ABSTRACT

One of the most critical elements in diesel engine design is the selection and matching of the fuel injection system. The injection largely controls the combustion process, and with it also a wide range of related issues, such as: fuel efficiency, emissions, startability, load acceptance (acceleration) and combustion noise. Simulation has been a valuable tool for the engine design engineer to predict and optimize key parameters of the fuel injection system. This is a problem that spans a number of subsystems. Historically, simulations of these subsystems (hydraulics, gas dynamics, engine performance and 3-D CFD cylinder modeling) have typically been done in isolation. Recently, a simulation tool has been developed, which models the different subsystems in an integrated manner. This simulation tool combines a 1-D simulation tool for modeling of hydraulic and gas dynamics systems, with 3-D CFD code for modeling the in-cylinder combustion and emissions. This required the development of specialized methodology to handle the differing time scales between hydraulics and gas dynamics, and also the 3-D computations of atomization and combustion in the cylinder.

2. INTRODUCTION

Fuel injection system selection is a critical task in diesel engine design. The injection largely controls the combustion process, and with it also a wide range of related issues, such as: fuel efficiency, emissions, startability, load acceptance (acceleration) and combustion noise. It is largely due to the improvements in fuel injection systems, that diesel engines have shown a steady growth in market penetration. This already happened some time ago in heavy duty applications, and more recently (and strikingly) in automobiles in Europe.

2. MULTI-DISCIPLINARY SIMULATION TOOL

To address these issues, a simulation tool has been developed which models the different subsystems in an integrated manner. This simulation tool combines a 1-D simulation tool (GT-SUITE) for modeling of hydraulic and gas dynamics systems, with 3-D CFD (KIVA-3V code) for modeling the in-cylinder combustion and emissions. This required the development of specialized methodology to handle the differing time scales between hydraulics and gas dynamics, and also the 3-D computations of atomization and combustion in the cylinder.

2.1 ENGINE CYCLE SIMULATION

The engine and gas dynamics calculations were done with the GT-POWER module of GT-SUITE (Ciesla et al. [1]). This module is in principle an extended engine cycle simulation tool. It contains models for intake and exhaust piping, for in-cylinder processes and for numerous other components and control elements.

When applying this tool to diesel engines, a detailed diesel combustion model is often used. This model is based on the work of Hiroyasu [2], as extended by Morel and Wahiduzzaman [3]. It represents the in-cylinder processes of fuel jet formation, the break-up of the jet into droplets, entrainment of air, evaporation of fuel droplets, mixing of air with fuel vapor, ignition, combustion, NOx emissions and soot emissions. The core of the approach is the division of the jet plume into hundreds of zones tracking in time the evolution of small packets of fuel issuing from the injector holes; up to 1500 hundred zones are tracked simultaneously (see schematic in Figure 1). The main simplification residing in this model is that fuel and jet motion is prescribed by a correlation, rather than solved by full flow-field calculations. Taking as input the fuel injection pressure (or mass flow rate), this model predicts the rate of burn, as well emissions of NOx and soot. While the NOx predictions obtained using this model can be reasonably accurate (Weiss [4], Figure 2), soot predictions give trends at best, and at the present time seem to demand 3-D flow modeling.

Figure 1: Schematic of Fuel Packets’ Tracking in the Detailed Diesel Combustion Model
Since practically all diesel engines are turbo-charged, a turbocharger model is essential. The turbocharger model is used to represent the compressor, turbine, turbo-assembly inertia, and aftercooler. Mechanical models are used to calculate the moving parts of the engine (pistons, conrods, crankshaft, flywheel), in order to obtain the engine torque and acceleration. Also, controls models are included, to allow calculations of sensors and actuators employed in feedback control schemes (e.g. injection pulse width or turbocharger wastegate control).

2.2 FUEL INJECTION SIMULATION

The fuel injection system calculations were performed using GT-FUEL, a module of GT-SUITE used for thermal-hydraulic system simulation. The flow modeling utilizes one-dimensional flow elements to model piping and quasi-3D elements for arbitrarily-shaped volumes. The module also contains libraries for mechanical and thermal modeling. These libraries form the base for the thermal-hydraulic modeling capabilities.

