MARMARA UNIVERSITY
F A C U L T Y O F E N G I N E E R I N G

1D AND 3D ENGINE MODELING, OPTIMIZATION AND ANALYSIS OF ONE CYLINDER DIESEL ENGINE

GÜLENAY KARAYAPRAK-150412012

CEM İLTER-150412025

ENES ERKAN-150412027

MUSTAFA KARA-150412006

GRADUATION PROJECT REPORT

Department of Mechanical Engineering

Supervisor

Asist.Prof. Mustafa Yılmaz

ISTANBUL, 2014

CONTENTS

ABSTRACT

Nowadays internal combustion engines development is widely supported by 1D and 3D codes. On the other hand, an extensive experimental activity at test bench involves increased development cost and time-to-market of a new engine. For this reason, a numerical methodology capable to provide a reliable estimation of engine performance starting from a reduced set of measured data represents a very promising approach.

In this paper, a hierarchical 1D/3D modeling procedure is proposed with reference to a compression ignition engine to predict its performance even in absence of experimental data.

For this purpose, we evaluate main softwares for 1D and 3D modeling seperatly. On the other hand we focused on GT Power for 1D and Ansys Forte for 3D anlysis.

1. ENGINE MODELING

In science and engineering, may be generally regarded as the process of describing the physical phenomena in a particular system with the help of mathematical equations (subject to “reasonable” assumptions) and solving the same to understand more about the nature of such phenomena. Usually, engineering models help in designing better devices by understanding more about the fundamental physical processes occurring therein. Engine modeling activities, at least in recent decades, have largely been concentrated in the direction of designing better performing engines with lower emissions. In this regard, modeling of engine combustion processes assumes importance.

The various engine combustion models that have been developed to date may be grouped into two categories:

1. Zero dimensional models

2. Three dimensional models

In the above classification, although the level of detail and proximity to physical reality increases as one proceeds downward, so does the complexity of creating and using those models.

1.1.The Zero Dimensional Model

When the control volume(s) is taken as the combustion chamber in the integral formulation, we have a zero dimensional model. This type of model is commonly referred to as a thermodynamic model or a phenomenology model. Most properties are averaged over the total cylinder volume and no spatial information is available. A survey of the thermodynamics models are presented by Blumberg, Lavoie and Tabaczynski, Mattavi and Heywood among others. These models rely on some understanding of the physics involved and try to capture the main features of the processes, by including the description of the most important aspects the models have performed surprisingly well and are ideally suited for parametric studies

Models in use of this type are implemented as multi-zone models in the recognition of the spatial distribution of properties. During combustion the charge in each control volume is divided into two zones one with unburned gases are assumed and one with burned gases. Before and after the combustion period the gases are assumed to be uniform over the control volume. The simplest model of this type has one control volume as the whole combustion chamber, with at most two zone

present. Several different models have been proposed and they show reasonable agreement with experimental data, when compared with overall measurements such as the pressure. Extending the model to contain two control volumes, the mean flow can be calculated as a non-solid body rotation with the possibility of doing the same for the turbulence field. Other zones can be added for more realistic computation of the wall heat transfer rates or a configuration with pre-chamber.

1.1.1.Two zone Model

In order to perform parametric simulation studies on engine combustion, a simple thermodynamic model, which incorporates two zones, viz. an unburned zone and a burned zone is desirable. Naturally, it is important to understand the suitability (or lack thereof) of the two zone model to predict heat release rates in diesel and spark-ignited engine combustion. Whereas in spark-ignited engines relatively well-defined (continuous) flame propagation might occur, diesel engine combustion might consist of several distinct burned and unburned regions scattered throughout the cylinder. Despite the differences in the nature of combustion, the two zone model would still be applicable for both diesel and spark-ignited engine combustion since it is basically a zero dimensional model that considers only unburned and burned zones without any consideration of the spatial location of such zones. The basic idea behind the two-zone model is to utilize the conservation of mass and energy (First Law of Thermodynamics) and also the ideal gas equations in obtaining an "apparent" rate of heat release curve or "apparent" mass burned fraction curve. The adjective "apparent" should be stressed in order to realize the fact that whatever be the heat release or mass burned fraction obtained indirectly from measuring in-cylinder pressure, its accuracy is limited by both the assumptions of the model as well as the accuracy of the pressure data.

