ONE-DIMENSIONAL ENGINE MODELING
AND VALIDATION
USING RICARDO WAVE

Final Report
KLK434-B
N07-09

National Institute for Advanced Transportation Technology
University of Idaho

Dan Cordon, Charles Dean, Judith Steciak and Steven Beyerlein
September 2007
DISCLAIMER

The contents of this report reflect the views of the authors, who are responsible for the facts and the accuracy of the information presented herein. This document is disseminated under the sponsorship of the Department of Transportation, University Transportation Centers Program, in the interest of information exchange. The U.S. Government assumes no liability for the contents or use thereof.
The use of one-dimensional CFD engine simulation is an essential tool to the engine development process. Engine design through simulation can drastically reduce time needed to perform engine experiments and prototyping, as most engine experiments can be simulated within the software. As long as a model can be validated to high degree of accuracy (i.e. +/- 5%), the model can be used with a high level of confidence to optimize engine parameters. This work has used only a small fraction of WAVE’s potential. More sophistication is possible in the areas of combustion kinetics, computational fluid dynamics, and emissions abatement through the use of advanced features and co-simulation.

The intention of this research was to learn the process of one-dimensional CFD engine simulation and document what has been learned about the process as well as the potential of the software for future work. The research opportunities available with this software are endless and far-reaching. The software can also be used as an essential engine technology learning tool in technical elective courses such as ME433 – Combustion Engine Systems. Any questions that may arise about standard engine operation or combustion processes can be investigated with WAVE.
# TABLE OF CONTENTS

## EXECUTIVE SUMMARY .............................................................................................................. 3

## 1. INTRODUCTION .................................................................................................................. 4

## 2. APPROACH .......................................................................................................................... 5

## 3 METHODOLOGY ..................................................................................................................... 6

- Model Setup .......................................................................................................................... 9
- Model Development .............................................................................................................. 14
- Parametric Studies ................................................................................................................ 16
- Optimization ......................................................................................................................... 17

## 4. FINDINGS ............................................................................................................................. 22

- Engine Package Design ........................................................................................................... 24

## 4. FINDINGS ................................................................................................................................

- Model Setup .......................................................................................................................... 24
- Model Validation .................................................................................................................... 27

## Sub-System Case Studies ........................................................................................................ 29

- Intake Manifold Design .......................................................................................................... 29
- Exhaust Header Design .......................................................................................................... 33
- Exhaust System Design Recommendations ........................................................................... 38
- Compression Ratio Study ........................................................................................................ 39
- Throttle Body Study ................................................................................................................ 40

## 5. CONCLUSIONS/RECOMMENDATIONS ............................................................................. 42

## 6. REFERENCES ...................................................................................................................... 45
EXECUTIVE SUMMARY

The use of one-dimensional CFD engine simulation is an essential tool to the engine development process. Engine design through simulation can drastically reduce time needed to perform engine experiments and prototyping, as most engine experiments can be simulated within the software. As long as a model can be validated to high degree of accuracy (i.e. +/- 5%), the model can be used with a high level of confidence to optimize engine parameters. After that, engine testing is only needed to validate the final prototype. This work has used only a small fraction of WAVE’s potential. More sophistication is possible in the areas of combustion kinetics, computational fluid dynamics, and emissions abatement through the use of advanced features and co-simulation.

The intention of this research was to learn the process of one-dimensional CFD engine simulation and document what has been learned about the process as well as the potential of the software for future work. Students considering graduate school here at the University of Idaho should be presented with the opportunity to use this software for engine research projects. The research opportunities available with this software are endless and far-reaching. The software can also be used as an essential engine technology learning tool in technical elective courses such as ME 433. Any questions that may arise about standard engine operation or combustion processes can be investigated with WAVE.
1. INTRODUCTION

The use of one-dimensional computation fluid dynamic (1D CFD) engine simulation software is widespread throughout the engine development industry. This simulation method allows for characterizing engine operation without the need for high-end processing and time-intensive computations. Various commercial software packages are available; however, software costs generally prohibit use in small organizations, primarily making them industry-specific software packages. There is a great value in being able to use such engine development methods at the collegiate level. The Ricardo Company has made this possible through Formula SAE sponsorship. This thesis researched professional simulation methods and used Ricardo’s WAVE software to research potential engine packages for the Formula SAE competition.

Simulation work has been applied in a case study of a naturally aspirated 600 cc motorcycle engine. The objective of this case study was to maximize engine torque output at the most useable engine speeds based on simulation predictions. Specifically, physical engine parameters such as intake and exhaust geometries were varied to discover performance trends. While no full-system optimization was performed, individual component parameters were investigated to reveal their sensitivity to engine output. Upon completion of these component studies, a complete engine package design was suggested based on the simulation predictions.
2. APPROACH

There are two primary engine simulation software packages used in the industry today: Ricardo WAVE and GT-Power. Both software packages are similar in purpose and functionality. These 1D simulation packages require detailed input parameters to simulate engine operation as a whole rather than using models that target certain engine sub-systems. Being that all engine components work together as a system, it is advantageous to model the entire engine system rather than individual subsystems. Ricardo WAVE was used for this research as it was provided by Ricardo through Formulas SAE sponsorship.

Ricardo WAVE is more than just an engine simulation program. It is an engineering code designed to analyze the dynamics of pressure waves, mass flows, energy losses in ducts, plenums, and the manifolds of various systems and machines. When further-detailed analysis is desired, WAVE can be coupled with various industry-standard software packages such as CFD analysis programs and statistical analysis software. Ricardo also offers various external programs that can be coupled with WAVE. [WAVE, 2007]

The basic operation of the WAVE code analyzes flow networks composed of ducts, junctions, and orifices. Within this network of plumbing, engine cylinders, turbochargers / superchargers, compressors, and pumps can be inserted. WAVE can simulate internal combustion engines as well as other compressible-fluid flow systems. Once a simulation has completed, the post-processing program, WavePost, allows for detailed analysis of the simulated engine operation. The overall capabilities of WAVE extend beyond the simulation of engine operation. The software bundle has a program specifically dedicated to acoustic and noise analysis. Other secondary software uses include: Hybrid 1D/3D CFD engine simulation, thermal analysis, controls systems, and combustion and emissions simulation. [WAVE, 2007]
3 METHODOLOGY

The use of Ricardo WAVE to develop engine simulations requires several computer programs that vary in purpose from model setup and 3D modeling, to statistical analysis of model output. The four programs used in this research are: WaveBuild, WaveMesher, WAVE, and WavePost. WaveBuild is the pre-processor and GUI that is used to define all engine and simulation parameters. All model and simulation parameters are defined from within this program with the exception of external CAD geometries. A screen-capture of the GUI layout is shown in Figure 1. The left portion of the screen lists the elements available to be inserted into the model such as engine cylinders, ducts, and orifices. The right side of the screen is the visual portrayal of the engine model layout in which all parameters can be modified. There are some geometries that can be difficult and time consuming to construct with the elements available.

