

Modeling and Simulation for Design and Testing of Direct Injection Gaseous Fuel Systems for Medium-Speed Engines

2011-01-2401

Published
12/15/2011

Vilmar Aesoy
Aalesund University College (AaUC)

Eilif Pedersen
NTNU - Trondheim

Copyright © 2011 SAE International

doi:[10.4271/2011-01-2401](https://doi.org/10.4271/2011-01-2401)

ABSTRACT

The purpose of this study is to develop a modeling and simulation environment for the early design and testing for high pressure injection of alternative fuels in a medium-speed engine application. The proper injection of fuel into the cylinder at the correct timing and with the desired rate is a key to high combustion efficiency. The fuel spray and gas jet characteristics are governed by the injection pressure and nozzle geometry. Incorrect injection causes a reduced efficiency and increasing concentration of harmful emissions. In addition to efficient and clean combustion, safety issues are also important design features for fuel injection systems. In particular, the handling of volatile alternative fuels such as natural gas requires special safety functions to prevent hazardous incidents. Efficient and safe fuel injection is obtained through an increased understanding of the dynamic behavior of the complex fuel injection systems, which involves mechatronics, hydraulics and gas dynamics. The objective of this research is to perform an early design evaluation and testing of these complex fuel injection systems to better optimize design parameters, thereby reducing the need for costly prototypes. This simulation-based testing also includes safety functions and failure mode modeling, both of which are extremely difficult and hazardous to test in full scale. Despite the development of powerful numerical methods and computer capability, the modeling of these multi physics systems is not a trivial task, particularly when the systems are non-linear and involve several energy domains. The proper mathematical model representation for mechanical components, hydraulics and compressible gas

must be linked together in an interactive model. In this study, the general modeling and simulation software 20-sim has proven to be an excellent tool for this type of analysis. Based on conservation principles, each component of the fuel injection system is modeled separately in a model library and then assembled into a system model.

This paper includes application examples in which a system is designed and tested using these simulation models and software tools.

INTRODUCTION

Natural gas-fuelled medium-speed engines have been developed over the past 25 years for stationary power generation. These highly efficient and environmentally friendly engines are now entering maritime applications, offering significant emission reductions, both from a local and global perspective [1],[2]. Converting engines from diesel fuels to gaseous fuels requires separate and specially designed fuel injection systems and different safety philosophies compared to traditional liquid fuels. Since the late 1980s, high pressure natural gas injection systems have been developed and tested at MARINTEK [3]. Wärtsilä Diesel have been using this technology on their Gas-Diesel engines since 1990 [4], and MAN/B&W are developing and testing high pressure natural gas injection on their slow speed 2-stroke engines [5]. Additionally, Cummins has developed a high pressure direct injection natural gas engine (HPDI) [6] for automotive applications in cooperation with Westport Innovations.

High pressure injection of natural gas requires a relatively complex fuel system. To obtain an efficient and clean combustion process, an optimum gas-jet formation in the cylinder must be achieved and the necessary ignition secured. As for natural gas, the auto-ignition temperature is too high for normal compression ignition. Therefore, an auxiliary ignition source is required, which is normally achieved using a pilot diesel injection (2-5%). The pilot diesel system is also normally designed to offer full fuel flexibility, meaning that the engine can operate on 100% liquid fuel when needed. Figure 1 illustrates the principles of a dual fuel engine for the high pressure direct injection (HPDI) of natural gas. These highly specialized multi-fuel systems are designed as a combination of traditional injection systems and common rail technology.

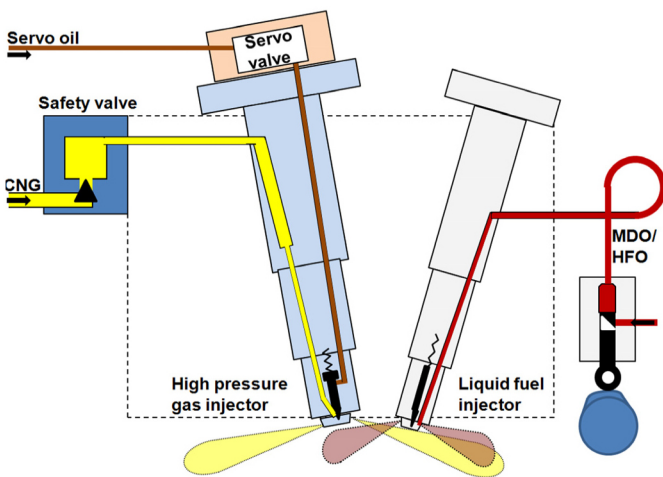


Figure 1. Fuel system principles of a dual-fuel engine with high pressure natural gas injection