The conservation of mass inside the cylinder is;

$$m + m_u = m_b \quad (1)$$

where;

m : Total mass of charge inside the cylinder (m= m_{fuel} + m_{air})

m_u : Mass of unburned charge

m_b: Mass of burned charge

Also, since the mass inside the cylinder is assumed to be constant in any given engine cycle;

$$\dot{m_b} = -\dot{m}_u \quad (2)$$

Since the unburned and burned zones together constitute the total cylinder charge volume at any instant, we have;

$$V_u + V_b = V \quad (3)$$

where;

V : Total volume of charge inside the cylinder

V_u : Volume of unburned charge

V_b: Volume of burned charge

The ideal gas equations for the unburned and burned zones are

$$P \times V_u = m_u \times R_u \times T_u \quad (4)$$

$$P \times V_b = m_b \times R_u \times T_b \quad (5)$$

Now, considering the above expression for the unburned zone in terms of the respective rates of the energy terms with crank angle;

$$\frac{\delta Q}{dt} - \frac{\delta W}{dt} + \sum_{in} \dot{m} \times h - \sum_{out} \dot{m} \times h = \frac{dEcv}{dt} \quad (6)$$

Now, considering the above expression for the unburned zone in terms of the respective rates of the energy terms with crank angle;

$$-\dot{Q}_{ht,u} - \left(P \times \dot{V}_b\right) - \dot{m}_u \times h_u = \dot{m}_u \times u_v \quad (7)$$

Similarly, applying the first law (Equation 5) for the burned zone and remembering that the total energy input into the burned zone is due to the chemical energy (released by the fuel when it burns), we get;

$$\dot{Q}_{ch} - \dot{Q}_{ht,b} - P \times \dot{V}_b - \dot{m}_u \times h_u = \dot{m}_b \times u_b \quad (8)$$

The governing equations for the two zone model are Equations (10-14) and (16-17). These equations are solved simultaneously each time (or crank angle) step to determine the unknown quantities T_u, T_b, V_u, V_b,, m_u, m_b, and heat release rate from the known cylinder pressure P and total volume V.

1.1.2.Single zone Model

Applying the First Law of Thermodynamics to the system (single zone);

$$\frac{\delta Q_{ch}}{d\theta} - \frac{\delta Q_{ht}}{d\theta} - \frac{\delta W}{d\theta} + \frac{dm_f}{d\theta} \times h_f = \frac{dU}{d\theta} \quad (9)$$

where:

$\frac{\delta Q_{ch}}{d\theta}$ = Apparent rate of chemical energy (or heat) release

$\frac{\delta Q_{ht}}{d\theta}$ = Rate of heat transfer out of the system

$\frac{\delta W}{d\theta}$ = Rate of work transfer out of the system

$\frac{dm_f}{d\theta} \times h_f$ = Rate of enthalpy inflow with the fuel

$\frac{dU}{d\theta}$ = Rate of change of internal energy of the system

All the energy rates are expressed with respect to the crank angle θ.

The sum of the work transfer term and the internal energy change of the working fluid term may be expressed in terms of P and V alone as follows:

$$\frac{\delta W}{d\theta} + \frac{dU}{d\theta} = P \times \frac{dV}{d\theta} + \frac{d(m \times c_v \times T)}{d\theta} \quad (10)$$

where;

m : Total mass of in-cylinder gases (the system) in kg

c_v : Specific heat of the working fluid at constant volume in kJ/kg. K

From the ideal gas equation;

$$P \times V = m \times R \times T \quad (11)$$

where;

R: Characteristic gas constant of the cylinder gases in kJ/kg. K

Upon logarithmic differentiation, the above equation becomes

$$\frac{dP}{P} + \frac{dV}{V} = \frac{dm}{m} + \frac{dR}{R} + \frac{dT}{T} \quad (12)$$

However, considering the fact that the mass in the control volume remains constant (neglecting crevice losses and quantity of fuel injected) and also assuming that the gas constant remains constant throughout the combustion process, we get;

$$\frac{dP}{P} + \frac{dV}{V} = \frac{dT}{T} \quad (13)$$

Substituting for m (from Equation 11) and $\frac{dT}{T}$ (from Equation 13) in Equation 10, we get;

$$\frac{dW}{d\theta} + \frac{dU}{d\theta} = P \times \frac{dV}{d\theta} + \left(\frac{P \times V}{R}\right) \times c_v \times \frac{\left(\frac{dP}{P} + \frac{dV}{V}\right)}{d\theta} \quad (14)$$

Rearranging and simplifying;

$$\frac{dW}{d\theta} + \frac{dU}{d\theta} = \left(1 + \frac{c_v}{R}\right) \times P \times \frac{dV}{d\theta} + \left(\frac{c_v}{R}\right) \times V \times \frac{dP}{d\theta} \quad (15)$$

Now for an ideal gas;

$$c_p - c_v = R \quad (16)$$

$$\frac{c_p}{c_v} = \gamma \quad (17)$$

where;

C_p: Specific heat of the working fluid at constant pressure in kJ/kg. K

γ: Ratio of specific heats at constant pressure and constant volume

Using Equations (16) and (17) and simplifying, Equation (15) becomes ;

$$\frac{dW}{d\theta} + \frac{dU}{d\theta} = \left(\frac{\gamma}{\gamma-1}\right) \times P \times \frac{dV}{d\theta} + \left(\frac{1}{\gamma-1}\right) \times V \times \frac{dP}{d\theta} \quad (18)$$