Figure 1: WaveBuild GUI
For this reason, WaveBuild allows complex geometries to be imported into the model from 3D CAD files using a program called WaveMesher. WaveMesher allows the end-user to manipulate 3D cad models of complex geometries such as intake manifolds and exhaust headers, so that the 3D models can be broken-down into a usable one-dimensional form for the WAVE processor. Figure 2 is a screen capture of the WaveMesher program in which an intake manifold has been meshed for use in WaveBuild.

![WaveMesher](image)

Figure 2: WaveMesher

Once the model is setup in WaveBuild and 3D models have been meshed, a simulation can be started by running the WAVE code. WAVE is the solver that performs all the calculations needed to simulate engine operation. It is a non-interactive program that runs in a DOS window while streaming certain output data and simulation progress. The output data shown during simulation runtime can be customized to show parameters of interest. These parameters can be
used as indicators to show whether or not the simulation is producing reasonable results, allowing the simulation to be prematurely stopped if the model is not functioning properly.

Figure 3: WAVE Solver

Once a simulation is finished, a large output file is created that contains all data needed to analyze the simulated engine operation. WavePost is the post-processor for WAVE simulations that allows interpretation of simulation results. It allows for the creation of: time plots, sweep plots, spatial plots (animated), and TCMAP plots (for turbines and compressors). Figure 4 is a screen-capture of the WavePost post-processing software. A sample sweep plot is shown.
The engine model is setup by defining some relatively basic inputs and then some more advanced inputs that require some engine testing. The basic inputs are composed of engine geometry and boundary conditions. Therefore, all dimensions from the intake and exhaust ducting must be recorded and input. Likewise, manufacturer specifications for internal engine geometry such as bore, stroke, connector rod length, wrist-pin offset and compression ratio must be input. Initial conditions such as exhaust temperatures, intake temperatures, and wall temperatures need to be input as reasonable values. These can be modified to higher accuracy once actual engine measurements become available. A general flow diagram for the development of a model is shown in Figure 5.
Within the WaveBuild GUI there are four basic junctions used to connect ducts [WAVE, 2007]:

- **Orifice**: A mass-less and zero-length junction used to directly connect two ducts. The diameter of the orifice defaults to match the connecting ducts but can be input independently when necessary.

- **Ambient**: Defines a condition in which one end of a duct is open to atmospheric pressure. They are commonly used to define air filter openings and muffler outlets.

- **Engine Cylinder**: Represents an internal combustion engine cylinder of either the 2-stroke or 4-stroke type. Allows for modeling of advanced concepts such as 6 and 8 stroke engines as well.

- **Y-Junctions**: Defines a junction that contains more than two openings or other geometries that cannot be adequately modeled with ducts and orifices. Used to model complex geometries such as intake manifolds, mufflers, air filters, and exhaust collectors.
Some basic modeling conventions must be well understood to properly create a model using the above-mentioned basic junctions. Y-Junctions should be used anytime that a desired geometry cannot be achieved through the use of ducts and orifices. Any flow junction that has a branched flow (i.e. manifolds and collectors) must be modeled with Y-Junctions. Two forms of Y-Junctions exist: the simple Y-junction and the complex Y-Junctions. Simple Y-junctions are represented by spherical volumes that distribute flow to multiple ducts. Simple Y-junctions are rarely used because they are essentially an over-simplified complex y-junction. Complex Y-junctions are represented by an arbitrarily shaped volume. A three-dimensional view allows for the arrangement of ducts relative to the junction to be modeled appropriately. Additionally, length properties from the duct opening to the back wall of the volume, discharge coefficients, and area change parameters can be modified.

Once the basic inputs are defined, the advanced inputs that define the cylinder head need to be defined. These inputs are: port flow coefficients, valve lift per crankshaft rotation, and combustion modeling. Because the flow through the engine is highly dependent on the flow through cylinder head ports, it is recommended that this information is measured directly from the cylinder head as it will achieve higher accuracy than what would be expected from 1D CFD modeling. These port flows can be measured on a flowbench. Typically, a pressure drop of 28” of water is applied across one cylinder and volumetric flow (measured in CFM) through the valves can be measured at incremental valve lifts. A spreadsheet provided by Ricardo converts these measured flow values to useable port-flow coefficients. The second advanced input within the cylinder head is the valve lift per crankshaft rotation. The valve lift per camshaft rotation can be measured by incrementally rotating the camshaft and measuring valve lift with a dial indicator. Then the opening points of the intake and exhaust valves relative to TDC must be known to reference the camshaft rotation data to the crankshaft.

The third advanced input and one of the most important datasets needed to characterize an engine model and validate simulation results comes from in-cylinder pressure measurements. Once crank-resolved pressure traces are obtained for complete engine cycles at each operating point, friction correlations and combustion models can then be determined. Unfortunately, obtaining in-cylinder pressure data is not a simple task. Once a pressure sensor is physically
implemented into the cylinder head, special equipment is still needed to interpret the very rapid sensor output. The University of Idaho Small Engine Research Facility currently does not have this ability; thus in-cylinder pressure measurements were not available for this modeling research.