Due to its importance, a considerable amount of effort has been devoted to achieving a thorough understanding of the dynamic behavior of fuel injection systems. Traditional design processes rely almost exclusively on experience and “know-how.” The result has often been the use of extensive and expensive prototype testing in fuel injection system development. The availability of computer capability has helped to create an increased effort in utilizing mathematical models and computer-aided simulation in the system development process.

Previous research papers have presented modeling and simulation of high pressure liquid fuel injection systems [7], as well as hydraulic power control of common rail fuel injection systems [8][9]. The objective of this project has been to expand the model library to handle more complex multi-fuel injection systems, and to integrate these models in an efficient modeling and simulation environment. Here, complex fuel injection system models can be built and analyzed to gain a thorough understanding of the dynamic

behavior that involves interactive gas, hydraulic and mechanical systems.

The process of designing a new fuel injection system for a medium-speed engine can be described in the following main steps, as illustrated in Figure 2. First, a preliminary design is done in relation to choosing the main concept and establishing the functional and safety objectives. The next step is the conceptual design to establish the main geometry and process parameters within the limitations given by the interacting components and systems. At this point, the system modeling starts and the main parameters are tuned to find the optimum performance within the given limits. The next step is the detail design, in which the safety functions and all production-specific details must be included. In parallel with the detail design, the simulation model is modified to include and test specific details and to further optimize the main parameters. When the detail design is established, the first prototypes can be manufactured for testing and the fine tuning of system and model can be done in parallel. Through a combination of physical testing and simulations, a very thorough understanding of the behavior is achieved, and this knowledge can be used for modifications and new prototypes, as well as feedback for the conceptual design of new systems.

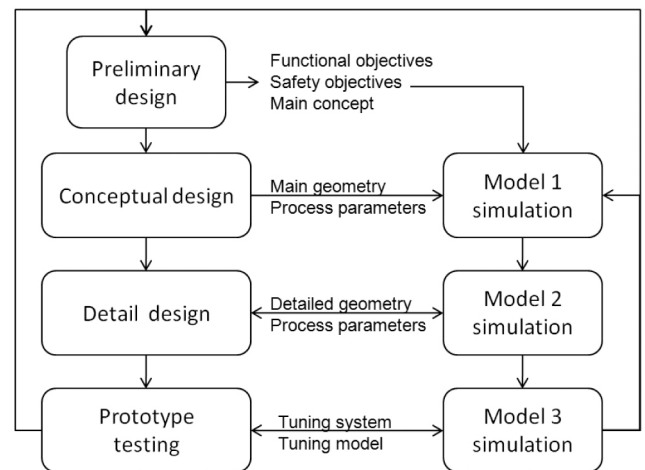


Figure 2. Design process using modeling and simulation

MODELS OF INJECTION SYSTEMS

As problem solving methodology, the modeling and simulation can be characterized at three distinct levels:

- Overall system model
- Sub-systems models
- Physical processes

A proper modeling approach starts with the overall system in order to identify the system's boundaries, further splitting the system into convenient sub-systems, and then finally identifying the physical processes involved. Through this top-

down procedure the problems are identified and the fundamental modeling can start at the bottom, building the model block by block while generating the mathematical relations. In this project, the high pressure gaseous fuel injection system illustrated in [Figure 1](#) was designed and modeled to illustrate this methodology. The system in question is rather complex to model, in that it involves the coupling of unsteady gas dynamics and the servo oil flow to the operation of mechanical systems in valves and injectors. Hence, this system is an example of the coupling of a lumped element model and a one-dimensional continuous flow model. It is also an example of a system in which several energy domains are coupled in the same system. Mathematically, the system is represented by ordinary differential equations for the spring-mechanical elements and partial differential equations for the fluid elements. The primary challenge for the modeler is to obtain a *unified dynamic model* of the total system, built from sub-models:

