbulent flows. The idea is to separate the flow quantities in the Navier-Stokes equations (4) into a steady and a fluctuating part, \( u_i = U_i + u'_i \) and \( p = P + p' \). \( U_i, P \) are the mean flow quantities and \( u'_i, p' \) are the fluctuating flow quantities. The equations are then time averaged and simplified using the quantities \( \overline{P} \equiv P, \overline{U} \equiv U, \overline{p'} \equiv 0 \) and \( \overline{u'} \equiv 0 \). Written in Einstein notation, the expression is simplified into:

\[
\rho U_j \frac{\partial U_i}{\partial x_j} = \rho \overline{T}_i + \frac{\partial}{\partial x_j} \left[ -P \delta_{ij} + \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \rho u'_i u'_j \right] \quad (14)
\]

The only dependence of the fluctuating quantities in the average equations stems from the last term often referred to as the Reynolds stress term. \(-\rho u'_i u'_j\) is a symmetric tensor of second order thus contains 6 unknowns. Counting the additional 3 velocity components along with the pressure field, there are a total of 10 unknown parameters. There are only 4 equations including the continuity equation. The problem described is referred to as the closure problem in turbulence\[28\]. Modeling the Reynolds stress term is the approach to make the system of equations solvable. All turbulent stress components are approximated as relations of the other quantities \( U_i \) and \( P \) therefore replacing 6 unknowns for less unknowns in the model. There are lots of different models within the field. How accurately a selected model can predict depends on the flow itself. For example, some models show better results in free flow whilst others show more accurate results in viscous regions. Also the computational effort and numerical stability varies with different models. This have made the choice of turbulence model an important feature in CFD and extensive research have been conducted to evaluating models and coefficient calibration for numerous types of flows. We mention Andreas Håkansson et al. as one of many as they evaluated predictions of different models in a high pressure valve\[13\].

The basic \( \epsilon - k \) model is often considered the standard turbulence model in CFD. It was developed by Lauder and Jones as early as in 1972\[16\]. The model is simple meanwhile it produces good results for free and isotropic flow. There are limitations and various corrections for strong pressure gradients, rotating flow, highly anisotropic flow or other more complex types of flow. The standard \( \epsilon - k \) is a two equation model is based on the Boussinesq hypothesis. Hence assumes that turbulent stresses can be described with a turbulent viscosity \( \mu_t \) and the mean velocity analogous to viscous stress\[13\]:

\[
-\rho u'_i u'_j = -\rho K \delta_{ij} + \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (15)
\]

Which can be compared to the exact viscous stress:

\[
-\rho \delta_{ij} + \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (16)
\]
One difference is that $\mu$ which is a property of the fluid and $\mu_t$ is a flow property. You can see that the approximation may fail in rotating flows because expression (5) has no explicit dependence on the rotation strain rate tensor $\Omega_{ij} = (\frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i})$. The two equation model estimates the unknown $\mu_t$ according to the following relations [16]:

$$\nu_t = \frac{\mu_t}{\rho} \approx C_{\nu} \frac{k^2}{\epsilon}$$  \hspace{1cm} (17)

$C_{\nu}$ is a calibration constant, typically around 0.09. $k$ and $\epsilon$ are determined by the two eddy viscosity differential equations.

$$\frac{\partial K}{\partial t} = \frac{P - \epsilon}{K} + \frac{\partial}{\partial x_j} \left( \left( \nu + \frac{\mu_t}{\sigma_K} \right) \frac{\partial K}{\partial x_j} \right)$$  \hspace{1cm} (18)

Another popular two-equation model is the $K - \omega$ model. A similar model utilizing the specific dissipation $\omega = \frac{\epsilon}{\epsilon K}$ instead of $\epsilon$. Generally it predicts separation in boundary layers better than the $k - \epsilon$ [11]. The drawback is poor results in free flow simulations. There are more accurate models than the standard two equation turbulence models. The Reynolds stress equation model (RSM) is regarded as a precise model with a higher level of accuracy in most flow predictions. It involves an additional 7 equations instead of 2, one for each component in the stress tensor and one for dissipation. Results are particularly reliability in rotating and anisotropic flows [15]. The extra equations increases the computational time.