Without in-cylinder pressure measurements, the combustion model had to be predicted based on typical forced induction Wiebe function parameters. WAVE allows for three parameters in the Wiebe correlation to be input: 10-90 percent burn duration, 50 percent burn point, and the Wiebe exponent. [WAVE, 2007]

The Wiebe combustion model is defined by:

\[
W = 1 - e^{-\left( -AWI \left( \frac{\Delta \theta}{BDUR} \right)^{WEXP+1} \right)}
\]

(Equation 1)

where,

- \( W \) = Cumulative mass fraction burned
- \( \Delta \theta \) = Crank degrees past start of combustion
- \( BDUR \) = User-entered 10-90 percent burn duration in crank angle degrees
- \( WEXP \) = User-entered Wiebe exponent
- \( AWI \) = Internally calculated parameter to allow BDUR to cover the 10-90 percent range
- \( CA50 \) = User-entered 50 percent burn location in crank angle degrees after top-dead center
Table 1: Wiebe Function Parameters

<table>
<thead>
<tr>
<th>Data Group</th>
<th>Engine Speed (RPM)</th>
<th>BDUR (Degrees)</th>
<th>WEXP</th>
<th>CA50 (ATDC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Recommended NA* Values [Blair, 1999]</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1200</td>
<td>14</td>
<td>1.72</td>
<td>20</td>
<td></td>
</tr>
<tr>
<td>2400</td>
<td>15</td>
<td>1.93</td>
<td>25</td>
<td></td>
</tr>
<tr>
<td>3600</td>
<td>17</td>
<td>1.82</td>
<td>25</td>
<td></td>
</tr>
<tr>
<td>6000</td>
<td>21</td>
<td>1.64</td>
<td>32</td>
<td></td>
</tr>
<tr>
<td>NA* Input Values</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3000</td>
<td>16</td>
<td>1.85</td>
<td>25</td>
<td></td>
</tr>
<tr>
<td>4000</td>
<td>17</td>
<td>1.75</td>
<td>27</td>
<td></td>
</tr>
<tr>
<td>5000</td>
<td>19</td>
<td>1.70</td>
<td>29</td>
<td></td>
</tr>
<tr>
<td>6000</td>
<td>21</td>
<td>1.60</td>
<td>32</td>
<td></td>
</tr>
<tr>
<td>7000</td>
<td>22</td>
<td>1.55</td>
<td>33</td>
<td></td>
</tr>
<tr>
<td>8000</td>
<td>23</td>
<td>1.50</td>
<td>34</td>
<td></td>
</tr>
<tr>
<td>9000</td>
<td>23</td>
<td>1.45</td>
<td>35</td>
<td></td>
</tr>
<tr>
<td>10000</td>
<td>23.5</td>
<td>1.45</td>
<td>35</td>
<td></td>
</tr>
<tr>
<td>11000</td>
<td>24</td>
<td>1.45</td>
<td>35</td>
<td></td>
</tr>
</tbody>
</table>

* NA = Naturally Aspirated

Table 1 shows the Wiebe parameters determined by Blair for naturally aspirated engine operation. Likewise, inputs based on Blair’s findings are included as well. The burn duration is relatively constant but does increase slightly with engine speed. Ignition timing advance increases with engine speed but the 50 percent burn location, CA50, remains relatively constant. Lastly, the Wiebe exponent, WEXP, tends to decrease with engine speed.

Once these basic and advanced inputs have been satisfied, the simulation format must be setup. WAVE allows for both transient and steady-state engine simulations. While transient simulations can provided a more detailed representation of engine performance, steady-state simulations are better suited for simulations in their early stages of development for various reasons. A significant advantage of steady-state simulations is that parameters can be modified independently for each engine speed simulated. Other important simulation parameters include: cycles to run for convergence, convergence detection, and convergence tolerances. These are some of many general simulation parameters. Another parameter allows for the flow-field to be reinitialized in between simulation cases. This means that for each case the simulation runs, it
will start with the user-imposed initial conditions such as wall temperature and ambient conditions of the gas. By default, this feature is turned off, as quite often the final gas conditions from a prior case are closer to the next case than those initial conditions guessed by the user. Another important general parameter is convergence detection. Convergence detection allows the WAVE code to move on to the next case once a user-specified tolerance for convergence (default is 1 percent) towards a solution is attained on the active case. Simulation duration is another important parameter. This parameter sets the number of cycles for the engine to run before settling on a solution. If auto-convergence is enabled, WAVE will move on to the next case once convergence is reached regardless of the number of cycles specified. If convergence is not reached by the number of specified cycles, the WAVE code will output a warning to let the user know that convergence was not reached within desired tolerances and will continue on to the next case.

There are certain limitations associated with 1D CFD flow analysis that require special attention. Ricardo has provided a very comprehensive set of input parameters that allow users to highly customize any flow model. One such limitation comes into play when trying to model ducts that have significant geometric changes over very small length increments. An example of such geometry is the bell-mouth of an intake runner. The 3-D flow phenomenon that occurs in such geometry can not be accurately modeled by 1D code. In this case, discharge coefficients can be used based on published findings for bell-mouth discharge coefficients. So, essentially flow losses can be imposed on geometric instances that can’t be accurately modeled with 1D code. Furthermore, when modeling tapered ducts, it is recommended to keep angles smaller than 7 degrees. This is due to the fact that flow separation, a phenomena not modeled in 1D code, typically occurs at flow angles of 7 degrees or more. One such instance that was encountered in this research was in the modeling of the flow restrictor that had a converging angle of over 20 degrees. If the restrictor is modeled exactly from its physical geometry, WAVE will over-predict power output due to the inability to calculate flow losses due to flow separation.

Model Development
A model is only as accurate as the input data. This is a principle that must always be kept in mind when attempting to validate a model. Likewise, it important to keep in mind that 1D
simulation is not a 100 percent accurate prediction of true gas dynamics. While good simulation results can be within 5 percent of actual performance, the amount of time spent troubleshooting a model while trying to perfectly match empirical data to simulation predictions should be limited. One of the reasons why 1D simulation is the mainstream form of engine simulation is because it allows for rapid development whereas a full-blown 3D CFD simulation could take weeks to calculate engine operation at a single engine speed. Another thing that should be kept in mind is that two identical engines will exhibit slightly different performance due to different conditions such as: bearing wear, piston ring seal, valve seating, carbon build-up, and fuel composition. These are all reasons why it is important to not expect perfect-matching predictions. What is more important is that the simulation accurately predicts the shape of the torque curve as opposed to the exact magnitudes. [Gurney, 2001]