2.2.1. Basic Principles

The flow formulation and solution is very similar to GT-POWER, described above. The main difference is that GT-FUEL allows for the extensive thermodynamic treatment of compressible liquids. The code employs a novel density equation of state model that exhibits observed trends of measurable fluid properties such as wave propagation speed. Cavitation and void fraction transport are allowed to occur in the piping sub-volumes and cell faces, respectively. The formulation is based on a homogenous equilibrium model. The code models the effect of pipe wall compliance on wave speed by utilizing a quasi-dynamic thick-shell model. The details of the sub-models employed in this module are reported in [5]. The application of the code to typical fuel injection systems is described below.

The tool contains higher-level models of injectors and pumps that are built from the primitive models. These higher-level models account for the complexities of the modeling of the components while encapsulating the details. Schematics of the common rail injector and the magnetic valve, that is part of the injector, are shown in Figures 3 and 4, respectively. The exploded models of the common rail injector and magnetic valve are shown in Figures 5 and 6, respectively.

Figure 2: Comparisons of NOx Predictions of the Diesel Combustion Model and Experiments

Figure 3: Schematic of the Common Rail Injector

Figure 4: Schematic of the Magnetic Valve

Figure 5: Model of Common Rail Injector
2.2.2. Application to Pump-Line-Nozzle System

Some specialized models are necessary when the tool is utilized in pump-line-nozzle fuel injection system modeling. One of these models is used to predict the fuel leakage rate exiting the high-pressure pump. The leakage exiting the high-pressure pump affects the peak pressure predicted by the simulation tool. The leakage model takes into account the eccentricity of the pump plunger with respect to the chamber and how the asymmetry of the path increases the leakage flow rate. These specialized models are encapsulated within the higher-level models that the tool utilizes when simulating pump-line-nozzle (PLN) injection systems. Utilizing these higher-level models makes the system model building process faster.

Figure 6: Model of the Magnetic Valve

Figure 7: Normalized Pressure Plot in the High Pressure Line of a PLN Injection System

Figure 7 above is a normalized pressure plot in the high-pressure line in a pump-line-nozzle fuel injection system. The high-pressure line is located downstream of the high-pressure pump and upstream of the injector. Important details of the pressure evolution can be seen in this plot. At about 10 degrees, the solenoid valve shuts and there is an increase in pressure in the line. When the needle valve fully opens at approximately 14 degrees, there is a dip in the pressure due to acceleration of the injected fuel. The plot also shows pressure wave reflections in the high-pressure line.

2.2.3. Application to Common-Rail System

Application of the tool to common rail (CR) injection systems is very similar to the application to PLN systems. In modeling common rail systems, however, there might be a need to model very small volumes, depending on the detail of the model. The smallest of these volumes are in the order of a tenth of a cubic millimeter. Figure 8, below, shows the pressure evolution and velocity outflow from the control chamber, through the A-Nozzle, during an injection pulse of an example simulation of a common rail injection system. Figure 9 shows the pressure and the velocity inflow through the Z-Nozzle for the same simulation. In this example, the common rail system was simulated for 40ms. There are four injection pulses and the common rail pressure is 1350 bars. The control volume is located on the backside of the injector needle. The pressure in this volume controls the lift of the injector needle and thus the injection rate. The control volume is upstream of the pre-A-nozzle chamber and the A-nozzle. Therefore, the correct modeling of these components is critical to the accuracy of the tools’ predictions of the common rail injector performance.

Figure 8: Pressure and Velocity Outflow in CR Injector Control Chamber (A-Nozzle)

Figure 9: Pressure and Velocity Inflow in CR Injector Control Chamber (Z-Nozzle)
2.3 CFD (3-D) SIMULATION

The 3-D CFD cylinder flow and combustion and emissions calculations were performed with KIVA-3V, complemented with spray and combustion models developed at the University of Wisconsin's Engine Research Center (ERC).