### 2.3.3 Large Eddy Simulation

LES is best described as an intermediate step between a DNS and a RANS. DNS fully resolves all turbulent scales down to eddies in the kolmogorov length scale. An eddy is the chaotic swirl or circulation in turbulent flow. Inside the large eddies there are smaller swirls with less energy. RANS equation models all this complex behavior and all its effects entirely according to models such as the $\epsilon - K$ described earlier. LES has started to be utilized in technical problems within the ICE and various other fields, in problems with small enough Reynolds numbers. The LES resolves only the largest scales of the turbulence and models the smallest eddies. It has been shown that small eddies are in general more isotropic and homogeneous [23]. While large scale eddy model are impossible generalize, small scale eddies are less dependent on geometry and flow conditions. Therefor small scale effects are more suitable for modeling while it also allows a lower grid resolution. Capturing only the large eddies lessens requirements and tolerates larger grid elements.

A spatial filtering technique is applied to the original Navier-Stokes equations. Filtering distance can be adjusted to change the minimum size where motion is resolved. Setting this size to the kolmogorov length will basically result in a DNS.
2.4 1D Flow Models

1D flow calculations have long been the number one tool for flow system prediction. In contrast to the 3D CFD, 1D models do not discretize the domain in all directions. A 1D model is often a combination attained from solving simplified Navier-Stokes in one direction with modifications from experimental observations. As discussed earlier, 1D models are very fast. Drawbacks are that useful details for development are missing. 1D flow models are typically only valid within its targeted application and needs extensive calibration[20]. As an example of the low generality in 1D models, consider a model developed to describe the flow inside a straight pipe. The same model might not accurately describe the flow in something as similar as the flow inside a equally long, although bent pipe. The software GT-POWER will provide a framework for the 1D flow models. GT-Power is a simulation software, specialized in engine simulations. The program offers models for components that are common in engines. The users can adjust parameters of the components to better reflect their engine. Typically by adjusting dimensions or use available experimental values for some quantities. Fortunately Scania will share their workbench used for developing this investigated geometry. With a system used for engine development, one can expect realistic output from the 1D models used in the thesis. There are however no previous calibration of the manifold in the model.

In consideration of the obvious periodicity of engine strokes, Periodic Steady State simulations are the standard in engine flow analysis and corresponds to the behavior during one cycle at a chosen Revolutions Per Minute (RPM). Because engines runs at RPM in the magnitude of $10^3$, fractions of seconds at constant speed is enough for the system to converge to a periodically steady behavior. Therefor, periodic solutions are suitable for engine calculations. The model allows different RPMs, injection pressures ect. Developers working with engine simulations or testing have experience of expected effects when changing a certain parameter. One can thus evaluate the simulation by its ability to capture known effects in tests.

2.5 Coupling Conditions

In the GT Suite coupling manual for Star-CCM+[2], There is a rather comprehensive instruction for running GT-Power in conjunction with 3D codes. A Separate interface surface is created in the GUI that will be used for communicating the updated boundary conditions in each time step. Either coupling is implemented in one direction through the interface surface or through both directions. In one direction coupling, information is exclusively transported along the flow direction, from upstream to downstream. This can be seen analogous to a structure-fluid coupling where the vibrations of the structure causes pressure fluctuations in the fluid but the pressure fluctuations have negligible effects on the structure. The described phenomena is often modeled by a single way coupling. Figure 3 below illustrates a one way coupling from an upstream 3D domain to a
1D model.

Figure 3: One direction coupling.

The quantities $Q$ and $p$ represent how flow rate and/or pressure information is transferred. Compare this with Figure 4 which represents two-direction 1D-3D flow coupling.

Figure 4: Two direction coupling.

The arrows in Figure 4 visualizes how information is transferred in each time step with help of the interface surface. Advantages with a two-way coupling is how the 3D manifold replacement also can affect the 1D model upstream. In general, one must assume that the downstream conditions of subsonic flow affects the upstream conditions[4]. One way coupling fails to account for this effects which might be considerable for some flows. The downside with two way coupling is a more unsteady system and reaching convergence has shown more troublesome.

A solution to common oscillations in coupled simulations are to use the uncoupled 1D solution as an initial condition to the co-simulation. The GT-Power model has this option as a standard procedure in co-simulations. Before information is exchanged with the CFD replacement 3D manifold, a number of engine cycles are simulated in the original 1D system. When a converged solution is found, this solution is saved as a initial condition for the co-simulations. The final solution is not expected to be drastically different from the initial condition. This removes the risk of divergence due to bad initialization.