It is recommended that initial validation and calibration of the model should be performed by comparing predicted volumetric efficiency to that measured on the dynamometer. In order to obtain volumetric efficiency calculations, an accurate air-mass flow sensor must be interfaced with the engine and dynamometer data acquisition. The mass flow sensor that is equipped with the dynamometer used for this research has calibration problems and could not be used for initial model validation and calibration. Instead, torque output was the parameter used to validate the model. While volumetric efficiency validation is recommended, torque validation could be considered the next best method. Torque output validation is performed by comparing torque output measured on a dynamometer to predicted torque output. Discrepancies between predictions and measurements can arise from several sources. For example, if the model is not accurately predicting the location of the pressure-wave-induced intake / exhaust torque peaks, then the dimensions of the intake / exhaust geometry and intake / exhaust valve opening inputs should be checked for accuracy. If the overall torque magnitude is offset a certain amount across the entire engine speed range, engine parameters such as the combustion model, heat transfer and friction correlations should be checked for accuracy. It should also be kept in mind when troubleshooting a model that the erroneous input is most often due to an input that was guessed due to a lack of data. If this is the case, the data needs to be acquired or modified to match empirical results.
Once the model seems to be predicting reasonable results, it can be further refined. Boundary conditions and initial conditions can be input to match temperatures and pressures measured at different junctions during actual engine testing. This helps improve model accuracy and decreases computational time needed for convergence.

**Parametric Studies**

Once a model has been validated, it can be used as a most valuable engine development and design tool. While WAVE’s features extend far beyond performance development, this discussion will primarily focus on the use of WAVE to develop a high-performance engine package. A good starting point is to parameterize engine dimensions of interest such as intake runner length and exhaust header primary length. By performing a parametric study on one geometric dimension at a time, trends can be revealed that indicate near-optimal dimensions for a desired torque output. Later studies can implement design of experiment (DOE) methods for full-system optimization. The below figure shows an example of a geometric parameter study; in this case showing torque output based on varied engine compression ratios.

![Compression Ratio Study](image)

*Figure 6: Example of Parametric Study Results*
Examining one parameter at a time is valuable for investigating trends, but does not necessarily tell the full story. This is especially true for parameters that have effects that are engine speed dependent such as intake and exhaust duct lengths. For example, having an intake resonance occur at the same engine speed as a poorly-timed exhaust pressure wave from the headers will severely hinder the effect of the intake resonance. On the contrary, parameters such as tubing diameter tend to have effects that are less dependent on other parameters.

**Optimization**

Before attempting to optimize an entire package, general trends, sensitivities, and near-optimal values should be investigated. Additionally, it is good practice to implement physical constraints on an optimization so that it will produce usable results. Such a constraint could be on header primary lengths. If a short and lightweight package is demanded by the application, length restrictions should be implemented in the experiment’s boundary conditions. WAVE allows optimization through its experiment feature. The experiment feature allows you to choose as many parameters as you want to be varied according to minimum and maximum acceptable values (boundary conditions). Two basic DOE methods are available for WAVE experiments: Full/Half Factorial (2-level) and central composite (3-level).

Once an experiment has been completed, any parameter can be minimized or maximized. For example; if the user wants to optimize camshaft parameters, the lift multipliers, lobe anchors, and duration multipliers could be set as varied parameters within the experiment. The experiment post-processing screen is shown in Figure 7. The parameters included in the experiment are located in the lower left of the screen and can be modified with the slide bars. Engine performance graphs on the right will reflect changes in real-time as the slider bars are adjusted. The list on the left side of the screen is a comprehensive collection of output graphs available. As an example, the user can click on a torque output graph and have torque output maximized, with WAVE indicating the experiment parameters that lead to that optimal performance. Conversely, the user could attempt to minimize an output such as trapped residuals and determine what camshaft parameters lead to that minimum. [WAVE, 2007]
Figure 7: WAVE Experiment Panel

Some parameters such as exhaust tubing diameter do not necessitate optimization due to the limited sizes available. In other words an exact dimension produced from the optimization is not
useful if the tubing is only available in four different practical sizes. These are cases where basic parametric studies may be best implemented.

The experiment feature does have some limitations. The parameter table that is used to vary parameters against engine speed is not used in experiment mode. So if a control system is modeled on the engine using the parameter table, it will not be implemented in experiment mode. The best method of optimization with WAVE requires co-simulation. Co-simulation is the modeling practice of simultaneously using two or more major software packages to come to one result. Examples of this include: WAVE with Fluent CFD, WAVE with Reaction Design’s KINETICS (combustion analysis), and WAVE with LMS/OPTIMUS. The WAVE code can be combined with a second program for higher-level modeling; in this case an optimization program (i.e. LMS OPTIMUS) to run comprehensive parametric experiments to optimize a complete engine system. Co-simulation typically requires state-of-the-art computing power to be used practically. However, the results of using co-simulation optimization allow for a truly state-of-the-art engine development program.

**Experimental Verification**

In order to properly validate an engine simulation model, the engine must be instrumented with a wide array of sensors with data being logged through data acquisition. For example, if the model assumes the same air-fuel ratio in each cylinder, air-fuel ratio should be logged for each cylinder and fueling should be modified within the engine control unit (ECU) to match the model or vice versa. The implementation of numerous sensors not only allows for simulation validation but also acts as a form of engine diagnostics to provide confirmation that the engine is operating as intended.

The dynamometer used for this research was a Land & Sea water-brake engine dynamometer (dyno). This type of dyno is primarily used for steady-state testing which pairs up nicely with engine simulation work. While eddy-current dynos have more precise control, a water-brake dyno is adequate for many applications. The dyno is interfaced to Land & Sea dyno controls and data acquisition. The controls are operated from a control room overlooking the engine test cell equipped with Dynomax dyno control software. Cooling is performed through the use of a
cooling cart. The cooling cart provides enough cooling to do prolonged steady-state testing. However, it is not able to exactly maintain constant coolant temperatures, which is often desired for accurate benchmarking. Likewise, ambient conditions are not regulated and will typically fluctuate slightly throughout testing sessions.

![Engine Dynamometer Test Stand](image)

**Figure 8: Engine Dynamometer Test Stand**

To allow for frequent engine testing on the dynamometer, a portable engine stand, shown in Figure 8, was adapted to plug directly into the dyno controls and data acquisition systems. The engine was coupled to the dyno at the transmission output shaft by a direct connection constrained by two pillow block bearings.

