Users Guide

For Dynomation6 and DynoSim6

On Windows 7, 8, 10 & 11

Motion Software Inc.

Wave-Action Filling & Emptying Simulations Version 6

Dynomation6™ and DynoSim6™ Engine Simulations

MOTION SOFTWARE, INC. SOFTWARE LICENSE

ACKNOWLEDGMENTS, ETC.

CONTENTS

CONTENTS

CONTENTS

CONTENTS

CONTENTS

CONTENTS

Wave-Action Filling & Emptying Simulations Version 6

INTRODUCTION

Note: *If you can't wait to start using this engine simulation, feel free to jump ahead to* **INSTALLATION** (or review the 16-page QuickStart Guide supplied with your software), *but don't forget to review this manual when you have time. Also, please complete the* **Registration Form** *that appears when you first start your software. It entitles you to receive tech support, obtain free automatic program updates and more.*

And make sure you have a look at our growing list of tech videos on YouTube: **youtube.com/@motionsoftwaresim**

Thank you for purchasing Dynomation6™ or DynoSim6™ for Windows from Motion Software, Inc. This software is the result of many years of simulation development and programming effort that included program optimizations developed over twenty years of on-going user feedback. This simulation was built, from the ground up, to be easy-to-use, yet it is fully capable of advanced engine-simulation analysis that can reveal WHY the IC (Internal Combustion) engine functions as it does. You can view pressure waves and mass flow that take place inside the intake and exhaust passages. You can easily change components and measure their affect on these pressure waves and on engine power. And you can learn more about how engines work than you ever thought possible. Not only that, we are confident that you will find this simulation to be an intuitive, easy-to-use program that makes engine testing and analysis simple and fun! By eliminating the frustration that is common with many engineering-oriented

Part No. DYM6   www.MotionSoftware.com

DYNOMATION6
P R O F E S S I O N A L
Easy-To-Use Wave-Action Engine Simulation

Includes Powerful Simulation Features:
• View Intake & Exhaust Pressure Waves
• Advanced Camshaft & Lobe Modeling
• *Fitter*™ Code Guarantees Accuracy
• Detailed Forced-Induction Modeling
• Diesel Modeling, *Iterator*™ Testing
• And Much More!

Explore and optimize intake and exhaust flow inside a running engine!

Used By Professional Engine Builders and Race Teams Worldwide

Dynomation6™ (the full Wave-Action simulation) is the most comprehensive yet easy-to-use engine simulation software package ever offered to performance engine designers, builders and enthusiasts. Dynomation6 provides full detail simulatoin analysis, optimum accuracy and a robust modeling toolset.
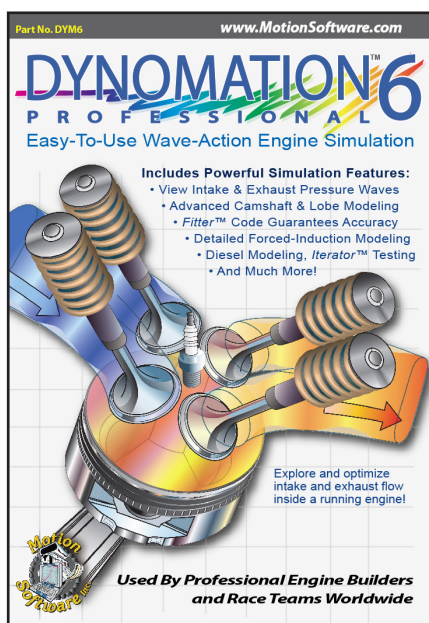
Introduction To Version6 Simulations

Main Program Screen
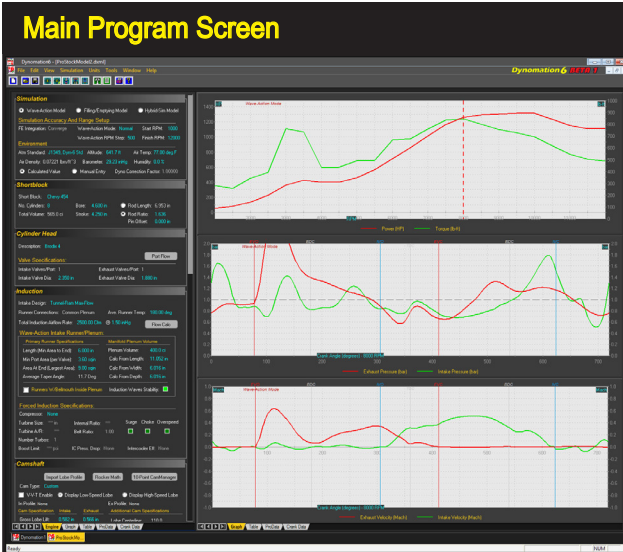
The simulation incorporates a completely unique, intuitive user interface (shown using one of several program color schemes). If you wish to change an engine component, simply click on any component field on the left side of the screen and select a new specification from the drop-down list or enter custom values. Engine components are shared between both the Filling-And-Emptying (FE) and Wave-Action (WA) simulation models. Results can be displayed in a wide variety of tables and graphs.

simulations (or the huge expense involved in long dyno-test sessions), you are free to "play," using your imagination to uncover power secrets for single or multiple-cylinder, four-stroke engines for automotive, racing, or a myriad of other applications.

What Are DynoSim6 and Dynomation6?

At the core of Dynomation6 and DynoSim6 are mathematical models that simulate four-stroke, internal combustion (IC) engines with remarkable accuracy. These simulations incorporate two distinct models: 1) A *Filling-And-Emptying* (FE) method, provided in both DynoSim6 and Dynomation6, that can calculate a complete engine simulation with fewer data iinputs, making this technique a fast and accurate way to optimize engine designs, and 2) A full *Wave-Action* (WA) method exclusively in Dynomation6 that calculates and predicts the complex pressure-wave dynamics and particle flows in intake and exhaust
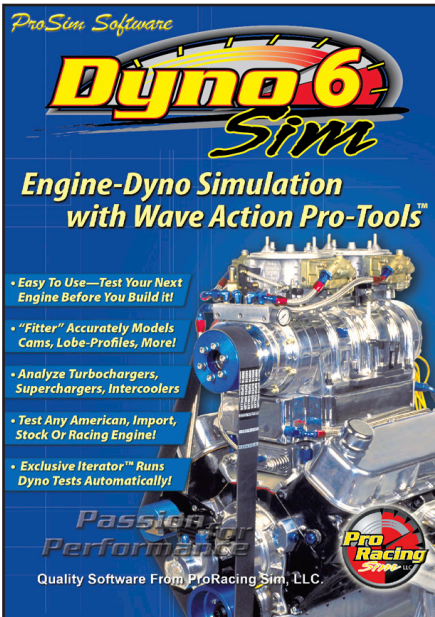
DynoSim6™ uses FE (Filling-and-Emptying model with pressure-wave analysis) simulation that offers ease of use and excellent accuracy. It includes a comprehensive toolset (developed in years of Dynomation6 developmenet) that add powerful calculators, cam-timing and analysis tools and much more.

passages. The *Wave-Action* model picks up where the *Filling-And-Emptying* method leaves off and "homes in" on the best port sizes and shapes, runner lengths, header configuration and tubing dimensions, cam timing, valve motion, and other engine parameters, providing unprecedented accuracy for the serious engine designer and builder.

Both DynoSim6 and Dynomation6 use a carefully-crafted *Direct-Click™* user interface that lets you select parts with a single mouse click. Multiple graphs and tables display power, torque, engine pressures and many other engine variables both independently and in comparison views.

You'll also find hundreds of other features that have become Motion Software's trademarks, including: 1) rapid *Iterative™* (fully automatic) testing, 2) comprehensive camshaft, cylinder-head-flow, and engine-file imports, 3) built-in calculators for compression ratio, airflow conversion, rocker-arm and cam-timing math, 4) side-by-side graph *QuickComparisons™*, 5) professional multi-page *ProPrint™* dyno-test reports, and more! We are confident that you will find Dynomation6 and DynoSim6 to be the most capable engine simulation software packages available for engine professionals and serious enthusiasts.

How Dynomation6 And DynoSim6 Work

Dynomation6 and DynoSim6 are Windows 7, 8, 10 and 11, 32- and 64-bit, engine-dynamometer simulations that utilize *full-cycle* analysis, meaning that they calculate the complete fluid-dynamic, thermodynamic, wave-dynamic, and frictional conditions that exist inside each cylinder throughout the entire 720 degrees of the four-cycle process.

Other simulation programs may simply calculate the volumetric efficiency (VE) and then derive an estimate of torque and horsepower. There are many shortcomings to this and similar "shortcut" techniques. The two greatest drawbacks are: 1) since cylinder pressure is not precisely determined, it is impossible to predict the pressure

Iterative Testing™ **is a power-ful feature included in both the** *Filling-And-Emptying* **and** *Wave-Action* **models. This screen illustrates a test evaluating a series of components (over 200 dyno tests have been performed). Using this powerful tool it is possible to automatically run thousands or even hundreds of thousands of tests to find the best combinations. The simulation keeps track of all the results and displays the best matches to your test criterion.**
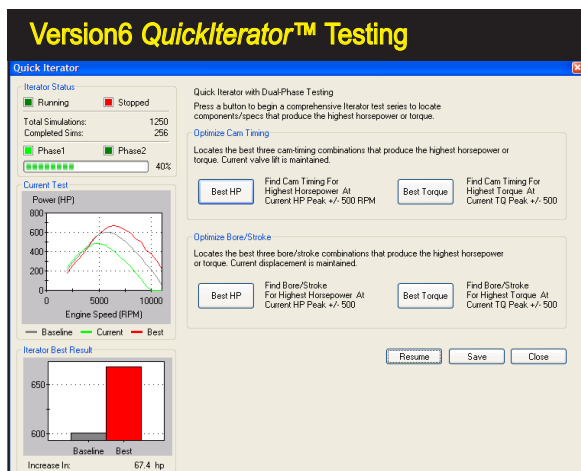
on the exhaust valve and the subsequent mass flow through the port when the exhaust valve opens, and 2) the inability to accurately determine pumping horsepower losses (energy needed to move gasses into and out of the engine) substantially reduce accuracy and applicability across a wide range of engines.

Version6 simulations incorporate models that include frictional, charge flow, and pumping-loss calculations. Extensive computation is required for each power point, in fact, the simulations perform several HUNDRED MILLION calculations at each rpm point throughout the rpm test range. However, due to the advanced mathematical analysis possible with modern Giga-Hertz-speed computers, most engine simulations can be completed in about one second in DynoSim6 and, typically, 1 to 10 seconds in Dynomation6!

The in-depth analysis provided by version6 simulations can be applied to a vast range of engine designs. Both DynoSim6 and Dynomation6 have been successfully used to model single-cylinder "lawn-mower" engines, light aircraft engines, automotive engines, 2-, 3-, 4-, and 5-valves-per-cylinder, high-performance racing engines, ProStock drag-racing powerplants, Formula-1 engines, multi-thousand horsepower supercharged, nitrous-oxide injected "mountain motors," and many other engine designs.

WHAT'S NEW IN Version6

If you have used earlier versions of our simulations, you will find version-6 simulations offer a significant upgrade. During the development cycle for these products, many aspects of the program were improved, including cam-timing and lift-curve modeling with the use of all-new *Fitter* (accuracy improving math) routines, enhanced user interface functionality, new graph-and-table results displays, more accurate forced-induction modeling with improved Compressor Map and Engine Demand Line displays, plus substantial improvements in exhaust-system modeling, both in DynoSim6 and Dynomation6. Here's a short list of some of the new features and changes in version6 simulations:

- Exhaust modeling in WA supports three primary sections (pipes), two collector sections, and a megaphone from which you can design any header system.
- Exhaust modeling in FE now includes stepped headers, Tri-Ys, and more.
- "Fitter" math routines ensure optimum valve-motion accuracy.
- New fuels and Diesel modeling supported; CRs extended to 30:1.
- *Nitrous-Oxide* injection now uses step-wise analysis of Nitrous charge.
- Improved *Latent Heat Of Vaporization* fuel modeling.
- Intake runner temperature modelling.
- WA "optimum cycle count" automatically determined.
- Pressure-Wave Stability indicators give at-a-glance warning of issues.
- WA Manifold-Plenum and Runner-Bellmouth modeling.
- FE displays pressure waves and particle flow velocities, similar to WA.
- FE includes new exhaust models and tubing-size prediction.
- Automatic step-by-step analysis of optimum exhaust pipe lengths.

Introduction To Version6 Simulations

- WA Iterator finds best Intake Runner-Length and Runner-Areas.
- Improved Forced-Induction modeling and Turbo Map displays.
- Model up to four compressors (any type) on any engine at once.
- Simulation Log feedback helps diagnose modeling issues.
- Graph Coefficient of Discharge (CD) and Curtain Areas for all valves.
- Tables display Mass Flow, in Lb/min and Kg/sec, at each RPM step.
- Enter custom conversion factors for Brake-to-Wheel Horsepower.
- Improved Units handling for US and Metric; new Hybrid Unit modes!
- Motion has released a series of YouTube Videos (with more to come) that are intended to provide tech tips and support for Version6 and earlier simulations. Check them out here: ***www.youtube.com/@motionsoftwaresim***
- Our videos will help you find even more simulation tools and features!

The display and analysis of pressure waves in the intake and exhaust ports also has been improved in Version 6. Click on the power/torque graph to select any rpm point; you will instantly see the status of pressure waves and mass flow for that engine speed for **both** the WA and FE simulations. Drag the reticule line on the power/torque graph; the pressures and flow dynamics will instantly update as you scan through the rpm range. This capability provides an unprecedented view of pressure and particle flow within a "running" engine for both WA and FE simulations.

Enhancements were added to *Iterative Testing*,™ an exclusive feature of Motion Software simulations. *Iterative Testing* allows you to automatically perform thousands of dyno tests, keep track of all the results, and locate the best component combination that matches your search criterion. In addition to cam valve-event timing, intake manifold selection, and bore and stroke values, you can now *Iterate* cam centerline timing and intake runner lengths and areas (port length and area iteration applies only to the WA simulation).

Version6 also offers many additional improvements. Diesel modeling is now supported with compression ratios as high as 30.0:1. *Latent Heat of Vaporization* modelling better simulates intake-charge temperatures, particularly with supercharges and racing fuels, like methanol. Manifold runner wall temperature can be modeled to directly support "air-gap" manifolds and other unique manifold/runner ambient conditions. Cam *Dynamic Stability Indicators* have been included to alert the user if the current cam timing values have caused stability issues. Additional Dynomation6 indicators in the Induction and Exhaust categories show potential issues (such as shock waves) in the intake and exhaust runners.

But most important is what hasn't changed: *Our top-tier tech support*. Motion Software, Inc., development staff will be available to help you if you run into issues that you can't solve on your own. We are always interested in helping you succeed with our simulation software. Let us know how we can help!

Program Requirements

Here are the basic hardware and software requirements needed to run our simu-

Introduction To Version6 Simulations

Dynomation6 and **DynoSim6** use two distinct simulation models: The *Filling-And-Emptying* (used in DynoSim6) and the *Wave-Action* (used in Dynomation6). Each of these simulation methods have their strong points, and just like selecting the right tool for the job, selecting the best simulation model will help you find optimum component combinations in the shortest time and at the lowest cost. Refer to page 264 for further details about each Simulation's Capability (the numbered items in this chart).

Motion Engine Simulation Capabilities	Filling-And-Emptying Simulation DynoSim6 and Dynomation6	Wave-Action Simulation Dynomation6 Only
Dynomation6 Wave-Action And DynoSim6 Filling-And-Emptying Functional Differences		
Dyno-Testing RPM Range	1000 to 14500 rpm	1000 to 14500 rpm
Bore Range Limits	2.00 to 7.00-in	2.00 to 7.00-in
Stroke Range Limits	1.50 to 7.00-in	1.50 to 7.00-in
Forced-Induction Modeling (1)	Yes	Yes
Extended-Data Displays (2)	Yes	Yes
ProPrinting™ Extended Dyno Reports (2)	Yes	Yes
ProData™ Tables (4)	Yes	Yes
Graph DataZones™ (5)	Yes	Yes
Prolterator™ Testing (6)	Yes	Yes
Analyze Area Under Power And Torque Curves (7)	Yes	Yes
Full Wave-Action Simulation (8)	No	Yes
Analyze Induction & Exhaust Pressure Waves (9)	Simplified Modelling	Full Analysis
Model Intake & Exhaust Runner Lengths & Taper Angles (10)	No	Yes
Megaphone/Normal Headers (11)	No	Yes
Transparent 3D-Engine Pressure-Analysis Display (12)	Yes	Yes
Graph Reticle Synchronized With 3D Engine Analysis (13)	Yes	Yes
Import Earlier Engines, Port Flow And CamFiles (14)	Yes	Yes
Import FlowPro Flow-Bench Files (15)	Yes	Yes
Import CamPro+, S96, CamDoctor Lobe-Profile Files (16)	Yes	Yes
Supplied With 3500+ Lobe-Profile Library (17)	Optional	Included
View 3D-Engine Mass-Flow Analysis (18)	Yes	Yes

lations:

- A Windows-compatible PC (with a CD-ROM drive if you purchase our products on CD, however, a CD not required for *Direct Download* software delivery).
- A USB Port for the supplied Security Key when running Dynomation6 (see *INSTALLATION* for more information on program security).
  Note: The Dynomation6 Security Key is *not* compatible with Microsoft *Insider Program* versions of Windows.
- A minimum of 4GB of RAM (random access memory) for Windows 7, 8, 10 and 11. More memory than this can improve program performance.
- Windows 7, 8 , 10 or 11 (32- or 64-bit). *Note: Version6 software **may** run on earlier versions of Windows (such as XP and Vista), but these installations are not supported.*
- An Internet connection is required to obtain *Free* program updates (you can receive periodic updates, all free for registered users of Version 6 software).
  Note: Automatic updates over the Internet are not supported in Windows 2000 installations.
- A video system capable of 1280 x 1024 or higher will optimize screen display of engine components and graphics.
- A fast system processor (the faster the better) will improve processing speeds; especially helpful for *Wave-Action* and *Iterative* testing. However, Version6 simulations will operate on any Windows qualified system, regardless of processor speed.

Introduction To Version6 Simulations

- A mouse/mousepad/trackball is required for cursor movement and component selection.
- Any Windows compatible printer (to obtain dyno-test printouts).
- In Dynomation6, you can export simulation data in a format that is readable by Microsoft Excel (spreadsheet). Excel (or a compatible product) is needed to utilize this feature.

Additional System Requirements
And Considerations

Windows 7, 8, 10 and 11: This software is fully compatible with Windows 7, 8, 10 and 11 (either 32- or 64-bit versions). Make sure to install all the latest service packs and updates (use the Microsoft Windows Update feature to update your operating system).

Windows 2000/XP/Vista: Our testing shows that Version6 simulations will usually operate properly on WindowsXP and Windows Vista. Make sure to install all the latest service packs and updates (use the Microsoft *Windows Update* feature or visit www.microsoft.com to locate updates and service packs for your operating system).

    Note-1: *Windows95 is <u>not</u> supported.*

    Note-2: We recommend that you run our simulations on Windows 10 or 11 machines. Our testing shows that these operating-system versions are the most sophisticated and reliable (and best supported).

Video Graphics Card And Monitor: An 800 x 600 resolution monitor/video card is required to use version6 simulations. Systems with 1280 x 1024 (HD) resolution will obtain more screen "real estate," and this is very helpful in component selection and power/pressure-curve analysis. Screens of 19-inches or larger with 1280 x 1024 resolution or higher are optimum for simulation testing.

System Processor: Version6 engine simulations are extremely calculation-intensive. Over 2 billion mathematical operations are performed for each complete power-curve simulation. While the program has been written to optimize speed, a faster processor will improve data analysis capabilities. Furthermore, our simulations incorporate powerful *Iterative Testing* functions that perform an analysis of hundreds or thousands of simulation runs. To reduce calculation times and extend the modeling capabilities of the program, use the fastest processor practicable.

Mouse: A mouse (trackball, or other cursor control device) is required to use this software. While many component selections can be performed with the keyboard, several operations within this simulation require the use of a mouse.

Printer: Dynomation6 and DynoSim6 can print a comprehensive *ProPrint™* "Dyno-

Introduction To Version6 Simulations

Test Report" of the simulated engine on any Windows-compatible printer. If you use a color printer, the data curves and component information will print in color.

Wave-Action
Filling & Emptying
Simulations

Version **6**

INSTALLATION

Note: In most cases, you can install Motion Software products simply by following on-screen prompts. As additional help, the following steps 1 through 8 provide information that will ensure a trouble-free installation. If you skip this part, we recommend all users review post-installation instructions later in this chapter:

Installation Overview:

• **Dynomation6—**As the purchaser/owner of Dynomation6, you are granted a license to install the simulation on as many computers as you wish, however, Dynomation will only startup and run on a computer that has the *USB Security Key* plugged into a USB port on your system (the USB Key is supplied in your software package).

• **DynoSim6—**As the purchaser/owner of this DynoSim6, you are granted a license to install this simulation on two computers for your personal use. Installation on more systems, or "loaning" this software to friends, is a violation of the Motion Software License agreement.

• *Make sure all other applications are closed before you begin this installation.* The best way to do that is to restart your system, then begin this installation <u>before you start any other programs</u>. If other programs are using system resources during Dynomation installation, your computer may appear to "lock-up" when in fact, another application ("hiding" in the background) has taken focus away from the Dynomation installer.

• **Note:** *You must have Administrator Rights to properly install Dynomation under Windows 7, 8 and 10. Administrator Rights are also required to receive automatic updates over the Internet.*

Program Installation Steps

1) Close all other applications before you begin this installation.

Installing & Starting Version6 Simulations

2) Insert the simulation CD-ROM into your CD drive.

3) A Software Installation Menu will open on your Desktop within 5 to 30 seconds. Click the *Install* option.
Note: If the Software Installation Menu does not automatically appear on your

Dynomation6 *Install* Menu

From the options provided in the Installation Menu, select *Install Dynomation6* or *DynoSim6.* If the Menu does not appear, see the Note in Step 3.

desktop within 30 to 60 seconds, view the contents of the install CD using *Windows Explorer* (not *Internet Explorer*) and double-click on *Dynomation6_InstallMenu.exe* or *DynoSim6_InstallMenu.exe* to begin program installation.

4) After you select an *Install*, allow up to two minutes for the program installer to read files from the CD and display an opening window. Click *Next* to view the Motion Software License Agreement. Read the Agreement and if you agree with the terms, click *I Accept....*, then click *Next* to continue the installation.

5) A Readme file is now displayed that includes information about installation and program updates. After you have reviewed the Readme, click *Next* to proceed with the installation.

6) *Important Note:* This software will be installed on its default path (on your boot drive, C:\, in the root). This location is essential to ensure that future updates install properly.

7) The installer *Start Installation* display gives you a chance to review the license agreement or readme info. Press *Install* to begin installation.

8) This step is for **Dynomation6 Only:**

Installing The USB Security Key

Plug the USB Security Key (the small USB device supplied with Dynomation6) into any available USB port on your computer. This key is licensed to you, the purchaser of this software, and will allow you to run Dynomation6 on any of your computer systems. You are licensed to install Dynomation6 on as many computers as you wish, however, Dynomation6 will only run on one system at a time; the computer with the Security Key installed.

Installing & Starting Version6 Simulations

The latest Sentinel-HASP USB Security Key driver will install next. This driver is required for Dynomation6 to communicate with the USB Security Key provided in your software package.

Note: If you do not have an available USB port (your computer must have at least one free USB port to run Dynomation6), you can install a USB Card or Hub to extend the number of available USB ports. The Dynomation6 Security Key functions properly with most external USB Hubs.

Next, a dialog box <u>may</u> appear and ask for permission to install Microsoft DirectX on your system (DirectX is required for 3D animations within the software). If you have the same or a newer version of DirectX already installed, the installer will detect its presence and will not overwrite newer files.

Solving USB Security Key Issues

If Dynomation6 displays an error message that the Security Key (or HASP) is missing, here are some quick steps you can follow to isolate and correct this problem:

a) Restart Windows after you install Dynomation6 to make sure the USB Key drivers are loaded and running.

b) Make sure the Security Key is, in fact, properly connected to a functioning USB port on your computer or has been plugged in a USB hub that is connected to your computer. If you plugged the Key into a hub (rather than into a USB port on the computer), try connecting it directly to a port on your computer system.
Note: The Security Key contains a small red LED that illuminates when it is properly connected and communicating with its software drivers.

c) Disconnect all other USB devices from your system, then reconnect the Dynomation6 Security Key (try a different port if possible).

d) Try reinstalling the Security Key drivers by reinstalling Dynomation6 from the program CD (you do not need to un-install first), or install the latest driver posted on our Support page (www.motionsoftware.com/support.htm).

e) If your computer is experiencing technical difficulties, such as non-functional devices, spontaneous rebooting, numerous system messages, etc., the device drivers for the Security Key may not function properly on your system. You must have a stable computer system and a clean, virus-free Windows installation to properly use this simulation.

f) As a "last resort," try installing the simulation (and the Security Key) on a second

computer system to determine if your original computer has compatibility issues.

POST-INSTALLATION SETUP
For Both DynoSim6 and Dynomation6

Installing A 10-Point CamDisk Library

9) CamDisks are additional libraries of 10-Point camfiles (much more information on 10-Point camfiles is provided later in this User Guide). If you wish to install a 10-point library (from a separate install CD or it can be included on the program installation CD), click the *Install* option on the Program Installation Menu.
Note-1: CamDisk camfiles can only be installed <u>after</u> the simulation has been successfully installed on your system.
Note-2: 10-Point camfiles are NOT the same as Lobe-Profile files; refer to the *Camshaft Category* later in this User Guide for more information on the differences between 10-Point valve timing and lobe-profile specifications.

Installing A Lobe-Profile Library

10) Motion Software offers libraries of cam-lobe profiles that allow version6 simulations to model exact valve motion and predict engine power with the highest accuracy. Lobe-Profile files consist of data that "maps" the entire shape of the lobe, not simply the valve opening, closing, and maximum lift points used in 10-Point Camfiles. If you wish to install a Lobe-Profile library, click on the Install option on the Program Installation Menu that will appear on your desktop after you insert the Profile CD into your CDROM drive.
Note: Profile Libraries can only be installed after a version6 simulation has been successfully installed on your system.

Starting The Simulation
For The First Time

11) To start the simulation, double-click the *Dynomation6* or *DynoSim6* program icon that was installed on your Desktop. Alternatively, you can open the Windows START menu, select All Programs or Apps, then choose *Motion-Dynomation6 Engine Sim*, or *Motion-DynoSim6 Engine Sim* and click on the *software* icon displayed in that folder.
Dynomation6 Only: If Dynomation6 displays an error message indicating that the Security Key (HASP) is missing or cannot be found, refer to the information on the previous page (*Solving USB Security Key Issues*).

RegisteringYour Software

12) When you first start the simulation, a Registration dialog will be displayed.

Installing & Starting Version6 Simulations

Please fill in the requested information, including your Serial Number found on the QuickStart Guide provided in your software package.

Note: If you are installing software that you have downloaded from the Motion Online Store, you will find a temporary registration number visible just above the Registration Number data-entry fields in the Registration Dialog. Enter this temp reg number until you receive your permanent number from Motion Software...you will only have to enter your permanent number once.

Press the *Register Now!* button to send your registration information to Motion's servers; an Internet connection is required for data submission. If you do not have an Internet connection, you will be presented with other registration options. If you do not register this simulation, you may not qualify for tech support or free updates.

If you move or change your email address, you can update your registration information at any time simply by selecting *Registration* from the HELP menu. **IMPORTANT:** Keep up to date with the latest program advances by keeping your registration information current.

Automatic Program Updates

13) This simulation incorporates an automatic program updater that will keep your software current with the latest simulation developments. Before you put the simulation to work, <u>make sure</u> you allow the *Motion Updater* to check our servers and install the latest program updates (requires Internet connection). The Motion Updater will run automatically after program installation and then approximately every 30 days thereafter. You can check for a new update at any time by selecting *Check For Newer Version...* from the HELP menu within the simulation.

Important: Don't assume you are running the latest software version if you just installed the software from the installation CD. CD's are NOT updated each time new releases are issued. The ONLY way to make sure you are running the latest version is to use the *Check For Latest Version* feature (in the HELP menu). If you cannot obtain/install program updates using our Updater, contact technical support at: support@motionsoftware.com.

Un-Installing The Simulation

You can un-install this simulation by either: 1) Use the program removal feature in Windows or 2) Use the *Uninstaller* placed in the Program folder (**Start** menu, select **All Programs** or **Apps**, then choose **Motion-Dynomation6 Engine Sim** or **Motion-DynoSim6 Engine Sim,** and finally click on the **UNInstall***).*

Solving Other Installation And Operational Problems

Installing & Starting Version6 Simulations

Important Note: *You can review tech-support updates* by visiting (www.Motion-Software.com). *Contact our Tech Support staff directly by sending an email to support@motionsoftware.com.*

If you experience problems installing or using this simulation, please review the information presented in this Users Manual, including the information earlier in this chapter, the FAQs later in this manual. You can also run *Check For Latest Version* features (in the HELP menu) to get the latest simulation updates.

Primary contact information:

Tech Support Email: support@motionsoftware.com
(Preferred Contact Method)

Voice Line: 714-231-3801
Web: www.MotionSoftware.com

And make sure you have a look at our growing list of tech videos on YouTube: *youtube.com/@motionsoftwaresim*

Tech Support Email: support@motionsoftware.com This is the best way to obtain tech support quickly. Always include your email address and attach any .DXML engine files that may help Motion diagnose problems. Include a thorough explanation of the steps that lead up to the fault.

Tech support will only be provided to registered users. Please complete the *Registration Form* that appears when you first start your software to qualify for technical support from the Motion Software staff (or select *Registration* from the HELP menu in the program).
If you need to update your address or any other personal information, simply select *Registration* from the **Help** menu, make any necessary address changes, then press *Register Now!* to transmit your updated info.

Send mail correspondence to:

Motion Software, Inc.
222 South Raspberry Lane
Anaheim, CA 92808-2268

Wave-Action Filling & Emptying Simulations 6

Version 6

OVERVIEW

(2) Program Menu Bar

(1) Title Bar

(19) Pop-Up DirectClick™ Menu

(18) Windows Size Buttons

(17) Graph Reticule

(3) Tool Bar

(16) Power & Torque Curves

(4) Simulation Selection/Setup

(15) Right-Click Graph Options

(5) Engine Component Categories With Roll-Up Menus

(14) Port Pressure Curves And
(13) Port Velocity Curves

(6a) Left Pane Display Tabs

(12) Crank Angle SimData Window

(7) Engine Selection Tabs

(6b) Right Pane Display Tabs

(8) Range Limits And Status Bar

(9) Quick-Access Buttons™

(10) Vertical Screen Divider Resizes Left/Right Panes

(11) Simulation Calc Progress

THE MAIN PROGRAM SCREEN

The left side of the **Main Program Screen** includes component categories that you can use to select simulation specifications and enter engine components and dimensions. The right side of the screen displays simulation results consisting of graphs, charts and tables. The Main Program Screen is composed of the following elements (matches numbers in the photo, above):

1) The **Title Bar** displays the program name followed by the name of the currently-selected engine.

2) The **Program Menu Bar** contains pull-down menus that control overall program

Program Overview

File Edit View Simulation Units Tools Window Help
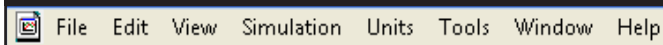
Program Menu Bar contains nine pull-down menus that control overall program function.

function. Here is an overview of these control menus, from left to right:

File—*Opens* and *Saves* version6 (.DXML) files; *Imports* other Motion simulation files, including previous version Dynomation4 and 5 engine files, DynoSim4 and 5 and even various DOS-based engine files; *Exports* crank-angle and rpm-based engine test results data in an Excel-importable format (feature not available in DynoSim6); *ProPrint™* generates a comprehensive report of engine components and simulation results, finally, the bottom of the *File* menu displays the most recently used simulation files, and contains a program-exit function.

Edit—Provides a *Clear-All* components from the currently-selected engine (the current engine is highlighted on the *Engine Selection Tabs* at the bottom of the screen; see *Engine Selection Tabs*, later in this chapter).

View—Allows you to display the *Toolbar(3)*, *Status Bar(8)* and *Workbook (see the Workbook note, below)* layout features. The View menu also includes a choice to reset all graphs to program default axis values.

> **Note:** The *Workbook* features activate the *Engine Selection Tabs(7)* at the bottom of the screen, allowing rapid switching between "open" simulated engines. If you have limited screen space (or you only simu-late one engine at a time), you can turn off the Workbook features and access multiple simulated engines from the *Windows* menu.

Simulation—*Run* forces an update (recalculation) of the current simulation. *Auto Run* enables or disables (toggles) automatic simulation updates every time a component is changed. You can also display a *Simulation Log* that contains diagnostic information about the simulation just completed. The
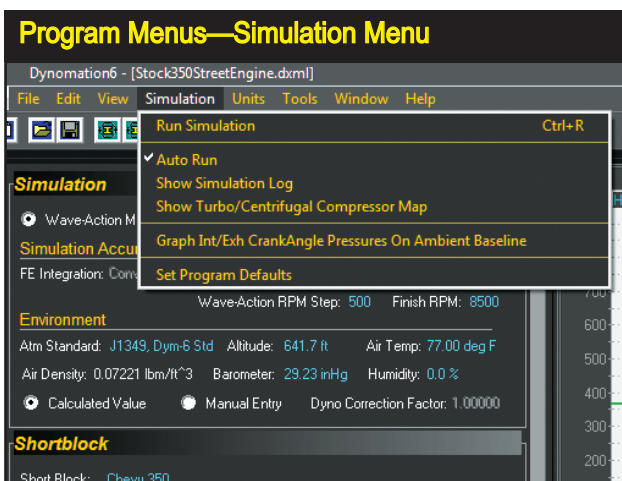
Each Program Menu pro-vides access to overall program features. Here the *Simulation Menu* lets you to choose when the simulation is performed and whether the SimLog and Blower-Map dia-logs are displayed. You can also open a *Program Default* dialog where startup charac-teristics for the software can be modified.

Program Overview
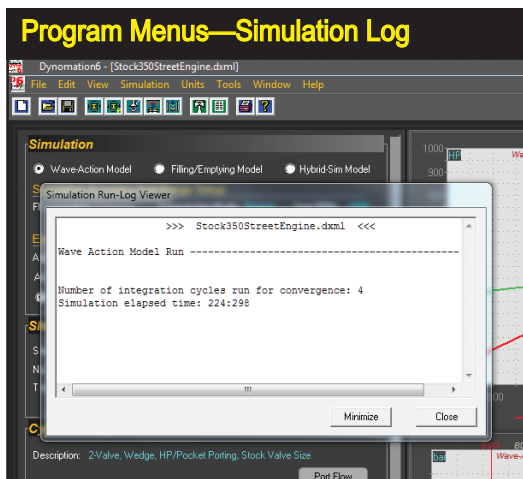
A Program Run Log can be displayed using the *Simulation* drop-down menu. This provides info about the just-completed simulation process, including execution times. Errors, like the lack of convergence or intake restriction warnings (if generated), can be helpful in diagnosing component-setup and other simulation issues.