Substituting Equation (18) in Equation (9), neglecting the fuel enthalpy term, and rearranging;

$$\frac{\mathrm{dQ}_n}{d\theta} = \frac{\mathrm{dQ}_{ch}}{d\theta} - \frac{\mathrm{dQ}_{ht}}{d\theta} = \left(\frac{\gamma}{\gamma-1}\right) \times P \times \frac{dV}{d\theta} + \left(\frac{1}{\gamma-1}\right) \times V \times \frac{dP}{d\theta} \quad (19)$$

where;

$\frac{\mathrm{dQ}_n}{d\theta}$: Net apparent rate of heat release

1.2.Three Dimensional Modelling

In 3D computer graphics, 3D modeling (or three-dimensional modeling) is the process of developing a mathematical representation of any three-dimensional surface of an object (either inanimate or living) via specialized software. The product is called a 3D model. It can be displayed as a two-dimensional image through a process called 3D rendering or used in a computer simulation of physical phenomena. The model can also be physically created using 3D printing devices.

Models may be created automatically or manually. The manual modeling process of preparing geometric data for 3D computer graphics is similar to plastic arts such as sculpting.

3D modeling software is a class of 3D computer graphics software used to produce 3D models. Individual programs of this class are called modeling applications or modelers.

2.ENGINE MODELING SOFTWARES

2.1.Zero Dimensional Softwares

2.1.1. GT-Power

(1) Instead of testing every operating point on a dynamometer, a computer model of an engine is used to simulate the engine. One program used to simulate an engine is Gamma Technology's GT-Power. Like most engine simulations, the physical dimensions of the complete powertrain system must be entered. The process of creating a GT-Power model begins with dividing the powertrain into its components. The major components of the powertrain are the throttle, the intake manifold, the fuel injectors, the engine, the exhaust system and the catalyst. To model the intake manifold, the

most important aspect is to model all of the pipe bends and flow splits. GT-Power has preset components for straight pipes, bent pipes and flow splits. Each component is defined by several parameters such as discharge coefficients, cross sectional area and lengths.

2.1.1.1.How GT-Power Generates a Solution?

GT-Power is based on one-dimensional gas dynamics which account for fluid flows and heat transfer. Each component in a GT-Power model is discretized or separated in many smaller components. These subcomponents have very small volumes and the fluid's scalar properties in these volumes are assumed to be constant. The scalar properties of a fluid include pressure, temperature, density and internal energy. Each volume also has vector properties that can be transferred across its boundaries. These properties include mass flux and fluid velocity. Figure illustrates the difference between vector and scalar properties. GT-Power determines the change in the scalar properties by solving simultaneous one-dimensional equation.

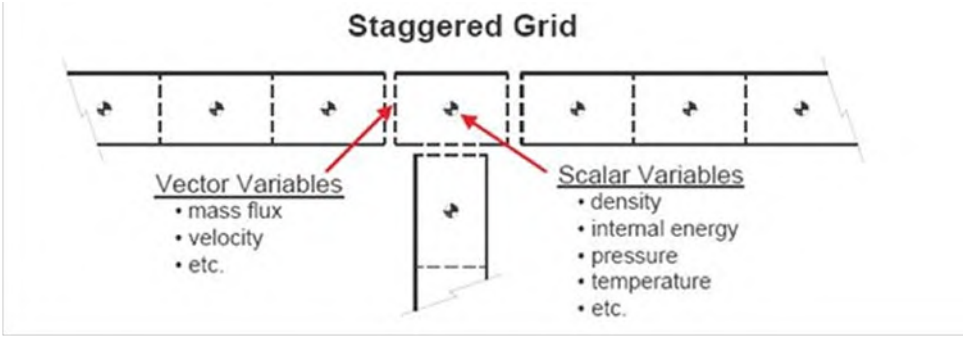

Figure 1 : Illustration of Component Discretization (GT-Power Manual, 2004)

The first equation ensures the conservation of mass as seen in Equation 20. Related to the mass equation is the conservation of momentum relationship as seen in Equation 21. From these two equations the trapped air and residuals masses can be found. The equations are also used to determine the fuel dynamics and mass flow rates. GT-Power also uses Equation 22 which ensures that energy is conserved. The final equation is an exergy balance as shown in Equation 23 which is implicitly solved. Using these equations, the heat transfer from a volume to the walls or another volume can be determined.