The engine used for this research is controlled using the Motec M4 Pro standalone ECU. This ECU can be adapted to practically any engine and is one of the most customizable ECU’s on the market with an abundance of features. Advanced features used in this research include: cylinder-to-cylinder fueling, fuel auto-calibration (“quick lambda”), and datalogging. Due to the speed range of the dyno head, testing is typically performed at engine speeds from 4500 rpm to 12000 rpm. Engine coolant and exhaust temperatures are allowed to stabilize before
loading the engine and taking measurements. Torque measurements and sensor outputs are
recorded at 10,000 Hz while engine speed is slowly swept from low to high and then back to low
engine speed. A typical sweep would start at 5000 rpm and wide open throttle (WOT). From the
control room, the dyno controls are used to slowly increase engine speed all the way to 12,000
rpm. The dyno then slowly sweeps back to 5000 rpm. By sweeping through the engine speed
range in both directions, transient error effects can be revealed. For example, if the dyno operator
increments engine speed too rapidly, the dyno will report higher torque output then actual steady
state output. After running an engine speed sweep, the data can then be exported to a spreadsheet
program such as Excel for analysis. Additionally, the software allows for a replay animation that
graphically displays the sensor outputs and test conditions exactly as they were during the test
session.

The controls available from the control room include: throttle actuation, engine speed control,
and emergency shutdown. The dyno equipment is equipped with a data acquisition unit capable
of recording a large array of sensors during engine testing. Typical operating parameters
measured include: intake temperatures, exhaust gas temperatures, manifold pressure, exhaust
back-pressure, air mass flow, engine knock, torque, shaft RPM, throttle position, and air-fuel
eratio. The system is able to control speed to roughly +/- 50 RPM. Due mostly to RPM
fluctuations, a torque variation of about +/- 0.3 ft-lb is seen during testing at a give test point.
4. FINDINGS

The WAVE simulation software was provided through Formula SAE sponsorship and thus all simulation work could only be directly applied to Formula SAE related engine development per Ricardo’s software license. The competition engine used for the past three years at the University of Idaho is the Suzuki GSX-R600. It is a high performance race-inspired sport-bike engine. Formula SAE (FSAE) rules mandate that engine displacement does not exceed 600cc; making 600cc sports-bike engines a popular choice for Formula SAE competition vehicles. In order to maintain competitive competition among teams, Formula SAE officials mandate that the intake system must breathe through a single 20 mm diameter restrictor. This air restriction largely limits the possible power produced and complicates the engine system design. The required layout of the intake system according to official Formula SAE rules is shown in Figure 9. Only one throttle body is allowed, and it must be located upstream of the flow restrictor. Formula SAE rules also have a regulation for the exhaust system in that it must not produce more than 110 db when measured at a distance of 0.5 meters from the exhaust outlet, with the sound meter oriented at an angle of 45 degrees from the exhaust outlet axis on the horizontal plane. A schematic for the sound test setup is shown in Figure 10. This exhaust rule also limits overall power output.
Figure 9: Formula SAE Intake Design Regulations

Figure 10: Formula SAE Sound Test Layout
Engine Package Design

The primary design objective for all FSAE engines is maximum performance (maximum torque at engine speeds where high torque output is desired). Optimal engine performance can be very subjective. Often, the most desirable torque curve is determined by the type of course that the vehicle will be racing on as well as driver preference. If a driver feels that more power is needed at corner-exit, then it may be advantageous to increase torque at lower engine speeds at the expense of some top-end power. Likewise, if the racecourse is very tight and gearing forces the vehicle to drive at lower engine speeds more often than higher engine speeds, it may again be advantageous for the engine to aim for high torque at lower engine speeds. Past experience with the Formula SAE competition and Autocross racing experience suggests that the Formula SAE courses are too small / tight to necessitate targeting torque increase at very high engine speeds. Therefore, design objectives for this naturally aspirated engine package will target torque increase at lower engine speeds more so than higher engine speeds.

Model Setup

The first basic inputs that need to be satisfied lie within the engine itself. Quantities such as bore, stroke, compression ratio, and cylinder-head clearance must be defined. These inputs can typically be found from manufacturer specifications published in service manuals. A method used for the initial model setup involved starting with the engine itself and moving outwards; from the port geometry, to the manifold / headers, to the exhaust outlet and intake inlet. The nearest connection to the engine’s cylinder bores are the intake and exhaust ports. The centerline distance from the valve to top of the ports had to be measured for both the intake and exhaust sides. This is a difficult measurement to make due to the small port size and complex geometry. This centerline distance was approximated using a piece of bent wire. Because the flow through the ports will be defined by the valve flow coefficients determined in flowbench tests, only the inlet / outlet diameter and centerline length must be defined. Once the port lengths are defined, all external intake and exhaust components need to be precisely measured and input into the model. Length measurements are critical for the intake and exhaust components as they affect the pressure wave timing in the ducting. This is especially true for the intake runners and exhaust header primaries. For this application, all dimensions of the intake and exhaust components were retrieved from the original solid models (CAD models) used for manufacturing. Components
such as exhaust headers can be difficult to measure directly from the components, making solid model measurements very valuable.

The intake manifold is one component that is often easier and more reliable to model as a 3D mesh rather than using one-dimensional parameters. Because the modeling of complex y-junctions can be cumbersome, a WAVE Mesher model was created to automatically calculate junction parameters based on the imported CAD model. Without the CAD model, certain volume calculations and area ratios at each manifold junction would have had to be calculated. The flow restrictor is venturi design that has a large converging angle and a small diverging angle. The restrictor was modeled as two tapered ducts joined by a very short 20 mm diameter duct in the center. The throttle plate is represented by an orifice between two ducts with a parameterized diameter. Each throttle position could then be represented by an effective diameter. For this research, wide-open throttle (WOT) performance is only of interest; so this parameter remained constant for most studies.

The air filter and muffler are modeled based on the supporting documentation that was packaged with the WAVE software. The air filter is modeled by inserting a zero-length duct that connects two y-junction volumes. The diameter of this duct is input as smaller than the actual physical diameter to produce a certain pressure drop between the two volumes, often between 1-2 kPa for most air filters. Precise modeling of the restrictor would require flowbench testing or pressure drop measurement on the dyno to accurately model the pressure drop. Resonators and mufflers are modeled similarly to their physical construction. The muffler designed for 2007 engine is a tri-pass design. The tubes in the muffler are represented by ducts while the muffler canister (the outer shell) is represented by a series of y-junction volumes. The volumes attach to the ducts by orifices and baffles within the muffler. The WAVE documentation provides detailed methodology for modeling mufflers.