Graph...On Baseline selection will re-scale pressure graphs to position the curves on the baseline (helpful when modeling forced-induction engines). A *Blower Map Window* can be displayed that shows the compressor map and engine demand line for Turbo and Centrifugally supercharged engines; essential in diagnosing compressor selection, surge or choke issues.

> **Note:** A compressor map will only be displayed when a map is available for the selected compressor. Most compressors featured in version6 simulations are modeled with typical compressor maps (over 300 maps are included with the simulation as of 2024), however, there are a few compressors for which maps were not available and it was possible to use a "pure math" approach to model compressor performance.

The Simulation menu also includes a **Program Setup** selection that lets you personalize simulation startup and operational characteristics.

Units—Selects between US/Domestic and Foreign/Metric units. Hybrid units systems are also available that display component values in Metric units with Power and Airflow in US units.

Tools—Opens *Quick* and *ProIterator™* testing windows, the *Cam Manager™* dialog, or one of several built-in, cam/airflow/conversion calculators.

Window—A standard Windows menu for arranging and selecting engine display windows.

Help—Gives access to this *Users Guide*, *Program Registration*, *Program Updates* and related program features and information.

3) The **Tool Bar,** above the component categories, contains a series of icons that speed up the selection of various program functions and features. The **Tool Bar** contains the following icons/functions: 1) Create New Engine, 2) Open Saved Engine, 3) Save Current Engine, 4) Open Quick Iterator, 5) Open *Pro Iterator™*, 6) Open Port Flow Dialog, 7) Open Compression-Ratio Calculator, 8) Open

Program Overview

Component Category Status

Simulation
- Wave-Action Model   Filling/Emptying Model
Simulation Accuracy And Range Setup
FE Integration: Converge   Wave-Action Mode: Normal
                            Wave-Action RPM Step: 500
Environment
Atm Standard: J1349, Dyn-6 Std   Altitude: 641.7 ft   Air Te
Air Density: 0.07221 lbm/ft^3   Barometer: 29.23 inHg   Humi
- Calculated Value   Manual Entry   Dyno Correction

Shortblock
Short Block: Chevy 465
No. Cylinders: 8   Bore: 4.440 in   Rod Len
Total Volume: 464.5 ci Stroke: 3.750 in   Rod Rat
                                            Pin Offse

Cylinder Head
Description: ***
Valve Specifications:
Intake Valves/Port: ***   Exhaust Valves/Port: *
Int Valve Test Dia: 2.000 in Exh Valve Run Dia:
Int Valve Run Dia: *** in   Exh Valve Run Dia:

Induction
Camshaft
Combustion
Exhaust
Notes

All Required Data Entered

Category Incomplete

The status of the Component Categories are indicated by the color of the *Category Title Bars*. The Title Bars are either red tone, indicating that required data has not been entered (inhibiting a simulation run), or dark tone indicating that all required data in that category has been entered. The simulation determines whether a category is "complete" based on the simulation method selected. For example, the *Filling-And-Emptying* model requires fewer data values than the *Wave-Action* model. As a result, categories may indicate dark when *Filling-And-Emptying* is active, but may switch to red when the *Wave-Action* model is activated, indicating that additional data in those categories is required.

Airflow-Conversion Calculator, 9) Open Piston Animation, 10) Open Crank-Angle SimData™ Window, 11) Generate A ProPrintout™ Of The Current Engine, 12) Display Program "About Box."

4) The **Simulation Category** is the topmost *Engine Component Category(5)*, and is titled *Simulation*. Use this group to select a simulation model (in Dynomation6 only) and make simulation-specific setup choices, such as engine-test rpm range, ambient atmospheric conditions, etc.

5) The remaining **Engine Component Categories** are made up of the following groups:

ShortBlock—Select the bore, stroke, number of cylinders, pin-or-piston offset, rod length and/or rod ratio in this category. Dynomation6 also provides an *Engine Friction* selection that models internal engine friction for various engine buildups.

Cylinder Head—Select the cylinder head type from generic choices in the menu or enter custom data using the Port-Flow dialog (click the Port-Flow button).

Induction and Forced Induction—Select intake manifold designs, airflow rates, pressure drop and runner temperatures for the induction system. Also select intake runner and plenum dimensions for the Wave-Action simulation. Select "Baseline" modeling to begin your engine development.

At the bottom of the *Induction* category you can activate and configure a *Forced-Induction Systems* for any engine. Roots and Screw blowers, Centrifugal superchargers, and Turbochargers are supported. Each engine can support up to four attached superchargers (1 to 4 compressors).

Camshaft—Select the camshaft type, activate V-V-T (variable valve timing

Program Overview

Incomplete Component Fields

Component fields that do not yet contain valid entries are marked with a series of asterisks. This indicates that the field is empty and can accept data input. Most numeric fields accept direct keyboard entry and/or selections from the provided drop-down menus. Some selection fields (like the *Cylinder Heads Type* menu) only accept selections from the associated drop-down menu. When a valid selection has been made, it will replace the asterisks.

similar to Honda's VTEC; see the Camshaft chapter for more information), set various cam timing/specifications, and displays *True Timing* used by the simulation (measured during the actual simulation run). Buttons open the *CamManager™*, *Rocker-Math™* dialog, and the *Lobe-Profile Import* dialog.
Combustion—Selects the compression ratio, the type of fuel, air/fuel ratio, nitrous flow rate, combustion chamber design, and ignition timing.
Exhaust—Selects the exhaust-system configuration. Runner and tubing dimensions and interconnection specifications are available in Dynomation6.
Notes—Enter any comments about the current simulation. Your comments are saved with the engine *.DXML* file.

Note: Each component category indicates its status by the color of the **Category Title Bar**. The Title Bars have either a red tone, indicating that the category is not complete (inhibiting a simulation run), or a **dark-tone** indicating that all components in that category have been selected. When all component categories indicate complete, a simulation can be performed (calculation will begin automatically if the *Auto Run* feature is selected in the *Simulation Menu)*.

The Main Program Screen window is divided into two panes. The left and right portions of each contain a set of *Screen Display Tabs* located at the bottom of the panes (left-most tab is highlighted in the photo). Use these tabs to switch the display to *Engine* component lists, *Tables*, *Graphs*, or other data displays.

Program Screen Display Tabs

Program Overview

The simulation can work with several engines at once. Switch between "open" engines by choosing from the *Engine Selection Tabs*, located just above the Status Bar at the bottom-left of the main program screen (right-most tab is selected in this photo).
Note: the Status Bar (7 & 8 below) displays "Ready" (to run sim).

Engine Selection Tabs

6a & 6b) The Main Program Screen window is divided into two panes. The left and right panes each provide a set of **Screen Display Tabs** at the bottom. Use these tabs to switch the left and right panes to tables, graphs, or other simulation displays.

7) This software can simulate several engines at once (supports multiple open documents). Switch between "active" engines (documents) by selecting any open engine from the **Engine Selection Tabs**, located just above the **Status Bar** at the bottom of the main program screen (engine selection can also be made from the *Windows Menu*). The current engine is highlighted on a foreground Tab. The name of the currently-selected engine is also displayed in the program **Title Bar**.

8) All Component Category menus allow either direct numeric entry or menu-selection choices, and some accept both. During data entry, the range of acceptable values and other helpful information will be displayed in a **Range Limit Line** within the **Status Bar** at the bottom-left corner of the main program window.

9) Several component categories contain **QuickAccess Buttons**™ that give "one-click" access to important data-entry functions and calculators. The **CYLINDER HEAD** category contains a *PortFlow* button that opens the Port-Airflow dialog box, allowing direct entry of flowbench data; the **COMPRESSION** category contains a *CR Calc* button that opens the Compression-Ratio Calculator, a tool that can save time and improve accuracy in determining engine compression ratio; the **INDUCTION** category contains a *FlowCalc* button that allows quick airflow calculations from throttle diameter, etc.; and the **CAMSHAFT** category contains the *CamManager™, Rocker Math™, and Import Profile* buttons that give ready access to important tools that help you with cam simulation and valve-timing modifications.

10) The widths of all program panes are adjustable. Simply drag the **Vertical** or **Horizontal Screen Divider** to re-size the Component-Selection and Graphics-Display panes. **Horizontal Dividers** are located between the right-hand graphs. By adjusting the position of these dividers, you can increase the display size of

Program Overview

SimData™ Window

The *Crank-Angle SimData™* Window displays the exact values of port pressures, flow rates and other simulation data at various rpm and crank-angle values. Click on the rpm graph to set the rpm reticule line, then click on any crank-angle graph to display a crankangle reticule. As you drag the reticule left and right, exact data values at the reticule intersections will be displayed in the SimData™ Window (open the *SimData Window* from the *Tools* drop-down menu).
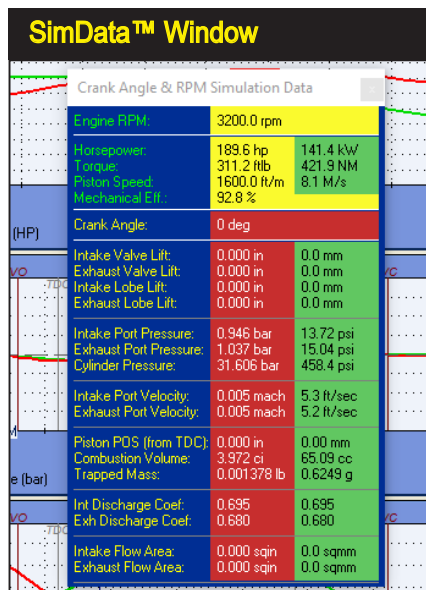
the power-curve and pressure/flow displays to suit your analysis.

11) The **Simulation Progress Indicator** (appears in the **Status Bar** when a simulation calculation is underway) shows the progress of calculations for the selected simulation model. Each "step" of the progress bar indicates the completion of a simulation at one rpm point. **Note:** Some engine simulations, particularly with forced induction, may require multiple passes of the Progress Indicator to complete the simulation process.

12) The **Crank-Angle SimData™ Window** (see photo, next page) displays the exact values of port pressures, flow rates, horsepower, and more at various rpm and crank-angle points. Click on the rpm graph to set the rpm reticule line, then click on any crank-angle graph to display a crankangle reticule (see 17, below). As you drag the reticule left and right across the graph, exact data values under the reticule intersection will be displayed in the SimData™ Window (open the *SimData Window* from the **Tools** drop-down menu).

13) The **Port Velocity** lower graph displays port flow rates at various rpm and crank-angle values (velocity is default display; it can be customized in the right-click Options menu). The displayed velocity values are calculated at the location of minimum cross-sectional area in the port. Click the top horsepower/torque graph to display a reticule line and establish the rpm for viewing flow and pressure data. Drag the reticule left and right on the **RPM** graph to establish the port flow data (in the lower graph) at each of the selected engine speeds. Click on the **Port Velocity** (lower) graph to display a crank-angle reticule line; the exact data at the reticule intersection are displayed in the **SimData™ Window** (see 12, above).

14) The **Port Pressures** are displayed in the middle graph (pressure is the default display; can be customized with a right-click *Options* menu). The displayed pressures are calculated at the location of the minimum cross-sectional area in the port. Click on the top RPM-based horsepower/torque graph to display a reticule line and establish the rpm for viewing pressure and velocity data. Drag the reticule left and right on the **RPM** graph to establish the port pressure data (in the center graph) at each of the selected engine speeds. Click on the **Port Pressures** graph to display a *crank-angle* reticule line; the exact values at the reticule intersection are displayed in the **SimData™ Window** (see 12, above).

15) The various graphs display horsepower, torque, port pressures, flow rates, valve lift, and more for the currently-selected engine. These graphic displays can be customized to display additional data in many formats using the **Graph Options Box**. To display the **Options Box**, *right-click* on any graph; reassign the **X**, **Y1** (left axis) and **Y2** (right axis) curves to any of the data sets provided in the submenu. **Optimize** functions quickly setup curves for best visual resolution. **QuickCompare™** will define sim "baselines" to help evaluate changes or establish comparisons with other engine files (discussed in detail later in this manual). Finally, use the **Properties** choice at the bottom of the menu to open the *Graph Properties* box where you can assign custom axis values and setup multiple engine-to-engine comparisons.

16) The **Horsepower And Torque** top *engine-rpm* graph displays engine output throughout the simulation speed range (HP and Torque are the default displays; the graphs can be customized using the right-click menu). Click on the graph to display a reticule line. The exact data at the reticule intersection can be viewed in the **SimData™ Window** (see 12, and photo on previous page). In addition, horsepower and torque results (and much more) are displayed in the data tables; click the **Table**, **ProData,** or the **Crank Data** *Screen Display Tabs* near the

The Graph Options Box allow you to customize any graph. Right-Click to *Reassign* the X, Y1 and Y2 curves to different data sets, to set *Optimize* functions, to setup *Quick Comparsions,* to assign custom axis *Properties* and setup multiple engine-to-engine comparisons.

Graph Options Box
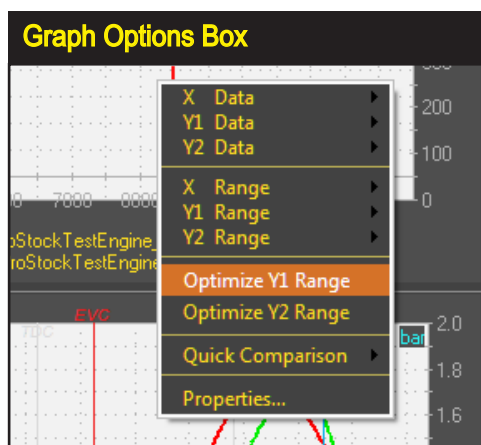
Program Overview

The DirectClick™ Component Menus allow you to select components and specifications for each Component Category item (Combustion-Chamber design is shown here). Click on any component specification to open its menu. The menu will close when a selection is complete (or accept the current selection by clicking on the green checkmark). If you wish to close the menu before making a new selection, click the red X next to the drop-down box, click anywhere outside the menu, or press the Escape key until the menu closes.

DirectClick™ Component Menus

bottom left and right of the Main Program Screen (see **6a** & **6b**, earlier in this section).

17) Each of the four graphs (the fourth is located "under" the Component Categories; click on a **Graph Tab** at the bottom of the left main-program window) incorporate a **Reticule Line** that appears when you left-click on the graph. You can "drag" the **Reticule Line** left and right across the graph between the lowest and the highest test rpm (for the horsepower and torque graphs), or between 0- and 720-degrees of crankshaft rotation (for the middle and lower port pressure/velocity graphs). The exact values of the underlying data at the reticule intersections is displayed in the **SimData™ Window** (see **12**, earlier in this section).

18) The Main Program Screen incorporates **Windows Size Buttons**. These buttons provide standard maximizing, minimizing, and closing functions common to all

Fields Accepting Direct Input

White Background:
Numeric input accepted.
Enter value or make selection from drop-down menu.
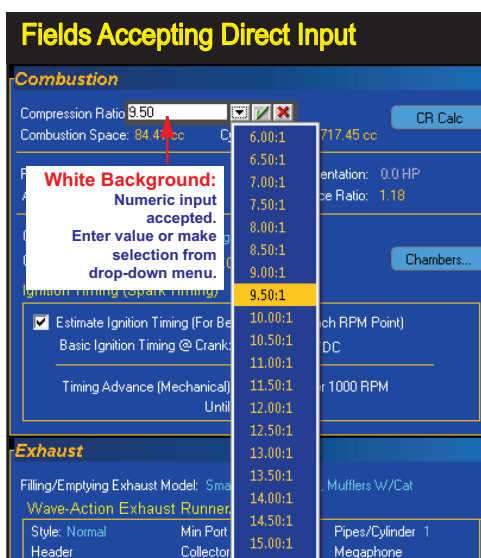
Component fields that support direct numeric entry have white bounding boxes (left). When the only selection possible is a choice from the drop-down menu, the bounding box will have a gray interior (below).

Fields Not Accepting Direct Input

Gray Background:
No numeric input accepted.
Make selection from drop-down menu.
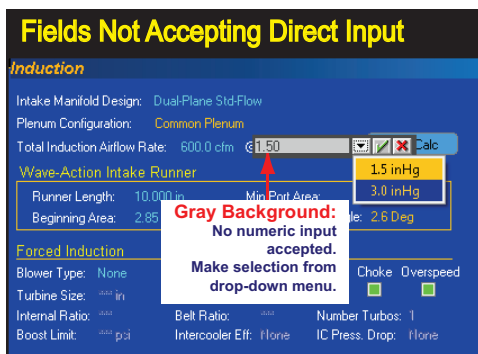
Program Overview

Windows applications. Refer to Windows documentation for more information on the use of these buttons.

19) The **Pop-Up _DirectClick™_ Component Menus** allow the selection of components and specifications within each of the Component Categories. Click on any component specification to open its menu. The menu will close when a selection is complete. If you wish to close the menu before making a new selection, click the red **X** next to the drop-down box, click anywhere outside the menu box, or press the **Escape** key until the menu closes.

Direct-Input vs. Menu-Input Component Categories

Component menus fall into three categories: 1) Those that accept direct user input (custom values), 2) those that only accept a selection from their attached drop-down menu, and 3) those that accept input from either direct input or menu selections. For example, the **Number Of Cylinders** menu in the _ShortBlock_ category will accept direct input (any integer value from 1 to 16), and you can also select typical cylinder combinations from the drop-down menu. When a menu supports direct-data entry, the component-entry bounding box will have a white interior (see photos). On the other hand, the **Pressure Drop** menu associated with the _Total Induction Flow Rate,_ will only accept one of the two selections from its attached menu (1.5- or 3.0-inHg). The data-entry boxes for these input fields have a light-gray interior (rather than white).

> **Note-1:** Almost all custom-input categories include an attached menu from which common data inputs are also provided.
> **Note-2:** Data entry into any component field is limited to values over which this software can accurately predict power. The range limits are displayed in the **Range Limit Line** within the **Status Bar** at the bottom-left of the Main Program Screen. If you enter an invalid number, the simulation will sound the Windows error tone and wait for new input.

Keyboard Selections And Shortcuts

The following keyboard "speed tips" will help you make the process of data entry and "what-if" testing more efficient. None of these operations are essential, and if you prefer to only use a mouse, you can skip this section.

> **Cursor Arrow Keys Move Reticule Lines**—You can move the reticule lines on any of the graphs in small increments by using the _Cursor Arrow_ keys. First, click on a graph to "select" it, then use the **Right-Arrow** key to the move the reticule line to the right; use the **Left-Arrow** key to move it to the left. If you hold the **SHIFT** key, the increment of movement on the crank-angle graphs increases to 10-degrees.

Program Overview

Opening Menu-Bar Menus—Press and hold the **Alt** key, then press the **F** key to highlight and open the File menu (in the Menu-Bar at the top of the program screen). Use the cursor-arrow keys to move through menu selections; press the **Right-Arrow** key to open submenus and the **Left-Arrow** key to close submenus. With any menu open, you can also use the right-and-left arrow keys to activate other menu-bar menus—e.g., *Edit*, *View*, *Simulation*, etc. When you have highlighted the menu selection you wish, press Enter to accept the selection. Press **ESC** (the escape key) repeatedly to close the menus without making a selection.

Note: You can also use the **Alt** key combination with the **E**, **V**, **S**, **U**, **T**, **W**, and **H** keys to directly open the other program drop-down menus.

Moving Through Component Fields—After you have selected any field within a component category, you can press the **TAB** key to move the highlight from field to field within that category. Tab-key presses will not open fields for data entry, but rather will move the focus highlight from field to field. Tabbing keeps field selection within the current category.

Note: The **Shift Tab** key combination will move the highlight box backwards to the previous component field.

Entering Data In Component Fields—When you've highlighted a component field, press the **Enter** key once to convert the highlight to a data-entry bounding box. Then key in your data (or open the drop-down menu and make a selection), finally press the **Enter** key again to close the bounding box and accept the data. To move to the next field, you can press the Tab key, or you can again press **Enter**. Using Enter instead of the Tab key not only moves the highlight to the next field, but also opens the data-entry bounding box in one step. So by

The colors used on the component-selection screen provide information about what fields are currently being used and whether you can modify these components. This is the *Dynomation6* (default) color scheme applied to DynoSim6; White/Cyan indicates components that can be entered and modified, while white/light-gray are fields that indicate "display-only" (like *Runner Connections* and *Average Taper Angle*, shown in the Induction category.).

Program Overview

pressing keys in the sequence "Enter, Enter, (data entry), Enter Enter," you can quickly enter (or jump past) data in component fields within a specific component category.

THE MEANING OF SCREEN COLORS

The colors used on the component-selection screen provide information about various engine components and specifications. Here is a quick reference to screen color functionality; these descriptions apply to the default color scheme:

White: Most engine <u>component field names</u> (not the component values) are displayed in white. This indicates that their associated data fields are <u>required</u> for a simulation to be performed.

Light Blue (Cyan): Engine component <u>values</u> displayed in light blue can be entered into the program and/or modified. For example, the value of ***Total Induction Airflow*** in the Induction Category is Cyan and can be changed to any value within the limits of the simulation.

Light Gray: These "display only" <u>values</u> have been automatically calculated by program and cannot be directly modified. For example, the ***Average Taper Angle*** value in the Induction Category is a display-only value that is determined by the minimum and maximum port areas (values that you *can* enter in the program).

Note: The basic color functionality described here applies to the DynoSim6 default color scheme. While other color schemes may use some of these colors, not every color scheme uses the same default colors for data inputs. However, the differences in display colors within each color scheme will always reflect the same functionality described here.

Wave-Action Filling & Emptying Simulations 6
Version 6

FIVE-MINUTE TUTORIAL

Building And Testing An Engine
Using *DynoSim6* And *Dynomation6* Simulations

This tutorial is primarily intended for first-time users of Dynomation6 and DynoSim6 simulations. However, anyone wishing to get an overview of basic data entry, simulation setup, and results analysis can benefit from reviewing these step-by-step examples.
Also, make sure you have a look at our growing list of tech videos on YouTube:
youtube.com/@motionsoftwaresim
We are sure they will help you get the most from our Version 6 Simulations.

The most common method of starting an engine-simulation project in this simulation is to "assemble" a test engine from component parts. In this tutorial, we'll quickly build a basic engine using the *Filling-and-Emptying-Wave-Assisted* simulation (this simulation model is used in DynoSim6 and it is also selectable in Dynomation6 in addition to the full *Wave-Action Model*).

After obtaining initial simulation results later in this section, we'll modify a few components to determine how they affect engine output. Finally, we'll switch to the *Full-Wave-Action* model (for Dynomation6 users) and continue to fine-tuning engine components. While this may take you more than five-minutes to complete, once you are familiar with the simulation, you'll be able to do similar engine buildups and basic testing in *less than five minutes!*

Note: Engine power values and other simulation results shown in this tutorial may vary as the software is updated and/or new features are incorporated over time. The results shown here reflect the current version of the simulation at the time this chapter was last updated (the software version at publication is shown next to the page numbers, at the bottom of the pages).

Getting Started: Building An Engine From Scratch

1) If necessary, start the simulation program and select **New** from the **File** menu. Component categories on the left side of the program screen begin empty, indi-

Five-Minute, Engine-Buildup Tutorial

The DirectClick™ Component Menus allow you to enter component values and specifications in the simulation. Click on any component specification to open its menu. Click on the down-pointing triangle symbol inside the bounding box to open a drop-down menu of possible choices. The menu will close when a selection is complete (or manually accept the current selection by clicking on the green checkmark). If you wish to close the menu before making a new selection, click the red X next to the drop-down box, or press the Escape key until the menu closes.

Selecting The *Start RPM*

Start RPM: 2500	1000 RPM
Finish RPM: 6500	1500 RPM
	2000 RPM
Air Temp: 77.00 deg F	
Humidity: 0.0 %	✓ 2500 RPM Default
Correction Factor: 1.00000	
	3000 RPM
	3500 RPM
	4000 RPM
	5000 RPM
Rod Length: 5.700 in	6000 RPM
Rod Ratio: 1.636	7000 RPM
Pin Offset: 0.000 in	8000 RPM
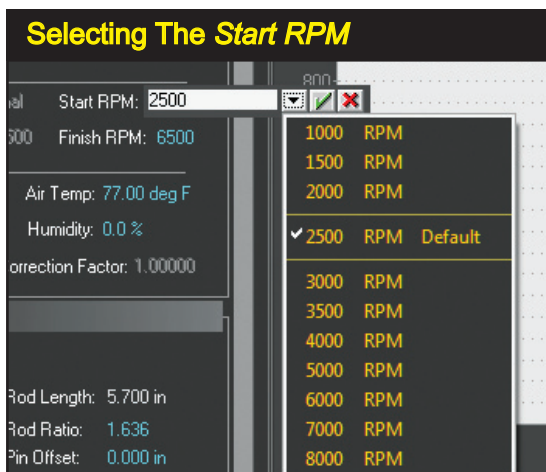
cated by a string of asterisks (****) next to each incomplete data field.

2) Data fields in the Component Categories accept your inputs within a "bounding box" that appears after you click on a data field value. Click on the down-pointing triangle symbol inside the bounding box to open a drop-down menu.

3) Begin the engine build by moving your mouse into the **Simulation** category and left-clicking the *Filling-And-Emptying-Wave-Assisted* radio button, activating the *FE* simulation method (this is not necessary for DynoSim6 users, as the FE method is selected by default). Next, select the *Advanced* mode (if not already shown in the **FE Integration** menu) which provides a good balance between speed and accuracy in simulation calculations. Finally, choose the *RPM Range* values for the simulation as shown here:

The *Simulation Category Choices:*
- **(WA-Only) Simulation Radio Button:** Click *Filling-And-Emptying Sim***.**
- **FE Integration:** Select *Advanced Converge***.**
- **Simulation RPM Step:** 500**.**
- **Start RPM:** Select *1000-rpm* from the drop-down menu.
- **Finish RPM:** Select *6500-rpm* from the drop-down menu.

4) Leave the atmospheric fields in the Simulation Category with their default values. The simulation will use the *J1394 Atmospheric Standard* as a baseline. This standard sets the *Altitude* to 641.7 feet, the *Air Temperature* to 77 F, the *Barometer* to 29.23 inches of mercury, and the *Humidity* to 0%; common atmospheric conditions used to evaluate IC engines.

5) Now move to the **Shortblock** category (see photo, next page). Click on the as-

terisks in the **Short Block** field (the asterisks indicate no selection has yet been made). Review the menu choices and select **American**. When the submenu opens, select **Chevy**, **8 Cylinder SB**, and finally click the left mouse button on **350 V8**.

Shortblock Category Choice:
- **Short Block:** Choose the *Chevy 350 V8* shortblock**.**

6) The **Short Block** name, **Bore**, **Stroke**, **Number of Cylinders**, **Rod Length**, and **Pin Offset** are now displayed in the *Shortblock* category.

Navigating The *Short Block* Menu
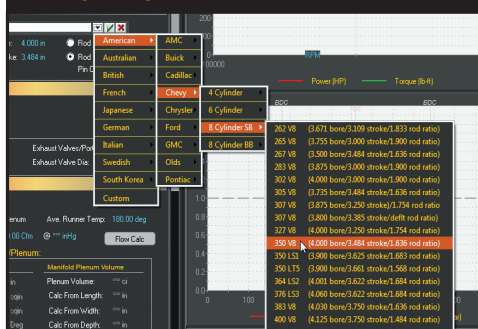
Pause over each selection to open a series of submenus. When you reach *350 V8*, click on it to load that selection into the simulation.

Note: Each component category has colored *Title Bars*. They are either a **red-tone**, indicating that the category is incomplete (inhibiting a simulation run), or a **dark-tone** indicating that all components in that category have been selected and are valid. Now that you have completed the **Short Block** category, notice that the **red** *Title Bar* switched to a **dark-gray-tone**.

7) Next, move to the *Cylinder Head* component category and click on the asterisks (********) next to the **Description** field. Select **Domestic Cylinder Heads**, then from the submenu, right click on **2-Valve Wedge, HP/Stock Ports And Valves** (top of the second group). Notice that after you have made your selection, the category *Title Bar* remains red. This is because Intake and exhaust valve diameters still need to be entered. To simplify this selection (and illustrate another program feature), let the simulation make a calculated "guess" of the valve diameters for you. Open either the **Intake Valve Diameter** or **Exhaust Valve Diameter** menu and select **Auto Calculate Valve Sizes**. This selection will calculate appropriate

valve sizes based on bore size, cylinder-head flow, etc. This completes the *Cylinder Head* category; the *Title Bar* should now a **dark-tone**.

Cylinder Head Category:
- **Description:** *Domestic Cylinder Heads / 2-Valve Wedge, HP-Stock Ports And Valves.*

Selecting Intake and Exhaust Port Flow
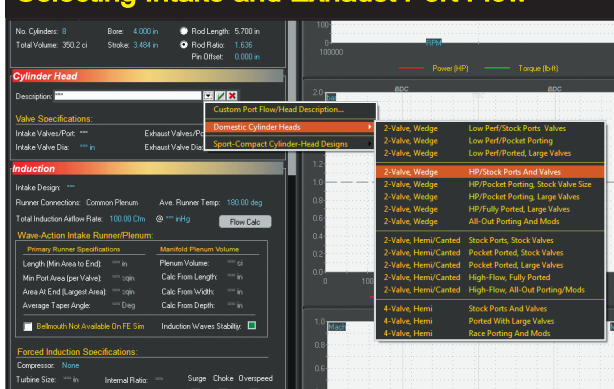
Five-Minute, Engine-Buildup Tutorial

- **Intake Valve Dia:** *Auto Calculate Valve Sizes.*

You'll notice that the simulation has assigned 1.94-inch intake and 1.50-inch exhaust valves to the cylinder heads. These sizes were calculated based on the cylinder head flow, engine bore diameter, and other engine specifications.

8) Continue moving down through the component categories and make the following selections:

Induction Category Choices:
- **Intake Manifold Design:** *Baseline Common Plenum.*
- **Runner Taper Angle:** *1.5 Degrees* (loaded automatically)*.*
- **Average Runner Temperature:** *180-degrees F.*
- **Total Induction Airflow:** *650 CFM (for a 4/8barrel carb/injection).*
- **Pressure Drop** (indicated by the "@" symbol)*: 1.5-InHg.*

Camshaft Category Choices:
- **Cam Type:** *Generic Cams For 2-Valve Engines / Stock Street/Economy.*
- **Rocker Ratio:** Set both Intake and Exhaust Rocker Ratios to *1.5:1.*
- **Valve Lash:** Set both Intake and Exhaust Valve Lash to *0.000.*

Combustion Category Choice:
- **Compression-Ratio:** Select *9.5:1.*
- **Fuel Type:** Select Basic Pump-*Gasoline.*

Exhaust Category Choice:
- **Filling/Emptying Exhaust Menu:** *HP Manifolds, Muffler* Under *With Catalytic Converter.*
- **Exhaust Tune RPM:** 5000*.*

The Main Graph Displays
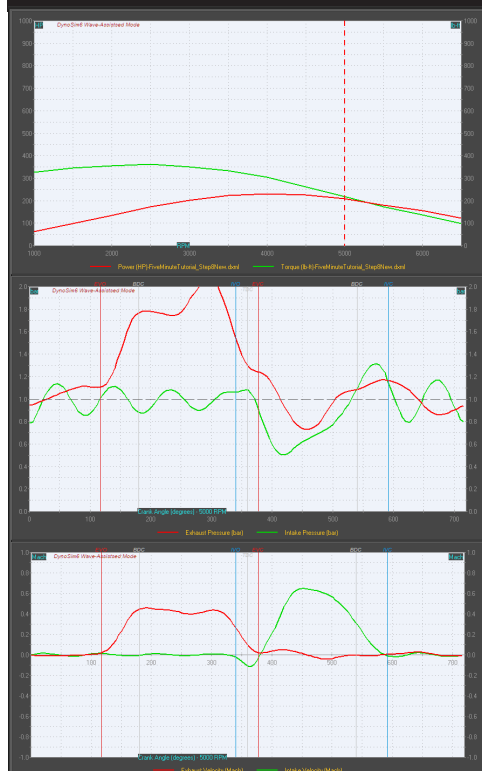
These are typical results curves. Rpm-based data is shown on the top graph (horsepower, torque, etc.); crank-angle based data on the center and lower graphs (pressures, particle velocities). Use *Data Selection Tabs* (at the bottom of the graph displays) to show results in tabular form.

Five-Minute, Engine-Buildup Tutorial

Table Display Of Simulation Results

Calculated Power and Engine Pressures

Engine RPM	Power (Fly) (HP)	Torque (Fly) (LB FT)	Power (Wheel) (HP)	Torque (Wheel) (LB FT)	Indicated Power (HP)	Brake Power (HP)	Frictional Power (HP)	Pumping Power (HP)	Piston Force(avg) (LBSavg)	Piston Speed (FT/MIN)	Ignition Timing (deg)	Vol. Eff. (%)	Mech. Eff. (%)	Fuel ConvEff (%)
1000	61	320	49	256	63.7	60.9	2.3	0.5	1809	580.7	25.7	81.4	95.6	28.0
1500	96	337	77	269	101.3	96.2	4.2	0.9	1926	871.0	26.9	84.1	95.0	28.6
2000	133	349	106	279	141.5	133.0	6.8	1.7	2009	1161.3	28.1	87.3	94.0	28.5
2500	169	356	136	285	183.1	169.5	9.8	3.8	2097	1451.7	29.2	92.2	92.6	27.7
3000	199	349	159	279	219.9	199.3	13.5	7.1	2088	1742.0	31.1	94.4	90.7	26.4
3500	222	332	177	266	250.5	221.5	17.7	11.3	2041	2032.3	32.2	94.9	88.4	25.1
4000	232	305	186	244	271.3	232.4	22.4	16.5	1929	2322.7	33.0	92.9	85.7	23.4
4500	230	269	184	215	279.1	230.2	27.8	21.1	1766	2613.0	33.3	88.0	82.5	21.8
5000	215	225	172	180	273.5	214.6	33.7	25.2	1556	2903.3	34.0	80.6	78.5	19.9
5500	189	181	151	145	257.7	189.2	40.2	28.3	1327	3193.7	34.4	71.9	73.4	17.8
6000	167	146	134	117	246.0	167.2	47.2	31.6	1165	3484.0	34.7	65.8	68.0	15.9
6500	130	105	104	84	219.0	130.1	54.9	34.1	957	3774.3	34.8	57.6	59.4	13.0
Averages Based On RPM Range Selected In Simulation Category:														
	170	273	136	218	209	170	23	15	1723	2178	31.4	82.6	83.6	23.0

Click the *Table Tab* **at the bottom of the graph pane (see photo on previous page) to switch to a table view of exact power values and other simulation results.**

> **Dynomation6 User Note:** Since you are performing a *Filling-And-Emptying (FE)* simulation, the component fields that are only related to the Full *Wave-Action (WA)* simulation are dimmed and do not require data entry.

> **Simulation Results:** At this point, all the *Category Title Bars* should be **dark-tone**, and the simulation will display power and torque curves. The engine produces about 230-hp at 4000 to 4500rpm; typical of 1960's era, V8 street engines with mufflers and restrictive exhaust systems.

>> **Problems?:** If the simulation did not run, first check the *Category Title Bars*. Have they all switched to **dark-tone**. Next, make sure that *Auto Run* is checked in the *Simulation Menu* (located at the top of the main program screen). Finally, check each data-entry field to confirm that the correct values were entered.