2.3.1. Basic Principles

The 3-D CFD code is a multi-dimensional, multi-phase, multi-component code for solving transient chemically reactive flows with sprays. The gas-phase solution employs the Arbitrary Langrangian Eularian (ALE) methodology. The code is applicable to turbulent and laminar flow, as well as subsonic and supersonic flow. The code utilizes the k-ε and Re-Normalization Groups (RNG) k-ε turbulence models.

2.3.2. Application to Diesel Engines

The 3-D CFD code was enhanced with University of Wisconsin’s ERC’s model library. The model library includes advanced multi-dimensional models specifically developed for diesel engine simulation. The typical models utilized in diesel engine simulations are: Kelvin-Helmholtz model for fuel jet break up, the Rayleigh-Taylor model for fuel droplet breakup, the shell ignition model, the characteristic-time combustion model, the extended Zel'dovich model for NOx prediction, and competing formation and oxidation rate models for soot production and consumption. Modeling diesel engines using this 3-D CFD code has been extensively validated. The details of the models and validation of the enhanced 3-D CFD code are discussed in [7-10].

2.4 INTEGRATED FUEL INJECTION AND ENGINE CYCLE SIMULATION

In the simulation of fuel injection systems, the cylinder pressure is typically applied as the boundary condition for the fuel flow calculations. The engine cylinder pressure is not readily known at the equipment selection phase of the engine design. Moreover, this parameter is hard to quantify when modeling transient phenomena in the fuel injection system. The effect of the incorrect pressure boundary conditions on the accuracy of the injection rate predictions can be important, especially for lower injection pressure systems with maximum injection pressures around 700 bars.

The trend in the industry is towards higher injection pressures by utilizing common rail injection systems, with typical injection pressures are around 1400 bars. Even at these high pressures, imposing the correct boundary condition plays a significant role in the accuracy of emission predictions. Moreover, accurate cavitation prediction, which is important in higher-fidelity nozzle flow modeling and transient performance, depends on the downstream cylinder boundary condition [6]. Even more important, however, is the need to couple the calculations for the purposes of transient simulations, when the boundary conditions can change considerably from cycle to cycle. This need for coupling in transient simulations is reported in [13].

2.4.1. Handling of Different Time-scales

A major consideration in the integrated simulation of fuel injection and engine performance is the difference in the timescales of the subsystems. In the explicit solution of fluid dynamic equations, the time step size, defined by the Courant condition, is linearly proportional to the spatial discretization employed in the solution and inversely proportional to the speed of sound in the fluid. The spatial discretization utilized in fuel injection simulation is about five times finer than that employed in engine performance simulation. Also, the speed of sound in diesel fuel is about four times larger than the speed of sound in air. Thus, the time step size required for a sub-volume in an engine performance simulation is about twenty times larger than the time step size required for a sub-volume in a fuel injection simulation. This difference is even larger in higher-fidelity fuel injection simulations where 0.25mm-length volumes are modeled. An example of such a critical component is the pre-A-nozzle chamber in CR injectors [6]. Imposing such a small time step size on an integrated engine performance simulation, which typically have significantly larger number of volumes, may result in impractical simulation times.

To solve this difficulty, the integrated tool employs a circuit-based time stepping methodology:

1. The time step size required by each circuit/subsystem is the minimum time step size required by its components. This would be the simulation time step size if only that subsystem is being modeled.
2. These circuits’ time step sizes may be modified if a maximum time step ratio between the circuits is imposed. In this case, the minimum circuit time step size is unchanged and the ratios of the other circuits’ time step size with respect to the minimum may not exceed the imposed ratio.
3. The largest time step size is denoted as the master time step and the circuit it belongs to the master circuit.
4. The time step sizes used in the subsystems will be integer ratios of the master time step (the subsystems will not need to take a smaller time step in order to match the master solution time). Calculations are performed for one step of the master circuit and integer number of steps of the other circuits and information is exchanged at the end of this process.
5. There is a small possibility that the required time step size in any of the other subsystems, as dictated by the Courant condition, may
become lower than the modified original time step size within the master time step. Provision is made in the methodology to recalculate a new time step size and an integer number of steps for that circuit for the remaining time within the master time step. Then the calculations proceed as before.