- Valves and nozzles
- Pipes - gas and fluid
- Compressible volumes in pump, pipes and junctions
- Boundary conditions

Today, a wide range of software is available for the modeling and simulation of dynamic systems, both general simulation tools and more specialized fuel injection system software. In this project, the general modeling and simulation software 20-sim was chosen. The current version of 20-sim was developed by Control Lab Products and originates from the 20-sim, MAX, CAMAS and TUTSIM prototypes that were developed by the Control Laboratory of the University of Twente [10]. This software was mainly chosen because of its flexibility in handling multi domain systems and its bond graph model interface. A more detailed description of a bond graph modeling technique is included in [Appendix A](#) and in [11] and [12].

GAS FLOW MODELS

Developing mathematical models for a natural gas injection system requires that models for mechanical, control, thermodynamic and hydraulic domains are combined to form a complete system model. What is especially challenging is the development and integration of thermo-fluid models in general, and compressible fluid flow models in particular. Modeling thermo-fluid systems is not trivial since the choice of state equations and state variables, as well as the calculation of thermodynamic properties for the gas composition, are closely linked. Fluid flow models are well established with numerous options for different properties and accuracy, but including such models in a complete system model requires that boundary conditions, i.e. the interfacing with other models, are taken special care of. It may even be necessary to reformulate such models for

application in system analysis. This is the approach taken here, and in this paper we outline how the bond graph method [11],[12] can be applied for the simulation of natural gas injection systems.

The bond graph modeling technique adopted by the authors in this work does not, in either its pure or classic form, handle thermo-fluid systems. However, in [14] Karnopp introduces the pseudo-bond graph concept for modeling thermodynamic systems. Here, he introduces two pseudo-bond graph bonds to represent the energy flow between systems or control volumes, and introduces the proper variables for bond graph efforts, flows and state variables.

In the following sections, we outline the basics of the main thermodynamic models, i.e. the building blocks necessary to put together models for the simulation of natural gas injection systems.

Volumes/Accumulators

To establish a proper mathematical model formulation for a general gas phase thermodynamic control volume with any number of inflow and outflow ports, the pseudo-bond graph concept is used [14]. The general conservation laws for mass and energy are utilized to form the state equations. The efforts, flows and state variables for a gas phase pseudo-bond graph control volume model then become:

Efforts(*e*):

P - pressure (Pa)

T - temperature (K)

Flows(*f*):

$\dot{m}$ - mass flow rate(kg/s)

$\dot{E}$ - total energy flow rate (J/s)

State variables (*q*):

m - total mass (kg)

E - total energy (J)

Likewise, the state equations for an accumulator become:

$$\frac{dm}{dt} = \dot{m}_{in} - \dot{m}_{out} \quad (1)$$

$$\frac{dE}{dt} = \dot{m}_{in} h_{in} - \dot{m}_{out} h_{out} \quad (2)$$

and the constitutive laws for the control volume model, i.e. the connections between the state variables and the efforts become:

$$\begin{aligned} P &= \Phi_P(m, E, V) = m^{id\ gas} RT / V \\ T &= \Phi_T(m, E, V) = \frac{E}{mc_v(T)} \end{aligned} \quad (3)$$

where $\Phi_P(\dots)$ and $\Phi_T(\dots)$ are functions derived from a real gas equation of state and the corresponding heat capacity equation for the actual gas, respectively. Very often, simplifications can be made utilizing the ideal or perfect gas equation of state in a particular case, and the rightmost expression in [Eq. \(3\)](#) shows the actual expression assuming ideal gas.

The general thermodynamic control volume model having one inflow and one outflow port is given in [Figure 3](#). This pseudo-bond graph expresses through its 0-junction structure the conservation of mass and energy and state, unambiguously demonstrating how to interface other thermodynamic component models to this particular component. This basic model can also easily be extended to include heat loss, variable size control volumes [\[14\]](#) and to multi-component single or multiple phase systems following the approach in [\[13\]](#). This basic model then often acts as the key building block for the modeling of thermodynamic control volume systems, maintaining the strength of the bond graph method when it comes to connecting models of different origins.