>> **Helpful Engine Files:** Engine files that match this engine at several points in the tutorial are included with your simulation and can be found in the default engine files directory *C\:Dynomation6\Engine Files\Engine-Files (.DXML Dynomation-6) or C\:DynoSim6\Engine Files\EngineFiles (.DXML DynoSim-6)*. For example, the file ***FiveMinuteTutorial_Step8.dxml*** should be identical to your engine at this point in the tutorial (providing the software versions are identical). You can always load one of the tutorial engine files to check your component selections and simulation results.

9) Now let's make a few component changes, but before we do, it's helpful to establish baseline curves on the graph so that we can easily compare changes in power and torque. Called **QuickCompare**,™ activate this function by right-clicking anywhere in the Power/Torque graph, then from the pop-up *Options Menu*, select *QuickCompare™*. Finally, select *Make From Current Test*.

Five-Minute, Engine-Buildup Tutorial

RPM-Based Graph Option (right-click graph for options menu):
- **QuickCompare™:** *Make From Current Test.*

This creates a new, identical engine in the simulation called ***CompareRun1*** (a new **Engine Tab** will be shown at the bottom-left of the screen)*.* At the same time, a "side-by-side" comparison is established on the current graph with data from the newly created baseline engine (the comparison curves lie directly underneath the existing power and torque curves on the graph...the curves exactly match because the new baseline engine, *CompareRun1*, is identical, so far).

10) Now, change the cam to the next choice from the ***Cam Type*** menu: A High-Performance Street grind:

Camshaft Category Choice:
- **Cam Type:** *Generic Cams For 2-Valve Engines / High-Performance Street*

    The new cam creates more power; the engine now produces about 250-hp at 5000-rpm. But there has been a drop in torque below 3000-rpm.

    Click on the Power/Torque graph to activate a *reticule line* that marks the rpm point at which data is obtained for the center and lower graph displays of intake and exhaust pressure waves and flow velocities. Drag the reticule line back and forth across the rpm graph. The changing pressure and velocity curves were recorded at each rpm point during the simulation.

    Notice that exhaust pressure (the red curve on the center, crank-angle graph) rises above 2.0-bar (that's more than 15-psi backpressure in the exhaust system at 5000-rpm). To get a better feel for the capabilities of this engine, let's see how it responds to a more free-flowing exhaust system. Install a set of small-tube headers with open collectors (a typical dyno setup):

Exhaust Category Choice*:*
- **Filling/Emptying Exhaust Model:** *Small-Tube Headers, Open Collector*
- **Filling/Emptying Exhaust Tune RPM:** *6000.*

Side-by-Side Results Comparison
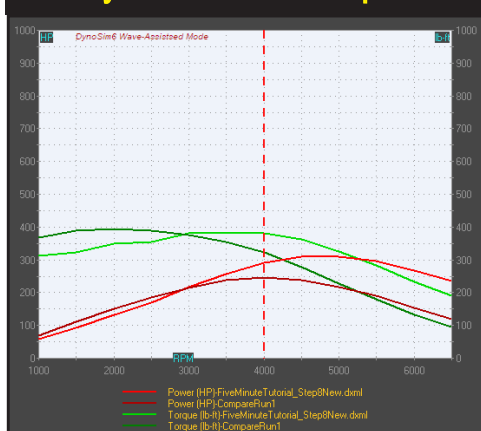
Here is the Side-By-Side comparison of the 350 engine after the HP Street Profile cam and small-tube headers were installed. The significant boost in higher rpm power was obtained with little loss in low-speed torque.

11) The comparison curves now show the improvements in power and torque above 3000rpm. The engine now produces about 290-hp at 4500 to 5000-rpm. Lower-speed torque (below 3000rpm) is still lower than with the stock cam. These results show that exhaust backpressure and a lack of cylinder scavenging (caused primarily from muffler restriction) significantly limited the potential of this engine.

We've also learned that the cylinder heads are able to supply substantial additional flow; if this were not true, the power increase from open headers would have been much less.

This is starting to get interesting! Before we proceed, let's perform another *QuickCompare™* to set current results as a new baseline: Select **QuickCompare** in the **RPM-Based Graph Option Menu** (right-click menu):

- **QuickCompare™:** *Make From Current Test*

Note: The engine configuration at this point in the tutorial was saved as **FiveMinuteTutorial_Step11.dxml**.

12) To determine if this engine has more airflow potential with the current cam timing and cylinder heads, let's improve the induction system by testing a tuned intake manifold. Since this is still a street engine, we'll select a manifold suitable for this application.

Induction Category Choices:
- **Intake Manifold Design:** *Dual-Plane Std-Flow (*from the *Plenum, Domestic, Dual Plane* sub menus). Notice that the **Runner Taper Angle** changed from 1.5 to 1.0 (a value loaded with the manifold; this tunes the simulation to better model a standard Dual-Plane design).

This Dual-Plane manifold is designed to boost torque within the rpm range of a typical street engine. While the power remains nearly the same, we see a boost in torque across the rpm range.

13) Let's try a couple of additional improvements. This time we'll install high-performance OEM-type cylinder heads that provide improved flow (primarily from pocket porting techniques, performance valves, seats, etc.).

Cylinder Head Category Choices:
- **Description:** *2-Valve, Wedge, HP/Pocket Porting, Stock Valve Size.*

Notice that the additional port flow generated virtually no negative affects on torque (thanks, in part, to the Dual-Plane manifold), however, the horsepower jumped to about 380hp at 5500-rpm. Improved induction flow also boosted pressure-wave strength in the exhaust system and this further assisted cylinder filling.

Five-Minute, Engine-Buildup Tutorial
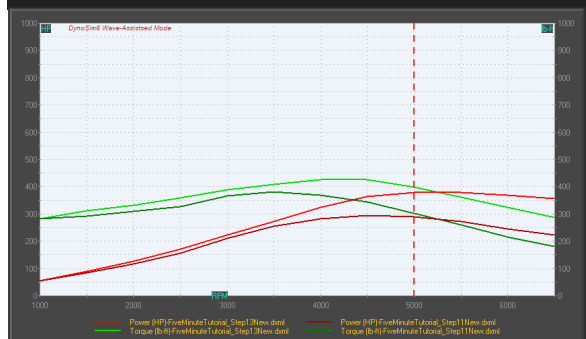
After the installation of a Dual-Plane manifold and cylinder heads with improved port flow, the engine performance is obviously much better across the entire RPM range, despite the restriction from mufflers added to the collectors of the small-tube headers.

You can adjust when these pressure pulses return to the cylinders by changing exhaust tubing lengths. Try increasing the tuning speed from 6000rpm to higher speeds (to shorten pipe lengths). Note that as power rises, the engine torque curve is slightly reduced (engine tuning is almost always a give-and-take process).

We are still simulating open headers. Let's see how much power we can expect to loose if mufflers are installed after the headers.

Exhaust Category Choice:
- **Filling/Emptying Exhaust Model:** *Small-Tube Headers, Mufflers, W/O Cat*

Based on the exhaust pressure estimates within the simulation, the power dropped about 25 hp. The engine still produces about 350hp at 5000rpm (with an exhaust tuning speed of 6000rpm); reasonable for this inexpensive basic street-rod buildup.

Note1: Some engine builders are confident that a well-designed high-flow exhaust system would reduce power by less than the amount simulated in this example.
Note2: The power values predicted by simulations including mufflers are based on the best calculation of back-pressure in the exhaust system. Keep in mind that mufflers and connecting-pipe designs will affect these pressures and engine performance characteristics in "real world" operation.

14) Next, change the Exhaust Selection back to Open Headers, then do another *QuickCompare™* to set current results as a new baseline.

Exhaust Category Choice:
- **Filling/Emptying Exhaust Model:** *Small-Tube Headers, Open Collector*

Five-Minute, Engine-Buildup Tutorial

RPM-Based Graph Option (right-click menu):
- **QuickCompare™:** *Make From Current Test*

Next we'll make a few small changes to explore possible improvements in engine performance. First, change the Intake Rocker Ratio to 1.65 from 1.50.

Camshaft Category Selections:
- **Intake Rocker Ratio:** *1.65* (from 1.50)

This increases intake lift about 0.050-inch. Notice the power increase and a new torque dip around 3000rpm. (Try increasing the Exhaust Rocker Ratio; the results are somewhat disappointing because this slightly reduces intake scavenging before EVC, and that reduces fuel flow into the engine.) To address the loss in torque at 3000rpm, change the Intake Centerline from 110 degrees to 105.

Camshaft Category Selections:
- **Intake Centerline:** *105 degrees* (from 110)

This reduces intake charge reversion (backward flow) just before IVC. That smoothed out low-speed torque with little negative affect on peak power. Finally, let's bump the compression ratio from 9.5:1 to 11:1.

Combustion Category Selections:
- **Compression Ratio:** *11:1* (from 9.5:1)

This final change increased the power to more then 400hp @ 5500rpm.

The setup at this point was saved as *FiveMinuteTutorial_Step13.dxml.*

14) **USING Dynomation6 Wave-Action Simulation: The remaining portion of this tutorial covers intake and exhaust port and tubing modifications; features available only in Dynomation6, Motion Software's professional Wave-Action simulation.**

Now we'll look into some of the capabilities of the *Wave-Action* simulation to delve deeper into the power potential of this engine. After you have started Dynomation6, make sure that the **Wave-Action Model** is selected in the *Simulation Category*. If necessary, load the previously saved engine file (***FiveMinuteTutorial_Step13***) and setup a new comparison.

Simulation Category Choice:
- **Sim Model Radio Button:** *Wave-Action Model*

Five-Minute, Engine-Buildup Tutorial

15) You may have noticed that the *Title Bars* of the *Induction* and *Exhaust* Component categories have now switched from a **dark-tone** to **red**, indicating that the Wave-Action model lacks sufficient data in these categories to complete the simulation. Fill in the missing component data with the following values:

Induction Category Choices:
- **Runner Length:** *10.5 inches*
- **Minimum Port Area:** *1.65 sq-inches*
- **Port Entry Area:** *2.1 sq-inches*
- **Plenum Volume:** *100 cubic-inches* (when you enter plenum volume, the program will display default plenum dimensions; these are for reference only, the simulation only uses *volume* in its calculations).

Exhaust Category Choices:
- **WA Exhaust Type:** *Basic Header--(Pipes Merge At Collector)*
- **Min Port Area (for each exhaust port):** *1.1-sq-inches*
- **Primary Pipes Per Cylinder:** *1 pipe-per-cylinder*
- **Port/Primary Length:** *32-inches*
- **Primary Inlet Diameter:** *1.500-inches*
- **Primary Exit Diameter:** *1.500-inches*
- **Collector Length:** *10-inches*
- **Collector Inlet Diameter:** *3.5-inches*
- **Collector Exit Diameter:** *3.5-inches*

16) The *Wave-Action* power and torque curves are similar to the FE, but they have a slightly different shape. This is due to the more extensive pressure-wave modeling used in the wave-action sim.
Note: These results would not have matched if the FE exhaust system used one

Intake Runner Dimensions
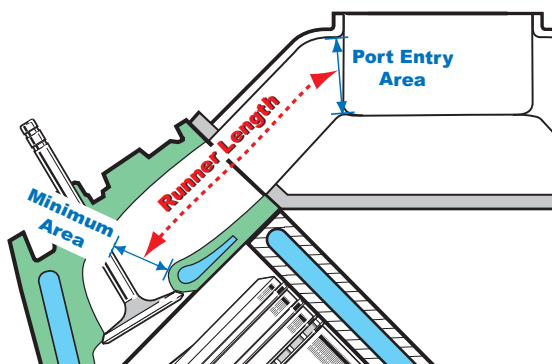
Intake *Runner Length* is defined as the distance from the minimum port area (typically near the valve-seat position) to the point in the port where the area substantially increases, typically the *Port Entry Area*. This area is measured at the port entrance junction with the plenum or just inside the runner bell-mouth transition to the atmosphere.

of the muffler models, since the WA does not model directly model backpressure created by mufflers or catalytic converters.

17) Now we'll dig a little deeper into pressure waves in the intake runners by testing the effects of *Intake Port Areas*. The **Minimum Port Area** is the smallest cross-section in the intake port, and is typically located just after the valve seat, near the valve guide. Currently we are using a value of 1.65-in^2 for this restriction. Try changing this value to 2.0-in^2.

Induction Category Choice:
- **Minimum Port Area (per valve):** *2.0-in^2; then find the area that makes peak power.*

You will see the power curve change shape with the larger minimum intake area of 2.0-in^2. Now decrease the area in 0.1-in^2 steps until you locate a size that produces optimum power (you can switch to the table display to view the exact power values; click **Table** in the *Selection Tabs* at the bottom of the graphs).
　　You should find that optimum power occurs at about 1.65-in^2. The power should be about 400-hp at 5500-rpm (back to nearly the same power values we obtained with the FE model in step 13). Notice how the power drops on either side of the optimum Min Port Area. This indicates that a port that is either too small or too large for the current port flow and cam timing.

18) Let's modify the **Port Entry Area** from its current value of 2.1-in^2. This area is the *maximum* port cross-sectional area, located where the port begins inside the intake manifold plenum or where it opens to the atmosphere with individual-runner (or injector) stacks.

Induction Category Selection:
- **Port Entry Area:** *5.0-, 4.0-, 2.0-, and 1.75-in^2; then find the area that generates what you consider to be the most usable power curve.*

As the *Port Entry Area* decreases (lower numerical values), the port taper angle also decreases, tuning the runner/manifold to a lower rpm. A reduced taper angle (port expands less from valve to manifold) tends to boost peak torque and reduce peak power. Larger Port Entry Areas typically tune the engine for higher rpm, boosting horsepower at the expense of mid-range torque.

Did you locate a port area that produces a good compromise between torque and horsepower? An area of about 2.1-in^2 produces a taper angle of about 1.0-degree. As mentioned earlier in the FE tutorial, lower values are best for OEM or engines where optimum torque and smooth driving are important. Angles of 3.0- to 5.0-degrees are best for high performance engines, where power over low-end torque is the goal (optimum taper angles depend on the desired rpm range of

the engine, cam timing, cylinderhead flow, etc.).

Before we proceed, let's setup a new comparison curve as we continue to explore intake runner tuning.

RPM-Based Graph Option (right-click menu):
- **QuickCompare™:** *Make From Current Test*

Overview: Cylinder heads with intake ports that are too big or too small do not allow the engine to reach peak performance. In a few minutes of testing, you have discovered this fact and explored optimum port shapes that can, otherwise, take weeks of work and thousands of dollars to discover!

19) Next, let's explore how *Runner Length* further tunes the engine power band. Increase the runner length from 10.5-inches to 18-inches and, then, shorten it to 6-inches. Watch the power and torque shift up and down the rpm range.

Induction Category Selection:
- **Runner Length:** *Increase to 18-inches, shorten to 6-inches, then return to 10.5-inches.*

As runner length increases, pressure-wave tuning is altered so that the returning *positive pressure intake wave* (the GREEN pressure wave on the center graph) arrives at the intake valve just before valve closing, when the piston is moving **_up_** in the cylinder). This helps prevent intake back-flow (reversion) into the intake manifold, boosting low-speed power. This phenomenon is visible on both the center (pressure) and lower (velocity) graphs. On the other hand, with runner lengths shorter than about 10-inches, intake pressure-pulse tuning occurs at engine speeds too high to be of much use on this basic street engine. 10.5-inch intake runners have a practical length and are close to optimum. Reset to the following values and we'll try some more exhaust tuning:

Induction Category Selection:
- **Runner Length:** *Return to 10.5-inches.*
- **Minimum Port Area (per valve):** *1.65-in^2.*
- **Port Entry Area:** *2.10-in^2.*

The setup at this point was saved as *FiveMinuteTutorial_Step19.dxml.* Now, set the current engine design as the new baseline.

20) There is a lot more we could explore in intake tuning, but for this introductory tutorial, let's experiment with some additional exhaust mods.

Currently, header primary plumbing is constructed from 32-inch long, 1.5-inch

diameter tubing and the collectors are 10-inches long and 3.5-inches in diameter.

Let's try increasing the *Primary Inlet Diameter* and *Exit Diameter* to 2-inches.

Exhaust Category Selections:
- **Primary Inlet Diameter:** *2.0-inches (Note: Dynomation will not allow the primary pipe inlet to be <u>larger</u> than the outlet. So, after you enter 2.0-inches for the Inlet, the program will display a message box and, when you click OK, the **Exit Diameter** will be set to the same larger size).*
- **Primary Exit Diameter:** *2.0-inches*

The slight boost in peak power (primarily from a reduction in exhaust restriction), comes at the expense of torque around 3500- to 4500rpm. Torque is lower due to reduced exhaust scavenging because of attenuated wave pressures within the header tubing. The weakened returning <u>*negative pressure wave*</u> provides less assistance to exhaust flow just before EVC. This increases cylinder pressure near EVC, and higher cylinder pressure increases restriction in intake flow.

21) Seems like that was one step forward and one step back! How can we maintain low-end torque while boosting power? The secret lies in harnessing the same phenomenon that is responsible the loss: *Exhaust Pressure Waves*. By installing an exhaust system that returns stronger negative pressure pulses, we can reduce cylinder pressure, and regain lost intake flow.

Let's try installing a header configuration with a reputation for helping low rpm torque <u>*and*</u> higher rpm horsepower at the same time:

Exhaust Category Selections:
- **WA Exhaust Type:** *Tri-Y With 2 Collectors*
- **Primary Length:** *20.0-inches*
- **Primary Inlet Diameter:** *1.625-inches*
- **Primary Exit Diameter:** *1.625-inches*
- **Collector One Length:** *6-inches*
- **Collector One Inlet Diameter:** *2.000-inches*
- **Collector OneExit Diameter:** *2.000-inches*
- **Collector Two Length:** *15-inches*
- **Collector Two Inlet Diameter:** *3.000-inches*
- **Collector Two Exit Diameter:** *3.000-inches*

This Tri-Y design of two collectors sends multiple return scavenging pulses back to cylinders to help reduce cylinder pressures and improve intake flow. To maintain as much low-end torque as possible we limited primary pipe size to 1.625 inches and transitioned to a larger secondary pipe (that reduces exhaust restriction). These choices help maintain cylinder scavenging at lower engine

Five-Minute, Engine-Buildup Tutorial

ProPrint™ Report

Motion Software, Inc.
222 South Raspberry Lane
Anaheim, CA 92808-2268
Fax: 714-974-5389
www.MotionSoftware.com

Numeric UNITS Are Selected in Dynomation6

Test Engine: FiveMinuteTutorial-Step19.dxml

Simulation Model Specs

Simulation Model Used: Wave Action

Start RPM: 1000	Finish RPM: 6500	Step RPM: 500
Wave Action Mode: Normal	Filling & Emptying Integration:	

Atmospheric Standard: J1349, Dym-6 Std

Altitude: 641.7 ft	Air Temp: 77.00 deg F	Humidity: 0.0 %
Barometer: 29.23 inHg	Air Density: 0.07221 lbm/ft^3	
Correction Application: Calculated Value	Correction Factor: 1.00000	

■ Engine Shortblock Specs

Short Block: Chevy 350		
Number Of Cylinders: 8	Bore: 4.000 in	Stroke: 3.484 in
Rod Length: 5.700 in	Rod Ratio: 1.636	Pin Offset: 0.000 in
Cylinder Swept Volume: 717.45 cc	Total Displaced Volume: 350.2 ci	

■ Cylinder Head Specs

Description: 2-Valve, Wedge, HP/Stock Ports And Valves	
Intake Valves Per Port: 1	Exhaust Valves Per Port: 1
Intake Valve Diameter: 1.940 in	Exhaust Valve Diameter: 1.500 in

■ Induction Specs

Induction: Simple Plenum Model	Runner Temp: 180.00 deg F
Basic Induction Type: Common Plenum	WA Plenum Volume: 100.0 ci
Overall Induction Flow: 650.00 Cfm @ 1.50 inHg	WA Runner Bellmouth: Not Used
Wave-Action Runner Length: 10.500 in	WA Runner Taper Angle: 1.3 Deg
WA Runner Entry Area: 2.00 sqin	WA Minimum Port Area: 1.45 sqin

Forced Induction Type: None	
Number of Turbos:	Turbine A/R Ratio
Turbine Size:	Blower Drive/Belt Ratio:
Intercooler Efficiency:	Blower Internal Ratio:
IC Pressure Drop	Maximum Boost Pressure:

This is just one of 17 pages included in the simulation ProPrint™ report. Keep this comprehensive analysis for your records or present to your customers with your engine buildups.
Note: While the benefits shown in the accompanying text for Tri-Y headers are relatively small, if exhaust pressure was higher (from greater compression ratio, different cam timing, larger engine displacement, higher engine speeds, etc.), pulse tuning in the exhaust system would yield more-obvious results.

speeds. The result: a small bump in hp and nearly the same torque we achieved with the original 1.5-inch headers.

Note: The effects of header tuning are more noticeable at higher engine speeds, with racing camshafts, larger valves, and higher flowing cylinder heads (these changes increase the strength of pressure waves discharged from the cylinders).

This final change increased the power to about 410hp @ 5500rpm.

The engine configuration at this point was saved as *FiveMinuteTutorial_Step21.dxml)*.

22) Before you quit this tutorial, generate a *ProPrint™* report for this engine: Select **ProPrint Preview** from the *FILE* menu. This will prepare a comprehensive *ProPrint™* report and load it into your default Internet browser. Use the scroll bar or page down keys in your browser to look through the report.

You can also send the document to your printer by selecting **Print** from your browser's *File* menu. Additional information about *ProPrinting™* is provided later in this manual.

Finally, make sure to chck out our growing list of tech videos on YouTube:
youtube.com/@motionsoftwaresim

COMPONENT MENUS

THE SIMULATION
COMPONENT CATEGORY

The **SIMULATION Category** is located at the top of the Component Selection Screen. Use this category to select the **Wave-Action**, and **Filling-And-Emptying** simulation models (*Dynomation6* offers all both models, while *DynoSim6* runs the Filling-And-Emptying model exclusively). These models represent the mathematical simulation techniques incorporated into version 6 simulations.

Just below the simulation model selection are the **Accuracy And Range** options. Here you can "fine tune" the models to deliver the accuracy and rpm range that meets your needs. In general, as you increase the accuracy settings and the rpm range included in the simulation process, the time it takes the software to complete its calculations will go up. In most cases, the *Normal* or *Fine* settings are the best options.

Finally, an **Environment** section includes variables required to establish ambient conditions during the simulation process. These include air temperature, density, and water content, plus altitude and air pressure maintained during the simulation process.

Two distinct engine-simulation mathematical models are used: **1)** A *Filling-And-Emptying (FE)* simulation method is used in both DynoSim6 and Dynomation6 and

Simulation Selection

The *Simulation Category* let's you select the overall engine simulation model. The WA is ideal for detail engine design, revealing port flow and pressure waves. Use the FE when you wish to have optimum intake and exhaust runner dimensions automatically determined by the simulation.

Simulation Category

The *Simulation Category* also includes menu choices for the *Start Rpm*, *Finish Rpm*, and *Step Rpm* used during the simulation. The *Wave-Action Mode* choice defines the "depth" or rigor for WA simulation calculations, and the Environment section establishes the Standards used for atmospheric correction.

it provides efficient and accurate mathematical solutions to engine physics, including port- and induction-flow calculations and automatic intake- and exhaust-manifold/ runner modeling, making the *Filling-And-Emptying* model an easy way to buildup and "ballpark" engine designs with excellent accuracy, and **2)** A full *Wave-Action (WA)* simulation used in Dynomation6 that precisely predicts the complex pressure-wave dynamics and particle flows in intake and exhaust ducting. This *Wave-Action* model picks up where the *Filling-And-Emptying* model ends and "homes in" on the best port sizes, shapes, runner lengths, header-tubing sizes, and valve motion, providing unprecedented accuracy for professional, detailed engine analysis.

Which Simulation Model?

Generally, if you need to determine how an engine will respond to cam timing changes, intake manifold designs, open-headers vs. mufflers, or you don't have access to all the required intake and exhaust port dimensions for a comprehensive *Wave-Action* model run, the *Filling-And-Emptying* simulation provides a quick and remarkably accurate way to build up and test an engine. However, if you are "digging into" the details of mass flow, port shapes and lengths, or exhaust tubing dimensions, the *Wave-Action* model will analyze these inputs and more, letting you home-in on optimum on optimum combinations with a few mouse-clicks.

FE vs WA Modeling Details: The *Filling-And-Emptying* model (in DynoSim6) predicts optimum port dimensions before the simulation process begins (these values are displayed in the Simulation Log in DynoSim6). In many engines, FE simulation results, using these dimensional predictions, will come very close to those from the WA simulation, especially when engine has more-or-less "standard" design. However, if you are modeling specialized engines, the Wave-Action's ability to test and zero-in on optimum intake and exhaust runner dimensions will result in higher overall accuracy.

Simulation Category

Simulation Run-Log Viewer

```
>>>  ProStockModel2.dxml  <<<

Filling/Emptying Wave-Assisted Model Run ------------------------------------

Approximate Exhaust Dimensions Used In FE Simulation:
Prmary Length:                  11.24 In,      285.5 mm
Prmary Diameter:                 2.42 In,       61.5 mm
Secondary Length:               16.86 In,      428.2 mm
Secondary Diameter:              2.67 In,       67.8 mm
Firat Collector Length:         14.00 In,      355.6 mm
First Collector DIamater:        4.47 In,      113.5 mm

Approximate Intake Dimensions Used In FE Simulation:
Intake Runner Length:            7.00 In,      177.8 mm
Intake Minimum Port Area:        3.85 In,       97.9 mm
Intake Maximum Port Area:        6.28 In,      159.5 mm

Number of integration cycles run for convergence: 3
Simulation elapsed time: 887:109 ms
```

Minimize   Close

The *Filling-And-Emptying* model precalculates port dimensions before the simulation process begins. These calculated dimensions are shown in the simulation Log in DynoSim6 for your reference.
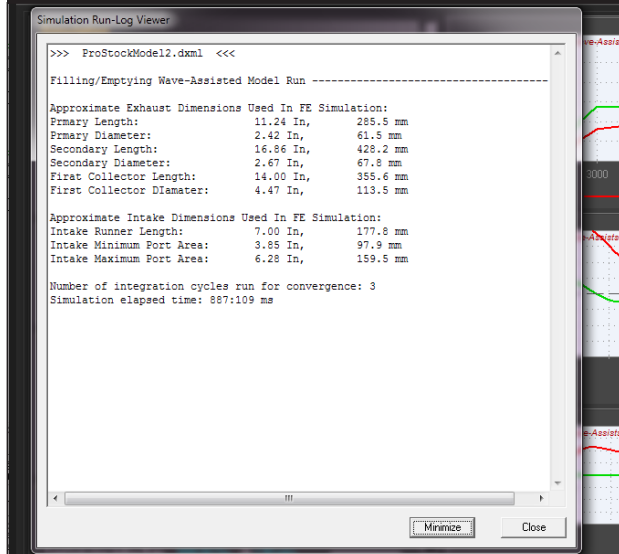
Both simulation models display port pressures and velocities throughout the 720-degrees of crankshaft rotation during the 4-stroke process. This data is displayed in graphs and tables and can help you make decisions about engine tuning (engine tuning is discussed in-depth later in this manual, especially in the **Wave Dynamics Analysis** chapter). However, keep in mind that pressure/velocity data calculated by the Wave-Action simulation is derived through a much more rigorous mathematical approach and should be considered the

Pressure and Particle Velocity Displays
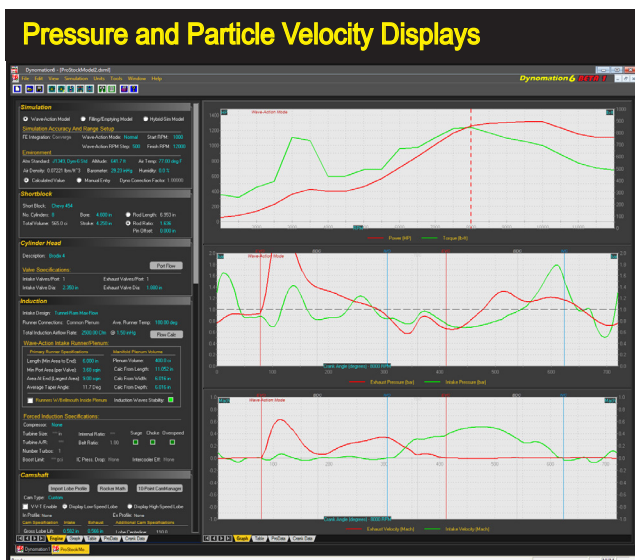
The simulation displays port pressure and velocity curves for the entire 720-degrees of crank rotation of the 4-stroke process (shown in the middle and lower graphs). This data can help you make critical decisions about valve-event timing and intake/exhaust tuning.

Simulation Category

most accurate representation of actual engine pressure-wave dynamics.

WA Note: In cases where you would like to run a quick test or evaluate a restrictive (muffler) exhaust system, run an FE Simulation. The simulation will show exhaust back pressure and other related info in the center and lower pressure-wave graphs.

"Cycles" And Convergence
In The FE Simulation

All Version 6 engine simulations perform an analysis of the gas-exchange process within engine passages. These processes are dependent on valve position, piston position, engine speed, surrounding atmospherics and much more. To ensure that these mathematical processes have reached a steady-state condition, several "integration" cycles may be necessary. These *cycles* should not be confused with engine cycles; they actually represent the number of times the simulation model must work through a gas-dynamics analysis to reach the *same* results, or stability. In some ways, this phenomenon is similar to the instability often seen in real-world dyno tests just before the engine "smooths out" at the measurement rpm. Some of this instability is due to the same gas-dynamic effects calculated within the simulation.

The FE simulation (used in both *Dynomation6* and *DynoSim6*) incorporates three user-selectable calculation modes that allow you set the mathematical depth of pressure-wave analysis. A **Normal** mode, which was the default in previous versions, does a fast analysis quite suitable for most stock and high-performance engines.
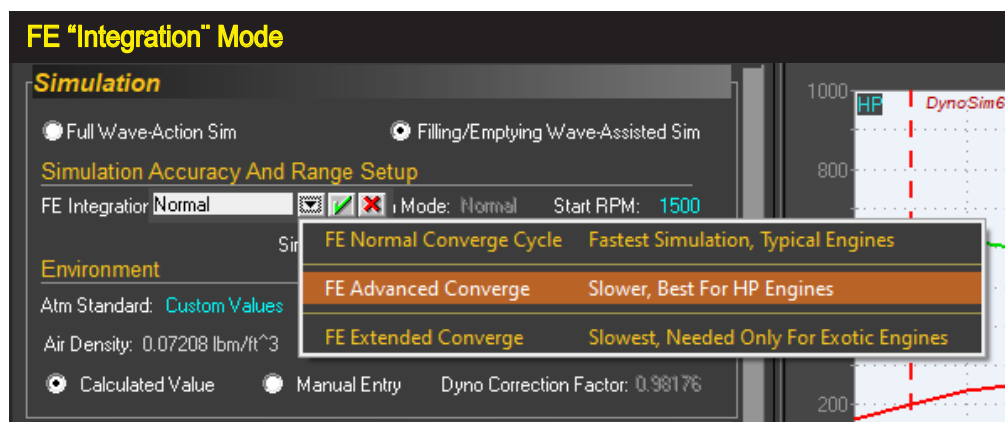
The *FE Simulation* used in both Dynomation6 and DynoSim6 provides an Integration menu that can help improve simulation accuracy on engines with complex pressure wave dynamics. The *Normal* and/or *Advanced* selections are usually sufficient to provide good simulation accuracy. But if you notice pressure irregularities (such as "ringing" pressure peaks) on specific engines, using the *Advanced* or even the *Extended Converge* selections may help clean up the pressure-wave displays.

Simulation Category

Version 6 also offers an *Advanced Converge* mode (the current program default) that runs the simulation multiple times to home-in on a solution with excellent accuracy for almost all engines. An *Extended Converge* mode is also available; designed specifically for unusual and/or very high-speed racing engines (e.g., ProStock, Formula 1, and similar). The *Extended* mode can increase calculation times. If you select the *Extended* mode and see very little change in the horsepower curve, you can switch back to the less calculation intensive modes.

The actual number of calculation cycles performed are displayed in the *Simulation Log* (open the Log using the *Simulation Menu* located at the top of the program screen).

Dynomation6 Only Simulation
"Mesh Mode"

Dynomation6 calculates gas-dynamics within engine passages by dividing the intake and exhaust passages into small, equal-size volumes, called meshes. The smaller the mesh size, the greater number of mesh volumes into which the passages will be divided. This can improve simulation accuracy, but at the expense of calculation times, since a larger number of mesh volumes require additional mathematical steps to process and integrate the results.

Dynomation6 incorporates three mesh sizes shown in the **Wave-Action Mode** field: *Normal, Fine* and *Optimized*. In most cases, the *Normal* mesh size is sufficient to accurately analyze gas dynamics within intake and exhaust passages. But, in some cases, a more rigorous analysis is required to overcome instabilities, shock waves in the exhaust runners, or odd pressure-wave interactions.

> **Important**: If you notice a pressure spikes or oscillations in the intake or exhaust pressure curves, it may be a calculation anomaly that will be minimized with more rigorous analysis offered in the *Fine* and *Optimized* modes. However, lacking pressure-trace anomalies or other unusual results, the *Normal* level

Wave-Action "Mesh" Mode
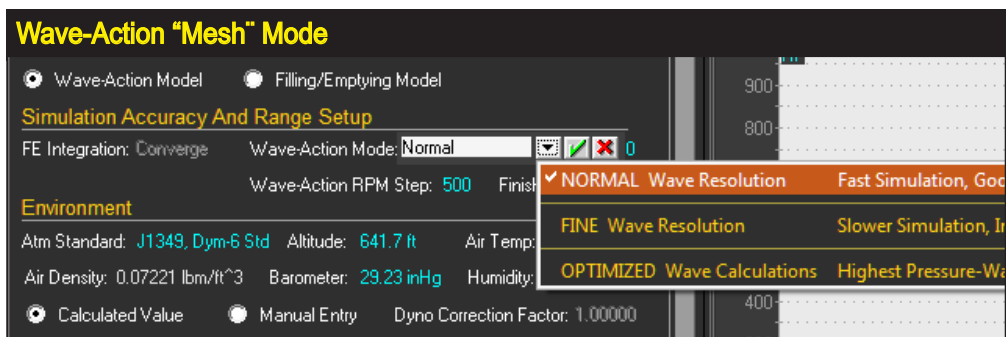
The Wave-Action simulation calculates gas-dynamics by dividing the intake and exhaust passages into small volumes, called meshes. Smaller mesh sizes can improve simulation accuracy, but at the expense of calculation times. The *Normal* mesh is usually sufficient to accurately analyze gas dynamics and provide good simulation accuracy.

Simulation Category

typically provides excellent simulation accuracy and fastest calculation times.

RPM Ranges And Atmospheric Standards

The SIMULATION Category also includes fields that specify the rpm range for the simulation (actual **Start & Finish RPM** points for mathematical analysis*)*. The narrower the range, the faster the simulation is performed, since fewer mathematical operations are required.

Note: If you are using a supercharging system and it's developing errors at higher engine speeds (like *overspeed* or *choke*), you can reduce the upper rpm limit to help prevent these errors. In addition, if you are running an *Iterative Test* (see the Quick and Pro Iterator chapters later in this User Guide), keeping the RPM range as narrow as possible will substantially reduce testing time.