A rule of thumb in 1D-3D co-simulation is to place the interface surfaces at locations where the flow is steady and well described by the 1D model[2]. In other words, limited directional differences between the interface surface and the velocity profile. The requirements are not unique for 1D-3D coupling methodology and is to some extent also appropriate in regular 3D CFD. To set fixed boundary conditions where the flow has spatial gradients or vorticies can remove these important effects. In co-simulation it can also bring numerical deficiencies when sampling 3D
domain averages to the 1D domain. An illustrative figure of a ill and a well placed boundary is presented below.

Figure 5: Velocity profiles before and after a pipe widening.

The left side velocity profile in Figure 5 represents a suitable location for a average boundary condition. The right velocity profile in Figure 5 is a perfect example where average boundary conditions should be avoided.

### 2.5.1 Gas species communication

Numerous gas species are participating in engine combustion, not the atmospheric composition alone. Burning diesel in air produces a mixture of mainly \( \text{CO}_2 \), \( \text{H}_2\text{O} \) and \( \text{O}_2 \). There are also small fractions of \( \text{N}_2 \), \( \text{Ar} \), \( \text{CO} \) and \( \text{NO} \) in the gas. The manifold is placed upstream of the cylinders. Despite that, there are moments when the fluid is moving from the cylinders back into the manifold. Typically after ignition when pressure rises. The described phenomena suggests for resolving the gas mixture inside the 3D manifold domain. Mixing fresh air with burned air influences the gas composition. Parameters such as oxygen fraction are of interest in engine performance. Some modern engines have an EGR pipe installed. A pipe that connects the exhaust back into the manifold. Considerable amounts of the exhaust gases recirculates back into the cylinders through the manifold. The mass fraction of burned gases are especially high for those engines. In such engines it is particularity important to account for mixing effects in the manifold domain. A 1D/3D coupling methodology can be argued as most beneficial for engine manifolds with a complex EGR gas mixing.

The first mentioned species \( \text{CO}_2 \), \( \text{H}_2\text{O} \) and \( \text{O}_2 \) are modeled in STAR as a multi-gas mixture along with the incoming ambient fresh air. These gases are not only tracked but also assumed to affect the flow with its properties. The small fractions of \( \text{N}_2 \), \( \text{Ar} \), \( \text{CO} \) and \( \text{NO} \) in GT are coupled as passive scalars. This means that the gas fractions are tracked within the CFD although the species are assumed to have no influence on the flow behavior. This is a reasonable assumption for these species with a mass fraction well below 1%.
3 Methodology

3.1 GT-Power Engine Model

Scania has provided a GT-Power engine model from a DC13-155-500 engine. 13 represents the total cylinder volume in liters and 500 is the power peak in horsepowers. The simulation model is used to make predictions of the engine during development. A new configuration or part of the engine can be tested virtually. A response such as in the power or in the exhaust gas composition can sometimes be captured without the need of producing the system in reality. Some details of the models are confidential and will not be reported to the reader. Fortunately, the engine overview is not confidential along with the aspects involving the flow system treated in this thesis. The figure below shows the engine model and marks the manifold that will be replaced with a 3D geometry.

![Figure 6: GT-Power model with markings for 3D replacement.](image)

Note how the manifold body is modeled as flow splits placed horizontally in the marked area. The splits are visualized as v-shapes. Only volumes, ambient temperatures and friction coefficients are given to the manifold parts. The 1D model cannot capture all flow phenomena occurring inside the manifold with only these features. Details such as outlet geometry and outlet placement along the manifold are important features for mitigating pressure losses and establishing an evenly distributed air flow to all cylinders[24]. The break valve and the baffle surface in figure 12 are examples of details that are not captured in the 1D manifold model.
Time step in engine simulations are often expressed in crank angle degrees, a consequence of the periodicity of engine strokes. The corresponding time step changes with the engine rpm. With 1900 rpm, that results in a time step of \(8.77 \times 10^{-5}\) s. Heat transfer in the 1D simulations is calculated with the heat transfer coefficient \(\lambda\) for air and the thermal conductivity \(k\) of aluminum. The manifold surface \(\Delta x\) is 4 mm thick. A simple expression for the heat transferred through the manifold surface per unit area is:

\[
q = \left( \frac{1}{\lambda_{\text{ambfluid}}} + \frac{1}{\lambda_{\text{intfluid}}} + \frac{1}{k} \right) \Delta T
\]

(19)

Effects such as radiation are neglected in relation (17) which is based on basic 1D multi-layer convection theory.