For a simulation that a maximum time step ratio is not imposed, the total simulation time of the integrated system will be a simple addition of the simulation times of the individual subsystems.

### 2.4.2. Application to Transient Operation

The integrated tool was used to study the transient response of a turbocharger in a six-cylinder diesel engine with pump-line-nozzle injection system. At full load the injection system was delivering 200mg/cycle of fuel. There was a step change in load from 20% load to full load. The delivery amount in the fuel injection system was controlled by the pulse width of the solenoid valve. Two cases were simulated: the fuel injection system was allowed to deliver the full load instantaneously in the first case and the injection was smoke-limited in the second case. For the second case, the injection rate was dependent on the amount of air in the cylinder, and hence on the performance of the turbocharger. The feedback nature of this problem (injection rate depends on volumetric efficiency of engine, the volumetric efficiency depends of the performance of the turbocharger, the performance of the turbocharger depends on the exhaust energy which depends on the injection rate and combustion) imposes the need for an integrated simulation approach. This is handled in the simulation by utilizing the control library to implement a realistic feedback control of the injection pulse width such that the objective of fixed fuel-air smoke limit is accomplished.

Figure 10: Engine Fuel Flow in a Smoke-Limited and non Smoke-Limited Diesel Engine

Figure 10 shows the compressor outlet pressure for the base case and the smoke-limited case. The consequence of the smoke-limiting is of course a reduced responsiveness (increased turbocharger lag). Also, note the gradual reduction of injected quantity in Figure 10 (for the non-smoke limited case). This is due to the effect of the increasing cylinder pressure on the injector performance.

Figure 11: Compressor Outlet Pressure in a Smoke-Limited and non Smoke-Limited Diesel Engine

### 2.5 INTEGRATED FUEL INJECTION, ENGINE CYCLE, AND 3-D CYLINDER COMBUSTION SIMULATION

Typically, 3-D CFD calculations of in-cylinder fuel spray, combustion, and emissions are performed at a single operating condition and the simulation is removed from the modeling of the other engine processes. For steady-state calculations, this methodology neglects the effect of the evolution of the fuel injection process on spray characteristics like cone angle and jet breakup length. The effect of the warm up of the cylinder structure is also neglected, unless an iterative modeling approach is employed. This engine cylinder modeling methodology is problematic when applied in a transient study.

In the present methodology, the full 3-D CFD code was integrated within the 1-D simulation tool. While the CFD code retained its full functionality, it gained added user-friendliness by employing the graphical interface of the 1-D simulation tool. Furthermore, most of the setup tasks such as mesh generation have been automated. This makes the integrated tool accessible to non-CFD specialist. 3-D cylinder modeling may be performed over multiple engine cycles with the integrated tool. The time marching scheme between the 1-D tool and the 3-D code is exactly similar to the time marching scheme employed in the integration of the fuel injection and engine performance simulation tool.

Extensive validation of the stand-alone 3-D CFD code exists in open literature [7-12]. The integrated tool, however, needs to be benchmarked with the stand-alone 3-D CFD code. To this end, simulation of a single-cylinder engine configuration was performed using the stand-alone code and the integrated tool. As expected, the agreement between the two tools is almost exact. Table 1 below shows the engine
geometry and injection information that were used in the simulations.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bore</td>
<td>137.16 mm</td>
</tr>
<tr>
<td>Stroke</td>
<td>165.1 mm</td>
</tr>
<tr>
<td>Connecting Rod Length</td>
<td>263 mm</td>
</tr>
<tr>
<td>Compression Ratio</td>
<td>15.6</td>
</tr>
<tr>
<td>Piston Crown</td>
<td>Mexican Hat</td>
</tr>
<tr>
<td>Intake Pressure</td>
<td>2 bars</td>
</tr>
<tr>
<td>Intake Temperature</td>
<td>352 K</td>
</tr>
<tr>
<td>Intake Valve Close Timing</td>
<td>147 deg. BTDC</td>
</tr>
<tr>
<td>Exh. Valve Opening Timing</td>
<td>135 deg. ATDC</td>
</tr>
<tr>
<td>Injection Quantity</td>
<td>159.25 mg</td>
</tr>
<tr>
<td>Nozzle Hole Diameter</td>
<td>0.259 mm</td>
</tr>
<tr>
<td>Number of Holes</td>
<td>6</td>
</tr>
<tr>
<td>Injection Timing</td>
<td>9 deg. BTDC</td>
</tr>
</tbody>
</table>

Table 1: Engine Geometry and Injection Information

The computational grid used in the simulation is shown below in Figure 12.