You can also set ambient atmospheric conditions (or Standards) for the simulation by selecting one of the built-in models available from the *Atmospheric Standard* menu. You can apply SAE, ISO, and other world standards commonly used for dynamometer testing. The default used in Dynomation6 and DynoSim6 is different from Dynomation5/DynoSim5, and is based on the SAE J1394 v3 standard widely used in modern dyno-test facilities. The available Standards selections are:

• **Standard Atmosphere:**	**Temp (F): 59.0**	**Humidity: 0%**	**Barometer(in): 29.921**
• **AE J606/7:**	**Temp (F): 60.0**	**Humidity: 0%**	**Barometer(in): 29.921**
• **ISO 1585:**	**Temp (F): 60.0**	**Humidity: 0%**	**Barometer(in): 29.530**
• **JIS D 1001:**	**Temp (F): 68.0**	**Humidity: 0%**	**Barometer(in): 29.234**
• **Dynomation-5 Standard:**	**Temp (F): 68.0**	**Humidity: 0%**	**Barometer(in): 29.919**
• **SAE J1349 v3, Ver-6 Std:**	**Temp (F): 77.0**	**Humidity: 0%**	**Barometer(in): 29.234**
• **SAE J1394 v1:**	**Temp (F): 77.0**	**Humidity: 0%**	**Barometer(in): 29.318**
• **SAE J1394 v2:**	**Temp (F): 77.0**	**Humidity: 0%**	**Barometer(in): 29.380**
• **SAE J1394 v3:**	**Temp (F): 77.0**	**Humidity: 0%**	**Barometer(in): 29.234**
• **SAE J1995:**	**Temp (F): 77.0**	**Humidity: 0%**	**Barometer(in): 29.530**

In addition to the available standards, you may enter any **Altitude**, **Air Temperature**, **Barometer**, and **Humidity** value you wish. The changes in environmental variables are applied to the simulation through a "correction factor." This value is normally calculated by Dynomation after you make a menu selection or enter specific atmospheric values. However, you can click **Manual Entry** and directly enter any dyno correction factor you wish.

ShortBlock Category

Short Block Component Menu
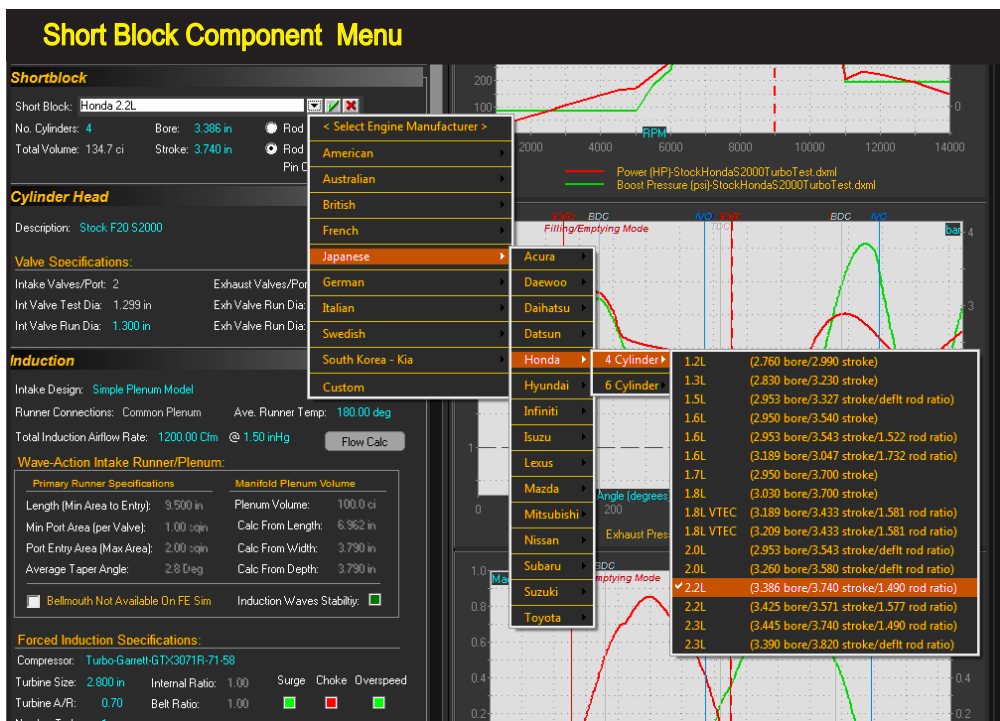
The *Short Block* component menu contains over 500 bore and stroke combinations of popular domestic and import engines that you can instantly use in any engine simulation. In addition, you can directly enter custom specifications for any engine into the *Short Block* component field.

THE SHORTBLOCK COMPONENT CATEGORY

The **ShortBlock** menu is located in the **SHORTBLOCK** *Component Category*. By opening this menu, you are presented with a variety of domestic and import "predefined" shortblock configurations. If any one of these choices is selected, the appropriate bore, stroke, number of cylinders, rod ratio, and pin offset (see entering *Rod Ratio, Rod Length And Pin Offset*, next) will be loaded into the *ShortBlock* category. In addition to the predefined shortblocks in the menu, you can directly enter any shortblock name (description) into the **ShortBlock** field, then enter custom values for **Stroke**, **Bore**, **Number Of Cylinders** (also **Rod Ratio** and **Piston Pin Offset**) within the acceptable range limits of the program (as indicated at the range limit line at the bottom of the screen).

> **Note:** The *Short Block* menu can be considered a "handy" list of common engine bores, strokes, rod-length and offset values. When you select a shortblock, you are not selecting any "hidden" characteristics, like material composition (aluminum vs. cast iron) or other component assumptions (such as typical cylinder

ShortBlock Category

ShortBlock Component Menu

Shortblock

Short Block: Acura 3.0L
No. Cylinders: 6   Bore: 3.540 in   ○ Rod Length: 5.023 in
Swept Volume: 181.3 ci   Stroke: 3.070 in   ● Rod Ratio: 1.636 *Estimate*
Engine Friction: Dynomation-6 Default   Pin Offset: 0.000 in

Rod Length & PinOffset:

Note: Rod Ratio will default to 1.636 if actual value is not available in the Short Block menu.

Pin Offset is set to 0.000-inch for all engines loaded from menu.

"Estimate"
Appears when Rod Length from ShortBlock Menu is an estimated value.

The *Short Block* component menu (shown on previous page) contains over 500 bore and stroke combinations. Selections also include rod-ratio, however if factory data could not be located, a default value of 1.636 is used and *Estimate* is displayed. Pin offset specifications are set to 0.000-inch for all engines loaded from the Short Block menu.

head type). The **Short Block** menu only loads the *Bore*, *Stroke*, *Rod Ratio*, *Pin Offset*, and the *Number Of Cylinders* into the simulation.

Entering Rod Ratio, Rod Length And Piston-Pin Offset

In addition to the *Bore*, *Stroke*, and *Number Of Cylinder* specifications, **Short Block** menu selections will load the **Rod Ratio** (or if the factory values could not be found, a default value of 1.636 will be used and "***Estimate***" will be displayed. The **Rod Ratio** is the length of the connecting rod divided by the stroke length and is commonly used to evaluate rod angularity and piston-to-cylinderwall friction, a significant source of frictional losses within the engine.

You may also manually enter values for *Rod Length* or *Rod Ratio* within program range limits (displayed at the bottom of the screen in the Status Bar). To enter *Rod Ratio*, select the "radio" button next to the **Rod Ratio** field, then enter the

The **Piston-Pin Offset is a measure of how far off-center the piston pin is located within the piston. The simulation models these changes in pin offset when you enter positive values for a pin location shifted toward the *Major Thrust Face* and negative values (for offsets toward the *Minor Thrust Face)*.

Piston-Pin Offset Values

Major Thrust Face

Minor Thrust Face

Enter Positive Values

Enter Negative Values
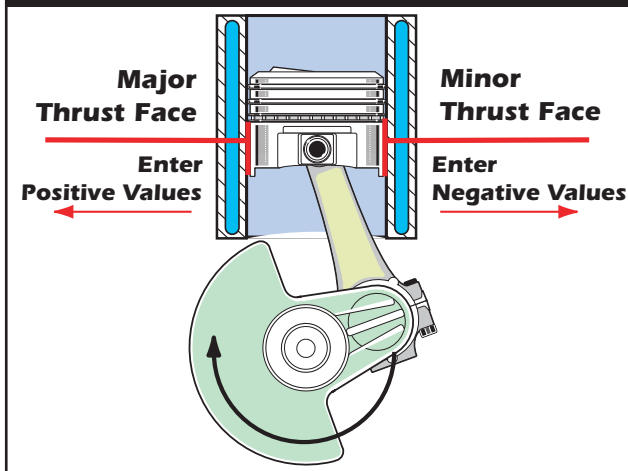
desired value. the simulation will display the <u>calculated value for **Rod Length**</u>. Or, by selecting the **Rod Length** radio button, you can directly enter a length value; <u>*Rod Ratio* will then be calculated</u> and displayed in the **Rod Ratio** field.

The **SHORTBLOCK** category also includes a field for **Piston-Pin Offset**. This is a measure of how far off-center (from the rod-to-crank centerline) the piston pin is located within the piston. Enter positive values to indicate a shift toward the *Major Thrust Face* and negative values to shift the pin toward the *Minor Thrust Face*.

> **Note1:** *Piston-Pin Offset* is set to 0.000-inch for all engines loaded from **Short-Block** menu.
> **Note2:** You can use Piston-Pin Offset data input to experiment with <u>*offset cylinder bores*</u>, where the piston and the cylinder are offset from the crank centerline.

Engine Friction Modeling In Dynomation6
(Not Included In DynoSim6)

Frictional losses from the motion of internal and external engine components are well understood and can be accurately simulated and predicted. However, the nearly endless variety of choices available to the engine designer/builder for machining and surface-finishes, metal alloys, clearances, lubricants, friction-reducing coatings, use of roller assemblies (like roller lifters, roller rocker-arm pivots and tips), oil types and temperatures of lubricants, piston materials, ring selections, and even oil seal types can interact and combine to modify frictional losses from predicted values. To allow the simulation to accommodate a wide range of engine designs, you can alter the frictional models to better suit individual applications.

The *Engine Friction* menu is located in the **SHORTBLOCK** component category (not offered in DynoSim6). The menu includes the following choices that alter the frictional characteristics of the simulation in relation to the baseline "default" models:

- **Extremely Low Friction:**   **Reduction: -0.5 (approx. 30% reduction)**
- **Extensive Friction Reduction:**   **Reduction: -0.4 (approx. 25% reduction)**
- **Dynomation5 Default:**   **Reduction: -0.3 (approx. 20% reduction)**
- **Pro Race Design & Prep:**   **Reduction: -0.2 (approx. 15% reduction)**
- **Typical Race Prep:**   **Reduction: -0.1 (approx. 10% reduction)**
- **Dynomation6 Default:**   **Default Frictional Model (Baseline)**
- **Blueprinted Buildup:**   **Increase: 0.1 (approx. 5% increase)**
- **Basic HP Street Buildup:**   **Increase: 0.2 (approx. 10% increase)**
- **Stock OEM Street Engine:**   **Increase: 0.3 (approx. 15% increase)**
- **High Frictional Losses:**   **Increase: 0.4 (approx. 20% increase)**

The frictional losses of the Internal Combustion engine fall into specific categories that are more or less consistent across a wide range of engine designs. Within four-stroke engines, the friction attributable to the Piston assembly, the Crankshaft, the Connecting Rods, and the Valvetrain fall within the following ranges:

ShortBlock Category

- **Piston/Rings:**   **35 to 60% of total engine friction**
- **Crankshaft:**   **15 to 20% of total engine friction**
- **Connecting Rods:**   **15 to 20% of total engine friction**
- **Valvetrain:**   **8 to 20% of total engine friction**

When you alter friction analysis by choosing other than the *Default Dynomation6* method (from the ***Engine Friction*** menu), the piston-to-cylinderwall and ring-to-cylinderwall contact surfaces contribute most significantly to changes in frictional characteristics. The result of increasing or decreasing friction in the simulation will have increasing effects at higher engine speeds.

In addition to modifying simulation results to better match specific engine designs, changing frictional models can be useful technique to slightly "tilt" engine power and torque curves to better match the results of real-world dyno tests. Since frictional modifications have no (or very little) effect on engine flow or the energy produced in the cylinders from combustion, frictional adjustments will not affect cam timing, intake or exhaust tuning, or other interrelated aspects of engine design. In most cases, frictional modeling can be considered (nearly) isolated from other aspects of engine design.

Additional Notes On Engine Friction

Frictional losses are created from several distinct physical processes that are commonly labeled: *Solid Contact, Boundary, Mixed, and Hydrodynamic.* Without introducing too much detail (which can be readily found on the Internet), the frictional losses in crankshaft, camshaft, and rod bearings principally fall into the hydrodynamic category. Under these conditions, metal surfaces do not touch and are held apart by a film of lubricant. The main frictional losses in this regime are due to viscous motion of the lubricant within bearing clearances. This characteristic changes with temperature and oil viscosity, and remarkably does not reach a minimum with the lowest oil viscosity, since very low viscosity can allow metal-to-metal contact on highly loaded surfaces. In general, a 30 to 40 weight oil, warmed to operating temperatures, produces the best horsepower on high-output racing engines.

On the other hand, ring, piston, (and to some degree) lifter-lobe contact surfaces spend time in *Boundary* and *Mixed* lubrication regimes. Here, the greatest friction reduction comes from component surface prep (with the goal to maintain the thickest lubricant film between sliding surfaces), and the selection of optimum materials and designs for component rigidity to maintain uniform clearances between sliding surfaces.

Cylinder-Head Category

Cylinder-Head Type Menu
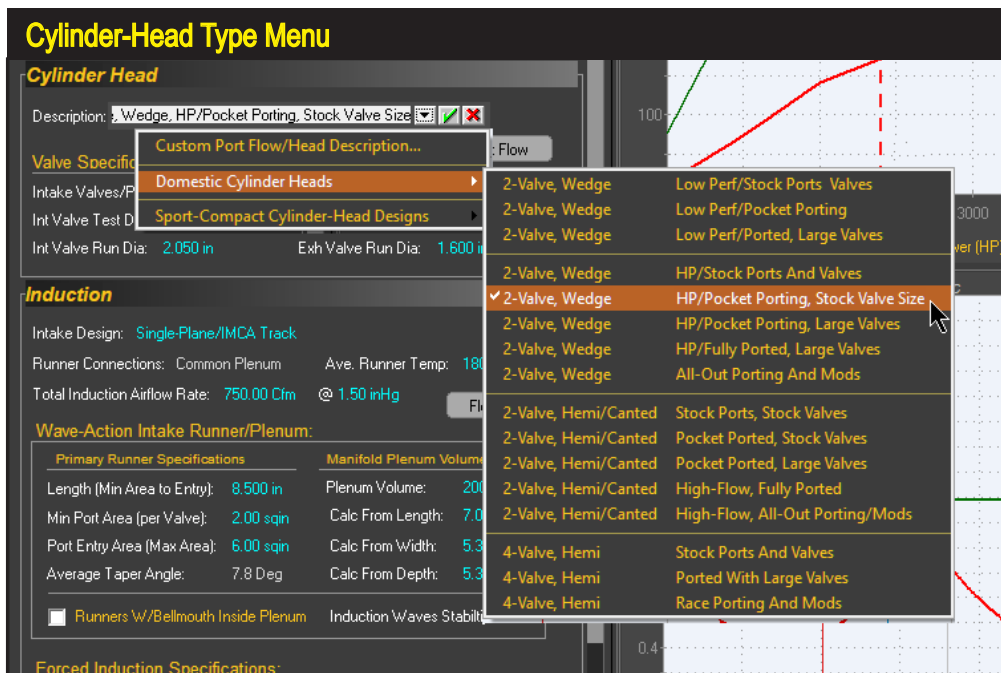
The *Cylinder Head Type (Description)* **menu contains a wide range of head/port choices. The main menu is divided into two groups:** *Domestic Cylinder Heads* **model typical 4-cylinder through V8 passenger car engines. The** *Sport-Compact Cylinder Heads* **selections primarily model newer, multiple-valve-per-port head designs.**

THE CYLINDER-HEAD COMPONENT CATEGORY

The **Cylinder-Head Type** pull-down menu is located in the **CYLINDER HEAD** category. Selections from this menu allow the simulation to model various "generic" intake and exhaust port airflow characteristics for both *Domestic* and *Sport-Compact* applications. Built-in selections range from restrictive low-performance ports to high-performance four-valve cylinder heads. Each grouping of head/port designs includes several stages of modifications.

In addition to the provided choices, the **Custom Port Flow** selection at the bottom of the *Cylinder Head Type* menu lets you directly enter flowbench data, allowing the simulation to model any cylinder head for which airflow test data is available (for convenience, a **Port Flow** button in the *Cylinder Head* category provides fast access to this same custom-airflow, data-entry dialog). Custom port flow data entry will be detailed after the following brief descriptions of the built-in menu choices.

Valve Diameters And Basic Flow Theory

A selection from the *Cylinder Head* menu is the first part of the process that establishes a baseline for the simulation of cylinder-head airflow. A *Cylinder-Head*

Cylinder-Head Category

Selecting Valve Sizes

Selecting a specific valve size fixes the theoretical peak flow of each port. Cylinder heads flow from about 50% to as high as 90% of this value. This percentage, called the *discharge coefficient*, has proven to be an effective link between flow-bench data and the calculation of mass flow moving into and out of the cylinders. In other words, the valve size and the derived discharge coefficient help establish a practical framework to accurately simulate mass flow within a running engine.
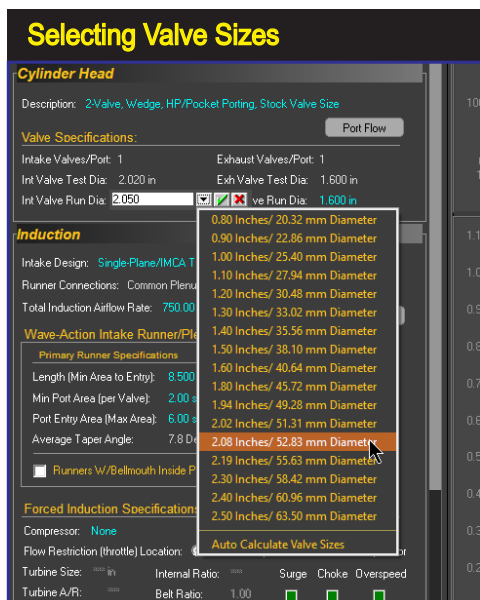
Description selection (or entering *Custom Port Flow* values) dictates the airflow restriction generated by the valves and ports. That is, flow data determines *how much airflow, below the theoretical maximum peak flow for a straight pipe of the same area, will pass through the valve orifice at all valve lift heights*. The percentage of ideal flow is called the *Coefficient of Discharge (CD)* and is always less than 100% (less than 1.0). The CD values are calculated and displayed in the *Port-Flow Dialog* as you enter flow-bench data (described later).

> **Induction Tuning Note:** The *INDUCTION* Category contains a *MinPortArea* menu (in Dynomation6) or an *Average Runner Taper Angle* menu (in DynoSim6) that help you establish the dimensional characteristics of the intake runners needed for pressure-wave analysis. As you change these specifications, you will "tune" the intake runners for lower or higher rpm ranges.

The additional data point needed to characterize overall engine airflow is the *Total Induction Airflow Rate* (also located in the *Induction Category)*. This can be considered the rated flow of the carburetor or throttle plates/valves measured at a specific pressure drop. This establishes an overall flow-rate restriction for entire engine.

When this essential data has been entered, the program is a step closer to determining mass flow within the ports. Unfortunately, the flow in the ports of a running engine is vastly different than the steady-state flow measured on a flow bench. A running engine will generate rapidly and widely varying pressures that directly affect—in fact, they directly cause—the flow of air and exhaust gasses within the engine. To determine the instantaneous flow in all ports at any point during engine

Cylinder-Head Category
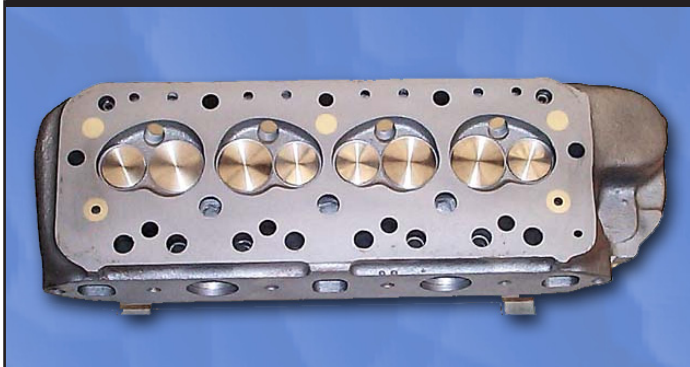
The *Low Performance* cylinder head choices are intended to model cylinder heads that have restrictive ports, valves, and combustion chambers. Heads of this type were often designed for low-speed, economy applications, with little concern for high-speed performance.

operation, the simulation must calculate all internal port and cylinder pressures at closely-spaced, small increments of time throughout the four-cycle process. Overall mass flow into and out of the cylinders is found from the sum of these instantaneous pressures, calculated gas densities, and flow restrictions throughout the runners. After a series of integration operations using this data, engine output and related "dyno" results can be determined. (Not magic, but close!)

Domestic Cylinder-Head Selections

Here is some basic information that will help you determine the appropriate cylinder-head selections (and port flow data) from the built-in menu selections for domestic engine applications.

Note: Each of the generic choices in the Cylinder Head menu has flowbench data associated with it. To view this test data simply select the cylinder head from the menu then click the **Port Flow** button.

Wedge-Chamber, 2-Valve Cylinder Heads

Domestic—2-Valve, Low Performance Cylinder Heads (Three variations: Stock, Ported, and Ported With Large Valves)

There are three *Low Performance* cylinder head selections listed at the top of the **Domestic Cylinder Head** menu. Each of these choices is intended to model cylinder heads that have small ports and valves relative to engine displacement. Heads of this type were often designed for low-speed, economy applications, with little concern for high-speed performance. Early 260 and 289 smallblock Ford and to a lessor degree early smallblock Chevy castings fall into this category. These choices have the lowest discharge coefficient of all the head configurations listed in this menu. Minimum port cross-sectional areas are approximately 60% of the valve areas, and if *Auto Calculate Valve Size* has been selected, relatively small (compared to the bore diameter) intake and exhaust valve diameters will be selected.

Cylinder-Head Category

The *2-Valve, HP Wedge Cylinder Head* selections model cylinder heads that have ports and valves sized with performance in mind, like this Edelbrock smallblock Chevy head.

Note: These low-performance cylinder-head choices are a good option if you wish to model flathead (L-head & H-head) and hybrid (F-head) engines for which you do not have actual flow data. While the port flow in these engines is quite restrictive, by selecting **Low-Performance** and manually entering the actual valve sizes, the simulation will provide a good approximation of power output from these early engine designs.

Domestic—2-Valve, Wedge Cylinder Heads (Three Variations: Stock, Ported, Large Valves)

The first three wedge cylinder-head selections model castings that have ports and valves sized with performance in mind. Ports are only somewhat restrictive for high-speed operation, and overall port and valve-pocket design offers a good compromise between low restriction and excessive flow velocity at higher engine speeds. The stock and pocket-ported choices are applicable to high-performance street to modest racing applications.

Domestic—2-Valve, Wedge/Fully Ported, Large Valves

The fourth wedge head moves away from street applications. This casting has improved discharge coefficients, greater port cross-sectional areas, and increased valve sizes. Consider this head to be a modified, high-performance, factory-type casting that has good flow for racing applications. It does not incorporate "exotic" alterations, like raised and/or welded ports requiring custom-fabricated intake manifolds.

Domestic—2-Valve, Wedge/ProStock Porting And Mods

The last choice in the wedge group is designed to model high-flow, professional drag-racing cylinder heads. These heads are designed for one thing: Maximum power. They usually require hand-fabricated intake manifolds, have high valve discharge coefficients, and the ports have the largest cross-sectional areas in the smallblock group. This head develops sufficient airflow speeds for good cylinder filling only at high engine rpm.

Cylinder-Head Category

The *Canted-Valve Cylinder Head* selections have ports with generous cross-sectional areas and valves that angle toward the port mouths. The first three menu choices model <u>oval-port</u> designs. The final two selections simulate performance <u>rectangular-port</u> heads. This L29 bigblock Chevy would be best modeled by the second or third menu choice—the fourth menu choice models a head with flow capacity beyond the capabilities of L29 castings.
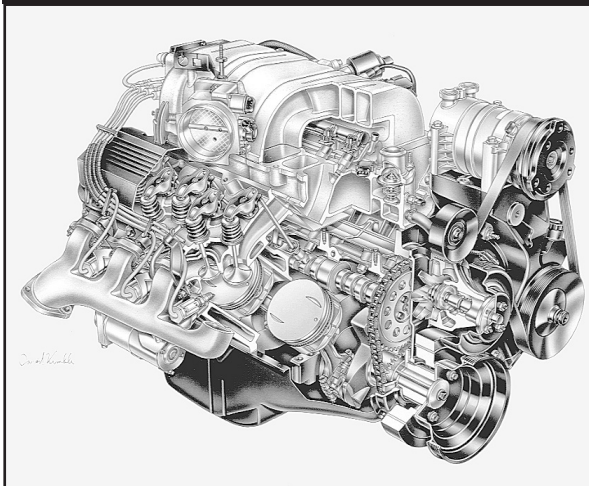
Hemi/Canted 2-Valve Cylinder Heads

The following *Hemi/Canted-Valve* selections are modeled after heads with valve stems tilted toward the inlet or discharge surfaces of the cylinder heads. This design reduces restriction at the valve, improves CD, and is particularly beneficial at higher engine speeds.

Domestic—2-Valve, Hemi/Canted-Valve Cylinder Heads (Stock, Ported, Large Valves)

The first three choices best model <u>oval-port</u> configurations. These smaller cross-sectional area ports provide a good compromise between low restriction and high flow velocity for larger displacement engines. The stock and pocket-ported choices are suitable for high-performance street to modest racing applications.

The next two selections best model extensively modified <u>rectangular-port</u> heads. These choices are primarily, all-out, bigblock heads, however, they also model other aggressive, high-performance designs, like the Chrysler Hemi head.

Domestic—2-Valve, Hemi/Canted-Valve, Rectangular Ports/Fully Ported

These <u>rectangular-port</u> heads have high discharge coefficients, large port cross-sectional areas, and increased valve sizes. This head is basically a factory-type casting but extensively improved. However, it does not incorporate "exotic" modifications, like raised and/or welded ports that require custom-fabricated manifolds.

Domestic—2-Valve, Hemi/Canted-Valve, Rectangular ProStock Ports/Mods

The last choice in the *Hemi/Canted-Valve* group is designed to model state-of-

Cylinder Head Category

The *2-Valve, Hemi, HP Cylinder Head* selections have ports with generous cross-sectional areas and valves that angle toward the port inlets (the Dodge 5.7L Hemi is shown here). These heads have improved discharge coefficients and their high-per-formance designs typically offer the highest flow capability for 2-valve cylinder heads. Heads of this type have the greatest potential of producing the high-est horsepower within the first three groups in the *Cylinder Head Type* menu.

the-art, ProStock (and Hemi) drag-racing cylinder heads. These custom pieces, like their wedge-design counterparts, are more-or-less built from the ground-up for maximum power. They require custom-fabricated intake manifolds, have optimum valve discharge coefficients, and the ports have the largest cross-sectional areas in the entire 2-valve **Cylinder Head** menu. These specially fabricated cylinder heads only develop sufficient airflow for good cylinder filling with large displacement engines at high engine speeds.

Domestic 4-Valve Cylinder Heads

The next three selections in the *Domestic Cylinder Head* submenu model 4-valve cylinder heads. These are very interesting choices since they simulate the effects of very low-restriction ports and valves used in many current high-performance applications. The individual ports in 4-valve heads begin as single, large openings, then divide into two Siamesed ports, each having a small (relatively) valve at the combustion chamber interface. Since there are two intake and two exhaust valves per cylinder, the valve flow area exposed as the valves lift off the seats (called *curtain* area) is considerably greater than in the largest single-valve-per-port designs. In fact, 4-valve heads can offer more than 1.5 times the curtain area of the largest 2-valve heads. This large flow area, combined with high-flow, low-restriction ports, improves air and fuel flow at low valve lifts *and* at high engine speeds. Unfortunately, the ports offer an equally low restriction to reverse flow (reversion of charge from the cylinder back into the intake port) that commonly occurs at low engine speeds when the piston moves up the cylinder from BDC to Intake Valve Closing (IVC) on the early portion of the compression stroke. For this reason, 4-valve heads, even when fitted with more conservative ports and valves, can be a poor choice for small-displacement, low-speed engines, unless camshaft timing is carefully de-

Cylinder Head Category
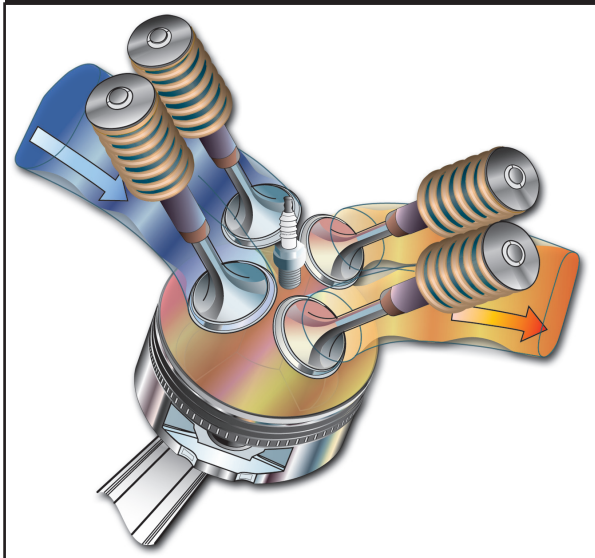
The *4-Valve Cylinder Head* selections model cylinder heads with two intake and two exhaust valves per cylinder. These heads can offer more than 1.5 times the curtain area of the largest 2-valve heads. This large valve flow area, combined with high-flow, low-restriction ports greatly improves air and fuel flow at higher engine speeds.

signed to complement the low-lift flow capabilities of these cylinder heads (using, for example, variable-valve timing...V-V-T). On the other hand, the outstanding flow characteristics of 4-valve heads puts them in another "league" when it comes to horsepower potential at high engine speeds.

Domestic—4-Valve, Hemi/Canted-Valve, Stock Ports And Valves

The first choice in the 4-valve group simulates a 4-valve cylinder head that would be "standard equipment" on factory high-performance engines. These heads offer power comparable to high-performance 2-valve castings equipped with large valves and porting work. However, because they still have relatively small ports, reasonably high port velocities, and good low-lift flow characteristics, they can show a boost in low-speed power over comparable 2-valve heads.

Domestic—4-Valve, Hemi/Canted-Valve, Ported With Large Valves

The next choice incorporates mild performance modifications. Larger valves have been installed and both intake and exhaust flow has been increased by improved port designs. However, care has been taken not to increase the minimum cross-sectional area of the ports. These changes provide a significant increase in power with only slightly slower port velocities. Reversion has increased slightly, but overall, these heads should show a power increase throughout the rpm range on most engines.

Domestic—4-Valve Hemi/Canted-Valve, Race Porting And Mods

The final choice, like the other *Race-Porting-And-Mod* choices in the **Cylinder Head** menu, models a racing cylinder head. This selection has the greatest power potential. The ports are considerably larger than the other choices, the valves are larger, and the discharge coefficients are the highest possible. These

heads suffer from the greatest reversion effects, especially with late IVC timing on small-displacement engines.

Domestic-Engine Simulation Tip: If you would like to know what "hidden" power is possible using any engine combination, try the *4-Valve Hemi/Canted-Valve, Race Porting And Mods* cylinder-head choice. It is safe to say that the only way to find more power, with everything else being equal, would be to use forced induction, nitrous-oxide injection, or exotic fuels.

Sport-Compact Cylinder Head Selections

The first three selections in the **Sport-Compact Cylinder Head Type** submenu model the low-restriction ports and valves used in modern, 4-valve cylinder-heads; the basic mainstay of the Sport-Compact enthusiast. The individual ports in 4-valve heads begin as single, relatively large openings, then neck down to two Siamesed ports, each having a small (relatively) valve at the combustion chamber interface. Since there are two intake and two exhaust valves per cylinder, valve curtain area (area exposed around an open valve through which air/fuel can pass) is considerably larger than the largest single-valve-per-port designs. In fact, 4-valve heads can offer more than 1.5 times the curtain area of the largest 2-valve heads. This large flow area, combined with the high-flow, low-restriction ports greatly improves air and fuel flow into the cylinders at low valve lifts and high engine speeds. However, the ports offer an equally low restriction to reverse flow (reversion) that can occur at low engine speeds when the piston moves up the cylinder from BDC to Intake Valve Closing (IVC). For this reason, 4-valve heads, even when fitted with more conservative ports and valves, can be a poor choice for small-displacement, low-speed engines, unless camshaft timing and valve lift is carefully designed to complement the flow capabilities of these cylinder heads.

Sport-Compact Cylinder-Head Menu Selections

Port Flow

-Head Designs

4-Valve, Pentroof, Stock Ports And Valves
4-Valve, Pentroof, Ported With Large Valves
4-Valve, Pentroof, Race Porting And Mods

2-Valve, Pentroof, Stock Ports, Stock Valves
2-Valve, Pentroof, Pocket Ported, Stock Valves
2-Valve, Pentroof, Pocket Ported, Large Valves
2-Valve, Pentroof, High-Flow, Fully Ported
2-Valve, Pentroof, High-Flow, All-Out Porting/Mods

Flow Calc

per Valve): 1.75 sqin
per Angle: 1.8 Deg

The *Sport-Compact Cylinder Head* selections are divided into two groups. The most common is the 4-Valve type with some version of a pentroof combustion chamber. The second group are 2-Valve heads, having good flow characteristics, but considerably lower power potential than the 4-Valve designs.

Cylinder Head Category

4-Valve, Pentroof, HP Cylinder Heads
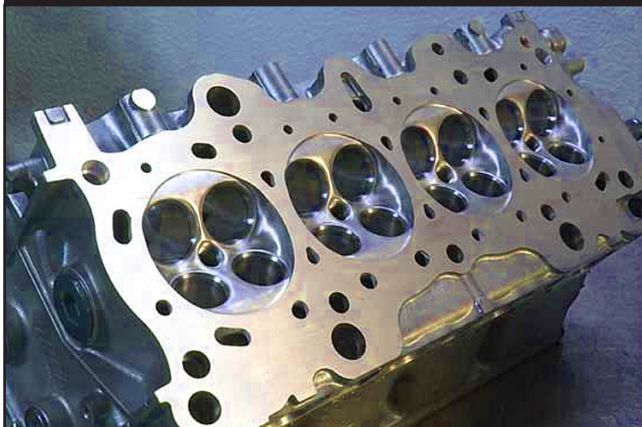
The *4-Valve Cylinder Head* selections model cylinder heads with two intake and two exhaust valves. These heads can offer more than 1.5 times the curtain area of the largest 2-valve heads. This large valve flow area, combined with high-flow, low-restriction ports greatly improves air and fuel flow into the cylinders at high engine speeds.