To define the operation of the cylinder head, the valve lift per incremental camshaft rotation had to be defined. The GSXR cylinder head is dual-overhead cam (DOHC) format, meaning that there is a separate camshaft for the intake and exhaust valves. To make the measurement, valve lift was measured on one camshaft at a time. The valve lift was measured by locating a dial indicator directly above the face of the cam follower. Cam rotation was handled by attaching a
degree wheel to the end of the camshaft and rotating it one degree at a time. These results of
valve lift versus incremental camshaft rotation were related to crankshaft rotation based on
published valve opening points with reference to crankshaft position. The resulting valve lift per
crank revolution is shown in Figure 11.

![Valve Lift vs Crank Angle](image)

Figure 11: Valve Lift per Crankshaft Rotation

To finish defining the cylinder head operation, flow versus valve lift had to be defined. The input
accepted by WAVE is in the form of flow coefficients or discharge coefficients. These
coefficients require empirical flowbench data. The cylinder head was adapted to a flowbench and
volumetric air flow (in CFM) was measured at a pressure drop of 28” of H2O, the standard
pressure drop measurement used on flowbenches in the performance development industry. The
flow data was then converted to a usable flow coefficient format using an excel spreadsheet
provided within the Ricardo WAVE Help documentation. The results are shown in Figure 12.
Figure 12: Cylinder Head Port-Flow

The third advanced input is the combustion model. Ideally, in-cylinder pressure data versus crankshaft rotation should be recorded to calculate Wiebe function parameters. Unfortunately, such data could not be obtained with currently available test equipment. The data published by Blair along with the data that was input into the model can be found in Table 1 in Chapter 2. Some intuition had to be used to determine the parameters due to the large difference in engine operating speed and difference in engine type.

**Model Validation**

There are numerous parameters that can be used to validate an engine model. Parameters such as torque output, volumetric efficiency, and air mass flow can all be used to evaluate overall engine operation. As mentioned, torque output was chosen as the parameter to use for validation due to test equipment limitations.

When using the torque output validation method, certain trends in the predicted torque curve can be validated without too much trouble. The engine testing results will show torque peaks and
valleys that correspond to either intake or exhaust geometry. Those peaks and valleys should occur at the same engine speeds that the simulation predicts. Because these pressure-wave-induced torque peaks result strictly from intake / exhaust geometry and the intake / exhaust valve lift profiles, it is important that these trends can be validated. If the predicted peaks and valleys are not occurring at the correct engine speeds, geometry and valve lift profiles should be investigated and modified for accuracy.

When a torque validation was attempted for this case study, the predictions could not be validated to a reasonable accuracy. To determine whether a given peak or valley was an effect of the intake or exhaust, prior intake runner length and exhaust primary length experiments were investigated to understand the associated trends. The modeled geometry of intake and exhaust ducting did not seem to be at fault. To further troubleshoot the problem, a sensitivity study was performed on intake valve opening location. The purpose of this sensitivity study was to determine the extent of change in torque curve shape due to change in intake valve opening location (in crank degrees). The valve opening point was originally input into the model based on engine specifications of questionable accuracy that were found online. Figure 13 shows how well the model output correlates to empirical results. The predictions are satisfactory considering the limited amount of model input data available. A model cannot be expected to be highly accurate without combustion analysis. The important thing is that this model can predict approximate torque output as well the locations of peaks and valleys in the torque curve. The model does predict the torque peaks seen at 5300 rpm, 8700 rpm, and 10300 rpm. If more empirical data was available, the model could be correlated to errors as low 1-3 percent.
Figure 13: Torque Validation for Naturally Aspirated Model

Sub-System Case Studies

In order to better understand the sensitivity of various engine design parameters, parametric case studies were used to reveal the performance trends associated with those parameters. These studies were run before the model was correlated to levels within 15 percent error. This was due to heat transfer and combustion modeling being incomplete. The overall torque output predicted should be viewed qualitatively because these studies were completed with default Wiebe function parameters. Nonetheless, the results from different studies can be directly compared and important lessons can be learned about how each parameter affects the shape of the torque curve.

Intake Manifold Design

Some engine components can be designed well using one-dimensional simulation while some components demand higher-level modeling to accurately predict performance. Parameters such as intake runner length and tubing diameter can be optimized through one-dimensional
simulation while plenum geometry is better designed through flowbench testing and 3D CFD studies. Design considerations such as cylinder-to-cylinder flow distribution and flow losses through the plenum are generally not predicted very well with one-dimensional simulation. Model predictions have not shown the major cylinder-to-cylinder flow variations that occur due to manifold plenum geometry. The orientation of the plenum inlet and the plenum shape highly affect the cylinder flow distribution.

**a) Runner Length Study**

The length of the intake runners (the tubing between the plenum and intake ports) has a dramatic effect on volumetric efficiency across the operating engine speed range. The unsteady flow nature of an engine creates pressure waves that continuously oscillate at frequencies that change with engine speed. At a given engine speed there is a certain intake runner length that will provide optimal volumetric efficiency. The gain in efficiency is due to a pressure pulse that arrives at the intake valve at or just before the intake valve closure. [Stone, 1999]

There are many simplified intake design models available to predict the engine speed at which the intake runners benefit most from pressure wave tuning, many of which are single equation models. Unfortunately, the simplified models don’t tell the whole story. Specifically, they don’t reveal the negative impacts of selecting certain parameters. For example, very long intake runners can significantly increase low-end torque; however, top-end power gets reduced dramatically. WAVE can show how intake runner length impacts the entire engine speed range so that an intake runner length can be chosen that produces the most desirable torque curve. An intake runner length experiment was performed in WAVE to determine how a given runner length affects the torque curve. Again, this experiment was performed before the model had been highly correlated; so overall output should be viewed qualitatively. This important thing to look at is the shape of the torque curve produced by the varied parameter. The results are shown in Figure 14.
Figure 14: Intake Runner Length Study

It can easily be seen that long-length intake runners significantly improve performance at engine speeds in the range of 6000-8000 rpm while causing severe losses above 10,000 rpm. The current intake runner length is 7.4". From this experiment, it was determined that an intake runner length of 9-10 inches would be best for overall torque output. The mid-range torque is high while high-rpm losses are minimal.

b) Intake Tubing Diameter

The intake tubing diameter also has a large effect on torque output, as shown in Figure 15. Current intake designs use the 1.625” diameter tubing. This tubing is sized best for high rpm torque. In order to improve mid-range torque, the tubing should be reduced to the 1.500” diameter. While smaller sizes present even larger mid-range torque increases, the top-end losses are too severe.
Figure 15: Intake Tubing Diameter Study

Figure 16: Intake Tubing Diameter Study with Tapered Runners
Figure 16 illustrates the effects of using tapered runners compared to the standard constant-diameter intake runners. The results indicate that a tapered runner will show very similar performance to that of constant-diameter runner with a diameter that is in between the diameters of the tapered runner. It can be seen that the 1.250” to 1.625” taper behaves very similarly to the 1.500” constant-diameter runner. Simulation results indicate that tapered runners are not necessary. However, flowbench results would be needed to validate this; as the 3D flow at bellmouth inlet in the intake manifold is not adequately characterized by 1D models.