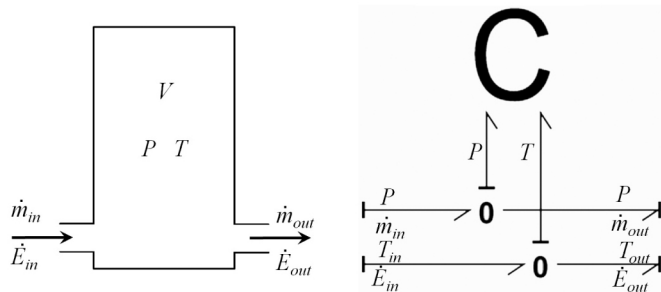


Figure 3. Control volume C-field model representation

Restrictions - Nozzles

In addition to the thermodynamic control volume model, a flow restriction model is necessary for modeling most thermodynamic systems. Flow restrictions are any component such as valves, nozzles and similar devices that can be assumed to have no accumulation effects, i.e. a small volume compared to connected control volumes up-and downstream of the restriction. For gas flows, the nozzle equation establishing the mass flow through a nozzle given the upstream pressure and temperature, as well as the pressure

ratio over the nozzle, is found to represent a large number of general restrictions found in such systems. The mass flow and energy flow through a nozzle is given as:

$$\dot{m} = C_d A \sqrt{\frac{P_u}{RT_u}} \Psi \left(\frac{P_d}{P_u} \right) \quad (4)$$

$$\dot{E} = \dot{m} h_u = \dot{m} c_p (T_u) T_u \quad (5)$$

where:

$$\begin{aligned} \Psi &= \sqrt{\frac{2\kappa^2}{\kappa-1} \left(\frac{P_d}{P_u} \right)^{2/\kappa} \left(1 - \left(\frac{P_d}{P_u} \right)^{(\kappa-1)/\kappa} \right)} \quad \text{when} \quad \frac{P_d}{P_u} > \left(\frac{2}{\kappa+1} \right)^{\frac{\kappa}{\kappa-1}} \\ &= \kappa \left(\frac{2}{\kappa+1} \right)^{\frac{\kappa+1}{2(\kappa-1)}} \quad \text{elsewhere} \end{aligned} \quad (6)$$

and where subscript u and d means the upstream and downstream condition, respectively, and κ is the ratio of specific heat. This implies that when the flow reverses, the assignments of the pressures and temperatures also have to be shifted accordingly.

This can be represented in bond graphs using the pseudo-bond graph R-field shown in [Figure 4](#). The structure of the model equations implies that the preferred causality for this model is an effort on all bonds, i.e. the model inputs should be the pressure and temperature on both sides of the restrictor. However, alternative causality options exist, although this requires a nontrivial reformulation of the nozzle equations, [Equations 4,5,6](#). In addition, the sub-model extension for handling multi-component fluids is given in [\[13\]](#).

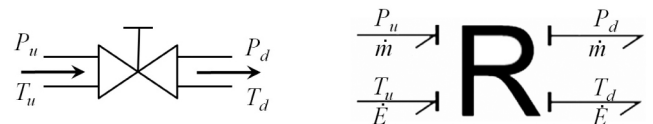


Figure 4. Restriction (R-field) bond graph model

Gas Pipe Flow

In contrast to the thermodynamic control volume model and the restrictor, which is actually a lumped modeling approach, the unsteady pipe flow model must include the distributed nature of the parameters, the thermodynamic properties and the velocity. Thus, in addition to the conservation of mass and energy, we now need to include the conservation of momentum principle as well to obtain a mathematical model of the unsteady pipe flow. For an unsteady pipe flow, a one-dimensional flow assumption is adopted. These conservation

equations, i.e. the Navier-Stokes equations for one-dimensional fluid flow, plus a thermodynamic equation of state and a corresponding heat capacity equation, serve as the theoretical foundation for the derivation of model equations for the unsteady pipe flow model.