As mentioned earlier, GT-Power is used to calculate the engine system behavior. One interesting feature that will be compared between the original system and the coupled system are the temperatures inside the cylinders. The temperature changes inside the cylinder during an engine cycle can be summarized as:

1. The relatively cold air enters the cylinder from the manifold.
2. The temperature increases when the cylinder compresses the air.
3. Fuel are injected and ignited at the minimum fluid volume. The temperature increases rapidly.
4. The gas expands and temperature decreases.
5. In this 4-stroke engine, a second cycle is made to remove the hot burned gases from the cylinder.

The interested reader can find more information about the Otto-cycle in the article [19]. The exact expression for how the temperature is calculated in GT-Power is not given to the public. It should depend on quantities such as the temperature of the incoming air from the manifold and the amount of fuel and air inside the cylinder at the ignition.

The manifold outlet pipes are named IM11, IM12, ..., IM62 in figure 6, where coupling intersections are outlined with a red line. The flow pipes connecting the manifold to the cylinders are made 7 cm shorter in the 1D model. The initial 7 cm of the manifold outlet pipes are instead included in the 3D manifold. These are included in the 3D domain to avoid coupling in the direct vicinity of the manifold, where the flow is unsteady. The 3D pipe parts are visualized in the upcoming section.

Several engine cycles are simulated in GT-POWER to establish a converged periodic solution. Typically, convergence is determined by several observables within a GT-Power engine model. Examples are engine power and fuel-air ratio. These are commonly used for this purpose. For the manifold co-simulation, additional convergence criteria are added for pressure and mass flow at the manifold inlet and outlets. The additional criteria are set to ensure convergence in the quantities used for
the coupling. It is important to have uncoupled 1D simulation results as an initialization for the co-simulations. A good initialization reduces the amount of cycles that have to be 1D/3D co-simulated. Figure 7 below shows the 1D convergence study for the initial solution. The simulation took 30 minutes for 87 engine cycles with the original 1D model. In the co-simulations, only 3 cycles required 20 hours on the same computer. A great deal of time is saved with the initial solution instead of running all cycles in co-simulations.

Figure 7: 1D model pressure convergence study. Illustrates the pressure at the manifold boundaries in the end of each engine cycle. A total of 87 engine cycles are simulated.

The maximum relative changes allowed are set in the range $0.5 - 2\%$ for all quantities. Convergence is obtained when the set criteria are fulfilled in 7 consecutive engine cycles. This early study indicates that at least 80 cycles are required for a satisfying 1D initialization. This value is important to remember for the upcoming co-simulation.

### 3.2 3D Manifold

The geometry is provided directly from the design team at Scania and is identical to the one used for manufacturing. Figure 8 represents an illustration of the geometry.
Figure 8: Truck engine manifold. Two outlets for each cylinder.

The gray section of the geometry represents what is normally considered the manifold. Additional pipes are included in the 3D domain to ensure a homogeneous flow at the coupling cross sections. Pipes are added both on the inlet side and on the outlet side of the manifold. These additional pipe sections are displayed in pink in Figure 8. The pink inlet system on the left part of figure 8 is considered too curved for a coupling intersection. This system must therefore also be included in the 3D domain. Upstream of the pink inlet system, there is a straight pipe where the flow can be considered homogeneous. The pipe that connects the compressor inter-cooler with the manifold inlet and is refereed to as Bellow-3 in figure 6.

The following figure shows the velocity magnitude and velocity vectors at the extended manifold outlets.
Figure 9: Velocity magnitude and velocity vectors at open outlets. The two outlets closest to the middle are open while the other two are closed. Green areas in the scalar field represent lower velocity. The two outlets closest to the middle are open.

CFD simulations of the manifold shows that there are small flow recirculations reaching to the middle of the outlet pipes. One can see the boundary layer detachment from the reduced velocities on the left side of the open outlets.

Information between the softwares are updated through a common interface surface. When data are collected from the CFD, an average is calculated from all cells inside a small volume. The figure 10 below visualizes the volumes used to produce the average data to the 1D model.

Figure 10: Outlet sampling volumes. The colors indicate 1D sampling volumes in the mesh.
The sampling volumes are 25-mm thick and are located outside of the \( \approx 4\text{cm} \) outlet recirculation distance. A homogeneous outlet flow is necessary in the sampling volume. The reason to extend the 3D outlet pipes with 7-cm is to avoid sampling sampling volumes in the recirculation area. The thermal CFD modeling will be set equal to expression (17) in the previous section to match the 1D model. In STAR-CCM+, surface roughness effects are captured by a modification in the log law described in the Background chapter. A roughness parameter is calculated from the surface roughness height. The roughness height is a material parameter for the cast iron used for producing the manifold and can easily be modified. More details about the roughness model can be found in the reference [5].