**Exhaust Header Design**

Unlike intake design, exhaust manifolds do not contain a plenum. The reason being, that the purpose of the exhaust manifold is to allow for the combustion products to rapidly exit the combustion chamber. If a plenum is used, a large volume of exhaust would become pressurized and prevent efficient cylinder scavenging of combustion products. Instead, long individual tubes are used to prevent one cylinder’s exhaust gas from pressurizing another cylinder. The size of the tubing is generally a compromise between mid-range and high-rpm power. Small diameter tubing maintains high exhaust gas velocities at lower mass flow rates, increasing exhaust scavenging efficiencies. At high higher engine mass flow rates, the exhaust headers will perform better with a slightly larger tubing diameter. The performance of exhaust headers are most largely affected by the diameter and length of its primary tubing. The secondary and tertiary (for 4-2-1 designs) geometry has noticeable effects but the primary tubing geometry is the dominant variable in the design. Figure 17 illustrates the two basic approaches to exhaust header design. Generally, a 4-into-1 design will produce the most power at the cost of: increased weight, packaging complications, and increased sound levels. The 4-2-1 will typically increase mid-range power at the cost of losing some top-end power. The design format used by the Formula SAE team has traditionally been the 4-1 design due to specific packaging and weight constraints. [Bell, 1998]
a) Exhaust Primary Length

The 2007 University of Idaho Formula SAE team made overall vehicle weight a primary design objective. This limited the options for exhaust header design. While it was determined that long-length headers could provide significant low-rpm torque increases, the gain in weight would conflict with the overall vehicle design objective. The figure below shows the torque curves for various header primary lengths. For packaging and weight-savings, a length constraint of 18-22 inches was applied. Predictions for a long-length design were included in the figure to illustrate the potential torque increases available. Based on these predictions shown in Figure 18, a 20 inch primary would be good for the length constraints given. The 22 inch length significantly reduces low-end torque and the 18 inch exhibits torque losses in the mid-range. The performance gains associated with a long-length design (40 inch primaries) are shown, but do not comply with the physical design constraints. A better torque curve could be achieved at the cost of almost twice the weight.
b) Exhaust Tubing Diameter

The tubing diameter of the header primary tubes also has a large effect on torque output. Because exhaust tubing is generally only manufactured in certain diameters, only those diameters (within practical limits) were investigated. While intuition may suggest that large-diameter header tubes would be best for a performance application, the 1D simulation reveals otherwise. The simulation shows that significant flow losses don’t occur until the tubing diameter is down-sized to a 1.000” outer diameter (OD). Figure 19 shows that both the 1.250” OD and 1.375” OD produce good torque curves, with the 1.250” OD increasing mid-range torque the most without significant high rpm losses. The 1.250” OD is recommended for future designs based on these findings.
The secondary tubing geometry does not have nearly the effect on torque output that the primary tube geometry has. Figure 20 indicates that a larger secondary tubing diameter slightly improves mid-range torque. However, the secondary tubing also has a large effect on sound output. The tubing diameter can be guided by the muffler design knowing that for each incremental downsize in tubing diameter, there is a small loss in mid-range torque. Figure 21 presents the effects of the secondary length. The secondary length has only minor effects on torque output and should be selected based on engine packaging constraints.
Figure 20: Secondary Tubing Diameter Study

Figure 21: Exhaust Secondary Length Study
Figure 22: Collector Length Study

An experiment was performed to determine if the y-junction length parameters for the exhaust collector have a significant effect on the pressure wave tuning of the engine. The results shown in Figure 22 indicated that those length parameters do in fact affect the pressure wave timing to a similar extent that was seen in primary length study. However, this study was only performed for model calibration reasons, not for engine design studies. The actual design of the collector should be based on minimizing flow losses without concern of component length.

Exhaust System Design Recommendations

The Formula SAE competition is a student engineering competition. Team designs are judged by industry professionals who are more concerned with why a particular design path was taken as opposed to the outcome. Designing an engine for optimal power independent of the vehicle itself is not what the competition is about. A great example of this is the exhaust headers. The judges
know that a 20” primary 4-1 design is not the optimal exhaust system. They want to know why a 20” length is chosen and why it’s better than alternative designs for the vehicle. With that said, an exhaust system design that is guided by the Formula SAE competition and the team’s recent goals for major weight reduction will be recommended. Gains available from a slightly heavier design will also be presented for comparison purposes. Table 2 shows the geometry of three designs for the exhaust headers: the current design on the 2007 engine, a recommended design for 2008, and long-length alternative. The relative torque increases for these header designs is presented at the end of this chapter.