Mass Continuity Equation;

$$\frac{dm}{dt} = \sum_{boundaries} m_{flux}$$

Conservation of Momentum Equation;

$$\frac{d(m_{flux})}{dt} = \frac{dpA + \sum_{boundaries}(m_{flux} \times u) - 4c_f \times \frac{\rho \times u^2}{2} \times \frac{d_x A}{D} - c_v \frac{1}{2} \times \rho \times u^2 \times A}{d_x}$$

Conservation of Energy Equation;

$$\frac{d(m_e)}{dt} = \rho \times \frac{dV}{dt} + \sum_{boundaries} m_{flux} \times H - h_g \times A \times (T_{gas} - T_{wall})$$

Implicitly Solved Exergy Equation;

$$\frac{d(\rho HV)}{dt} = \sum_{boundaries} \left(\rho \times u \times A_{eff} \times H\right) - V \times \frac{d\rho}{dt} - A \times (T_{gas} - T_{wall})$$

2.1.1.2.GT-Power Model Creation

GT-Power is an object oriented program with a logical user interface. To create a model, components are placed on a worksheet. Components are connected using lines to mirror the fluid paths. This process is similar to creating a block diagram or Simulink model. Figure 2 shows an example of difference components that are used to create a model and how they are connected. Several parameters must be entered into each component to specifically reflect the physical engine. To define these values, a user must double click on the object and enter the required values in a graphic user interface window. A picture of this window is shown in Figure. The information that must be entered for each component will be described in the next sections.

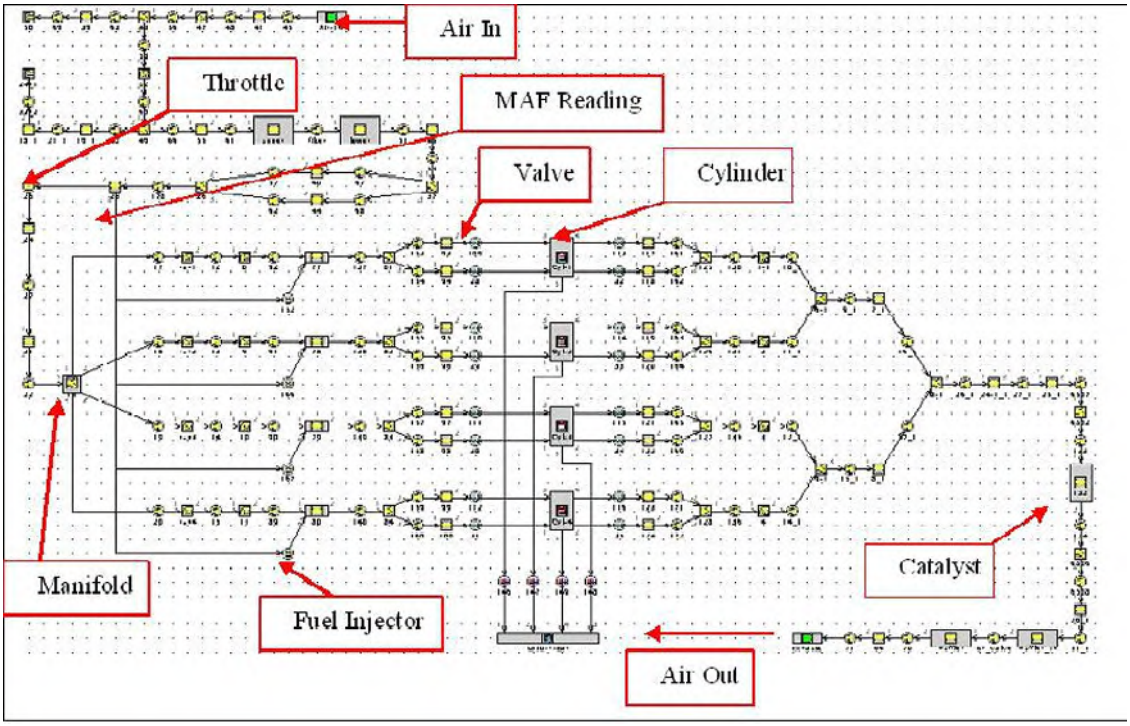

Figure 2: Example of a GT-Power Model

2.1.2.Lotus Simulation

Computer simulation is a key part of the automotive development process. Lotus Engineering Software has been developed by automotive engineers, using them on many powertrain and vehicle projects at Lotus over the past 15 years.

The philosophy under-pinning Lotus Engineering Software is to offer simulation tools which enable the user to generate models very quickly, using a mixture of embedded design criteria and well-structured interface functionality.

2.1.2.1.Lotus Engine Simulation (LES)

An easy-to-use engine cycle simulation tool, developed for Lotus' own engine research and development department.

The tool has been developed from our extensive experience of applying performance simulation to engine design projects. The tool can be used to simulate the performance of two and four stroke, gasoline and diesel, naturally aspirated or supercharged and turbocharged engines.

Lotus engine simulation models the gas dynamics in the engine manifolds and enables the complex operating modes used in modern engines to be simulated.