### 3.2.1 Uncoupled boundary conditions

The 3D CFD manifold replacement is evaluated by its sensitivity to variations in surface roughness, turbulence models and variations in the mesh size. If a number of manifold CFD setups gives similar results, we have an increased confidence in the manifold CFD. A table containing the various simulation setups is found in the upcoming results section. If manifold experiments were available, that would be a more convenient way to validate the CFD results.

Sensitivity studies for the CFD are developed in uncoupled CFD simulations. Uncoupled simulations run faster and are easier to set up than co-simulations. In the uncoupled CFD simulations, the boundary conditions are obtained from a table of instantaneous values in one engine cycle. Mass flow through the manifold is saved in the original GT-Power system with a 1D manifold. These values are set to the inlet and all the outlets in the manifold CFD. The corresponding pressure from the manifold CFD is compared to the pressure with the 1D manifold. The difference in pressure between the manifold models are interesting. It reflects the differences in the uncoupled and the coupled system.

Heat transfer values for the CFD boundaries are set from the 1D simulations according to relation (17). The time step in the CFD domain is set to match the time step in the 1D engine model with \( \Delta t = 8.77\text{E-5[s]} \).

### 3.3 Mesh

The manifold geometry is fortunately seemingly flat. There are few sharp edges and no rotating parts which usually is a difficult task for the automated mesh generator in Star-ccm+. Automated meshing in Star-ccm+ starts with setting a default grid size. This size is called the base size. Local refinements are later made with element sizes relative to the base size. Relative sizes are given in percentages of the base size. Altering only one parameter will therefore change the element size of the entire mesh. A total of 5 prismatic cell layers is placed along all boundaries of the manifold. The total thickness for the prism layer are 20% of the regular
element base size, to accurately deal with the viscous boundary layer at the walls. Figure 11 below displays the mesh in a plane view inside the manifold.

![Figure 11: Mesh view with a base size of 6 mm. Prism layers are shown along the boundaries.](image)

One might also notice how the automated mesh is finer close to boundaries or in tight areas such as the outlets displayed in the lower section of figure 11. This automatic feature is heavily appreciated and allows for less elements with a strategic placement. Additional, manual refinements are made on the valve inside the inlet and in the baffle due to the complex geometry of these parts.

![Figure 12: Refined mesh scene at valve and baffle surfaces](image)

Figure 12 displays a 3D view of the surface mesh at the valve and baffle
parts, both placed in the pipe section before the manifold. One can clearly see the refined mesh on the surfaces. The relative surface element length for the valve and baffle are set to 15% and 20% of the base size. Smaller elements on the surfaces also make the surrounding elements smaller. This allows for the calculations to better capture the wake effects in these areas.

### 3.3.1 Mesh independence study

A mesh independence study was carried out to determine the base size required in the 3D domain. The idea is to gradually lower the grid size thus gradually increasing the number of cells. Meanwhile, monitoring the pressure for each mesh. When an increase of the amount cells does not change the manifold pressure in a simulation, the simulation is considered mesh independent[30]. There is no value in increasing the number of cells further.

![Mesh independence study](image)

*Figure 13: Mesh independence study on the average outlet pressures. All boundary conditions are set to mass flow, taken from the last engine cycle in the uncoupled 1D model. The resulting CFD average outlet pressure is collected. All pressures are in relation to the reference pressure. The total outlet pressure is \( \approx 2.7 \text{ bar} \).*

Three full cycles are simulated. The pressure is gathered as an average during the last engine cycle in the uncoupled 3D simulation. As described earlier, outlet pressure is considered the important feature for the manifold CFD simulation. Pressure is the quantity sent to the 1D model during the coupling. Pressures are presented as the difference compared to the 1D manifold average pressures. The total pressure in the manifold is in the neighborhood of 2.7 [bar]. Figure 13 show that increasing the amount of cells above \( \approx 10^6 \) will have a weak effect on the pressure difference. The suitable base size of 10 [mm] is chosen. It corresponds to mesh with approximately 1.2 million cells. Figure 15 is presented below as a complement to Figure 13. Figure 15 presents the relation of the number
of cells in the mesh to the base size.

Figure 14: Grid size to cell amount relation.