Table 2: Exhaust Headers Design Specifications

<table>
<thead>
<tr>
<th>Headers Design</th>
<th>Headers Type</th>
<th>Primary Length (inches)</th>
<th>Primary Diameter (inches)</th>
<th>Secondary Length (inches)</th>
<th>Secondary Diameter (inches)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2007 Design</td>
<td>4-1</td>
<td>20</td>
<td>1.500</td>
<td>16</td>
<td>1.500</td>
</tr>
<tr>
<td>Recommended 2008 Design</td>
<td>4-1</td>
<td>20</td>
<td>1.250</td>
<td>*</td>
<td>*</td>
</tr>
<tr>
<td>2008 Design (long length alternative)</td>
<td>4-1</td>
<td>40</td>
<td>1.250</td>
<td>*</td>
<td>*</td>
</tr>
</tbody>
</table>

* Can be designed exclusively for packaging due to engine output showing low-sensitivity to these parameters

**Compression Ratio Study**

Modifying the compression ratio is another method of increasing overall torque output from the engine. Because the engine in this application is restricted, raising the compression ratio is a practical improvement to the engine. In its restricted operation, the engine does not operate near its performance limits where engine knock and detonation become issues. WiseCo manufactures high performance pistons for this engine that increase the compression ratio to 13.3:1 from 12.2:1. The higher the compression ratio, the more torque the engine can produce. However, there are diminishing returns for very-high compression ratios and knock sensitivity, detonation, and piston-to-valve clearance become issues. As seen in Figure 23, increasing the compression ratio (CR) from 8.0 to 10.0 provides a much higher torque increase than increasing from 13.3 to 15.0. Due to the complexity and marginal gains attributed to modifying the compression ratio higher than 13.3, it is recommended that no more changes be made.
Throttle Body Study

The design of a throttle body for an engine has a large effect on the overall feel and driveability of the vehicle. Because the throttle must be mounted upstream of the restrictor, having a large diameter throttle body will have only marginal gains. For this application, an oversized throttle body will provide poor throttle response with nearly 90 percent of peak torque occurring at throttle positions as low as 50 percent. Prior testing with a large diameter throttle body revealed this issue. To avoid the pitfalls of an oversized throttle body, torque predictions were run to predict torque output losses for various throttle body diameters. The results in Figure 24 show that throttle body performance maxes out with a 33-35 mm diameter bore. Diameters above 35 mm provide no significant gains and actually decrease performance due to the rapid transition from a large-bore diameter to a 20 mm diameter throat in the restrictor. Any diameter from 30-40 mm would be acceptable; however, throttle response decreases as the diameter gets larger than 30 mm. It is important to mention that this study only analyzes the throttle bore diameter. The
throttle plate is ignored in the model so the actual bore diameter may need to be slightly larger than predicted due to slight flow losses that may occur due to the throttle plate. A 35 mm diameter throttle body is recommended based on these model predictions and the above assumption.

Figure 24: Throttle Body Diameter Study
5. CONCLUSIONS/RECOMMENDATIONS

The objective of the naturally aspirated case study was to investigate the potential for performance increases for the 2007 Formula SAE competition vehicle, to be applied for the 2008 competition. Recommended engine parameter values have been compiled in Table 3. Some values are recommended for reasons that are practical for team implementation. For example, model results predict that a throttle body of 33-35 mm diameter would provide the best combination of throttle response and power output. The current throttle body has a 38 mm diameter. It is recommended that this design be reused as it is not practical for it to be redesigned for only very marginal performance gains. The performance gains associated with the new intake and exhaust system recommendations are shown in Figure 25. Line number 3 is the recommended engine package for 2008 and line number 1 is the current 2007 design.

![Engine Configuration Performance Comparison](image)

**Figure 25: Engine Performance Comparison**

(1) 1.500" OD Exhaust Tubing, 20° Primaries, 7" Intake Runners with 1.625" OD Tubing
(2) 1.250" OD Exhaust Tubing, 20° Primaries, 7" Intake Runners with 1.625" OD Tubing
(3) 1.250" OD Exhaust Tubing, 20° Primaries, 9" Intake Runners with 1.500" OD Tubing
(4) 1.250" OD Exhaust Tubing, 40° Primaries, 7" Intake Runners with 1.625" OD Tubing
Table 3: 2008 NA Engine Recommendations

<table>
<thead>
<tr>
<th>Intake System Design</th>
<th>Current Design</th>
<th>2008 Recommendations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Manifold Type</td>
<td>Side Inlet, Tapered-Log Type Plenum</td>
<td>Center Inlet, Trapezoidal Distribution Manifold</td>
</tr>
<tr>
<td>Runner Length</td>
<td>7.4&quot;</td>
<td>9.0&quot;</td>
</tr>
<tr>
<td>Runner Diameter</td>
<td>1.625&quot; OD</td>
<td>1.500&quot; OD</td>
</tr>
<tr>
<td>Compression Ratio</td>
<td>13.3:1</td>
<td>SAME</td>
</tr>
<tr>
<td>Throttle Body Diameter</td>
<td>38 mm</td>
<td>SAME</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Exhaust System Design</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Header Type</td>
<td>4-1, for weight savings and packaging</td>
<td>4-1, for weight savings and packaging</td>
</tr>
<tr>
<td>Primary Length</td>
<td>20.0&quot;</td>
<td>20.0&quot;</td>
</tr>
<tr>
<td>Secondary Length</td>
<td>16&quot;</td>
<td>*</td>
</tr>
<tr>
<td>Primary Tubing Diameter</td>
<td>1.500&quot; OD</td>
<td>1.250&quot; OD</td>
</tr>
<tr>
<td>Secondary Tubing Diameter</td>
<td>1.500&quot; OD</td>
<td>1.500&quot; OD</td>
</tr>
<tr>
<td>Muffler</td>
<td>Tri-pass</td>
<td>SAME</td>
</tr>
</tbody>
</table>

* Can be designed for packaging

As previously discussed, the flow losses and flow distribution in the intake manifold are not accurately modeled with the one-dimensional code unless flow-loss coefficients are input based on test measurements. The 2007 manifold design utilized a side-entry manifold type. While the one-dimensional code did not predict major flow-distribution problems, engine testing did. The flow variations were so severe that if fueling was not controlled on a cylinder-to-cylinder basis; one or more cylinders would not even operate due to the inaccurate fueling. Figure 26 shows the two manifold types. The side-entry manifold, shown on the left, is similar to design that was tested and currently installed on the 2007 vehicle. The center-inlet manifold, shown on the right, provides a much more direct passage to all cylinder runners. It is recommended that the 2008 engine package uses a manifold of this type to provide better flow distribution and better overall cylinder filling. The orientation of the inlet to the plenum can be modified and for sound test reasons, should be directed away from the exhaust outlet.
Future work on the naturally aspirated model should be focused on the configuration of crank-angle resolved measurements of in-cylinder pressure and pressure traces in the intake and exhaust system. All data needed to properly define the engine model should be acquired. Once the model is validated, the resulting accuracy of the model and the source of error should be documented.
6. REFERENCES


WAVE Knowledge Center v7.0p3. Ricardo Software. 2007.