In order to obtain the state equations, we decide to apply the method-of-lines approach. This involves dividing the pipe into a number of control volumes and applying any suitable integration method to advance the solution in time, i.e. to transfer the partial differential equations to a set of ordinary differential equations. The density, temperature and velocity of control volume i are assumed to be constant within the control volume. Integrating the conservation equations over a control volume of the pipe yields the following state equations:

$$\begin{aligned}\frac{dm_i}{dt} &= \dot{m}_{i-1/2} - \dot{m}_{i+1/2} \\ \frac{dp_i}{dt} &= \dot{m}_{i-1/2}u_{i-1/2} + P_{i-1/2}A - \dot{m}_{i+1/2}u_{i+1/2} - P_{i+1/2}A - \frac{\lambda}{2D}m_i|u_i|u_i \\ \frac{dE_i}{dt} &= \dot{m}_{i-1/2}(h_{i-1/2} + \frac{1}{2}u_{i-1/2}^2) - \dot{m}_{i+1/2}(h_{i+1/2} + \frac{1}{2}u_{i+1/2}^2) + \dot{Q}_i\end{aligned}\quad (7)$$

where $p_i = m_i u_i$ is the fluid momentum, E_i is the total energy, including the kinetic energy of the fluid, A is the cross-sectional area of the pipe, $\dot{Q}$ is the total rate of heat added to the control volume. For the momentum equation, the frictional forces are included using the Darcy's friction factor relation.

The subscripts i represent the average value of any property or variable inside the control volume, and $i-1/2$ and $i+1/2$ represent the value of the variable at the inflow (the left) and outflow (the right) boundaries of the control volume.

Having introduced the state vector $[m_i, p_i, E_i]^T$ for the individual pipe control volumes, the corresponding required efforts and flows, i.e. the pressure P_i , the temperature T_i and the velocity u_i for the individual pipe control volumes can be calculated from:

$$P_i = \frac{(\kappa-1)}{\Delta x_i A} \left(E_i - \frac{1}{2} \frac{p_i^2}{m_i} \right) \quad (8)$$

$$T_i = \frac{1}{c_v} \left(\frac{E_i}{m_i} - \frac{1}{2} \frac{p_i^2}{m_i^2} \right) \quad (9)$$

$$u_i = \frac{p_i}{m} \quad (10)$$

Equation 7 expresses the time rate for the change of mass, momentum and total energy in control volume i in terms of properties in the control volume and at its boundaries. Dividing the pipe into the N number of the control volumes yields the complete set of state equations for the unsteady pipe flow model. Nevertheless, expressions for the fluxes, i.e. the flow-field variables at the boundaries between the adjacent internal control volumes and the boundaries at the first and last control volume, still need to be given.

To derive expressions for the required fluxes, numerous options exist, and the general approach is to use the values at the adjacent control volumes. An approximation often used is the average of the value at the adjacent control volumes. This approximation, however, does not take into consideration the convective nature of the fluid flow. Mass, energy and momentum transport all occur at a given velocity, often referred to as the convection velocity. This means that it is physically obvious that the condition at the control volume boundary will be more affected by the condition upstream than the condition downstream, which will be more pronounced at higher convective velocities.

In [15], this upwind nature is taken care of using the Mach-modulated approximation:

$$g_{i+1/2} = \frac{1}{2}(g_i + g_{i+1}) + \frac{1}{2}f(M_u)sign(u_u)(g_i - g_{i+1}) \quad (11)$$

where M_u and u_u are the Mach-number and velocity upstream of the boundary and the function $f(M_u) = f(M_u) = M_u^r \in [0,1]$ where r is a positive number.

In this form, the Mach-modulated approximation approaches the mean value approximation for low velocity flows and the pure upwind condition for high velocity flows, i.e. well-known requirements from numerical fluid flow literature.

Using different values for r , the difference in the upwind nature for flow-field properties can be accounted for. In [15], the values for the r are set to [2.0, 2.0, 0.01, 0.0001] for the velocity, the pressure, the density and the temperature, respectively. These values are proposed based on an optimization of accuracy, computational speed and robustness within a wide range of applications. Compared to standard computational fluid dynamic schemes, this approach performs equally well or better than most first-order methods.

A Bond graph interpretation of a control volume i within the pipe is given in [Figure 5](#). Here, the conservation of mass and energy are as stated previously for a lumped control volume in a pseudo-bond graph formalism, but now the energy involves the total energy, including the fluid kinetic energy.