2.1.3.Matlab Simulink

Simulink, developed by MathWorks, is a graphical programming environment for modeling, simulating and analyzing multi domain dynamic systems. Its primary interface is a graphical block diagramming tool and a customizable set of block libraries. It offers tight integration with the rest of the MATLAB environment and can either drive MATLAB or be scripted from it. Simulink is widely used in automatic control and digital signal processing for multi domain simulation and Model-Based Design. (3)

2.1.4.Fortran

Fortran (formerly FORTRAN, derived from "Formula Translation") is a general purpose, imperative programming language that is especially suited to numeric computation and scientific computing. Originally developed by IBM in the 1950s for scientific and engineering applications, Fortran came to dominate this area of programming early on and has been in continuous use for over half a century in computationally intensive areas such as numerical weather prediction, finite element analysis, computational fluid dynamics, computational physics, crystallography and computational chemistry. It is a popular language for high-performance computing and is used for programs that benchmark and rank the world's fastest supercomputers.

Fortran encompasses a lineage of versions, each of which evolved to add extensions to the language while usually retaining compatibility with prior versions. Successive versions have added support for structured programming and processing of character-based data (Fortran 77) array programming, modular programming and generic programming (Fortran 90), high performance Fortran (Fortran 95), object-oriented programming (Fortran 2003) and concurrent programming (Fortran 2008). (4)

2.1.5.Ricardo Wave

WAVE is the technology leader in engine performance and NVH. This simulation software package is used worldwide in industry sectors including ground transportation, rail, motor sport, marine, and power generation for 1D engine and gas dynamics analysis. WAVE enables performance and acoustic simulations to be carried out based on virtually any intake, combustion and exhaust system configuration, and includes a drivetrain model to allow complete vehicle simulation. (5)

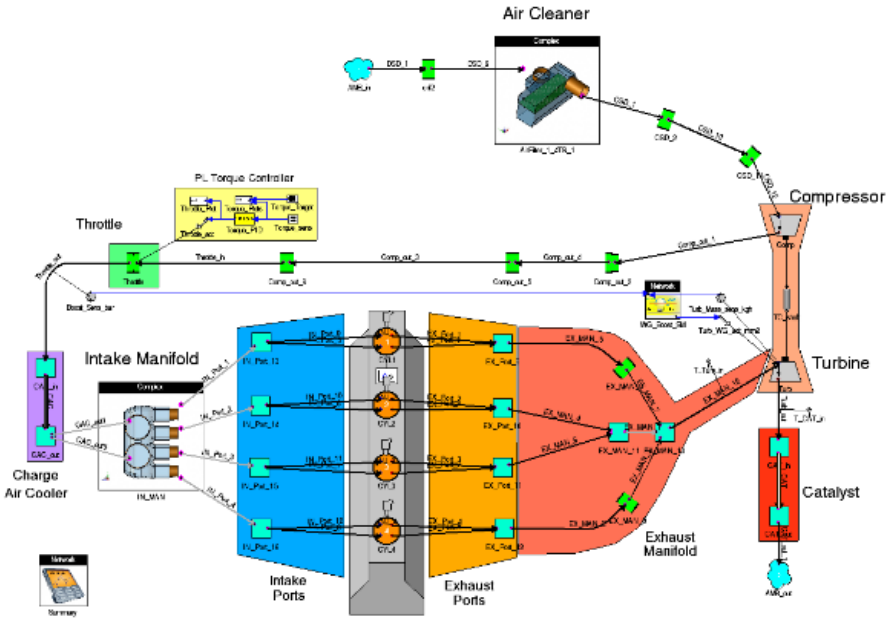

Figure 3: Example of Ricardo Wave Model

2.1.6.AVL Fire

AVL FIRE™ is the leading CFD simulation tool in the field of combustion analysis. It specializes in the accurate prediction of all IC Engine relevant processes including injection nozzle flow, fuel injection, combustion, emission and exhaust gas after treatment. The software also supports the development of electrified powertrains and drivelines. (6)

2.1.6.1.The AVL Solution

The automatic pre-processor is able to efficiently generate grids for geometries with variable volume and/or featuring moving parts such as, pistons, valves, shafts, gears, etc. Automation shifts the otherwise significant effort required for generating a simulation model from the user to the computer.

The solver utilizes a pressure based segregated solution algorithm. It incorporates implicit time discretization as well as accurate conservative and bounded differencing schemes. High-end turbulence models allow for the precise description of transient effects within an engine or on a significantly larger scale.

The integrated postprocessor offers visualization and supports detailed analysis of results online during the simulation and offline interactively or based on marcros and scripts. In-build features allow the fast generation of charts, colored plots and animations in different formats to support the interpretation of both steady and transient phenomena.

2.1.7.Chemkin

Chemkin is a proprietary software tool for solving complex chemical kinetics problems. It is used worldwide in the combustion, chemical processing, microelectronics and automotive industries, and also in atmospheric science. It was originally developed at Sandia National Laboratories and is now developed by a US company, Reaction Design.

Chemkin solves thousands of reaction combinations to develop a comprehensive understanding of a particular process, which might involve multiple chemical species, concentration ranges, and gas temperatures.