In other words, figure 15 translates the base size to the corresponding cell amount. Both can be handy features. The element size can be related to the numerical accuracy whilst the total cell amount is related to the computational effort[14]. Figure 15 motivates why no less than 10 [mm] were chosen as a suitable base size. Smaller values drastically increases the cells and figure 13 shows that no less is needed. Simulations are practical and fast with 10 [mm] on the 32 core in-house server system. The most demanding simulation is a LES that required 20 hours in wall clock time.
4 Results

As for the mesh independence, effects of changing turbulence models and surface roughness heights are analyzed in uncoupled 3D simulations. Major effects in the uncoupled 3D simulations have corresponding effects in the co-simulations. A similar argument can be made for minor effects in the uncoupled simulations. These are expected as minor in co-simulations as well. The $O_2$-fraction in the fluid that flows into the cylinders is also presented. Comparisons of the Co-simulations with the original pure 1D model will be presented in the following section. We first present the co-simulations for all three coupled cycles to motivate for periodic convergence. Then moving on with last cycle results in the manifold. Manifold flow quantities and some relevant quantities in engine simulations are considered. Differences in mass flow rate, outlet pressure, pressure drop, simulation run-times and cylinder temperatures due to the manifold replacement are investigated. The maximum combustion temperature is the key feature for generation of $NO_x$ and other unwanted products in the combustion process[21].

Instead of presenting the results in terms of manifold outlet 1,...,outlet 12, we will use the cylinder number and port number for the outlets. Each cylinder has 2 ports or inlets which corresponds to the manifold outlets. Cylinders and ports are more familiar to people with experience in engines. Outlet 1 corresponds to port 1 for the first cylinder and outlet 12 corresponds to the second port for the sixth cylinder. The figure below shows the translation from the outlet number to the cylinder inlets.

![Figure 15: Translation from manifold outlets to corresponding cylinder and inlet port number.](image)

Many of the coupled/uncoupled system comparisons are made with quantities inside the cylinders. This makes the conversion from the manifold outlet number to the corresponding cylinder inlet appropriate.
4.1 Turbulence model

As before for grid convergence, turbulence models will be evaluated in terms of outlet pressure values. The $\epsilon - k$ model is regarded as the default model in this thesis. Comparisons are made to 3 other models counting the LES as a model. LES, RSM and a Menter’s Shear Stress Transport model (SST) are used to evaluate the sensibility in turbulence modeling for the CFD. SST is a hybrid two equation model, using $\epsilon - K$ in free flow areas and the $K - \omega$ model close to boundaries. All mentioned models are regarded as superior compared to the default $\epsilon - K$. Table 1 below contains pressure values for one inlet per cylinder and for each turbulence model.

<table>
<thead>
<tr>
<th>Cylinder</th>
<th>Port</th>
<th>Pressure [kPa]</th>
<th>$K - \epsilon$</th>
<th>SST</th>
<th>RSM</th>
<th>LES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>5.107</td>
<td>5.093</td>
<td>5.229</td>
<td>4.881</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>5.039</td>
<td>5.028</td>
<td>5.118</td>
<td>4.979</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>5.108</td>
<td>5.149</td>
<td>5.149</td>
<td>5.030</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>5.049</td>
<td>5.098</td>
<td>5.120</td>
<td>5.077</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>5.094</td>
<td>5.088</td>
<td>5.140</td>
<td>5.093</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>5.183</td>
<td>5.177</td>
<td>5.247</td>
<td>5.234</td>
<td></td>
</tr>
</tbody>
</table>

Table 1: Average manifold pressure for various turbulence models. The pressure is presented as the difference to the reference pressure. The reference pressure is set as the average pressure in the original simulations with a 1D manifold. A smooth surface is used for all the turbulence models.

4.2 Surface roughness

Effects of surface roughness are also investigated. The manifold is made of cast iron with exception for the baffle which is always considered smooth. The baffle part is made of a fine plastic material. Roughness is calculated with a smooth grain size of 0.04[mm] according to the roughness model mentioned earlier. Also values of 0.28[mm] and a rough surface with grain size 0.56[mm] are tested. Cast iron Table 2 below contains pressure values for one inlet per cylinder and for each roughness height.
<table>
<thead>
<tr>
<th>Cylinder 1 Port 1</th>
<th>Smooth</th>
<th>0.28[mm]</th>
<th>0.56[mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.107</td>
<td>5.097</td>
<td>4.904</td>
<td></td>
</tr>
<tr>
<td>5.040</td>
<td>5.031</td>
<td>4.885</td>
<td></td>
</tr>
<tr>
<td>5.108</td>
<td>5.090</td>
<td>4.951</td>
<td></td>
</tr>
<tr>
<td>5.049</td>
<td>5.012</td>
<td>4.922</td>
<td></td>
</tr>
<tr>
<td>5.094</td>
<td>5.063</td>
<td>4.997</td>
<td></td>
</tr>
<tr>
<td>5.183</td>
<td>5.159</td>
<td>5.116</td>
<td></td>
</tr>
</tbody>
</table>