Sport-Compact 4-Valve Cylinder Heads

Sport-Compact—4-Valve, Pentroof, Stock Ports And Valves

Note: Each of the generic choices described below in the Cylinder Head menu has flowbench data associated with it. To view this test data simply select the cylinder head from the menu, then click the **Port Flow** button.
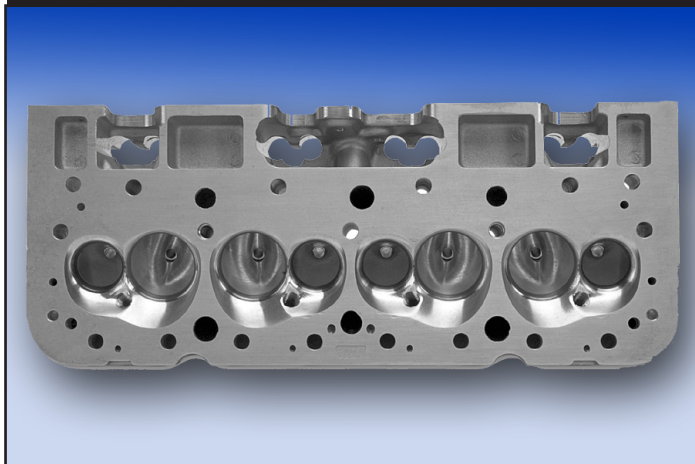
The first choice in the 4-valve group simulates a 4-valve cylinder head that would be "standard equipment" on factory high-performance, sport-compact engines. Although excellent for street performance, since they have relatively small ports, reasonably high port velocities, and good low-lift flow characteristics, with proper cam timing they often show a boost in low-speed power over comparable 2-valve heads (and at higher engine speeds, they will outperform all but the most highly modified 2-valve heads).

4-Valve Cylinder Heads are commonly used on Import high-performance street engines, like the Variable Valve Timing powerplants installed in many Honda vehicles. These engines offer a level of performance and driveability that is impossible with 2-valve cylinder heads.

Honda VTEC, 4-Valve, Pentroof Engine
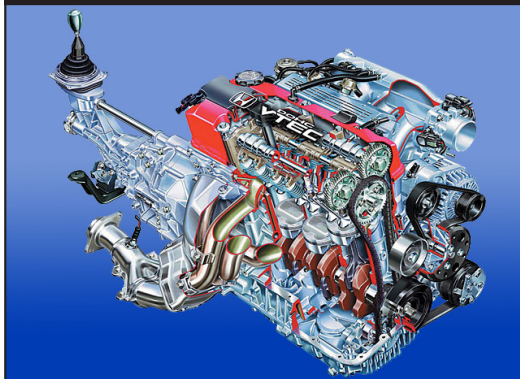
Cylinder Head Category

Sport-Compact—4-Valve, Pentroof, Ported With Large Valves

The next choice incorporates mild performance modifications. Larger valves have been installed and both intake and exhaust flow has increased from improved port designs. However, care has been taken not to increase the minimum cross-sectional area of the ports. These changes provide a significant increase in power with only slightly slower port velocities. Reversion has increased, but overall, these heads will show a power increase throughout most of the rpm range on many engines.

Sport-Compact—4-Valve, Pentroof, Race Porting And Mods

The final choice, like other Racing options in the **Cylinder Head** menu, models a very efficient, high-flowing cylinder head. This selection has the greatest power potential of all sport-compact menu choices. The ports are considerably larger, the valves are larger, and the discharge coefficients are the highest possible. These heads suffer from the greatest reversion effects at lower engine speeds, especially with late IVC timing on small-displacement engines.

Sport-Compact Simulation Tip: If you would like to know what "hidden" power is possible using any particular engine combination, try this cylinder head choice. It is safe to say that the only way to find more power, with everything else being equal, would be to add forced induction, nitrous-oxide injection, or use exotic fuels.

Sport-Compact-Design, 2-Valve Cylinder Heads

The next five choices in the Sport-Compact/Import *Cylinder Head* menu model pentroof-chamber, "canted-valve" heads with one valve per port, each tilted or canted toward the port inlet/discharge surface. This positioning improves discharge coefficients and overall airflow. All ports in this menu group have cross-sectional areas sized for performance.

Sport Compact/Import—2-Valve, Pentroof, HP Cylinder Heads (Stock, Ported, Large Valves)

The first three choices can be considered suitable for street/performance applications. These selections model smaller cross-sectional area ports (sometimes oval or round) that provide a good compromise between low restriction and high flow velocity for production engines.

Sport-Compact/Import—2-Valve, Pentroof, High-Flow, Fully Ported

The final two Pentroof, 2-valve selections simulate extensively modified, high-performance cylinder heads. These castings have high discharge coefficients, large port cross-sectional areas, and increased valve sizes. These heads are basically factory-type castings but incorporate improved porting and valve work. However, they do not use "exotic" modifications, like raised and/or welded ports that require custom-fabricated manifolds.

Cylinder Head Category

Sport-Compact/Import—2-Valve, Pentroof, High-Flow, All-Out Porting/Mods

The last choice in this group is designed to model state-of-the-art, 2-valve, drag-racing cylinder heads. These custom pieces are extensively modified for maximum power. They require hand-fabricated or custom intake manifolds, have optimum valve discharge coefficients, and the ports have the largest cross-sectional areas of the 2-valve head designs. These cylinder heads only develop sufficient airflow for good cylinder filling at high engine speeds.

Custom Port Flow And The Port-Flow Dialog

The simulation will accept flowbench data from a variety of sources. Basic flow-test data can be directly entered into the *Port Flow* dialog using any valve size, at any pressure drop. In addition, you can load version 5 and 6 airflow files (.DFW files). You can also *Import FlowPro* files containing individual-port-flow data obtained from flow-bench testing using Audie Technology *FlowPro* software. Open the *Port Flow* dialog by clicking the *Port Flow* button in the *Cylinder Head* component category or clicking the *Port AirFlow* icon in the Toolbar.

To manually enter custom flowbench data, first select a suitable name for the flow data and enter it in the *Description* field. This description will also be displayed in the *Cylinder Head* category *Description* field. Next, select the number of data points in your flowbench test data using the *Data Points* field (a minimum of 4

Port-Flow Dialog Data Import

You can enter flowbench data from a variety of sources. Open the *Port-Flow Dialog* by clicking the *Port Flow* button in the *Cylinder Head* component category. Basic flow-test data can be directly entered using any valve size at any pressure drop.

Cylinder Head Category

Port-Flow Dialog Data Import

Basic flow data can be entered manually or you can load flowbench data from external sources: Load *Dynomation Airflow* files (.DFW, version 5 and 6), or use the *Import* buttons to load *Flow Pro* files produced on a flowbench equipped with Audie Technology software.

points are required; click the up icon to increase, down to decrease). Also, enter the *Test Valve Diameters* (valve sizes used on the flow bench), and the *Pressure Drop* (in inches, or mm, of H_2O) used during testing. Enter the number of *Valves Per Port* for individual intake and exhaust ports. Finally, enter *Flow* and *Valve-Lift* data for each test point.

Flow Data Entry Tip-1: If you press the *Calc Others* button after entering at least two valve-lift heights, the simulation will fill in the remaining valve-lift fields with the same "step" value that was established in the first two fields.

Flow Data Entry Tip-2: If you have fewer than four test points for one of the valves, duplicate the highest measured flow rate at highest lift value to "flush out" the remaining empty fields. This technique will produce reasonable results, but it is best to have actual flow data that covers the entire valve-lift curve.

Flow Data Entry Tip-3: For cylinder heads with more than one valve per port, the flow values measured on the flowbench should be a measure of the

Select Number Of Valves Per Port

The simulation will accept flow data from cylinder heads using 1, 2, or 3 valves-per-port. Multiple valves-per-port cylinder heads are flow tested by opening *all valves in each port* to the same lift while recording the flow rate. The number of valves is used to calculate the overall flow area and the *Coefficient of Discharge*. Since the flow data is always linked to the number of valves, this value can only be selected/changed from within the *Port Flow* dialog.

Cylinder Head Category

Every simulated engine has two intake and two exhaust valve diameters: one pair of *Test Diameters* and one pair of *Running Diameters*. Normally, these valve diameters are the same sizes, but Dynomation allows you to enter different *Running Diameters* to test the effects of installing larger (or smaller) intake and/or exhaust valves without changing cylinder-head flow data. Keep in mind that the accuracy of the simulation is best if both *Test* and *Running* diameters are kept the same.

flow for **both** (or **all**) valves in each port opened to the same lift height. *The recorded flow must be the combined flow for all valves in each port.*

You can save flow data in a separate file (.DFW) by pressing the **Save As** button. However, even without doing this, <u>head flow data entered in the **Port Flow Dialog** will be saved</u> with engine data within the current **.DXML** engine file. It's easy to recall previously saved flow data (a .DFW file), you can also **Open** flow-data files created in other Motion Software engine simulations, including DynoSim4 and 5 (load .FLW files).

Pressing the **OK** button in the *Port Flow* dialog will transfer flow-test data into the engine simulation and display the flow-test data **Description** in the **CYLINDER HEAD** category.

TEST Valve Diameters

The intake and exhaust **Test Diameters** are entered in the Port-Flow Dialog. These are the diameters of the valves used in the cylinder head during flow-bench testing. Normally, these are the same diameters as the valves used in the simulated engine (the **Running Diameters**), but Dynomation allows you to enter a different Running Diameter to test the effects of installing larger (or even smaller) intake or exhaust valves without retesting the cylinder heads (flow data is "scaled" up or down with the valve diameters). The accuracy of the simulation is best if both **Test** and **Running** diameters are the same, but small differences between these diameters (no more than 5% of the **Test** valve size) will be modeled accurately by the simulation. The results will be as if the larger/smaller valves were installed in the cylinder heads, the heads were retested, and the new flow data was used in Dynomation.

Engine *RUNNING* Valve Diameters

The **Running Valve Diameter** menus are located in the *Cylinder Head* category.

Cylinder Head Category

Auto Calculate Valve Sizes
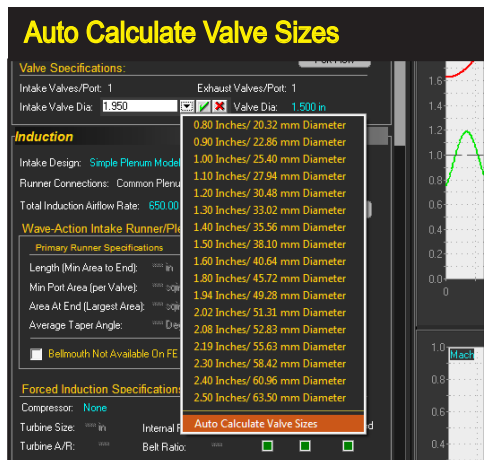
The last selection in the Valve Diameter Menus is *Auto Calculate Valve Sizes* (active only when a "generic" cylinder head has been selected from the *Cylinder Head Description* menu). This feature determines nominal intake and exhaust valve diameters. *Auto Calculate Valve Sizes* will always select valves of an appropriate diameter for the cylinder heads under test and ensure that valve sizes (including in multiple valves-per-port applications) are never too large for the current bore diameter. *Note: This selection is dimmed whenever custom airflow values have been entered in the Port Flow Dialog.*

This selection is the diameter of the valves that you wish to use in the simulated engine. As stated previously, the accuracy of the simulation is best if *Test* Diameters (entered in the *Port-Flow* dialog and used during flow-bench testing) are the same as the *Running* diameters. However, small differences between these diameters (no more than 5% of the *Test* valve diameter) can be modeled accurately by the simulation. If you use this feature, make sure the CD values (can be displayed on any crank-angle graph) have not exceeded 0.95.

The last selection in each menu is *Auto Calculate Valve Sizes*. This feature is functional only when a "generic" cylinder head was selected from the *Cylinder Head Description* menu. *Auto Calculate* instructs the simulation to determine the nominal intake and exhaust valve diameters for use with the current engine based

Selecting a specific valve size will disable the *Auto Calculate Valve Size* feature. Select from the provided sizes (displayed in both Metric and US measurements), or you can directly enter any valve dimension within the range limits of the simulation (range limits are shown in the *Status and Range Limit Line*, at the bottom of the main program screen).

Entering *Running* Valve Sizes

Cylinder Head Category

on an assessment of the bore diameter and cylinder-head selection. When the *Auto Calculate* function is activated, *Auto* will be displayed next to the calculated sizes, and it will remain active on the current engine until turned off (by choosing *Auto Calculate* a second time or by selecting a new diameter for either the intake or exhaust valve).

Auto Calculate will always select valves of an appropriate diameter for the engine under test (including multiple valves-per-port applications) and ensure that valve sizes are never too large for the current bore diameter.

> **Note-1:** *Auto Calculation* of valve size will only function with the generic cylinderhead choices provided in the *Cylinder Head Type* menu. If any modification is made to these heads, or other Port Flow files are loaded, The simulation assumes that the sizes of the valves are known by the user and *Auto Calculation* is turned off.

> **Note-2:** *Auto Calculation* is turned **OFF** by default when the simulation is started and whenever *Clear Components* is chosen from the **Edit** menu.

> **Note-3:** While *Auto Calculate Valve Sizes* can be helpful during quick back-to-back testing, it probably will not "guess" the precise valve sizes typically used by the engine/cylinder-head manufacturer.

Selecting Alternate Valve-Flow Area
Calculation Methods (Advanced Topic)

The *Port-Flow Dialog* in allows you to select alternate methods of calculating port flow areas and, subsequently, valve CDs. To understand how these different algorithms function and when to use them, a bit of background is helpful.

As we have discussed in this chapter, the method used to determine the restriction in the ports and the mass flow rates within an engine simulation are derived from cylinder-head flowbench test data. Many engine developers have concerns that this steady-state flow may not accurately represent the flow that occurs within the running engine, particularly in high-output, high-speed engines. The cyclical, highly-pulsed flow that occurs in the ports as charge moves toward and away from the valves may produce flow regimes across the valve openings that cannot be accurately modeled using steady-state flow measurements.

In addition, the changing shape and geometry of the openings created when the valves move off their seats may have also affect overall mass flow. And because most engines respond dramatically to changes in low-lift flow, alternate methods of calculating the "exposed areas" of partially-open valves have been developed. The flow areas, also known as the *Curtain Areas*, are the entire open area through which mass can move through each valve at every point in the lift curve. However, analysis has shown that changing flow geometries as the valve moves through low-lift into higher lift regimes can alter the _effective_ Curtain Areas. Since the Curtain Area is compared to the calculated flow within in a straight pipe of equal area (to determine the CD or Coefficient of Discharge), using precise Curtain Area values is fundamental in determining accurate flow rates within an engine simulation.

Cylinder Head Category

Alternate CD Calculation Methods

To simulate engines with unique cylinder heads or that have other unique modeling complexities, in particular, high-speed large-displacement engines, we have added two new methods of determining Curtain Areas and CD Values. These alternate algorithms may be most appropriate where high-energy pulsed flow may affect engine output. See text for more details.

To address these modeling complexities, we have added two additional mathematical algorithms in Dynomation6 that determine Curtain Areas and CD values using alternate methodologies. An explanation of each of these methods follows. Finally, this section concludes with additional tips and suggestions that can help you model unique powerplants that don't seem to fit the "mold."

Standard Dynomation Method (Default): This is the default method used in Dynomation 4, 5, and 6. It has been successful in modeling with excellent accuracy over a wide range of engines since it's introduction in 1995. We strongly suggest that you, at least, begin any engine design using this method of flow area determination. It does not emphasize low-lift flow through the valves, rather it maintains a consistent analysis of valve flow during low- and high-lift regimes (although Dynomation6 includes additional analysis of mass flow within the Wave-Action simulation and can apply a "compression" coefficient to CD values during low-lift flow to improve accuracy).

2-Step Gordon Blair Method: This algorithm was developed by the well-known engine expert Dr. Gordon P. Blair in the mid to late 1990's. It is based on two separate methods to determine flow areas depending on valve lift (the switch between methods is based primarily on valve dimensions). The overall effect

is to decrease calculated flow areas (more on this later) and increase CD values within low-lift values and, to a lessor extent, increase flow throughout the entire lift range. You'll likely see an increase in power at mid to high rpm, with some loss in torque at lower speeds on high-output engines.

3-Step Trenity Simpson Method: This algorithm was developed within a PhD thesis by Dr. Trenity Simpson at Iowa State University. It is based on a three-step method to isolate and individually analyze three flow regimes that occur during low- to mid-lift regions of valve lift. The overall effect is very similar to Blair's method, but with slightly more boost in CD values. This method often produces an increase in high-speed power and slight losses in torque at low engine speeds, although with lessor effect in low-performance engines.

<div align="center">

When to Apply
Alternate CD-Calc Methods (Advanced Topic)

</div>

Since it quite difficult to predict when alternate CD calculations would improve simulation accuracy. To help you make this decision, we have added an mathematical test to Dynomation6 Full Wave-Action simulation that helps determine when changes to CD values may help improve simulation accuracy (not included in DynoSim6, since direct changes to port dimensions is not possible). When the Wave-Action simulation determines that the values you have selected as the best minimum port areas for the intake or exhaust runners are somewhat larger than simulation-determined "optimum" values, it will place a message in the Simulation Log (which you can see by selecting **Show Simulation Log** in the **SIMULATION** menu at the top of the main program screen). This message (see photo) will recommend testing one of the alternate CD-calculation methods. You can check to see if this was an appropriate change by comparing the calculated minimum areas shown in the log to those you have entered in the simulation. If the calculated values are a closer match than they were with the original CD calculations, then the Alternate Method will likely improve simulation accuracy.

Note: If you are using DynoSim6 (or the FE model in Dynomation6), the simulation model cannot determine if alternate CD calculation methods would be helpful. This is not an arbitrary limitation, but is a basic restriction of the faster but somewhat pressure-wave limited *Filling & Emptying* (FE) model.

Before you experiment with Alternate CD Calculation methods (particularly if you have real dyno data to use as a baseline), you may wish to try increasing or decreasing the flowbench pressure-drop values entered in the *Port-Flow Dialog*. Here you can make subtle and/or dramatic adjustments to the slope of the power curves. Considering the uniqueness of each engine design and the way pressure waves and mass flow moves in the engine, if you are able to adjust how static flowbench data is used to better match true dyno output, you are, in effect, refin-

Cylinder Head Category

The Wave-Action simulation used in Dynomation6 checks the values entered for minimum port areas against simulation-determined "optimum" values. When appropriate, a message in the *Simulation Log* will recommend testing one of the *Alternate-CD Calculation* methods provided in the Port-Flow Dialog.

Simulation Run-Log Viewer

```
>>>  ProStockModel2.dxml  <<<

Dynomation6 Full Wave-Action Model Run ---------------------------------

Exhaust Flow Coeff > 0.999; Check port flow data, valve sizes, etc. Index 1, Lift:
Exhaust Flow Coeff > 0.999; Check port flow data, valve sizes, etc. Index 2, Lift:

Estimate Optimum Intake Area   --------------------------
  Intake Port Minimum Area:     2.80 Sqin,    71.0 Sqmm
Estimate Optimum Exhaust Area --------------------------
  Exhaust Port Minimum Area:    2.75 Sqin,    69.8 Sqmm

Simulation Runner-Area Recommendation   -----------------------1
  A Minimum Intake and/or minimum Exhaust Area entered in Dynomation6
  is somewhat larger than the calculated optimum shown above. If the
  area that you entered is known to be optimum, simulation accuracy
  MAY BE IMPROVED by switching to one of the Non-Default-CD Calculation
  methods in the Port-Flow-Dialog drop-down menu (press the Port Flow button)
  If this does not increase power, switch back to Standard CD Calc mode.

Number of integration cycles run for convergence: 4
Simulation elapsed time:  1:744:334 ms
```

Minimize   Close

ing the simulation model to better match actual test results. The more accurately the simulation is able to model your engine baseline, the more useful it will be in predicting the outcome of other engine changes, such as valve timing, exhaust tuning, etc.

Final Note: The alternate valve curtain-area algorithms discussed in this section effectively INCREASE the CDs of intake and exhaust valves. This may seem counter intuitive, since Blair's and Trenity's methods actually calculate SMALLER effective flow areas at low- to mid-lift heights. Since the measured airflow rate remains the same (flowbench values aren't changed), the same flow through a smaller calculated orifice size increases the CDs (which reflect improved flow efficiency). This results in the flow through the calculated (smaller) curtain areas increasing closer to the flow measured in a straight pipe of the same size (in other words the ports become less restrictive). The bottom line is greater mass flow into and out of the engine.

Induction Category

Induction

Intake Design: Baseline Plenum Model

FE Runner Taper Angle: 1.50

Runner Connections: Common Plenum   Ave. Runner Temp: 180.00 deg

Total Airflow Rate At Throttle: 750.00 CFM   @ 1.500 inHg   Flow Calc

Wave-Action Intake Runner/Plenum:

Primary Runner Specifications		Manifold Plenum Volume	
Length (Min Area to Entry):	9.500 in	Plenum Volume:	160.0 ci
Min Port Area (per Valve):	1.70 sqin	Calc From Length:	6.501 in
Port Entry Area (Max Area):	2.50 sqin	Calc From Width:	4.961 in
Average Taper Angle:	1.9 Deg	Calc From Depth:	4.961 in

☐ Runners W/Bellmouth Inside Plenum   Induction Waves Stabiltiy: ▣

Forced Induction Specifications:

Compressor: Turbo-Garrett-GTX4088R-88-54

Flow Restriction (throttle) Location: ◉ Before Compressor   ◉ After Compressor

Turbine Size:	3.031 in	Internal Ratio:	1.00	Surge	Choke	Overspeed
Turbine A/R:	0.72	Belt Ratio:	1.00	▣	▣	▣
Turbine Eff.:	75.0 %	Number Comp:	1			
Boost Limit:	10.0 psi	IC Press. Drop: None		Intercooler Eff: None		

The Induction system components are divided into four main groups: 1) *Intake Design,* **2)** *The Induction Airflow Rate* **and** *Pressure Drop***, 3)** *Wave-Action Specific* **intake runner specifications, including** *Runner Length***,** *Entry Area***, and** *Minimum Port Area***,** *Plenum Volume***, and 4)** *Forced Induction* **component selections (covered in the next section)***.*

THE INDUCTION CATEGORY

The **Induction** component category establishes the characteristics of the intake flow path from atmosphere to the valve seats. This includes everything upstream of the valves, including the port runners, intake manifold, plenums (if applicable), carburetor or fuel-injection, venturis (if used), any supercharger or turbocharger, and openings to the atmosphere. Induction menus are divided into four main groups: 1) **Intake Design,** 2) **Induction Airflow** and **Pressure Drop**, 3) *Wave-Action specific* intake runner specifications, including **Runner Length**, **Minimum Port Area**, **Area At Entry**, **Plenum Volume**, and **Runner Bellmouth**, finally 4) **Forced Induction** specifications are located at the bottom of the *Induction Category*.

The *Intake Design* Menu

The **Intake Design** menu consists of 32 intake manifold/system choices plus two "Baseline Models," a **Baseline Plenum** (Common Plenum) and a **Baseline IR** (Individual Runner) configuration; baseline choices are often used for initial engine development, since simulation results are not modified by specific intake manifold characteristics. All of these selections apply a unique tuning model to the induction system. While these manifolds are only a small sample of the intake manifolds available for IC engines, this list of 34 models simulate most of the manifolds and

Induction Category

The "Baseline" Induction choices remove any specific manifold modeling characteristics from simulation results. Engine developers use these choices (one for Common-Plenum manifolds and one for IR induction) when beginning initial engine development.

"Baseline" Induction Models
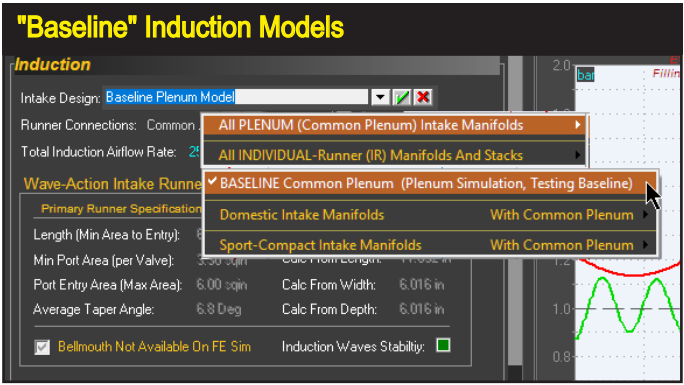
induction systems for both street and racing applications.

The manifold models are divided into four main groups: 1) **"Baseline" Intake Models (2 choices)**, 2) **Plenum—Domestic Intake Manifolds (14 choices)**, 3) **Plenum—Sport-Compact Manifolds(14 choices)**, and 4) **Individual-Runner Inductions(5 choices)**. These categories are only general classifications. In some cases, certain "domestic" engines (utilizing modern tuned-runner induction) are best modeled with selections from the Sport-Compact submenu. Finally, if you are interested in a manifold that falls "in between" two menu selections, you can use the *trend* method to estimate power for a hybrid design. For example, run a test simulation using manifold Type A, then set up a **Quick Comparison™** (right click on the rpm graph) and view the differences in power attributed to manifold Type B. The power differences will give you insight into how a hybrid manifold Type A/B *should* perform.

The Domestic Plenum Manifold menu consists of 13 intake manifold choices. Each of these selections applies a unique tuning model to the induction system. While these manifolds are only a small sample of the intake manifolds available for typical IC engines, these simulation models reproduce the characteristics of many manifolds and induction systems used today for both street and professional/amateur racing applications.

Domestic Manifold Selections
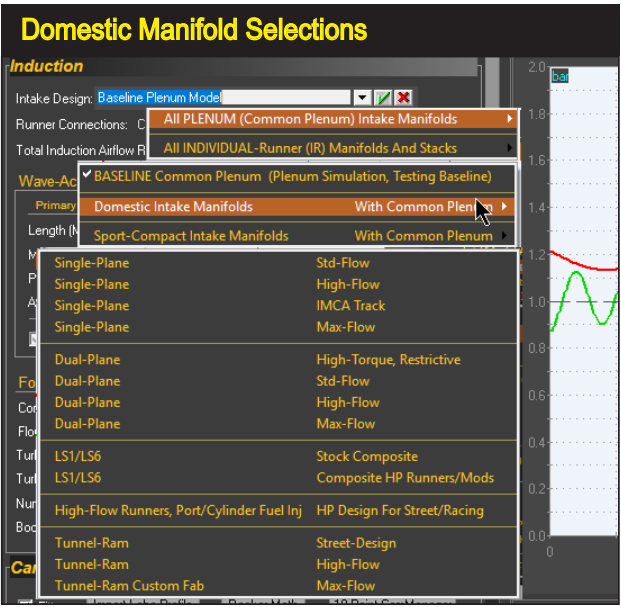
Induction Category

Baseline
Manifold Models

The *Baseline Induction* models provided in the **Intake Design** menu apply either no manifold algorithm or use the simplest model available for both the *Filling-And-Emptying* and *Wave-Action* simulations. These baseline models leave simulation results as "untouched" as possible by manifold influences, and this may improve the accuracy of "back-to-back" cam-timing and exhaust-system tuning efforts. In addition, consider using the *Baseline Induction* models if you are unsure of which specific manifold model to apply. The *Baseline Common Plenum* or *Baseline Individual-Runner* choices provide an excellent starting point on which to build further engine testing and development.

Domestic
Common-Plenum Manifolds

The following sections provide an overview of each *Domestic Common-Plenum* manifold models provided in the **Intake Design** menu. Here you'll find a brief description of the assumptions used in each model, and recommendations associated with that individual design.

Domestic—Dual-Plane, (Four versions: Maximum-Torque, Standard-Flow, High-Flow, and Max-Flow Manifolds)

The **Dual-Plane, Max-Torque Manifold** models dual-plane manifolds with smaller runners, such as those designed for heavy vehicles, economy and other high-torque, applications. The *Dual-Plane Max-Torque* model will tune for a lower peak engine speed, due mainly to runner and plenum restrictions. The **Standard-Flow, Dual-Plane Manifold** selection represents the majority of street-oriented dual-plane manifolds available to performance enthusiasts (including many OEM manifolds). The *Standard-Flow* model will accurately simulate street and performance engines with "as-cast" dual-plane manifolds. The **Dual-Plane, High-Flow** and **Max-Flow Manifold** selections model modified, large-port, (and even custom-fabricated) dual-plane manifolds, as used on high-performance and all-out racing engines.

Dual-Plane Manifold Theory

Remarkably, the well-known and apparently straightforward design of the dual-plane manifold is, arguably, one of the more functionally-complex manifolds to model. An intake manifold is considered to have an effective dual-plane configuration when 1) the intake runners can be divided into two groups, so that 2) each group alternately receives induction pulses, and 3) the pulses are spaced at even intervals. If all of these criteria are met, the manifold is said to have a 2nd degree

Induction Category

The Edelbrock Performer Q-Jet represents a typical dual-plane manifold design (of "Standard-Flow" capability). This manifold is said to have a 2nd degree of freedom. A powerful resonance multiplies the force of the pressures waves, simulating the effects of long runners, boosting low- and mid-range power.
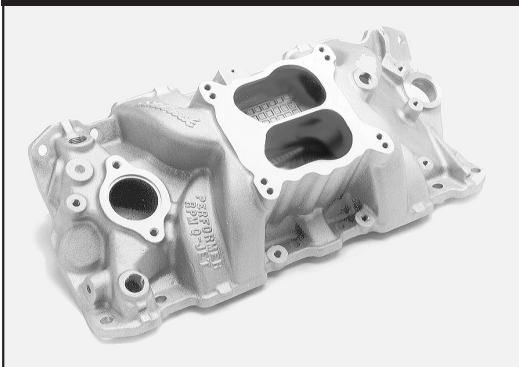
of freedom, allowing it to reach a unique resonance producing oscillations within the entire manifold. During this resonance, pressure readings taken throughout the manifold will be in "sync" with one another. Full-manifold resonance multiplies the force of the pressures waves, simulating the effects of long runners. Since longer runners tune at lower engine speeds, the dual-plane manifold is most known for its ability to boost lower-to-midrange rpm performance.

The divided plenum is another common feature of dual-plane manifolds that boosts low-end power with carbureted engines. Since each side of the plenum is connected to only one-half of the cylinders (4-cylinders in a V8), each cylinder is "exposed" to only one-half of the carburetor. This maximizes wave strength and improves low-speed fuel metering (these effects are less pronounced with throttle-body fuel-injection systems). However, the restriction inherent in a divided plenum can reduce peak power at higher speeds.

The main benefits of the dual-plane design are its low-speed torque-boosting capability, compact design, wide availability, and ease of use with both carburetors and injection systems. However, not all engines are capable of utilizing a dual-plane configuration. Typically, engines that do not have an even firing order or have too

Many dual-plane manifolds are hybrids. This Edelbrock dual-plane manifold is designed for the 440 Chrysler engine and has a partially open plenum. In manifold such as this, the opening adds mid-range and high-speed performance with, typically, a slight sacrifice in low-speed torque. Not all hybrid designs are as successful as this one. In situations where you are not familiar with specific engine or manifold characteristics, it may be worthwhile to stick with "plane-vanilla" designs.

many cylinders to generate a resonance effect will not benefit from a dual-plane manifold. While there are exceptions, engines having 2 or 4 cylinders work best with this manifold. Since most V8 engines are basically two 4-cylinder engines on a common crankshaft, even-firing V8s benefit from the resonance effects of the dual-plane manifold. The simulation does not prevent the selection of a dual-plane manifold on engines that will not develop a full resonance effect. For example, you can install a dual-plane manifold on a 5-cylinder engine, but the results—a low-end power boost—may not reproducible in the real world, since an effective dual-plane manifold cannot be built for this engine. The *Filling-And-Emptying* simulation is best utilized by modeling dual-plane manifolds combinations that already exist rather than testing theoretical fabrications.

Many dual-plane manifolds are hybrids, incorporating facets of other manifold designs. Especially common is the use of an undivided or open plenum typically associated with single-plane manifolds. These "mixed" designs are attempts at harnessing the best features of both manifolds while eliminating the drawbacks of each. Sometimes the combinations are successful, adding more performance without much sacrifice in low-speed driveability. With these designs, you can successfully use the "trend" method described earlier to estimate engine torque and power. Unfortunately, there is no shortage of manifolds that can reduce power without giving anything back in driveability or fuel economy. In fact, some of the worst designs

Dual-Plane vs. Single-Plane Design
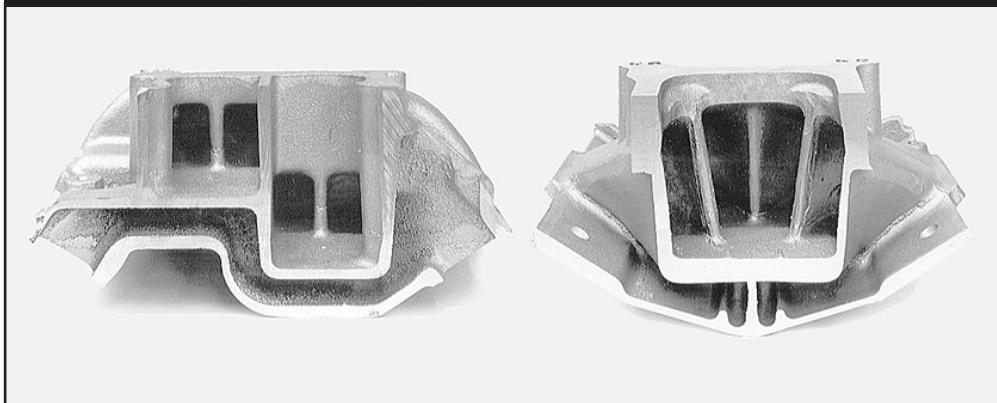
The basic differences between single- and dual-plane manifolds are clearly illustrated here. The dual-plane (left) divides the plenum in half, with the runners grouped by firing order. Each cylinder "sees" only one-half of the carburetor, transferring a strong signal to the venturis. This manifold design is said to have a 2nd degree of freedom, allowing it to reach a unique resonance that makes its short runners act as if they were longer and boosts low-speed power. The single-plane manifold (right) has short, nearly equal-length runners with a large open plenum, much like a tunnel ram laid flat across the top of the engine. The manifold has excellent high-speed performance, but its design prevents full-manifold resonance. That reduces low-speed torque, which can impair driveability and fuel economy.

are remarkably bad. It is impossible to determine which of these combo designs is better than others using Dynomation6 (a future multi-junction version of Dynomation, that models every segment of all intake passages, including the complex effects of multi-cylinder interference, will be able to perform this analysis). Unless you can perform actual dyno testing on these manifolds to determine what works and what doesn't, it may be worthwhile to stick with more "plain-vanilla" designs that produce predictable results.

Domestic—Single-Plane (Three Versions: Standard-Flow, High-Flow, and Max-Flow Manifolds)

The *Single-Plane, Standard-Flow Manifold* selection represents the majority of single-plane manifolds sold to performance enthusiasts. The *Standard-Flow* selection accurately simulates street and performance engines with "as-cast," single-plane manifolds.

The *Single-Plane IMCA Track* manifold is a high flow design with specific modifications to help boost torque in midrange engine speeds. This manifold was designed for IMCA dirt track racing but can be useful for any application needing near-max power with a improved midrange (available from Weiand, as model 7547).

The *Single-Plane High-Flow* and *Max-Flow* manifolds simulate modified, large-port, air-gap, and/or custom-fabricated single-plane manifolds, as used on high-performance and racing engines.