Chemical kinetics simulation software allows for a more time-efficient investigation of a potential new process compared to direct laboratory investigation.

One important driver for the development and use of Chemkin is the reduction of pollutants, such as NOx. As these pollutants become more tightly regulated through agreements by agencies such as the United States Environmental Protection Agency and the California Air Resource Board (CARB), researchers are making increasing use of simulation technology.

One limitation of Chemkin is that it assumes the reaction vessel has a relatively simple geometry, whereas sometimes this is not the case. For that reason, a related program called KINetics is often used in conjunction with Computational Fluid Dynamics tools. CFD programs are better able to account for geometric complexity, at the expense of being more limited in their treatment of the underlying chemistry of the reactive process being studied. (7)

2.1.8. SRM-Engine Suite

SRM Engine Suite is an engineering software tool used for simulating fuels, combustion and exhaust gas emissions in internal combustion engine (IC engine) applications. It is used worldwide by leading IC engine development organisations and fuel companies. The software is developed, maintained and supported by cmcl innovations, Cambridge, U.K.

The sofware uses a probability density function (PDF)-based stochastic reactor model (SRM). Detailed chemical kinetics is solved whilst accounting for inhomogeneity in composition and temperature space arising from on-going fuel injection, heat transfer and turbulence mixing events. Through this coupling,heat release profiles and in particular the associated exhaust gas emissions (Particulates, NOxi Carbon monoxide, Unburned Hyddrocarbon etc.) can be predicted more accurately than if using the more conventional approaches of standard homogenous and multi-zone reactor method. (8)

2.1.9. Diesel-RK

DIESEL-RK is a full cycle thermodynamic engine simulation software. One is designed for simulating and optimizing working processes of two-stroke and four-stroke internal combustion engines with all types of boosting. The program can be used for modeling the following types of engines: (9)

DI Diesel engines, including PCCI and engines fueled by bio-fuels.

- SI petrol engines.
- SI gas engines including prechamber systems, and engines fueled by different gases: Methane, Propane-Buthane, Biogas, Wood gas, Syngas, etc.
- Two-stroke engines with uniflow and loop scavenging, opposed piston engines (OP or Junkers engines) and OPOC engines.
- Dual fuel engines (engines having few independent fuel injection systems for different fuels). RCCI

The DIESEL-RK is a thermodynamic software: engine cylinders are considered as open thermodynamic systems.

Representative applications include:

- Fuel consumption prediction and optimization.
- Torque curve and other engine performances predictions.
- Combustion and emission analysis, including PCCI.
- Dual fuel engine mixture formation and combustion analysis.
- Knock prediction.
- Valve timing optimization, including VVA optimization for every operating mode.

- EGR analysis and optimization.
- Turbocharger and bypasses matching and optimization.
- Conversion of diesel engines into gas engines.
- Cooperation with different modeling tools: Simulink, IOSO NM, etc.: Diesel RK solver can be run under the control of other applications.

2.2.Three Dimensional Softwares

2.2.1.Ansys Fluent- Forte

The FORTE CFD Package is the only CFD simulation package for combustion engines that incorporates proven CHEMKIN-PRO solver technology – the gold standard for modeling and simulating gas-phase and surface chemistry. While legacy engine-combustion CFD simulations utilize chemistry solvers that are too slow to handle the chemistry details required for accurate predictions of ignition and emissions, FORTE enables multi-component fuel models to combine with comprehensive spray dynamics – without sacrificing simulation Time-to-Solution. FORTE robustly and accurately simulates IC engine combustion performance with nearly any fuel, helping engineers rapidly design cleaner burning, high-efficiency, fuel-flexible engines.

Features:

Efficient and Flexible Workflow;

Fluent is fully integrated into the ANSYS Workbench environment, a platform designed for efficient and flexible workflows, CAD associativity and powerful capabilities in geometry modeling and meshing. The built-in parameter manager makes it easy to rapidly explore multiple design options.

Built for Multiphysics;

Gain deeper insight into the complex, often counterintuitive interactions caused by multiple physics such as fluid–structure interaction (FSI). ANSYS Fluent is fully integrated with ANSYS Workbench to provide full two-way interactivity with ANSYS Mechanical, ANSYS Maxwell and other simulation technologies.

Solve Complex Models with Confidence;

ANSYS Fluent can solve your most sophisticated models for multiphase flows, chemical reaction and combustion. Even complicated viscous and turbulent, internal and external flows, flow-induced noise predictions, heat transfer with and without radiation can be modeled with ease.

Go Faster with High Performance Computing (HPC);

With HPC, ANSYS Fluent delivers CFD simulation solutions faster so that engineers and designers can make better decisions sooner in the design cycle. While ANSYS HPC provides linear scalability on systems with tens of thousands of processors, there is more to HPC than just the number of cores. ANSYS also optimizes processor architecture, algorithms for model partitioning, optimized communications and load balancing between processors to deliver results in breathtaking speed on a wide variety of simulation models.