Table 2: Average outlet pressure for various surface roughnesses. The pressure is presented as the difference to the reference pressure. The reference pressure is set as the average outlet pressure in original simulations with a 1D manifold. The $\epsilon - k$ model is used for all roughness heights.

4.3 co-simulation comparisons

Figure 16 shows the pressure in all three co-simulated engine cycles to investigate the periodic convergence. Figures 16-20 presents values for the last cycle. These figures compares flow quantities inside the manifold along with some interesting features in engine simulations. The original simulation with a 1D manifold model is compared to the co-simulation with a sophisticated CFD manifold.

![Figure 16: Pressure at the first manifold outlet during all three co-simulation engine cycles.](image)
Figure 17: Average fraction of O\textsubscript{2} molecules in the fluid. The fraction is collected at the manifold outlets in co-simulations. The value represents the average for all manifold outlets.

Figure 18: Mass flow through cylinder 1, port 1 and Cylinder 6, port 2 during one cycle. Data is collected from the last engine cycle, both in co-simulations and in the uncoupled 1D simulations.
Figure 19: Temperature in cylinders 1 and 6. The temperature peaks correspond to ignition inside the cylinders. Ignition point during the cycles varies for all cylinders. The cylinder temperatures are calculated in the GT-POWER model both with the 1D manifold and in co-simulations.

Figure 20: Close view at temperature peaks in Figure 19.
Figure 21: Manifold inlet mass flow. Last simulated engine cycle.

Table 3: Comparison for average pressure drop and average inlet mass flow rate from the last simulated cycle. Uncoupled 3D operates with 1D model mass flow values as BCs. Consequently, mass flow rates become identical in these cases. Run-times are included.
5 Analysis

Sensitivity studies were made on the CFD mesh and on the physics in terms of turbulence models and surface roughness. Glance at the Tables 1, 2 and figure 13. According to the Tables 1 and 2, the 3D simulation is nearly unaffected by the choice of turbulence model or surface roughness height. A rough surface decreases the average outlet pressure. The outlet pressures decreases for approximately 200[Pa] or 2[mbar] with the roughest surface compared to the smoothest surface. The largest deviation from the $K - \epsilon$ in Table 1 is for outlet 1 with the LES. The difference is close to 2[mbar]. That is a small number compared to the 50[mbar] difference between the coupled 1D/3D simulation and the original 1D simulation. It was earlier concluded that there are only minor changes in the pressure difference with more than $\approx 1.2E6$ cells. Various approaches leading to similar results generates a higher confidence in the default CFD model used for the co-simulations. Differences are within a few percentages for all outlets and models. It is unlikely that the small variations in pressure within these tables would drastically change the co-simulation system. Especially with the slight effects that were found in co-simulations compared to uncoupled 1D simulations.

Figure 16 visualizes how the pressure varies at the first outlet during all 3 co-simulated cycles. It is included only to validate co-simulation convergence. Some variations are visible when comparing cycle 1 and cycle 2 in Figure 16, especially at the earliest stages of the cycles. There are however no significant changes from cycle 2 to cycle 3. The simulation is at this stage considered converged enough.

Ambient air is entering the intake system with atmospheric conditions. The fraction of oxygen molecules is set to 23.3%. The $O_2$ fraction from co-simulations are found in Figure 17. A sudden fall of oxygen occurs at the opening a cylinder as explained earlier. Deviations from ambient levels are considered negligible throughout the entire engine cycle. Deviations never exceed one percentage.

Acknowledge cylinder 1, port 1 in Figure 18. Coupled and uncoupled results are very similar. There are evidently less oscillations for the coupled simulation when the outlet is closed. Even though such similar results appear uninteresting, they indicate that the coupling methodology is correctly implemented. There are no indications of major errors in the implementation or signs of leakages at the manifold coupling sections. Mass flow at cylinder 6, Port 2 shows slight deviations when the manifold outlet valve is open. The coupled simulation suggests more mass flow. Cylinder 6, port 2 is placed at the end of the manifold, while cylinder 1, port 1 is the first manifold outlet. Larger deviations at the last outlet are expected. The fluid has traveled significantly longer within the 3D domain giving more time for differences in the models to affect the flow properties.