Figure 12: Computational Grid used in Simulation. The Azimuthal Sector is 60 degrees. The Number of Cells in the Azimuthal Direction is 30.

Figure 13: Comparison of Cylinder Pressure Predictions of the Integrated Tool and the Stand-Alone CFD Code

After the validation of the baseline case, the present 1-D/3-D integrated model was further applied to perform parametric studies. The conditions include both high (75%) and low (25%) loads using split injection schemes. Various start-of-injection timings were used to investigate the effects on soot and NOx emissions as listed in Table 2. The injection scheme 12-(6)-13 means the first 50% of fuel is injected within 12 crank angle degrees, followed by 6 CAD dwell, then the rest of fuel is injected within 13 CAD.

<table>
<thead>
<tr>
<th>Load</th>
<th>Speed (rpm)</th>
<th>Injection</th>
<th>SOI (ATDC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>75%</td>
<td>1600</td>
<td>Split 12-(6)-13</td>
<td>-7, -4, -1, 2, 5</td>
</tr>
<tr>
<td>25%</td>
<td>1690</td>
<td>Split 9-(8)-5.25</td>
<td>-9, -6, -3, 0, 3</td>
</tr>
</tbody>
</table>

Table 2 Engine Conditions for Parametric Study

Note that all the mesh and input files for the 3-D CFD simulations are automatically generated by integrated
tool. In addition, the same set of model constants are used for all the simulations.

Figure 16 shows the comparisons of measured and predicted cylinder pressure and heat release rate data for the high load, split injection, SOI=2 ATDC case. The soot and NOx results are also shown in Fig. 17. The soot-NOx trade-off results for cases with different injection timings are shown in Fig. 18.

Similarly, good levels of agreement in measured and predicted engine data are also obtained in low-load, split injection cases. Figure 19 shows the cylinder pressure and heat release rate results of SOI=-6 ATDC case. The soot-NOx trade-off results are shown in Fig. 20 for various injection timings.

In the past, 3-D CFD cylinder computations have been used to optimize fuel injection strategy [11], exhaust gas recirculation (EGR) fractions, and combustion chamber geometry [12]. The results of such optimization are valid for that operating condition. A better approach is to optimize these parameters in a transient simulation over a standard driving cycle. Such strategy is feasible with the integrated tool. 

Simulation duration is an issue that arises when multiple operating conditions are simulated in a 3-D CFD tool. The integrated tool has a novel technique for handling this issue. The 3-D code is run intermittently with a frequency that is specified a priori. During the engine cycles that the CFD code is not run, the integrated tool employs the earlier-described quasi-dimensional combustion and emission models.
or employs the burn rate curve that was generated when the CFD code was run last.

**SUMMARY**

An integrated simulation tool has been developed for modeling the transient behavior of a fuel injection system, combustion and overall engine performance. It is built on the basis of GT-SUITE (GT-POWER+GT-FUEL) and it incorporates KIVA-3V with well established advanced models of diesel engine combustion and emissions. The purpose of this development was to bring together the modeling components needed to predict and optimize key parameters of the fuel injection system and engine combustion and emissions. Among the advantages this modeling system offers are:

- Modeling of system transients
- Accounting for the interactions between injection system and combustion
- Single tool eliminates laborious and error-prone data transfers between different tools
- Higher user productivity

This paper discusses the development of the tool and the methodologies employed to this end. In future work, the application of the integrated tool to different stages of fuel injection equipment and engine design will be presented.

**REFERENCES**