Turbulence Modeling;

ANSYS Fluent software places special emphasis on providing a wide range of turbulence models to capture the effects of turbulence accurately and efficiently. Several innovative models such as the Menter–Langtry γ–θ laminar–turbulent transition model™ are available only in Fluent.

Heat Transfer & Radiation;

Fluent handles all types of radiative heat exchange in and between fluids and solids, from fully and semi-transparent to radiation, or opaque. You can choose from a variety of spectral models to account for wavelength dependencies in a simulation and to account for scattering effects.

Multiphase Flow;

A complete suite of models capture the interplay between multiple fluid phases like gasses and liquids, dispersed particles and droplets, and free surfaces.

Reacting Flow;

Whether simulating combustion design in gas turbines, automotive engines, or coal-fired furnaces, or assessing fire safety in and around buildings and other structures, ANSYS Fluent software provides a rich framework to model chemical reactions and combustion associated with fluid flow. ANSYS Fluent handles non-premixed, partially-premixed, or premixed combustion models to accurately predict parameters like the flame speed, flame location, and the post-flame temperature. (10)

2.2.2-)OpenFOAM

OpenFOAM (for "Open source Field Operation And Manipulation") is a C++ toolbox for the development of customized numerical solvers, and pre-/post-processing utilities for the solution of continuum mechanics problems, including computational fluid dynamics (CFD). The code is released as free and open source software under the GNU General Public License. (11)

The OpenFOAM name was registered by OpenCFD Ltd in 2007 and non-exclusively licensed to the OpenFOAM Foundation Ltd in 2011.

Advantages:

- Friendly syntax for partial differential equations
- Fully documented source code
- Unstructured polyhedral grid capabilities
- Automatic parallelization of applications written using OpenFOAM high-level syntax
- Wide range of applications and models ready to use
- Commercial support and training provided by the developers
- No license costs

Disadvantages:

- The development community suffers from fragmentation, giving rise to numerous forked projects, a portion of which are listed in the Forks and adaptations section below.
- Absence of an integrated graphical user interface (stand-alone Open Source and proprietary options are available)
- The Programmer's guide does not provide sufficient details, making the learning curve very steep if you need to write new applications or add functionality.

2.2.3.Kiva

KIVA is a family of Fortran-based Computational Fluid Dynamics software developed by Los Alamos National Laboratory (LANL). The software predicts complex fuel and air flows as well as ignition, combustion, and pollutant-formation processes in engines. The KIVA models have been used to understand combustion chemistry processes, such as auto-ignition of fuels, and to optimize diesel engines for high efficiency and low emissions. General Motors has used KIVA in the development of direct-injection, stratified charge gasoline engines as well as the fast burn, homogeneous-charge gasoline engine. Cummins reduced development time and cost by 10%–15% using KIVA to develop its high-efficiency 2007 ISB 6.7-L diesel engine that was able to meet 2010 emission standards in 2007. At the same time, the company realized a more robust design and improved fuel economy while meeting all environmental and customer constraints. (12)

Four types of KIVA programs exist;

- KIVA-3V
- KIVA-4
- KIVA-4mpi
- KIVA-EXEC

3.OPTIMIZATION

In mathematics, computer science and operations research, mathematical optimization, also spelled mathematical optimization, alternatively named mathematical programming or simply optimization or optimization, is the selection of a best element (with regard to some criterion) from some set of available alternatives.

In the simplest case, an optimization problem consists of maximizing or minimizing a real function by systematically choosing input values from within an allowed set and computing the value of the function. The generalization of optimization theory and techniques to other formulations comprises a large area of applied mathematics. More generally, optimization includes finding "best available" values of some objective function given a defined domain (or input), including a variety of different types of objective functions and different types of domains. (13)

3.1.MODE FRONTIER

Mode FRONTIER provides an innovative optimization environment with modular, profiled-based access. ESTECO's integration platform for multi-objective and multi-disciplinary optimization offers a seamless coupling with third party engineering tools, enables the automation of the design simulation process and facilitates analytic decision making.

Mode FRONTIER streamlines the design process with powerful workflows, innovative algorithms and sophisticated post-processing tools allowing the user to perform statistical analysis, data visualization and decision making. Its multidisciplinary design enabling technology, critical to successful new product development today, keeps it at the forefront of engineering technology.

User Profiles enable multidisciplinary engineering practices to consolidate specialized expertise and streamline teamwork by allocating software resources where needed. Depending on the step of the engineering problem at hand, it is now possible to access different functionalities within the same

installation through dedicated modules (modeSPACE and modePROCESS) or directly in Mode FRONTIER, according to the profile of the user. (14)

OPTIMIZATION WITH MODEFRONTIER

Mode FRONTIER multi objective optimization capabilities enables to robustly identify the set of best possible solutions even for highly constrained design problems. By combining opposing objectives and considering user-defined constraints, Mode FRONTIER helps manage the complexity faced in engineering projects, including multidisciplinary concerns.