Cylinder temperatures are calculated by the GT-Power model. The cylin-
der temperatures for both systems are presented in the Figure 19. No
significant differences can be seen for cylinder 1 and minor temperature
differences are found for cylinder 6. The minor coupled temperature in-
crease in cylinder 6 can be explained by the increased flow rate to the
cylinder seen previously. The temperature curves split at ignition after
air has filled the cylinder. The amount of air entering cylinder 6 before
ignition differs between the coupled and uncoupled simulations which antici-
patizes changes in the combustion temperature as well. The maximum
cylinder temperature at ignition is found in figure 20. The maximum
temperature is increased 15°C in cylinder 1 while increased more than
40°C in cylinder 6 when running co-simulations. Differences in the sixth
cylinder have a potential to influence $NO_x$ generated inside. [21].

Much emphasis have been put to the outlets since the mass flow into the
cylinders has direct effects on the engine performance. Figure 21 shows
the inlet mass flow rate instead. There are greater changes on the inlet
side of the manifold. The coupled simulation predicts smaller variations
in the mass flow during the cycle. We can see in Figure 21, that the red
curve has smaller differences between the highest and lowest values in the
cycle. On average, values are not dissimilar at all. The average mass flow
can be found in Table 3.

Predictions for the manifold pressure drop between coupled model and
the original 1D model are investigated. Pressure drop values are found
in Table 3. Pressure drop predictions for a individual component is not a
suitable task for the 1D model. Comparisons of pressure drop are made
with uncoupled 3D CFD simulations. CFD is often used as the industrial
tool when no experimental data is available [3][12]. The uncoupled and
coupled CFD generate no significant differences. 3.95[kPa] compared to
4.05[kPa] in the standard settings of roughness and a $k - \epsilon$ turbulence
model.

In terms of implementation, co-simulations with GT-Power and Star-
ccm+ are time consuming for the user. It is much easier to run a 1D
engine model and acquire boundary data to a separate CFD manifold sim-
ulation. Unless 1D/CFD coupling is required for a prediction, it should be
avoided. Additional settings are required to correctly set up communica-
tion between both software. With even further discomfort, the potential
errors are increased in coupled simulations. Error messages in the host
software STAR-CCM are not created for co-simulations. When a feature
is unobtainable to the receiving program, an error message is generated
by that program. It is then difficult for the user to determine whether
the error lies in generating or loading the specific feature. In turn, that
might force error search within both software. User knowledge in both
programs are a prerequisite to developing a coupled method. The compet-
ing methodology suggested in this thesis does not. One part, responsible
for the 1D engine model can request an uncoupled manifold CFD analy-
ysis from another part without detailed knowledge in any CFD software.
The same can be argued for how the CFD part need limited experience
in the 1D software. User effort along with insignificant differences in
results are the reasons to avoid implementation of a 1D/3D manifold coupling methodology as a optimization toolkit for engines similar to this DC13-155-500. Even though 1D/CFD coupling appear wasteful for the investigated system, it is not argued as a poor method. Successful examples were presented in the beginning. 3D Catalyst simulations with 1D models for monolith channels [27] and exhaust sound wave co-simulations by Tatiana et.al [7] are two examples.

5.1 Conclusions

Even though no experimental data was available, a judgment could be made for the usability of the method on the investigated engine model. Differences between the coupled system and the original system were slim, regardless of were the reality lies. The only potentially meaningful difference is the maximum temperature in cylinder 6. A satisfying discovery in regard for the thesis. With larger differences, experiments would be a natural continuation.

The simulation time is increased from 30 minutes to 16 hours when the manifold was replaced. This is a major increase that makes the system unable to run optimizations for engine development within a reasonable time. In the optimization process, a large number of setups are simulated to find the best configuration for a specific engine part. Predictions must therefor run fast for each of the setups.

With small differences in the predictions and the heavy increase in the computational cost, we conclude that the investigated method is not suitable for the investigated engine model. It would be interesting to see how results change with a EGR installed on the same engine. An EGR is a pipe that redirects a part of the exhaust gases back into the inlet manifold. A EGR further complicates mixing effects in the inlet manifold. There are also inlet manifolds with a more complex geometry, where the method might prove necessary.
References