Single-Plane Manifold Theory

In a very real sense, a single-plane manifold is simply a low-profile tunnel ram. The tunnel-ram manifold (discussed next) is a short-runner system combined with a large common plenum; a design that optimizes power on all-out racing engines

A single-plane manifold is simply a low-profile tunnel ram. The design combines short, nearly equal-length runners with an open plenum, but "lays" the entire configuration flat across the top of the engine. The single-plane manifold combines improved flow capacity, higher charge density, and short runners to build substantial horsepower at higher engine speeds.

Single-Plane Manifold
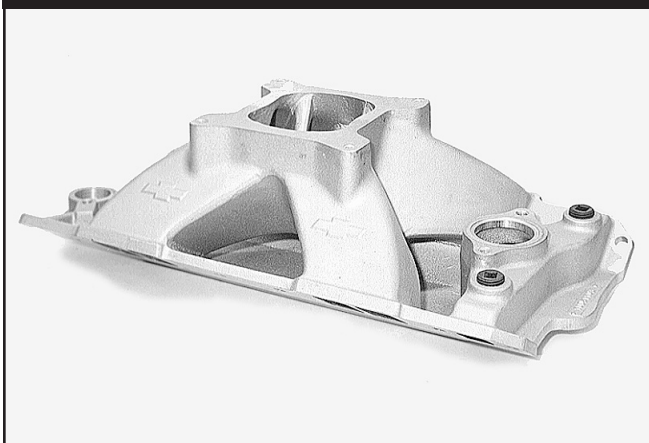
Induction Category
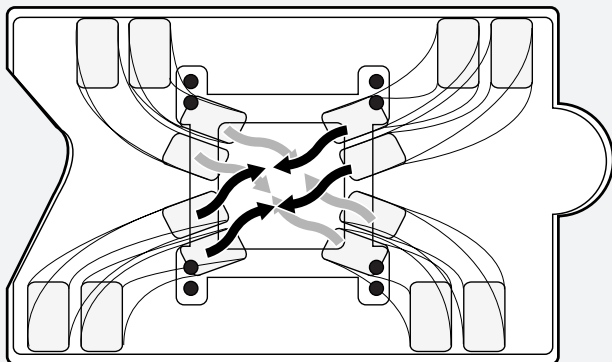
The typically compact, low-profile design of the single-plane manifold has some drawbacks. The runners are connected to a common plenum. This arrangement tends to create unpredict-able interference effects as pressure pulses moving through the runners meet in the plenum and stir up a complex brew, sometimes creating irregular fuel-distribution.

where engine-compartment clearance is not an issue. The single-plane manifold combines short, nearly equal-length runners with an open plenum, but "lays" the entire configuration flat across the top of the engine.

The single-plane runner design prevents full-manifold resonance (found in dual-plane manifolds). This reduces low-speed torque, and depending on the size of the plenum and runners, single-plane manifolds can also reduce driveability and fuel economy. Furthermore, large-volume, undivided plenums often contribute to low-speed performance problems by presenting every cylinder to all barrels of the carburetor, lowering venturi signal and low-speed fuel metering accuracy (again, this drawback is minimized on fuel-injection systems). On the other hand, the single-plane manifold (like the tunnel ram) combines improved flow capacity, potentially higher charge density, and short runner lengths to build substantially more horsepower at higher engine speeds.

As a high-performance, high-speed manifold, the single-plane design has many advantages, however, it's compact, low-profile design also has drawbacks. The runners are connected to a common plenum like spokes to the hub of a wheel. This arrangement tends to create unpredictable interference effects as pressure pulses moving through the runners meet in the plenum (or travel down the opposite runner) and stir up a complex brew. Large plenum volumes help cancel some these negative effects, but open-plenum, single-plane manifolds may produce unexpected anomalies in fuel distribution and pressure-wave tuning with specific camshafts, headers, or cylinder heads (to some degree, these effects are present in all manifold designs). Locating these will-o'-the-wisp anomalies requires dyno testing and the use of temperature and pressure probes and careful measurement fuel distribution accuracy throughout the rpm range.

Designers and engine testers have experimented with hybrid single-plane manifold designs that incorporate various dual-plane features. One common modification is dividing the plenum of a single-plane manifold into a pseudo dual-plane configura-

Induction Category

tion. While this does increase signal strength at the carburetor, uneven firing pulses presented at each side of the plenum do not allow 2nd degree of freedom resonance. This modification can cause sporadic resonances to occur throughout the rpm range with unpredictable results. Spacers between the carburetor and plenum are also commonly used with single-plane manifolds often with positive results, particularly in racing applications. Spacers typically increase power for two reasons: 1) By increasing plenum volume they tend to reduce unwanted pressure-wave interactions, and 2) a larger plenum improves airflow by reducing the angle at which air/fuel must negotiate a transition from "down" flow through the carburetor to "side" flow into the ports. While there is no way to use trend testing to evaluate the effects of a divided plenum, spacers can be simulated. The increase in plenum volume tends to transform the single-plane manifold into a "mini" tunnel ram, so horsepower gains tend to mimic those obtained by switching to a tunnel ram design (i.e., performance improvements, when found, usually occur at high rpm).

Single-Plane Rule Of Thumb: Since a single-plane manifold typically reduces low-speed torque and improves high-speed horsepower, it is often the best compact manifold design for applications where wide-open-throttle engine speed exceed 4000rpm. If the engine commonly runs through lower speeds, a dual-plane, individual runner, or similar high-torque system will usually provide better performance, driveability, and fuel economy.

Domestic—Tunnel-Ram (Three Versions: Standard, High-Flow, and Max-Flow Designs)

The *Standard Tunnel-Ram Manifold* selection represents a relatively small runner, tunnel-ram manifold with applications in high-performance street (roadster) and mild racing. This selection simulates performance applications with "as-cast," tunnel-rams with dual- or single-carburetors or throttle bodies.

The *High- And Max-Flow Custom Tunnel-Ram Manifold* models simulate extensively-modified, large-port, and/or custom-fabricated tunnel-ram manifolds, as seen on ProStock and other "exotic" racing engines. The advantages of

Tunnel-Ram Manifold

This Weiand/Holley BB Chevy tunnel ram manifold is a single-plane induction system designed to produce optimum power on all-out racing engines. It has a large common plenum and short, straight, large-volume runners. The tunnel ram manifold menu selections have the potential to produce the highest peak horsepower of all the naturally-aspirated manifolds listed in the *Induction* menu.

Induction Category

This custom-fabricated tunnel-ram manifold (by Jon Kaase Racing Engines) has the potential to produce the highest peak horsepower of all plenum-based, naturally-aspirated induction systems. Combined with the proper cam timing, compression, and other components, the large cross-sectional areas, straight runners, and short tuned lengths make this custom manifold a "no compromise" high-rpm racing design.
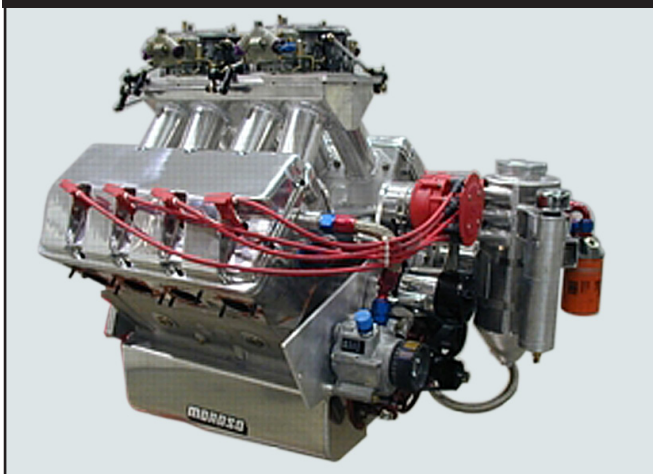
these tunnel rams derive from their combination of a large common plenum and short, straight, large-volume runners. The large plenum can accommodate one or two carburetors, potentially flowing up to 2200cfm or more. The large plenum also minimizes pressure-wave interaction and fuel distribution issues (especially with dual carburetors or fuel-injection throttle bodies). The short runners can be kept cooler than their lay-flat, single- and dual-plane counter-parts, and they offer a straight path into the ports, optimizing ram-tuning and minimizing flow restriction.

Tunnel-ram applications are limited because of their large physical size; vehicles using tunnel-ram manifolds usually require a hole in the hood and/or a hood scoop to provide manifold and carburetor clearance. While a protruding induction system may be a "sexy" addition to a street rod, in single-carburetor configurations, the tunnel ram offers very little potential power over a well-designed, single-plane manifold. Only at very high engine speeds, with multiple carburetors, will the advantages in the tunnel ram contribute substantially to power.

The **Standard** and **Max-Flow Tunnel Ram** manifold selections have the potential to produce the highest peak horsepower of all naturally-aspirated induction systems listed in the **Manifold Type** menu. The large cross-sectional areas, straight runners, and short tuned lengths make this manifold a "no compromise" racing design.

Domestic—LS1/LS6 Stock Composite and LS1/LS6 HP Runner/Mods Manifolds

Since the mid 1980's, engine simulation programs have been regularly used by engine designers and manufacturers to optimize wave dynamics developed inside the intake passages of the IC engine. These pressure waves can have a marked effect on engine performance. When unharnessed, they can pro-

Induction Category

Composite intake manifolds have become standard equipment on many modern IC engines. They allow designs that could not be easily replicated in aluminum, and some offer unique performance technologies. The stock GM LS1/LS6/Z06 induction, shown on the right, and the *FAST* version illustrates the changes in runner length and volumes that can improve power in HP applications.
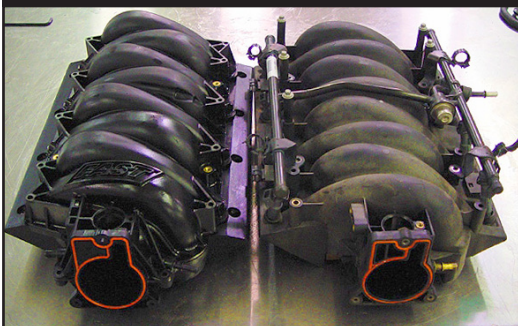
duce non-uniform fuel distribution, prevent cylinder filling, and adversely affect driveability. On the other hand, when an induction system has been carefully designed to harness pressure-wave dynamics, the engine can benefit from improved airflow and cylinder filling at the desired engine speeds. Using this technology, manifolds can produce "designer" power and torque curves to optimize overall engine performance within specific rpm ranges and within fuel economy and emissions requirements.

The fully composite manifold developed by GM for their "new" smallblock engine is a good example of this design. It is biased toward producing power at higher rpms, while maintaining good torque throughout the rpm range. Installed in many performance vehicles, like the Z06, 405hp Corvette, the new composite design allows high power while maintaining good driveability and low emissions. This manifold can be modeled in the simulation by selecting the *LS1/LS6 Composite Stock* from the Domestic selections provided in the **Manifold Type** menu. Despite the fact that this manifold packs its runners in a small package designed to fit under the low-profile hoods of modern vehicles, generous flow capacities produces good peak power.

Aftermarket companies have taken this concept a bit further to provide

Edelbrock Port Injection Manifold

The final selection in the *Domestic Plenum* manifold menu is a Direct Port Injection manifold with high-flow runners. The Edelbrock Pro-Flo XT is a good example of this design. Because of its tunnel-ram-like features, this manifold offers excellent performance for street and racing applications, yet fits under the hood of most street iron.

more performance for the automotive enthusiast. High-performance versions of this manifold have been shown to add from 3% to 12% more power with very little sacrifice in bottom-end torque. You can test a performance design with the *LS1/LS6 HP Runner/Mod* induction selection. This induction model was based on the *FAST™* manifold for the same engine.

Domestic—Direct Port Injection W/HP Runners & Plenum

This manifold category models modern designs that use direct port injection and incorporate free-flowing runners. This manifold is typically construction like a low-profile tunnel ram, designed for under-the-hood installations. The Edelbrock Pro-Flo XT pictured on the previous page is an example of this type of manifold.

Sport-Compact
Common-Plenum Manifolds

The following sections provide an overview of each of the 14 *Sport-Compact* manifold models in the **INDUCTION** category. Here you'll find a brief description of the assumptions used in each model, and recommendations associated with that individual design.

Sport-Compact—Non-Tuned Runner Manifolds (Three Versions: Stock, Ported, Race Ported)

The *Non-Tuned, Small Runner, Stock* selection models an intake manifold with

The Sport-Compact manifold submenu consists of 14 choices. Each selection applies a unique tuning model to the induction system. While this is far from a complete list of intake manifolds available for Sport-Compact engines, these generic models can simulate many of the manifolds and induction systems available for racing and street applications.
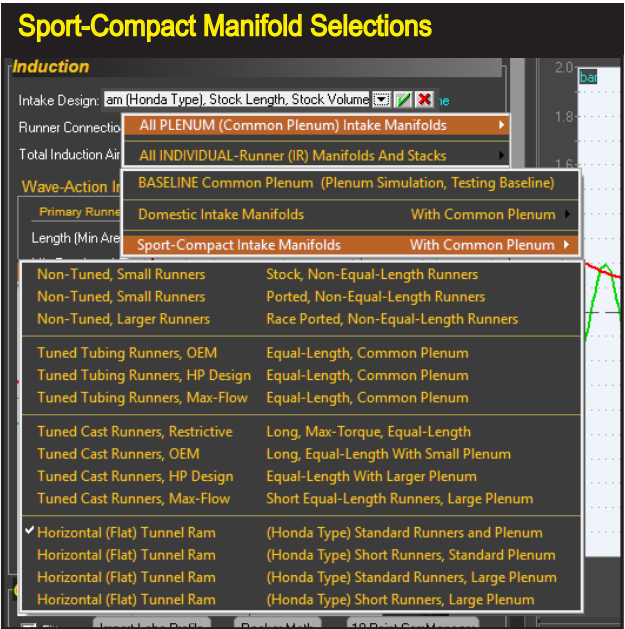
Induction Category

The *Non-Tuned Manifold* selections model intake manifolds with small-diameter runners that connect to a central plenum with little consideration for equal length or tuning characteristics.

small-diameter runners that connect to a central plenum with little consideration for equal length or tuning characteristics. This restrictive manifold was designed for low-performance economy vehicles or other "basic" engine configurations. In naturally-aspirated applications, this manifold produces the least power and offers the least boost in torque since there is very little runner tuning effects. The ***Non-Tuned, Small Runner, Ported Manifold*** selection provides a slight improvement in airflow that improves power at higher engine speeds. But the same non-tuned runners provide little pressure-wave tuning and low-speed torque boost.

The ***Non-Tuned, Larger Runner, Race Ported*** model represents an attempt to improve this simple manifold design as much as possible. By porting the runners, modifying the plenum, and even modifying and welding the runner passages, this is about as much as you can expect to improve this non-tuned intake manifold design. Even with this extensive work, however, this design simply cannot match the torque potential of a stock, OEM, tuned induction system. Primarily, this induction model has been included for comparison purposes, since such basic manifolds are rarely modified to this degree.

Porting the runners, modifying the plenum, and even modifying and welding the runner passages can improve performance on non-tuned manifolds. However, despite extensive modifications to non-tuned manifolds, a more modern OEM-type induction systems will generally produce more torque and horsepower.

Short, Non-Tuned Runners

Induction Category

Sport-Compact—Long Tubing Runners, Common-Plenum Manifolds (Three Versions: Basic, HP, & MaxFlow)

The *Long-Tubing Runners, Common-Plenum, Basic OEM Manifold* models the induction system typically used on a flat-four and flat-six engines, like the original VW "Beetle" and lower-performance Porsches. The runner lengths and their relatively small diameters tune for good lower-speed torque, but offer restriction at higher engine speeds. These manifolds are particularly restrictive when used on larger displacement engines (such as those with larger cylinder liners or stroker cranks).

The *Long-Tubing Runners, Common-Plenum, HP Manifold* selection introduces a considerably different *Filling-And-Emptying* induction model. The larger diameter runners offer much less restriction at high engine speeds, but the increased volume dampens low-rpm tuning, often reducing bottom-end torque on all but large-displacement engines. This model closely simulates many naturally-aspirated Porsche 911 induction systems.

Long-Tubing Runners, Common-Plenum, Max-Flow Manifold represents manifold designs used on very high-performance engines or in racing applications. The large runner cross-sectional area ensures cylinder filling at high engine speeds, however, the increased runner and plenum volume further reduces torque at low rpm.

Sport-Compact—Tuned Runner (Four Versions: Restrictive, OEM, High Perf., Max Flow)

All of the manifolds in this **Manifold Type** group model OEM (usually cast aluminum) induction systems. The longest runners produce copious torque at low to medium speeds, while the shortest runner manifolds are commonly used on high-performance, higher-rpm engines.

The *Equal-Length, Cast Long Runner, Max-Torque, Small Plenum* is

Long Tubing Runners With Common Plenum

The *Long-Tubing Basic OEM* manifold selection models induction systems often used on a flat-four and flat-six engines, like the VW and early opposed-six Porsche powerplants. The length of the runners and their relatively small diameters optimize torque, but restrict airflow at higher engine speeds.

Induction Category

This Porsche 993 uses a cast-aluminum equivalent of the steel-tubing manifolds of the past. While the length of these runners have positioned this induction into the *Tubing, Long-Runner* manifold category, *HP* or *Max-Flow* selections can be used with this free-flowing design.

the first selection in this group, and it models a manifold ideal for engines in heaver vehicles, like trucks and vans. The low-rpm tuning boosts efficiency, economy, and driveability in stock engines. However, while the runners are somewhat long, they are not overly restrictive and stock engines using this manifold will produce good power.

The next selections (downwards in the menu within this 3rd group) is the *OEM, High-Perf.,* and *Max Flow* versions of equal-runner length runner manifolds.

The *Equal-Length, Cast Long Runner, OEM, Small Plenum* version has slightly shorter runners are capable of producing more horsepower, but torque below 3500rpm can suffer somewhat. This is still excellent manifolds for heavier performance vehicles.

The *Equal-Length, Cast Medium Length Runner, HP Design, Medium Size Plenum* manifold is the first manifold in this group to offer a bias toward performance and higher engine speed. Manifolds of this type are commonly

The *Long-Tubing Runners, Common-Plenum, Max-Flow* selection models manifold designs used on high-performance engines or in racing applications, like this Porsche 928 V8. The large runner cross-sectional areas ensure cylinder filling at high engine speeds, however, the additional runner and plenum volume can reduce torque at low rpm, offset somewhat, in this case, by the length of the runners.

"Tubing," Max-Flow Manifold

Induction Category

The *Tuned-Runner Manifold* group (3rd group in Sport Compact menu) models induction systems that have slightly smaller plenums and runners than the more aggressive manifolds in the 4th group of *Flat* or *Horizontal Tunnel Ram* designs. This 3.0L V6 Honda is typical of engines that can be modeled by *Tuned-Runner Manifold* selections. The torque curves are broad and flat, while peak power occurs about 500 to 1000rpm lower than manifolds with shorter runners and larger cross-sectional areas.

Tuned-Runner Induction Systems
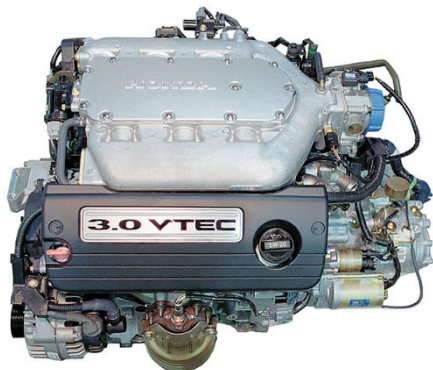
found on 4, V6, V8, and V12 engines in performance sedans and sports cars.

The most performance oriented manifold, the **Tuned, Short-Runner, Max-Flow, Large Plenum** is an excellent choice on lightweight, performance vehicles. The tuned runners offer good pressure-wave tuning, while low restriction and large plenum volume give excellent horsepower potential. This manifold design is used on many performance-oriented sports cars, like Aston Martin, Maseratti, and Ferrari. The only manifolds that are superior in performance to this selection are the "**Honda Type**" listed in the next group that have the largest runner and plenum volumes. However, even "Honda" manifolds may not provide as broad a range of torque and power as the manifolds in this group.

Sport-Compact—Honda Type (Four Versions: Standard And Short Runner,

The *Tuned Runner, Medium Length, Larger Plenum* manifold provides a bias toward performance and higher engine speed. Manifolds of this type are commonly found on 4, V6, V8, and V12 engines in performance sedans and sports cars.

Medium-Length, Tuned-Runner Induction

Induction Category

Small And Large Plenum Designs)

While the manifolds in this group are designed to model the Honda induction systems used on their 4-cylinder, high-performance engines, like the B16, B18, S2000 and others. This manifold model can be applied to any engine that uses straight, high-volume runners and a large-volume plenum. The power potential from this manifold model is similar to "all-out" induction systems, such as tunnel-ram manifolds. In fact, we refer to this manifold as a *"Flat Tunnel Ram,"* since it is basically a tunnel ram turned 90-degrees from vertical to horizontal.

The *Flat Tunnel Ram, Standard-Length Runner, Factory Plenum Volume* selection models the stock Honda manifold supplied on many of its B16A high-performance engines, like the Del Sol, Civic Si, the Integra GSR, and others. This induction system generates a characteristic broad and flat torque curve (with 4-valve heads), and its large runners and plenum volume will supply all but the largest engines with adequate airflow to well beyond 7500rpm. For many high-performance street vehicles, this is an excellent manifold with the only weak point being somewhat lower torque, usually below 3500- to 4000rpm, compared to the longer-runner manifolds in the previous group. But above these engine speeds, the *OEM Honda-Type* induction system with medium-length runners is virtually unbeatable.

The next "stage" in *Flat-Tunnel Ram* manifold designs is the shorter-runner

Large Plenum, Long And Short Runners (Honda-Type, Flat Tunnel Ram)

The Honda-Type manifold selections (referred to as *Flat Tunnel Rams*) in the simulation model Honda induction systems used on 4-cylinder, high-performance engines, like the B16, B18, S2000 and others. This manifold model can be applied to any engine that uses direct, high-volume runners and a large plenum. The stock-length runner designs (like the Edelbrock Performer X, shown on the left) generate a characteristic broad and flat torque curve, and its large runners and plenum volume will supply all but the largest engines with adequate airflow to well beyond 7500rpm. Manifolds with shorter runners will often lower torque below 4000- 5000-rpm and offer slight-to-significant power gains above 7000- to 8000rpm.

version of the stock B16A intake, the *Flat Tunnel Ram, Short Runner, Factory Plenum Volume*. Manifolds of this type are supplied on the Type R (originally on the 1.8L, B18C5 Honda engines) and are available in many forms from aftermarket manufacturers. They will often generate a slight reduction in torque below 4000- to 5000-rpm and offer a slight-to-significant gain in power above 7000- to 8000rpm. This is not necessarily the best manifold for a small-displacement street engine, but for a modified or "stroker" engine it can offer more power with little or no loss in torque.\

The third selection (working towards the bottom of the menu within this group), is the *Flat Tunnel Ram, Standard-Length Runner, Large Plenum Volume* manifold model. This choice simulates a manifold (only available from the aftermarket, such as the *Edelbrock Performer X* manifold) that has increased cross-sectional area runners of about factory length combined with a large-volume plenum. This manifold will show power gains above max-torque engine speeds, while the standard-length runners will usually maintain good torque at lower speeds. This may be the best manifold for a modified engine that will be operated on the street.

Finally, the *Flat Tunnel Ram, Short Runner, Large Plenum Volume* manifold model simulates many of the exotic induction systems used on racing all-motor and forced induction applications. This induction has the largest volume runners and plenum, and will reduce low-speed torque below about 5000rpm. However expect substantial gains on highly-modified engines, especially above 8000rpm.

Custom Racing Manifolds
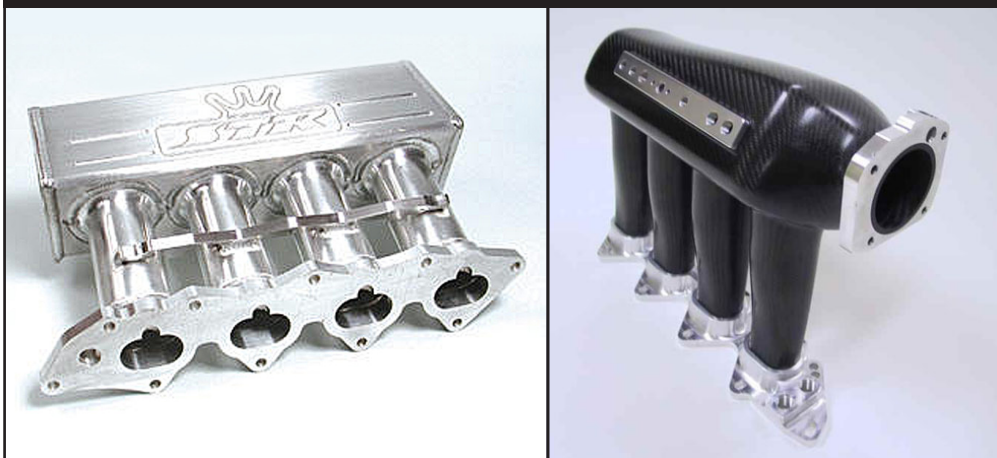
The *Honda-Type, Standard Or Short Runner, Large Volume* manifolds simulate inductions that have increased cross-sectional area runners combined with large plenum volumes. These manifolds will usually reduce low-speed torque below 5000rpm. However expect substantial gains on modified, high-speed engines, especially above 8000rpm. Notice the tapered runners on the manifold on the right.

Induction Category
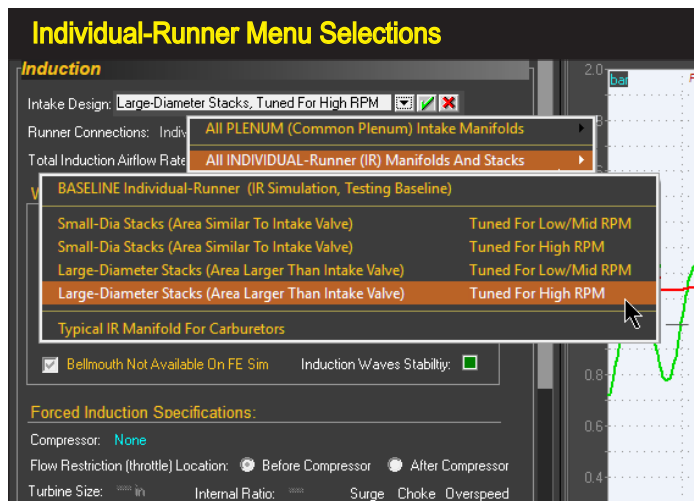
Individual- (or isolated) runner, I.R., induction systems, use separate tubes for each runner with their own individual throttle plates. These induction systems offer the potential for optimum power at high engine speeds.

Individual-Runner Induction Choices

Individual Runner, No Plenum (Large And Small Diameter, Long And Short Stacks)

For naturally-aspirated, professional racing applications, individual- (or isolated) runner (I.R.) induction systems, with separate tubes containing their own throttle plates for each cylinder, can offer the ultimate in power potential at high engine speeds. The single element that sets the I.R. system apart from any other induction models is that each "barrel" or individual "stack" *does not share flow with other stacks through interconnecting passages* (like a plenum). This characteristic means that the overall induction flow is divided between all barrels (or cylinders).

All of the manifolds in the **Manifold Type** submenus that we have discussed up to this point have shared-flow between cylinders, typically through a plenum.

When you select any manifold model from the *Intake Manifold Design* menu, the appropriate plenum model is displayed in the *Plenum Configuration* field. For example, if you select a *Single-Plane* manifold, the *Common Plenum* model is used to analyze total induction airflow. On the other hand, if you select an I.R. system, an *IR Runner* model is used to divide total induction airflow between all cylinders.
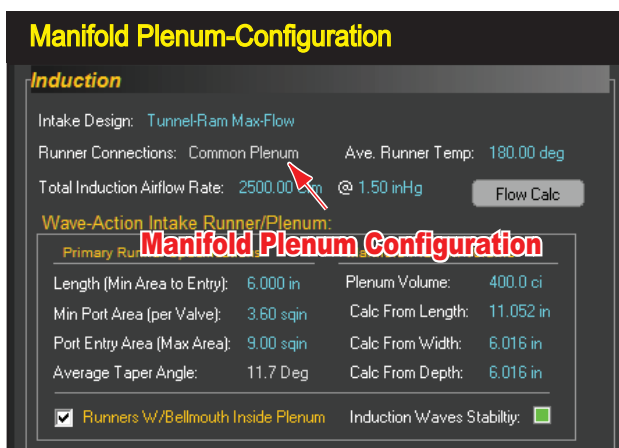
Induction Category

And the induction flow (***Total Induction Flow Rate***) in the **INDUCTION** category specifies the maximum airflow that passes through the restriction common to all cylinders: typically, the throttle body. Similarly, in an I.R. system the ***Flow Rate*** is the total airflow through all of the individual stacks. So, for example, if each I.R. stack has a rated flow of 100cfm, the correct induction ***Flow Rate*** you should enter in the Induction Category is 400cfm (100cfm x 4-cylinders).

> **Note:** The ***Total Induction Flow Rate*** is divided equally between all of the individual runners/stacks.

The ***Small Diameter, Longer Stacks*** model produces a power curve similar to the *Flat Tunnel Ram, Standard Runner, Factory Volume* manifold discussed in the previous section. However, the improved flow potential over the factory manifold at higher engine speeds offers improved power that will peak at a slightly higher rpm.

The ***Small Diameter, Short Stacks*** is the shorter-runner version of the previous I.R. selection. It produces a power curve similar to the *Honda Type, Short Runner, Factory Volume* manifold discussed earlier. The improved flow potential at higher engine speeds offers improved power that will peak at a slightly higher rpm. Expect this I.R. induction to produce good power levels beyond 8000rpm on a properly modified race engine.

The ***Large Diameter, Long Stacks*** selection produces a horsepower curve similar to the *Flat Tunnel Ram, Standard Runner, Large Volume* manifold. The improved flow potential at higher engine speeds offers improved power that will peak at a higher rpm. Expect this I.R. induction to produce good power

Individual Runner (Injector Stack) Induction System
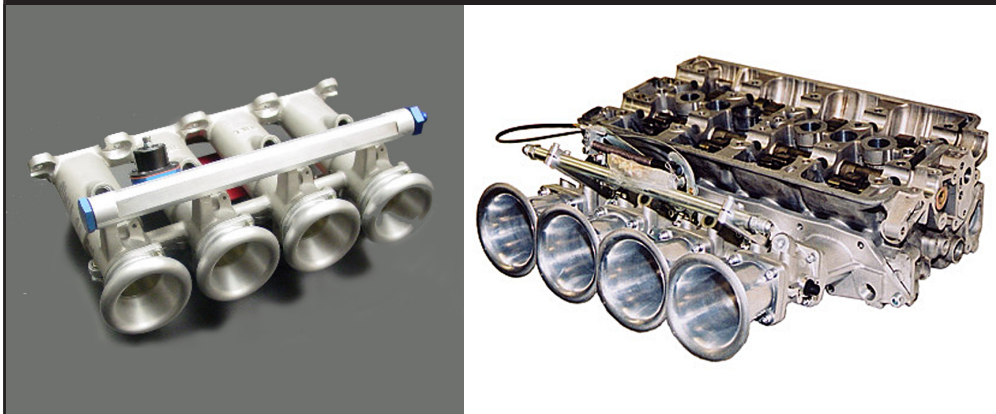
I.R. systems offer the lowest potential restriction of any induction system. The bell-mouth entrance can help tune the horsepower peak to the desired rpm range. Careful selection of stack size, length, bellmouth size, and flow rate are essential to building a winning I.R. induction. All of these elements can be tuned and optimized, including elliptical bellmouth runners.

Induction Category

For naturally-aspirated, professional racing applications, individual- (or isolated) runner (I.R.) induction systems offer the ultimate in flow potential and peak power at high engine speeds. Each runner/tube contains its own throttle plate or slide valve.
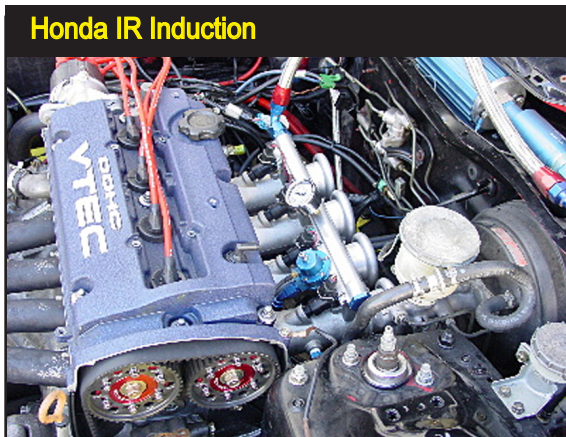
beyond 8000rpm on a properly modified race engine.

Finally, the **Large Diameter, Short Stacks** produces a horsepower curve similar to the *Flat Tunnel Ram, Short Runner, Large Volume* manifold. The improved flow potential at higher engine speeds offers improved power that will peak at a higher rpm. Expect this I.R. induction to produce excellent power beyond 9000rpm on a properly modified race engine.

Individual Runner Manifold For Carburetors

Similar to the previous I.R. induction systems, individual- (or isolated) runners connect each cylinder to one "barrel" of a (in this case) carburetor and *they do not share flow with other cylinders/barrels through interconnecting passages.* These systems, once commonplace in drag racing, are now mostly limited to specific classes or vintage street vehicles.

While IR systems with carburetors are now rare, they still look awesome. The flow potential of these four Webers is quite a bit less than injector stacks, however in street-rod applications, they produce good low- and mid-range power. The air screens on this setup are available from *Carbs Unlimited* (they offer a range of accessories for Webers and Dellortos carburetors).

Racing/Street IR Carburetion

Induction Category

FE Runner Taper Angle Menu

Induction

Intake Design: Baseline Plenum Model

FE Runner Taper Angle: 1.00

Runner Connections: Common Plenum

Total Airflow Rate At Throttle: 550.00 Cfm

Wave-Action Intake Runner/Plenum

Primary Runner Specifications

Length (Min Area to Entry):   in
Min Port Area (per Valve):   sqin
Port Entry Area (Max Area):   sqin
Average Taper Angle: Deg

☐ Bellmouth Not Available On FE Sim

Forced Induction Specifications:

Compressor: None

0.0 Degrees Taper	Tune For Low-Rpm Power Peak
0.5 Degrees Taper	
✓ 1.0 Degrees Taper	
1.5 Degrees Taper	
2.0 Degrees Taper	
2.5 Degrees Taper	Tune For Mid-Rpm Power Peak
3.0 Degrees Taper	
3.5 Degrees Taper	
4.0 Degrees Taper	
4.5 Degrees Taper	
5.0 Degrees Taper	Tune For High-Rpm Power Peak

Average (or FE) Intake Runner Taper Angle menu sets the progression of the area change in the intake port/manifold runner from the minimum area (near the valve stem) to the maximum area at the port entrance. Minimum taper increases low-speed torque and is commonly found in stock, OEM manifolds. For high-performance applications, taper angles of 3 to 5 degrees optimizes power at higher engine speeds.

FE Simulation Intake Runner Taper Angle (FE Only)

The *Average (or FE) Runner Taper Angle* menu sets the intake port taper in degrees as the port changes from it's *Minimum Area* (typically located at the valve stem position) to the *Maximum Area* at the atmospheric entrance or junction with the intake manifold plenum.