After exploring and understanding relations underlining the Design Space engineers can count upon a wide selection of innovative algorithms, shaped to respond to different design needs and able to tackle discrete or continuous variables to solve single and multi-objective problems.

Mode FRONTIER empowers the user to outline the appropriate optimization strategy, according to the design space boundaries and to the reliability and robustness sought. The algorithms families encompass both RSM-based and direct optimization, managing efficiently problem dimensions and the attainable computational resources.

The easy-to-use wizard allows to choose the best combination of local, global and hybrid optimization methods for reaching an enhanced and robust design. (15)

4. NUMERICAL ANALYSYS

Numerical analysis is the study of algorithms that use numerical approximation (as opposed to general symbolic manipulations) for the problems of mathematical analysis (as distinguished from discrete mathematics).

One of the earliest mathematical writings is a Babylonian tablet from the Yale Babylonian Collection (YBC 7289), which gives a sexagesimal numerical approximation of $\sqrt{2}$, the length of the diagonal in a unit square. Being able to compute the sides of a triangle (and hence, being able to compute square roots) is extremely important, for instance, in astronomy, carpentry and construction.

Numerical analysis continues this long tradition of practical mathematical calculations. Much like the Babylonian approximation of $\sqrt{2}$, modern numerical analysis does not seek exact answers,

because exact answers are often impossible to obtain in practice. Instead, much of numerical analysis is concerned with obtaining approximate solutions while maintaining reasonable bounds on errors.

Numerical analysis naturally finds applications in all fields of engineering and the physical sciences, but in the 21st century also the life sciences and even the arts have adopted elements of scientific computations. Ordinary differential equations appear in celestial mechanics (planets, stars and galaxies); numerical linear algebra is important for data analysis; stochastic differential equations and Markov chains are essential in simulating living cells for medicine and biology.

Before the advent of modern computers numerical methods often depended on hand interpolation in large printed tables. Since the mid 20th century, computers calculate the required functions instead. These same interpolation formulas nevertheless continue to be used as part of the software algorithms for solving differential equations.

4.1.CFD (Computational Fluid Dynamics)

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

5.BASIC DIFFERENCE BETWEEN 0D/1D AND 3D

1D calculations are generally used to simulate the complex behaviour of the whole engine system (fuel injection, gas exchange system, turbocharger, etc.). These 1D models can handle real time simulations for transient engine evolutions and can be used to develop relevant control strategies. 1D simulations are highly flexible and can be adapted for the complete range of operation points. However, they use phenomenological models and are not always predictive. These models contain several empirical parameters that need to be fitted with experimental data. 1D simulations can also be used to give initial (trapped mass) and boundaries conditions (wall temperature) needed by the 3D calculations.

On the other hand, 3D Computational Fluid Dynamics (CFD) calculations are used to investigate physical phenomena where 3D flow effects are prevailing. Optimisation of the geometry of complex engine components (such as intake and exhaust systems, and combustion chamber) requires the use of 3D calculations. Combustion and pollutant emission predictions require 3D simulations to take into account the fuel/air mixing process in the combustion chamber, the internal aerodynamic, the heterogeneous fuel distribution, and the chemical processes in the cylinder. All these 3D effects cannot be evaluated with 1D models alone. However, 3D simulations focus on a single component of the whole engine (the cylinder) without the dynamic interaction with the other components of the system. (16)

CONCLUSION

In this Project, we learnt about engine modeling, analysis and optimization after that we searched about the softwares which are used in this purposes. Then, we introduced these softwares and mentioned about their objectives.

References

Engine Modeling of an Internal Combustion Engine With Twin Independent Cam Phasing By Jason Meyer (1)

https://en.wikipedia.org/wiki/Simulink (3)

https://en.wikipedia.org/wiki/Fortran (4)

https://www.software.ricardo.com/Products/WAVE (5)

https://www.avl.com/fire2 (6)

https://en.wikipedia.org/wiki/CHEMKIN (7)

https://en.wikipedia.org/wiki/SRM_Engine_Suite (8)

http://www.diesel-rk.bmstu.ru/Eng/ (9)

http://symkom.pl/wp/wp-content/uploads/2016/05/ansys-forte-brochure1.pdf (10)

https://en.wikipedia.org/wiki/OpenFOAM (11)

https://en.wikipedia.org/wiki/KIVA_(software) (12)

https://en.wikipedia.org/wiki/Mathematical_optimization (13)

http://www.esteco.com/modefrontier (14)

http://www.esteco.com/modefrontier/optimization-modefrontier (15)

Simulation of a 4-Cylinder Turbocharged Gasoline Direct Injection Engine Using a Direct Temporal Coupling Between a 1D Simulation Software and a 3D Combustion Code- Julien Bohbot, Christos Chryssakis, Marjorie Miche (16)