The intake taper angle can have a significant effect on engine tuning. As runner taper angle increases, the returning pressure waves speed up, providing optimum tuning at higher engine speeds. Straight runners, with little or no taper, often provide more power at lower speeds and work quite well for stock engines. However, to obtain peak performance from high-speed engines, intake runners typically incorporate a taper of about 2 to 5 degrees.

There are several reasons why taper angles are used to tune high-performance engines: 1) As just mentioned, the pressure waves in a tapered port speed up

Average (or FE) Runner Taper Angle is the measure of the changes in port cross-sectional area from the minimum point (typically at the valve stem) to the maximum area at the port entrance.

Intake Runner Taper Angle

Port Entry Area

Port Taper Angle

Minimum Area
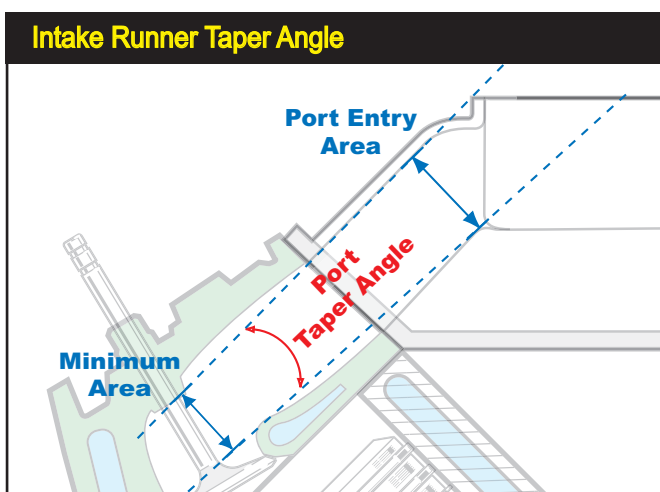
and arrive at the valve and cylinder sooner; providing a "ram effect" that is better tuned to higher engine speeds, 2) A tapered port has a greater volume to store air/fuel charge and feed the cylinder during the main portion of intake stroke, and 3) A taper angle acts somewhat like a stepped-header for the induction system, generating a series of positive pressure waves that return to the valve and assist cylinder filling.

Inlet-runner taper angles can be directly modified in the FE Simulation by selecting the taper angle in degrees from the **Average (FE) Runner Taper Angle** menu. As you modify this parameter on many engines, you will see the power curves shift higher and lower in the rpm range. The amount of shift varies depending on engine speeds, cam timing, and other engine tuning parameters. Try different values to gain a better understanding of how this important element affects engine output. After some testing and experimentation, you'll be able to build better-functioning induction systems.

> **Note:** A runner taper angle will be automatically loaded into the simulation when any intake manifold/configuration is selected from the **Intake Design Menu**. This value is only a suggested tuning point; you can set the taper angle to any value, effectively tuning the induction system to suit a wide range of applications.

Runner Wall Temperature

The **Average Runner Temperature** menu sets the temperature of the intake manifold and/or the runner passages leading from the plenum/atmosphere to the junction at the cylinder head. The default selections are 180-degrees(F) for plenum inductions and 120-degrees(F) for individual runner systems.

This temperature setting models manifolds or runners that are cooler (or hotter)

Average Runner Temperature menu sets the temperature of the intake manifold and/ or runners leading from the atmosphere or plenum to the junction at the cylinder head. This temperature lets you model intake systems that are cooler (or hotter) than the overall engine temperature (e.g., "air-gap" manifold designs).
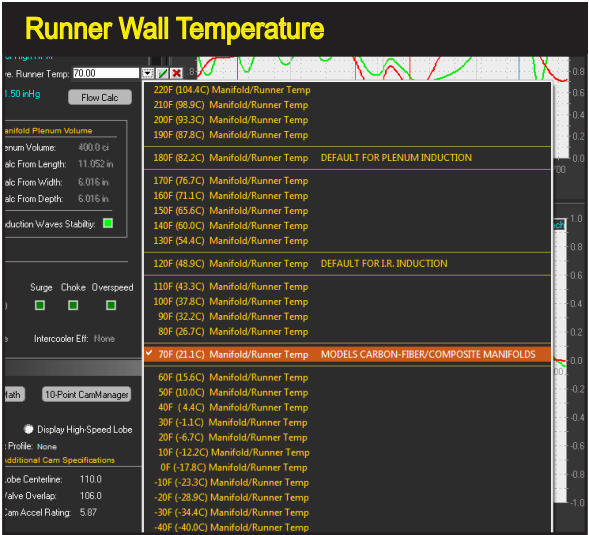
Induction Category
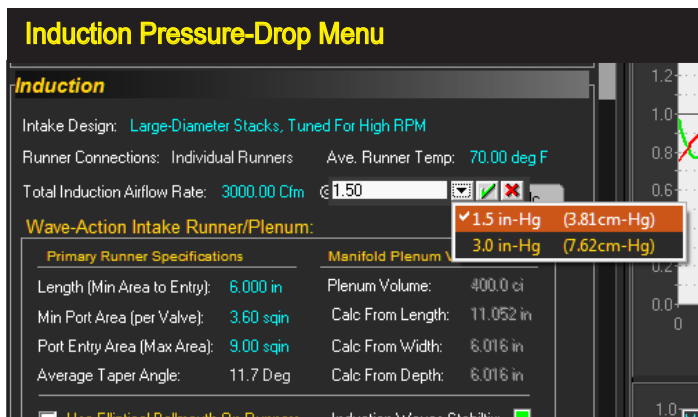
Use the *Induction Airflow Pressure Drop* **menu to select either 1.5-inches of mercury (1.5-in/Hg), a measurement standard for 4-barrel carburetors and injection systems, and the two-barrel carburetor standard of 3.0-inches of mercury (3.0-in/Hg).**

than the overall engine temperature (e.g., "air-gap" and other raised-runner designs). Keep in mind that average runner wall temperature only indirectly changes air/fuel charge temperatures. At lower engine speeds, slower runner flow interacts with the walls for a longer time, increasing the exchange of heat. At higher engine speeds, this interaction time decreases, reducing the effects of wall temperature on charge density.

> **Note:** When you select an intake manifold from the **Intake Design** menu, the *Average Runner Temperature* will *automatically revert to the default values*. Make sure to review (and change, if necessary) runner wall temperatures after selecting an induction system (manifold or IR).

Airflow Selection And Pressure Drop

The **Total Induction Airflow Rate** menus establish the rated airflow for the induction system and the pressure drop at which it's measured. The consists of *2-barrel-carburetor* and *4-barrel-carburetor/fuel-injection* choices. In addition, you can directly specify any rated airflow from 100 to 7000cfm.

> **Note-1:** The flow ratings for 2-barrel carburetors are measured at a pressure drop twice as high (3.0-in/HG) as the pressure used to rate 4-barrel carburetors and most fuel-injection systems (1.5-in/HG). The higher pressure drop increases the measurement resolution for smaller carburetors and "shifts" the flow numbers toward the range commonly found in automotive applications (roughly, 100 to 700cfm). The pressure drop is the pressure differential maintained across the system (e.g., carburetor or throttle body) during airflow measurement at wide-open throttle.

> **Note-2:** You can use the built-in **Airflow Pressure-Drop Conversion Calculator** (available in the *Tools Menu*) for quick conversions between airflow and pressure-drop values.

Airflow Rate Assumptions

Induction Category

Wave-Action-Only Induction Fields

Wave-Action Intake Runner/Plenum:

Primary Runner Specifications		Manifold Plenum Volume	
Length (Min Area to Entry):	6.000 in	Plenum Volume:	400.0 ci
Min Port Area (per Valve):	3.60 sqin	Calc From Length:	11.052 in
Port Entry Area (Max Area):	9.00 sqin	Calc From Width:	6.016 in
Average Taper Angle:	11.7 Deg	Calc From Depth:	6.016 in

☑ Runners W/Bellmouth Inside Plenum   Induction Waves Stabiltiy: ■

The *Induction* category includes data-input fields specifically for the *Wave-Action* simulation. These are *Runner Length*, *Port Entry Area*, *Minimum Port Area*, and *Plenum Volume*, plus a checkbox to activate an Elliptical Bellmouth model.

The **Total Induction AirFlow** menu selections "install" **2-bbl** carburetors from 300- to 600cfm or **4/8-Bbl Carb Or Inj.** systems from 300- to 7000cfm. The **4/8-BBL** label indicates that the induction system can consist of single or multiple carburetors (or fuel injection throttles.) that combine to produce of the rated airflow. An induction system equipped with twin 1100cfm carburetors would have a rated airflow of 2200cfm. If an air cleaner is used, total airflow (at the same pressure drop) should be adjusted to compensate for the increase in restriction at the atmospheric boundary.

> **Note:** The simulation makes *no assumption about the type of restriction* used in the induction system (any throttle configuration, any runner shapes or interconnections, etc.). The airflow rating is simply a means to model the greatest overall restriction of the entire induction system. As higher airflow levels are selected, the simulation lowers the restriction within the induction system.

<div align="center">

**Wave-Action Only
Intake-Runner Specifications**

</div>

The power of the *Wave-Action* simulation lies in its ability to predict and analyze the interaction of pressure waves and their influence on mass flow in the engine. By design, this simulation is sensitive to runner lengths, passage taper angles and volumes. The *Induction* category includes a group of data-input fields specifically for the *Wave-Action* simulation. These fields are **Runner Length**, **Port Entry Area**, **Minimum Port Area**, and **Plenum Volume**, plus a checkbox to activate an **Elliptical Bellmouth** model applied to the runner atmospheric boundary.

The following sections provide an overview of each of these *Wave-Action* intake-runner data-entry fields:

<div align="center">

Runner Length (Dynomation6 WA Only)

</div>

This is the distance from the point of minimum cross-sectional area to the location in the port where the area substantially increases. In a common-plenum manifold, like a single-plane or tunnel-ram, this point is usually located at the plenum junction. In the case of an individual-runner system, this transition occurs at the bellmouth

Induction Category

Measuring Runner Dimensions
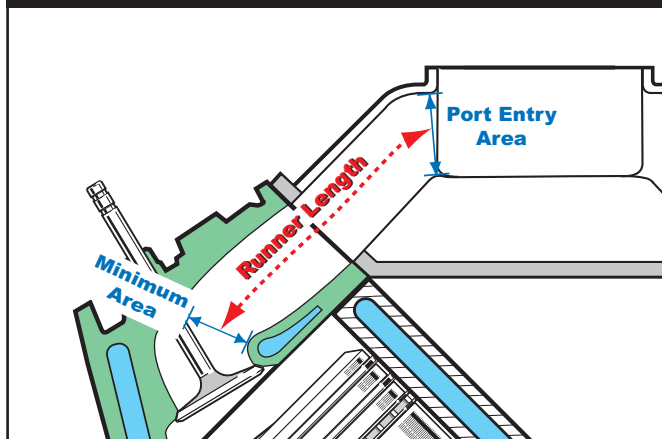
Intake *Runner Length* in the wave-action simulation is defined as the distance from the point of minimum area to the point where the port area substantially increases. *Port Entry Area* is measured at the port entrance junction with the plenum or just inside the runner bellmouth. And the *Minimum Port Area* usually occurs just above the valve seat, although it can occur at other locations in the port.

of the velocity stack.

Port Entry Area (WA Only)

This is the area at the entrance to the port runner (in in^2 or mm^2, depending on the **Units** currently selected). This value is the starting point for the port, and is used, along with the *Minimum Port Area*, to determine the included *Port Taper Angle*, a critical dimension analyzed in the Wave-Action simulation.

Minimum Port Area (WA Only)

This is the area at the point of *greatest restriction* in the port and is also the point at which runner length is measured. It is usually located just above the valve seat near the valve guide, however, it may also occur at other locations within the port. The *Minimum Port Area* is a very critical engine dimension; carefully measure and/or calculate this value to optimize simulation accuracy in Dynomation6. If you inaccurately measure this value, horsepower predictions and your tuning efforts will also be inaccurate.

As you test various engine configurations using the Wave-Action model, you will find that *Minimum Port Area* has a "resonant" value at the point of optimum power. For any particular engine, areas larger or smaller than this will produce less power. If your cylinder heads provide optimum port areas and *Taper Angles*, your engine can produce peak power.

When the *Minimum Port Area* is tuned to an optimum size, it generates a "sweet spot" between restriction that produces high port velocities, allowing pressure waves to impart sufficient energy to the intake charge and "ram tune" the cylinders, versus an oversize area that, although offering less restriction, depletes the strength of pressure-waves and consequently diminishes cylinder filling. Rapidly pinpointing

Induction Category

Intake Port Taper Angle
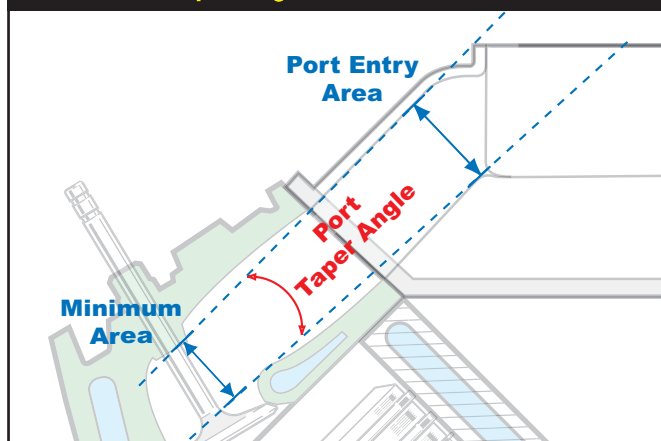
Intake runner *Minimum Port Area* and *Taper Angle* are critical dimensions in the induction system. If incorrect values are entered, power predictions will also be incorrect.

NOTE: The *Minimum Port Area* value used in Dynomation6 is the area *Per Valve.* If you are modeling multiple-valve-per-port cylinder heads (2 or 3 valves per runner), do not add these port areas together to determine Minimum Port Area!

the *Minimum Port Area* that produces peak performance is a terrific strength of the wave-action simulation.

Determining optimum *Minimum Port Area* is entirely dependent on engine displacement, cam timing, port flow, valve sizes, engine speed range, and many other variables that make this job extremely difficult without using a mathematical simulation. Dynomation6 analyzes all of these variables in every Wave-Action run. After just a few simulations, you can easily home-in on an optimum *Minimum Port Area* for any engine application!

> **Important-1:** This may seem obvious (but it's easy to forget), the *Port Entry Area* and *Minimum Port Area* are **areas**, not diameters of the ports/runners.
>
> **Important-2:** The *Minimum Port Area* can range from below 50% of the area of the intake valve, to as large a 75-to-80% or larger of the valve area in racing cylinder heads.
>
> **Important-3:** The *Minimum Port Area* is a dimension in in^2 or mm^2 (depending on the current **Units** selection) within the port of *Each Valve* of multiple-valve-per-port cylinder heads. ***Do Not Add*** minimum port areas together when you enter a *Minimum Port Area* for 2- or 3-valves-per-port applications.

Plenum Volume (WA Only)

The ***Plenum Volume*** models the *volume* (in^3 or mm^3) not the *shape*, of a plenum used in *Common-Plenum* induction systems. You can directly enter the volume of the plenum, or enter the approximate physical dimensions and Dynomation6 will calculate the volume. The minimum plenum volume is dependent on the *Port Entry Area* (Maximum Area of the port). The simulation checks to ensure a transition of sufficiently large volume occurs at the port/plenum boundary. If a larger plenum volume is required, a message box will indicate the issue and it will be corrected

Induction Category

An Elliptical Bellmouth may be added to the end of intake runners using the Wave-Action simulation. Here are the typical dimensional relationships modeled by the sim. An Elliptical profile typically outperforms a simple radius or airfoil shapes.

Elliptical Bellmouth Specifications

Pipe Begin Diameter =
2.13 x Tube Diameter

Elliptical Profile Stack

Same Height
As Tube Diameter

Tube Diameter
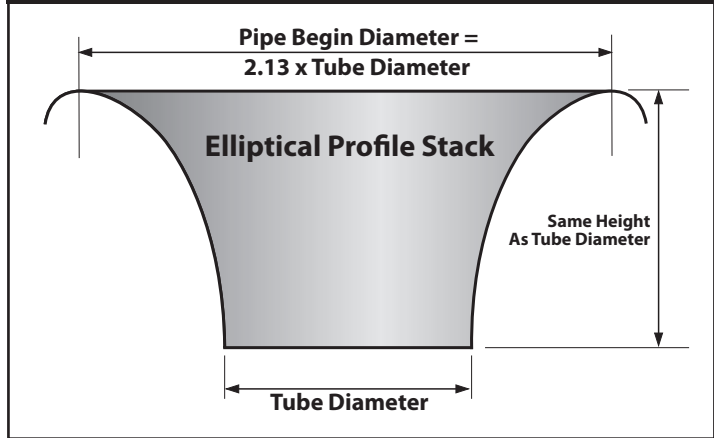
automatically when you click on **OK**.

> **Note:** Only the volume of the plenum is used in the simulation; height, width and depth are only used to calculate a volume and hold no further significance within the simulation.

Elliptical Bellmouth (WA Only)

A checkbox is provided in the *Induction* category to activate an ***Elliptical Bellmouth*** model that can be applied to the runners, either at their connection to a plenum or at the atmospheric interface. Principal bellmouth characteristics include a wide radius with an elliptical profile and a short overall height (equal to the diameter of the main stack tube). The effects on performance generally occur at higher rpm but depend on runner dimensions and overall engine design.

Induction Category

Induction Category (Forced Induction)

The *Induction* Category includes several *Forced-Induction* systems, including Roots & Screw Blowers, Centrifugal Blowers, and Turbochargers (any of which can be used with Nitrous Injection, discussed later in this manual). Fields that are not applicable to the current forced-induction method are dimmed, like the *Internal* and *Belt Ratios* when a turbocharger is used.

Forced-Induction Modeling And Specifications

The ***Forced Induction Selections*** included in the *Induction* category considerably expand the modeling power of Dynomation6 and DynoSim6. You can add a positive displacement Roots- or Screw-type blowers, a centrifugal blower (like a Rotrex or Vortech), or a turbocharger to any engine. In addition, you can vary maximum boost—or blow-off (wastegate) pressure—***Belt Ratios***, ***Turbine Size***, ***Turbine A/R*** ratio, ***Intercooler*** parameters, and more.

> **Note-1:** When you apply any of the forced-induction systems, keep in mind that you are adding forced induction to the intake manifold selected in the *Intake Design* menu.

> **Note-2:** Adding forced induction to *Individual-Runner* selections, while technically possible, converts the IR system into a common plenum configuration by the plumbing interconnections required to route airflow to each stack.

To select any of the forced-induction options, double-click the ***Compressor*** field to open a menu containing ***Turbocharger***, ***Centrifugal***, ***Roots***, and ***Screw*** blower choices. After a selection has been made, appropriate fields in the *Forced Induction* group will become active depending on the characteristics of the selected supercharger. You may modify these values at any time to determine their effect on engine power.

Here is a quick overview of the supercharger component fields, the supercharg-

Induction Category (Forced Induction)

Forced Induction Menus
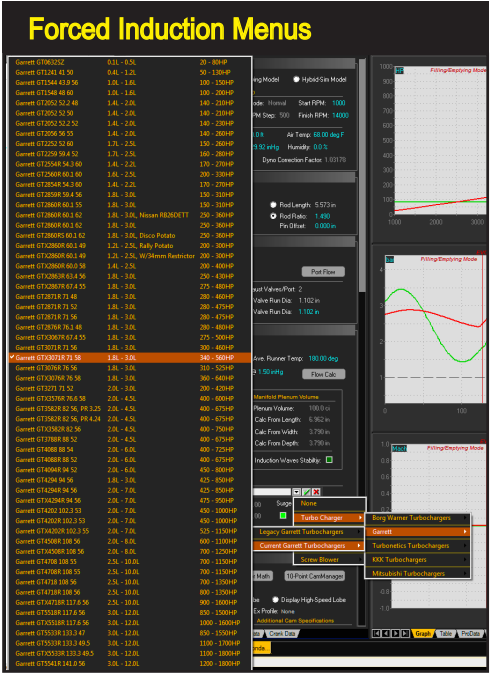
The simulation includes over 300 forced induction choices. Selecting a supercharger from any of the submenus will load that device into the *Induction Category* and transfer supercharger-map data into the simulation.

ers to which they apply, and how they affect forced-induction performance:

Flow Restriction (throttle) Location—The location of the primary intake restriction (commonly the throttle) can have a significant affect on forced induction engines. If the restriction is located *Before* the compressor, inducted air is *drawn-through* the restriction and (like all naturally aspirated engines) the maximum pressure drop (and resulting mass flow) through the restriction is limited to one atmosphere. However, if the restriction is positioned *After* the compressor, the pressure drop across the same restriction is limited by manifold pressure

Turbine & Compressor Impellers
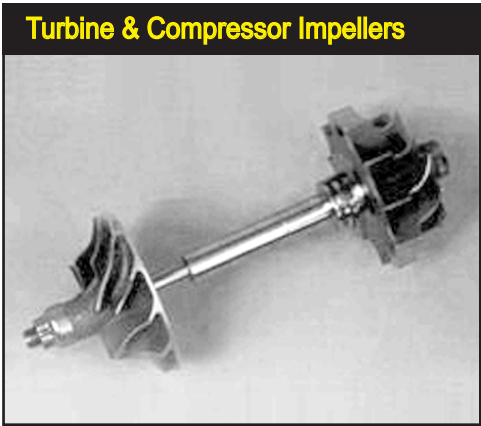
The turbine (impeller shown on the right end of the shaft) is placed in the exhaust stream of an engine and is driven by a combination of exhaust flow and exhaust pressure waves. Two parameters in the simulation adjust the performance of the turbine: the *Turbine Wheel Size* and the *Turbine Housing A/R Ratio*. Modifying the turbine wheel size will make large changes in the turbine speed; the smaller the wheel, the faster the turbine will rotate. The higher the turbine speed, the more airflow will be driven through the compressor (until the compressor blade tips reach supersonic speed).

Induction Category (Forced Induction)
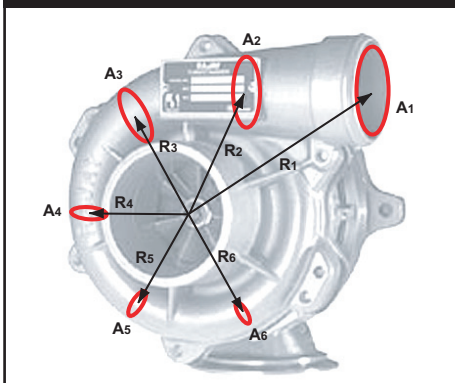
The *Turbine* (the exhaust-driven element) *Housing A/R* ratio is a comparison of the cross-sectional area of the housing inlet (the "A's" in the photo) to the distance the inlet passage is positioned from the center of the turbine (the radius, or "R's" in the photo). Unlike the *Turbine Size* (that has a dramatic affect on turbine speed), the *Turbine Housing A/R Ratio* fine-tunes turbine speed and helps establish where—in the engine rpm range—the turbo begins to produce boost pressure.

(as opposed to atmospheric pressure), potentially generating greater mass flow for the same size restriction. Many turbo systems position the throttle after the compressor, while Roots and Centrifugal compressors typically place the throttles before the compressor.

Turbine Size—(Turbos only) This is the diameter of the exhaust-driven impeller (in inches or millimeters). The turbine is placed in the exhaust stream of an engine and is driven by a combination of exhaust flow and exhaust pressure pulses. Two parameters adjust the performance of the turbine: in general, the *Turbine Wheel Size* and the *Turbine Housing A/R Ratio* (discussed next). Modifying the turbine wheel size will make relatively large changes in the turbine speed; the smaller the wheel, the faster the turbine (and the compressor) will rotate. The higher the turbine speed, the more airflow will be driven through the compressor. Smaller turbines generate boost earlier in the rpm range, however, if the turbine
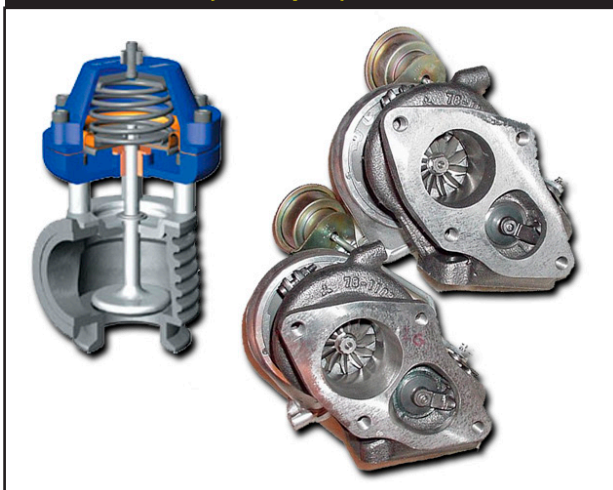
Boost Limit is an arbitrary maximum induction pressure established by the wastegate setting. It is not a measure of the capability of the supercharger, i.e., the blower may or may not be able to develop sufficient pressure to activate the wastegate. The simulation incorporates a wastegate model that modulates the size of its bypass passages when the *Boost Limit* is reached.

Induction Category (Forced Induction)

Belt Ratio

Both centrifugal and roots blowers are mechanically driven by the engine. The *Belt Gear Ratio* (external drive) is the mechanical connection between the engine crankshaft and blower input shaft. This Jackson Supercharger pulley setup provides an overdrive (an overall *Belt Ratio* of 1.20:1).

is too small for the application, the turbocharger shaft speed can exceed the manufacturer's recommendations (turning the *Overspeed* indicator from green to yellow, and if overspeed is sufficiently high, it will turn red).

Turbine Housing A/R Ratio—(Turbos only) This is a ratio of the cross-sectional area of the turbine housing inlet to the inlet passage distance (radius) from the center of the turbine wheel (see photo). As the *A/R Ratio* is increased, the overall size of the turbine housing and its internal passage become larger. Unlike *Turbine Size* that has a dramatic affect on turbine speed, the **Turbine A/R Ratio** fine-tunes turbine speed. Changing *A/R* has many effects. By using a larger **A/R Ratio**, the turbo produces less boost at lower engine speeds but develops more boost at a higher engine speeds, primarily due to the reduction in exhaust backpressure in larger turbine housings. Lower engine backpressure

Centrifugal superchargers are driven through an external *Belt Ratio*, but this speed increase is insufficient for most centrifugal superchargers to reach their optimum operating speeds (35,000rpm and higher). An internal gear train is commonly used to further increase rotational speed.

Internal Gear Ratio
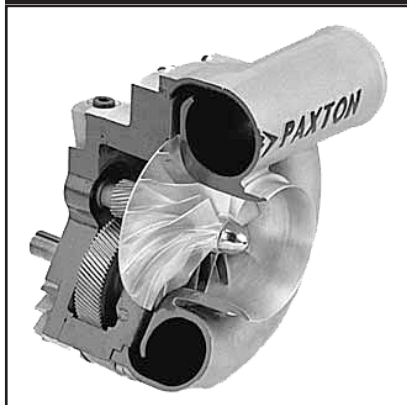
improves engine volumetric efficiency (VE), and that can result in an power increase throughout the usable rpm range. So, finding the optimum values for both the *A/R Ratio* and *Turbine Size* are important steps in building the most efficient and responsive turbocharger system.

Number Of Compressors—This selection divides the exhaust flow (for turbos) and intake flow restrictions equally between all compressors (up to four can be used in parallel on the same engine). Outflow from all compressors is directed into the induction system of the engine (through an intercooler, if selected) and is divided equally between all cylinders.

Turbocharger Note: Multiple turbo applications are most successful when the engine produces sufficient exhaust-gas volume to spin up all turbos; using small turbine sizes and housings can improves multiple-turbo applications.

Boost Limit—(Turbos, Centrifugals, Roots, Screw) This is the pressure at which the wastegate or blow-off valve is activated, maintaining maximum induction pressure at or below this value.

Note: Boost Limit is an arbitrary pressure, chosen by you, for the simulation. It is not a measure of the capability of the supercharger. In other words, because of component mismatch and other factors, the blower may not be able to develop sufficient airflow and boost to reach the wastegate relief pressure.

Belt Ratio—(Centrifugals, Roots, Screw) Centrifugal, roots, and screw superchargers are mechanically driven by the engine. The **Belt Ratio** (the external drive ratio) is the ratio of the mechanical connection between the engine crankshaft and blower input shaft. This value is multiplied by the **Internal Ratio** (internal gear ratios are commonly used on centrifugal superchargers) to determine final compressor/rotor speed.

Internal Ratio—(Centrifugal) Centrifugal superchargers are driven by a mechanical connection to the engine crankshaft (the external **Belt Ratio**). Input shaft speed, however, it usually insufficient for most centrifugal superchargers to reach their

The Forced Induction Specifications group includes three "indicators" that will alert you if *Surge*, *Choke*, or *Overspeed* are present during any portion of the test rpm range. See the text for a definition of each indicator and tips on how to correct problems. Also, refer to the *Simulation Log* for more information about the indicated problem.

optimum operating speeds (35,000rpm and higher), so an internal gear train is commonly used to further increase rotational speed. The ratio of this internal gearing determines how much faster the turbine rotates. To determine the final speed of a centrifugal turbine, multiply crankshaft rpm by the **Belt Ratio**, then multiply that by the **Internal Ratio**.

Operational Indicators—(Turbos, Centrifugals) The forced-induction portion of the **Induction** category includes three "indicators" that will help you select the correct turbocharger, turbine wheel size, and A/R ratio. Each indicator shows a potential operational problem by turning from green to yellow (minimal problem), or to red (should be corrected). Additional information about the cause and extent of the problem will be included in the Simulation Log (displayed using the *Simulation Menu*).

Surge: (Turbo Only) This condition occurs when mismatched components cause unstable airflow through the compressor. If a turbo is operated consistently within surge, additional loads can damage the turbines, shafts, and bearings. If surge is detected during a simulation run, the **Surge** indicator will display either yellow or red. A yellow *Surge* indicator means that the turbo has entered surge only a limited time during the full engine rpm range. This limited-surge operation is not considered unusual and is not normally associated with shortened turbocharger life. If the indicator turns red, the turbo has entered into surge often during the engine "dyno test." The turbocharger, *Turbine Size*, or *A/R* ratio should be changed to eliminate this condition.

Choke: (Centrifugals, Turbos) While **Choke** can apply to both the Compressor and Turbine impellers (in turbocharger applications), **Choke** most often refers to

This turbo compressor map, produced by the manufacturer, shows how the compressor turbine performs at various flow rates, pressure ratios, and rpms. If insufficient engine demand causes the turbo to operate "off the map" to the left, *Surge* can occur. If engine demand line pushes off the right side of the map, the turbo is said to be in a *Choke* condition. And if the turbocharger speed exceeds its maximum rpm, as shown on the map (in this case slightly above 200,000rpm), an *Overspeed* condition exists. Each of these abnormal operating conditions will turn a forced-induction indicators from green to yellow, and if the condition occurs at multiple rpm points or is well beyond the limit, the indicator will turn red.

Turbocharger Compressor Map
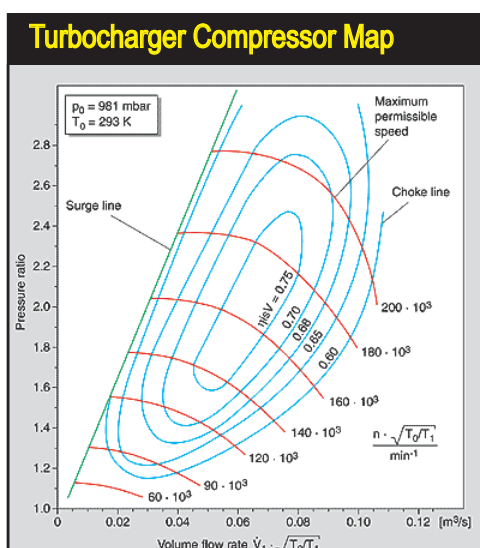
the point at which compressor wheel tips reach sonic velocity, preventing further flow. The choke line on a compressor map can be recognized by the steeply descending speed lines at the right side of the map. If choke is detected at any point during the engine rpm range, the **Choke Indicator** will turn from green to yellow, indicating a condition that should be corrected. However, choke problems are relatively easy to correct, often disappearing with a slight reduction in shaft speed (use a larger Turbo or *Turbine Size,* and/or increase the *A/R* ratio).

Overspeed: (Centrifugals, Turbos) If the shaft speed exceeds the manufacture's recommendations (the engine demand line travels off of the top of the compressor map), the **Overspeed Indicator** will change from green to yellow. If the overspeed condition exceeds the manufacturer's recommended speed by over 10%, the indicator will switch from yellow to red, indicating a potentially harmful condition to the turbocharger bearings and seals. If you are simulating a turbocharger system and you have a red *Overspeed* condition, increase the *Turbine Size* and/or the *Turbine Housing A/R* ratio.

Important Note: If overspeed occurs in a real supercharging system, you risk destroying the compressor and spraying metal particles into your engine.

Selecting The Best Supercharger For Your Application

The Version6 Compressor Map Window

Compressor Map: Garrett-GTX3076R_76_58.map

Pressure Ratio vs. Flow Rate (Lb/Min)
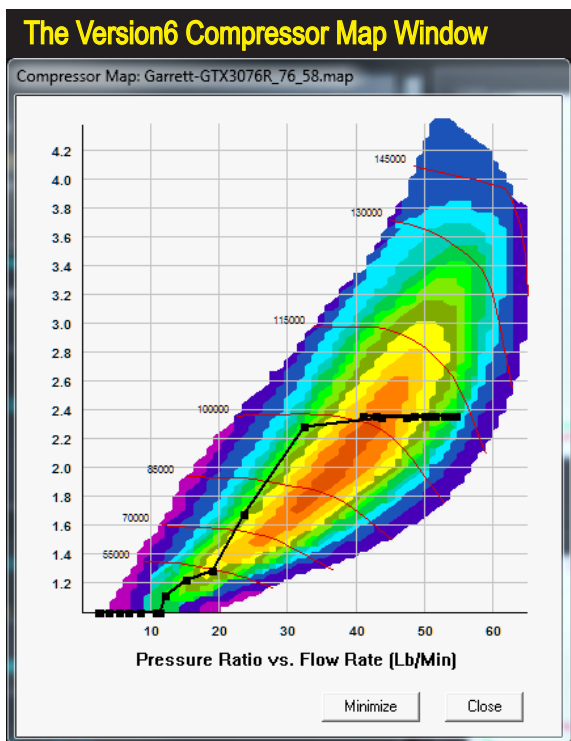
The compressor map display window in the simulation can be viewed by choosing *Show Turbo/Centrifugal Compressor Map* from the Simulation Menu. The left vertical axis shows the output pressure ratio (2.0 equals 2 atmospheres of pressure; about 15 psi of boost). The bottom horizontal axis is the mass flow through the compressor in pounds per minute (Lb/Min). The small numbers positioned along the left edge of the map are the rpm values for the red speed lines. The black line on the graph is the calculated engine demand line; the dots indicate the mass flow at each rpm test point (also displayed in the ProTools table). You can move the map window to any position on your desktop. Use the *Minimize* button to temporarily "park" the window to a tab that appears at the bottom of the program screen; to re-display the window, click the parking-tab.

Induction Category (Forced Induction)

Selecting a Turbocharger—A turbocharger consists of a compressor and a turbine. The "compressor side" characteristics are established when you choose the turbo from the main **Blower Type** menu. The compressor acts as pump that forces air into the induction system. The output of the compressor is determined by the speed of the impeller (inside the compressor housing), the shape of the housing, and the swallowing capacity of the engine. These variables are visually illustrated on a compressor map. The simulation incorporates more than 100 turbo maps within its library that cover a wide range of capacities.

The turbine side of the turbocharger is composed of a turbine wheel and the surrounding housing. The turbine is placed in the exhaust stream of the engine and is driven by a combination of exhaust flow and exhaust pressure waves. The turbine is connected to the compressor impeller through an interconnecting shaft. Two principal parameters modify the performance of the turbine: 1) the **Turbine Size** and, 2) the **Turbine A/R** ratio. As described earlier, the **Turbine Size** has a major effect on turbine and compressor speed. In general, the smaller the wheel, the faster the turbine will rotate. The **Turbine A/R** ratio acts to fine-tune the turbine speed. The smaller the **Turbine A/R**, the more quickly the turbo will "spool up."

To find the optimum turbocharger for your engine, first select a basic turbo/compressor (turbocharger) from the **Blower Type** menu. For help in making an initial selection, consult with the turbo manufacturer or visit the many performance websites that offer turbo kits and performance modifications. You will often find turbo recommendations and even suggestions for turbine-wheel diameters and A/R ratios. After you have selected the basic turbo, set the **Boost Limit** (wastegate pressure) to the desired level (use 10psi if you're not sure). Next, you'll need to analyze how much airflow the turbo is delivering to the engine by viewing the manifold pressure curve (right-click the rpm-based graph, located on the top-right of the main screen, and assign **Boost Pressure** to either the **Y1** or **Y2** axis).

Inspect the manifold pressure curve for the rpm point at which the pressure rises to a maximum. Make sure that the turbo you have selected, in fact, reaches the wastegate relief pressure chosen in the **Boost Limit** field.

> **Note-1:** The simulation can display either *Boost Pressure* and/or *Absolute Pressure*. If the graph pressure curve is an *absolute* pressure plot, remember to subtract ambient pressure to determine boost pressure (for example, an absolute pressure of 25psi would indicate a boost pressure of about 10psi—25psi absolute, minus 14.7psi ambient, equals 10.3psi boost).

> **Note-2:** The mass flow rate of the engine is displayed in the ProTools graph. This value is helpful in selecting a supercharger or a turbocharger, since the mass flow rate is also shown on the blower map and plotted as dots on the demand line. Pick a turbo that has a maximum flow somewhat greater than the highest value of engine demand, otherwise the turbo may enter a restrictive "choked flow" state at full load.

If the maximum-boost rpm point occurs considerably late in the rpm range, the

Induction Category (Forced Induction)

Centrifugal Supercharger
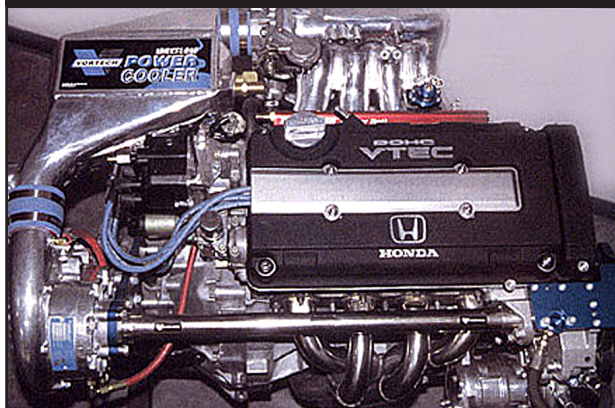
Like a turbocharger, a centrifugal supercharger uses a compressor wheel, but the driving force comes from an internal gear box connected to the engine crankshaft. Because the centrifugal compressor also must rotate at high speed, the internal "step-up" gear increases input pulley speed by up to four times.

boost pressure may never reach the *Boost Limit*, or if the **Choke** indicator is illuminated, installing a larger compressor may solve the restriction problem. On the other hand, if the boost pressure curve begins too early in the rpm range, or the **Surge** or **Overspeed** indicators are red, installing a larger turbine wheel or increasing the *A/R* ratio may solve the problem.

   Selecting a Centrifugal Supercharger—Like a turbocharger, a centrifugal supercharger consists of a compressor wheel and housing, but the driving force comes from the engine crankshaft through a drive belt or direct coupling and an internal gear box. Also like the turbocharger, the centrifugal compressor has performance characteristics defined in a "map" provided by the manufacturer. Because the centrifugal compressor turbine also must rotate at high speed, an internal "step-up" gear typically increases input pulley speed by several times. And input speed is often 1.5- to 3-times engine speed, determined by the appropriately sized pulleys. Final impeller speed is found by multiplying the *Internal Ratio* by the *Belt Ratio* and multiplying the result by engine *RPM*.

   Note: While the internal gear ratio is not usually changeable, the simulation gives you the freedom to experiment with this ratio in addition to all the other tunable elements in the **Forced Induction** category.

   Belt Ratio can be calculated using the following formulas:

   For Serpentine and V-Belts:

$$\text{Belt Ratio} = \frac{\text{Crank pulley diameter}}{\text{Supercharger pulley diameter}}$$

   For Cog Pulleys:

$$\text{Belt Ratio} = \frac{\text{Crank pulley teeth count}}{\text{Supercharger pulley teeth count}}$$

Induction Category (Forced Induction)

To find the optimum centrifugal supercharger for your engine, first select a centrifugal supercharger from the menu. For help in making an initial selection, consult with the manufacturer's website or visit the numerous performance sites that offer kits and performance modifications. You will often find supercharger recommendations and suggestions for belt ratios. After you have selected the basic blower, set the **Boost Limit** (wastegate pressure) to the desired level (use 10psi if you're not sure). Next select the **Belt Ratio** and set the **Internal Gear Ratio** to the values used by the manufacturer (many centrifugal blower run with approximately 2.5:1 external **Belt Ratios** and 3.5:1 **Internal Gear Ratios**). Finally, you'll need to analyze how much airflow the blower is delivering to the engine by viewing the manifold pressure curve (right-click the rpm-based graph on the top, right of the main program screen and assign *Boost Pressure* to either the **Y1** or **Y2** axis).

Inspect the manifold pressure curve for the rpm point at which the pressure reaches the maximum boost pressure. Check that the supercharger you have selected, in fact, reaches the wastegate relief pressure chosen in the **Boost Limit** field.

> **Note:** The simulation can display either *Boost Pressure* and/or *Absolute Pressure*. If the graph pressure curve is an *absolute* pressure plot, remember to subtract ambient pressure to determine the boost level (for example, an absolute pressure of 25 psi would indicate a boost pressure of about 10psi; 25psi absolute minus 14.7psi ambient equals 10.3psi boost).

If the maximum-boost rpm point occurs considerably late in the rpm range, or if the boost pressure never reaches the **Boost Limit**, increasing the *Belt Ratio* or installing a larger supercharger may solve the problem. On the other hand, if the pressure curve rises early in the rpm range, or the *Surge* or *Overspeed* indicators are lit, try decreasing the *Belt Ratio* or installing a smaller supercharger.

Inspect the manifold pressure curve (use the Graph Options Menu to assign boost pressure to either the Y1 or Y2 axis) for the rpm point at which the pressure rises to the maximum boost pressure. This display will show you if, in fact, the selected turbo reaches the wastegate relief pressure set in the *Boost Limit* field. Here, the green curve indicates that a boost pressure of about 19psi is reached at about 5000rpm.

Manifold Pressure Graph
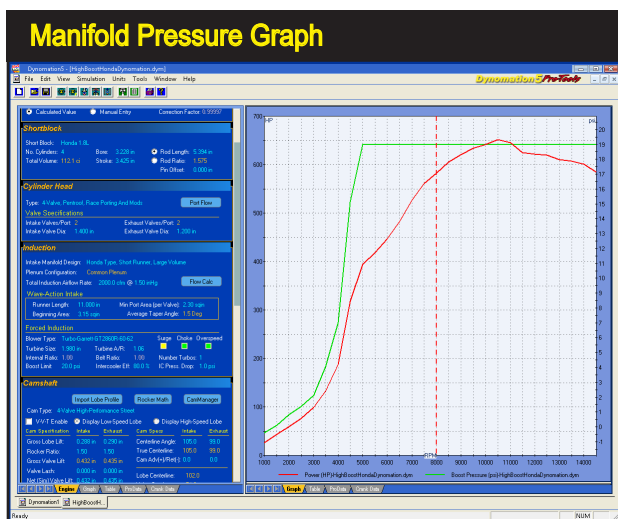
Induction Category (Forced Induction)

Roots Supercharger

The roots supercharger is a positive displacement "pump." It really doesn't compress air, rather it pushes "measured chunks" air into the engine. It employs 2 or 3 counter-rotating lobes to capture a fixed volume of air per revolution and deliver it to the intake manifold. Unlike the compressors of the turbocharger and centrifugal supercharger, a positive displacement supercharger pumps a steady amount of air for every revolution. When properly setup, it can deliver "instant boost" even at low speed. This compact setup is a Jackson high-boost system for the Honda Civic Si.

Selecting a Roots or Screw Supercharger—Roots or Screw-type supercharger selection involves only two parameters: the flow volume and the **Belt Ratio** (these superchargers usually do not have internal overdrive gear boxes). The compressor-flow volume is fixed in the design of the supercharger and is indicated in the blower menu as a volume-per-revolution value.

To find the optimum Roots/Screw supercharger for your engine, first select a supercharger from the menu. For help in making an initial selection, consult with the manufacturer's website or visit the numerous performance sites that offer kits and performance modifications. You will often find supercharger recommendations and suggestions for belt ratios. After you have selected the basic blower, set the **Boost Limit** to the desired level. Remember that the **Boost Limit** is simply the blowoff-valve pressure; the maximum pressure *limit* for the induction system. The *Boost Limit* is not the "guaranteed" pressure the blower will generate; improper sizing or belt ratios, even altitude will lower maximum blower pressures. Next select the **Belt Ratio** recommended by the manufacturer (many Roots blowers use *Belt Ratios* of 1.0- to 1.5:1, while Screw superchargers use approximately 1.5- to 2.0:1). Finally, you'll need to analyze how much airflow the blower is delivering to the engine by viewing the manifold pressure curve (right-click the rpm-based graph on the top-right of the main program screen and assign **Boost Pressure** to either the **Y1** or **Y2** axis).

Inspect the manifold pressure curve for the rpm point at which the pressure rises to the desired level. For optimum efficiency, make sure that the roots or

Induction Category (Forced Induction)

Screw Supercharger
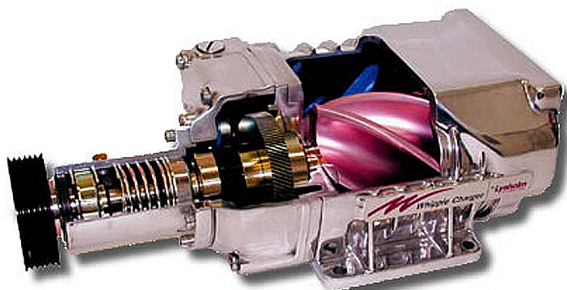
The screw compressor (like this model from *Whipple*) uses a positive-displacement design for excellent low-end torque as well as high efficiency for good top-end horsepower.

screw supercharger you have selected does not quite reach wastegate pressure (*Boost Limit*) within the usable rpm range of the engine (if it does, you may need to select a smaller blower or reduce the *Belt Ratio*). However, manifold pressure should reach a sufficient level to provide the desired performance improvement.

> **Note:** The simulation can display either *Boost Pressure* and/or *Ambient Pressure*. If the graph pressure curve is an *absolute* pressure plot, remember to subtract ambient pressure to determine boost pressure (for example, an absolute pressure of 25 psi would indicate a boost pressure of about 10psi—25psi absolute, minus 14.7psi ambient, equals 10.3psi boost).

If the desired boost pressure occurs considerably late in the rpm range or the desired pressure is never reached, try a larger supercharger or increase the *Belt Ratio*. On the other hand, if the pressure curve rises too quickly, or the *Boost Limit* pressure is reached within the usable rpm range of the engine, decrease the *Belt Ratio* or install a smaller supercharger.

> **Note:** The **Overspeed** indicator works with all superchargers (all turbos and other compressors that have published max-rpm data). **Surge** and **Choke** indicators are not active when Roots or Screw superchargers are used.

Intercoolers

One of the drawbacks to any method of supercharging is the resulting increase in induction-gas temperatures. High boost pressures can quickly raise charge temperatures more than 200-degrees(F)! These higher temperatures, common on blowers with pressure ratios of 2.0 or higher, cost more than just lost horsepower. Higher temperatures can lead to detonation, increase octane requirements, and usually require a reduction in ignition timing advance. While induction cooling can improve performance directly from increased charge density (more oxygen and fuel per unit volume of inducted charge), the additional benefits of reduced detonation and increased reliability make charge cooling an attractive addition to any super-

Induction Category (Forced Induction)

Intercooler And Pressure-Drop Menu

Forced Induction Specifications:

Compressor: Turbo-Garrett-GTX3071R-71-58
Turbine Size: 2.800 in   Internal Ratio: 1.00   Surge   Choke   Overspeed
Turbine A/R: 0.70   Belt Ratio: 1.00
Number Turbos: 1
Boost Limit: 20.0 psi   1.0

None
✓ 1 psi / 0.069 bar
2 psi / 0.138 bar
3 psi / 0.207 bar
4 psi / 0.276 bar
5 psi / 0.345 bar

None
10% Efficiency
20% Efficiency
30% Efficiency
40% Efficiency
50% Efficiency
60% Efficiency
70% Efficiency
80% Efficiency
90% Efficiency
✓ 100% Efficiency
110% Efficiency
120% Efficiency
130% Efficiency
140% Efficiency
150% Efficiency

Camshaft

☑ Filter   Import Lobe Profile
Cam Type: V-V-T Street Cam
☑ V-V-T Enable ◉ Display L
In Profile: None

Cam Specification	Intake	Exhaust	Additional Ca
Gross Lobe Lift:	0.220 in	0.203 in	Lobe CL (Int t
Rocker Ratio:	1.50	1.50	Valve Overlap
Gross Valve Lift:	0.330 in	0.305 in	Cam Accel Ra
Valve Lash:	0.020 in	0.020 in	
True Lift @ Valve:	0.310 in	0.285 in	Variable Valv
Modeled From:	10-Point	10-Point	V-V-T Activat

V-V-T LOW SP
Desired Cam Timing (Modeling Input)   IVO   IVC
(Valve)   Primary (0.006-in) Timing: -6.5   44.5   46.5   4.5

The simulation includes an *Intercooler* model that can be activated with any forced-induction system. An intercooler reduces high induction temperatures that can substantially reduce performance. The *Intercooler* is selected from a menu that indicates efficiency; 100% efficiency reduces induction temperature to ambient. An *IC PressureDrop* menu shows an estimated pressure drop value based on the Intercooler efficiency (due to increased charge density from charge cooling).

charged high-performance engine.

Charge cooling is accomplished in the same way that heat is removed from the engine. A radiator, called an intercooler, is placed in the pressurized air ducting between the supercharger and the intake manifold. The efficiency of an intercooler determines how much of the heat generated by charge compression is removed. The lower the efficiency of the intercooler, the less heat is removed from the induction charge. An efficiency of 100% removes all extra heat (brings charge temperature down to ambient). An efficiency of over 100% (reduces charge temperatures below ambient) is possible with water or ice. Everything from outside air to ice water and even evaporating pressurized liquefied gas (like Freon or nitrous oxide) have been

Intercoolers: The "Secret" To Supercharging Power

Higher induction temperatures, common on blowers with pressure ratios of 2.0 or higher, can cost horsepower and reliability. An Intercooler cools the inducted air the same way that heat is removed from the engine itself: with a radiator. Intercooling is the key to power and reliability in high-boost systems.

Induction Category (Forced Induction)

Intercooled VW Golf

Here's a great ride! This turbocharged Golf uses a generously-sized intercooler. Note the cold-air inlet ducted to the hood/cowl. The clear front end is awesome—this shot was taken with the hood closed!

used to remove heat from an intercooler. The ballpark efficiencies for these devices are:

Air-To-Air	**40%**	**Air-To-Cooler Ducted Air**	**50%**
Air-To-Water	**75%**	**Air-To-Cooled Water**	**100%**
Air-To-Ice Water	**120%**	**Air-To-Evaporating Liquid**	**120+%**

The simulation includes an intercooler model that can be activated with any forced induction system. Simply double-click on the **Intercooler** field and select an intercooler efficiency from the drop-down list (or directly enter a custom value).

Every intercooler will produce a pressure drop generated by its restriction and the increased charge density within the intercooler. This pressure drop is usually small—in the range of 1 to 3 psi. You can adjust the Intercooler Pressure Drop by selecting a value from the **IC Pressure Drop** menu, or let the simulation automatically calculate an estimated pressure drop value based on the selected Intercooler.

Note-1: The **IC Pressure Drop** occurs after the Intercooler, so the **Boost Limit** may need to be increased to obtain the desired boost pressure at intake valve.

Note-2: When methanol evaporates, it cools the intake charge more than gasoline (the latent heat of vaporization of methanol is much greater than gasoline and the volume of *methanol* used is also greater). Therefore, intercooling is less effective when methanol is used (especially if it is injected well before the intake valve in the intake runners, allowing the fuel to fully atomize and drop in temperature).

Camshaft Category

Camshaft

Import Lobe Profile   Rocker Math   10-Point CamManager

Cam Type: Imported Profile

☐ V-V-T Enable   ◉ Display Low-Speed Lobe   ◉ Display High-Speed Lobe

In Profile: 3632.ecp   Ex Profile: 3634.ecp

Cam Specification	Intake	Exhaust	Additional Cam Spec Measurements	
Gross Lobe Lift:	0.348 in	0.354 in	Lobe CL (Int to Exh):	112.0
Rocker Ratio:	1.600	1.600	Valve Overlap:	62.1
Gross Valve Lift:	0.557 in	0.566 in	Valvetrain Accel:	2.78
Valve Lash:	0.000 in	0.000 in		
True Lift @ Valve:	0.5568 in	0.5664 in	Variable Valve Timing Activation	
Modeled From:	Profile	Profile	V-V-T Activation: 5000 rpm	
	->10-pnt	->10-pnt		

INPUT Cam Timing (Zero Lash)	IVO	IVC	EVO	EVC	
(Valve) Primary (0.006-in) Timing:	30.8	70.9	77.8	31.2	
(Valve) Primary 720-Based Timing:	329.2	610.9	102.2	391.2	
(LOBE) Secondary (0.050-in) Timing:	4.0	40.0	51.8	0.1	->STS
(LOBE) Secondary 720-Based Timing:	356.0	580.0	128.2	360.1	
(Valve) Rocker-Math Timing Changes:	0.0	0.0	0.0	0.0	Reset
(LOBE) Centerline (Max Lift to TDC):		108.0		116.0	
(LOBE) Cam Advance(+)/Retard(-):		0.0		0.0	

TRUE Cam Timing (Running Values)	IVO	IVC	EVO	EVC
(Valve) Timing @ 0.006-in Valve Lift:	30.8	70.9	77.8	31.2
(LOBE) Timing @ 0.020-in Lobe Lift:	16.0	53.0	64.0	13.0
(LOBE) Timing @ 0.050-in Lobe Lift:	4.0	40.0	51.8	0.1
(LOBE) Camshaft Ramp Dynamics:	▦	▦	▦	▦
(Valve) Acceleration Rates:		2.78		2.79
(Valve) Centerline (0.006-in Timing):		108.0		116.0
(LOBE) Centerline (0.050-in Timing):		108.0		115.8

Other True Timing Measurements

(Valve) Duration @ 0.006-in Valve Lift:	281.8	289.1
(LOBE) Duration @ 0.050-in Lobe Lift:	224.0	231.9
(Valve) Duration @ 0.050-in Valve Lift:	238.0	246.0

Multi-Standard Lift Calculator   Calculate SAE Timing From Other Lobe-Lift Specs

(LOBE) Pri. Duration @ 0.006000 in	----	----	▦
(LOBE) Sec. Duration @ 0.05000 in	----	----	Calc OK
(Valve) Timing @ Alternate PRI Lift:	---- ----	----	----
(LOBE) Timing @ Alternate SEC Lift:	---- ----	----	----
(Valve) Use -> PRI @ 0.006-in Lift:	---- ----	----	-> Use
(LOBE) Use -> SEC @ 0.050-in Lift:	---- ----	----	

The *Camshaft Category* establishes one of the most important specifications in the engine: *Intake And Exhaust Valve-Event Timing* that dictates the beginning and ending of all four engine cycles. The *Camshaft Category* organizes and labels the numerous timing specifications that define cam design and helps you visualize their effect on engine performance. In addition, the simulation includes many features and tools that will help you find optimum cam-timing for just about any engine application.

THE CAMSHAFT CATEGORY

The *Camshaft* component category lets you select and modify one of the single most important components of the engine. For many enthusiasts and even professional engine builders, the subtleties of cam timing defy explanation. And, considering the various "standards" of measurement and pervasive advertising hype, the reason for the confusion is understandable.

The camshaft is often called the "brains" of the engine, directing the beginning and ending of all four engine cycles. Even with a good understanding of engine systems, the interrelatedness of the dynamic processes within the IC engine can make the results of cam timing changes read like a mystery. In many cases there are only two ways to determine the outcome of a modification: 1) run a real dyno test, or 2) run a simulation of the flow dynamics of the engine. Since the camshaft

directly affects several functions at once, e.g., the flow of the exhaust and intake gases, the reverse (or reversion) flow in intake and exhaust passages, cylinder pressures, etc., using a computer engine simulation is the only practical way to keep track of all these variables and accurately predict the outcome.

The *Camshaft Category*: A Quick Overview

The *Camshaft Category* in this Version-6 simulation has several enhancements over previous program versions. The most important changes are: 1) The use of new algorithms (collectively called the *"Fitter"*) that mathematically generate very precise valve lift data used by simulation, and 2) 0.050-inch valve timing (secondary timing points) are used, along with primary (seat-to-seat) timing values, as the "controller" of lifer and valve acceleration rates. If you wish to change the cam/lifter acceleration, you use the same methods as all cam manufacturer's: *by changing the primary and secondary valve timing.* As a result, valve acceleration rate fields are now measured values (will only change when cam timing is changed). The overall result of these changes is that entering and testing camshaft specs better represents real-world camshaft design and testing practices.

The *Camshaft Category* is divided into two functional groups: 1) *Cam Modeling Data-In, From User Input*, and 2) *Cam Modeling Results, Measured During the Simulation Run*. The *Data-In* group contains the input fields you use to specify the desired timing, lift and other values for the current camshaft (or multiple camshafts in VVT...*variable valve timing*...applications). These data-input fields are displayed in Cyan (in the default color scheme). In the *Results* group, the simulation displays the outcome of a mathematical analysis of input data and gives you precise feedback about possible problems or inconsistencies. If the *Data-In* matches the *Data-Out*, the camshaft was precisely modeled (to within 0.5 crank degrees) and this data was used to actuate valve motion within the simulation run. Details on how to get the most from the new *Camshaft Category* features are provided in this section of the User Manual.

Camshaft Category

We will explore all the features available the *Camshaft Category* in this chapter, however, let's begin by looking at basic cam data input and the analysis provided by the new *Fitter* algorithms.

The *Camshaft Category* is, overall, divided into two groups: 1) **Cam Modeling Data In (user input)** located in the upper portion of the category, and 2) **Cam Modeling Results (true timing measured during the simulation run)** shown in the lower part of the category. The *Data In* group displays the desired cam timing, lift and other values for the camshaft (or multiple camshafts in VVT...*variable valve timing*...applications). This camshaft data-inputs are displayed in Cyan (for the default color scheme). The *Results* group displays an analysis of camshaft data measured during the simulation run, including feedback about cam modeling accuracy and possible issues or inconsistencies. If the displayed *Data In* values match the *Modeling Results*, the camshaft was precisely simulated (to within 0.5 crank degrees) and the generated valve-lift curve will exactly pass through the primary and secondary timing points and the point of maximum lift.

The *Results* group shows the actual, measured timing and duration values for the currently simulated camshaft. If you compare the *(Valve)—Primary (0.006-in) Timing* values in the *Modeling Input* group with the measured *(Valve)—Timing @ 0.006-in Valve Lift* in the *Measured Values* group, the values should match (as should the 0.050-inch timing values**)**. If they do, it means that the requested cam timing was exactly simulated. If there are differences, one or more of the **Camshaft Ramp Dynamics** indicators may turn yellow or red (details of cam modeling problem(s)

The camshaft is a round shaft incorporating *cam lobes*. The *base circle diameter* is the smallest diameter of each lobe. *Clearance ramps* form the transition to the *acceleration (or flank) ramps*. The lifter accelerates up the clearance and acceleration ramps and continues to rise as it approaches the *nose*, then slows to a stop as it reaches maximum *lift*, often at the *lobe centerline*. Maximum *lifter rise* is determined by the height of the *toe* over the *heal*. *Valve-open duration* is the number of crankshaft degrees that the valve or lifter is held above a specified height (usually 0.006-, 0.020-, or 0.050-inch). A symmetric lobe has the same opening and closing lift profile.

Cam Lobe Basics
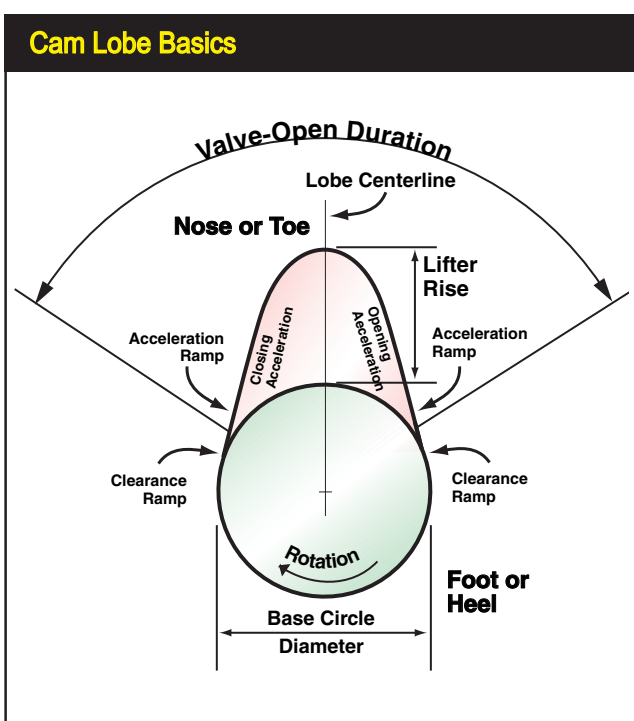
can be found in the *Simulation Log; open the Log from the **Simulation** menu; at the top of the main program screen*). Yellow (or in severe cases, red) indicates that it was not possible for the simulation to produce a lift curve with precisely the valve-event timing that was requested.

Before we delve more deeply into the *Camshaft Category* and cam modeling in the simulation, it's essential that we cover 10-point cam timing vs. lobe-profile timing.

Cam Basics

Most camshafts are steel or iron shafts with eccentric lobes. They are mechanically driven by the crankshaft, typically by a chain or gear train and are usually rotated at one-half crank speed. Lifters (or cam followers)—and for in-block cam locations, pushrods, and rockerarms—translate the rotary motion of the cam into an up-and-down motion that opens and closes the intake and exhaust valves. This entire assembly must function with high precision and high reliability. Street engines driven hundreds-of-thousands of miles operate their valvetrain components *multiple billions of cycles*. And after all that use, if the overall camshaft and valvetrain embody a solid design, a precision measurement will detect only negligible wear!

The camshaft controls valve opening and closing points by the shape and rotational "index" of the lobes (position relative to each other and the crankshaft). Most cams are ground to a precision well within one crankshaft degree, ensuring that valves actuate exactly when intended. However, timing variations of several degrees
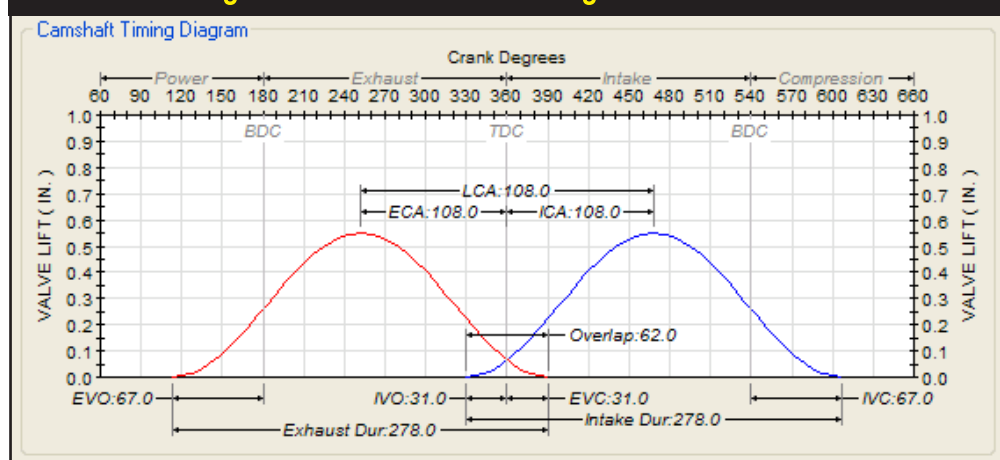
The best way to visualize camshaft timing in a running engine is to picture this "twin-hump" *Valve-Motion* **diagram (drawn "on the fly" in the** *CamManager™***). You can see valve motion for the exhaust lobe on the left and the intake lobe on the right. Also illustrated are the valve-timing points, duration, valve overlap, valve lift, centerlines, lobe center angle, and "ideal" engine-cycle timing, all relative to TDC at the center of the drawing. Study this picture and the text in this section to help you visualize the intricacies of cam-timing specifications.**

Camshaft Category

Long before engine simulations were widely used, cam manufacturers established a methodology for identifying and classifying camshafts. Unfortunately, some of these "catalog" specs can be misleading. But the trend is toward publishing exact valve-event timing for both seat-to-seat and 0.050-inch timing methods (and even at additional lift points, like 0.100 or 0.200). Many cam manufacturers publish timing data for their entire camshaft line on the web. In fact, since printed catalogs are typically updated yearly, web postings tend to be more timely, including their latest cam developments.

often develop in the cam drive, especially in chain-drive systems (racing gear drives typically reduce variations to within two crank degrees or less of indicated timing). Camshaft lobe height (heal-to-toe) and the multiplying ratio of rockerarms (if used) less any valve lash determines how far valves will lift off of the valve seats. The rates at which the valves are accelerated open and then returned to their seats are also "ground into" cam lobe profiles. Only a very specific range of contours will maintain stable valve motion, particularly with high-lift, racing profiles. Unstable profiles or excessive engine speed can force the valvetrain into "valve float" or other instabilities, resulting in component and/or engine failure.

Valve Events
And Event-Timing Standards

There are six "basic" cam timing events ground into cam lobes. These points control the following events:

1—Intake-Valve Opening (IVO)   **2—Intake-Valve Closing (IVC)**
3—Exhaust-Valve Opening (EVO) **4—Exhaust-Valve Closing (EVC)**
5—Intake-Valve Maximum Lift   **6—Exhaust-Valve Maximum Lift**

The first four basic timing points (IVO, IVC, EVO, EVC) pinpoint the "true" beginning and end of the four engine cycles and dictate when the phase of the piston/cylinder mechanism changes from intake to compression, compression to power, power to exhaust, and exhaust back to intake. These six points can be "adjusted" somewhat (we'll discuss which and how events can be altered in the next section), but for the most part they are fixed by the design of the cam.

Several other timing points are often used, but they are always derived from the above, six events. Common derivative events are (discussed more later):

Camshaft Category

7—Intake Duration	**8—Exhaust Duration**
9—Lobe-Center Angle (LCA)	**10—Valve Overlap**
11—Intake-Center Angle (ICA)	**12—Exhaust-Center Angle (ECA)**

Almost all of these valve-timing events can be measured and modeled in several ways. The most common methods are used in the simulation and are described below.

Seat-To-Seat (VALVE) Timing

This valve-event timing method measures the valve events in degrees—relative to piston position—when the valve has only just begun to rise or has *almost* completely returned to the valve seat. While, there is more than one "standard" seat-to-seat measuring method used in the automotive industry, these are the most common timing methods at which both intake and exhaust valve-events are said to begin or end:

0.004-inch LIFTER Rise (SAE Standard)
0.006-inch VALVE Rise (SAE Standard)
0.007-inch open/0.010-close VALVE Rise
0.010-inch VALVE Rise
0.020-inch LIFTER Rise (sometimes used in *solid-lifter* applications)

The valve-event timing measured using these methods is meant to approximate the actual valve "opening-and-closing" points that occur within the running engine. Because of this, seat-to-seat valve events are often called the *advertised* or *running* timing. The simulation uses this event information to fix the beginning and end of

Seat-to-seat timing measures valve-event timing—relative to piston position—when the valve has just begun to open (usually at 0.006-inch valve rise). This is measured during engine assembly by positioning dial indicators on the valve-spring retainers and measuring <u>valve rise</u>, which is the most common definition of seat-to-seat timing. Timing specs measured using these methods are meant to approximate the actual valve opening and closing points that occur within the running engine. Because of this, Seat-To-Seat valve events are often called the *running* (or *advertised*) timing.

Seat-To-Seat Timing Method
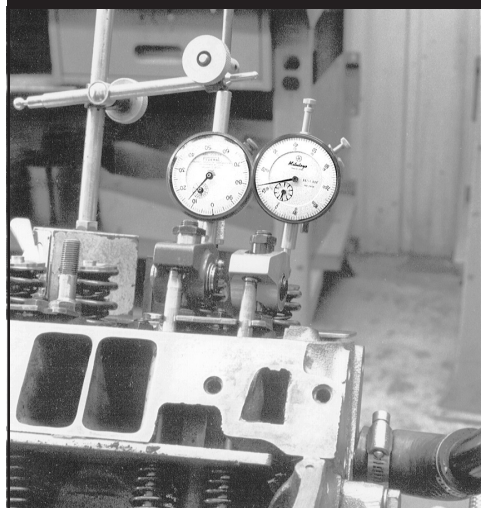
Camshaft Category

0.050-Inch Timing

The *0.050-inch Lifter (LOBE) Rise* valve-event timing measures timing (relative to the crankshaft) when the <u>lifter</u> has risen 0.050-inch off of the base circle of the cam. In the setup pictured here, a dial indicator is positioned on an intake lifter; the 0.050-inch timing point can now be read directly off of the degree wheel attached to the crankshaft. *Note that valve lash and rocker ratio play no part in LOBE Timing measurements.* Lobe-lift timing <u>was not originally meant</u> to approximate the valve opening and closing points, instead its purpose was to permit accurate cam installation. But now it is commonplace to use both Primary and Secondary timing to evaluate the appropriateness of camshaft for any application.

mass flow into and out of the ports and cylinders, a crucial step in the process of determining cylinder pressures and power output.

Note: While there is an accepted seat-to-seat measuring standard (the SAE *0.004-inch Lifter Rise* or *0.006-inch Valve Rise* methods), unfortunately there is also a great deal of variation from this standard among cam manufacturers for "cataloging" or "advertising" purposes. These variations can easily confuse anyone trying to determine actual timing specifications. If you use the seat-to-seat timing specs that fall into any of the categories discussed above, you should obtain accurate results. *Any <u>seat-to-seat timing</u> specifications obtained at substantially more or less than 0.006-inch Valve Rise will not produce ac-*

Primary (0.006-in) and Secondary (0.050-in) Cam Timing Data Entry

Desired Cam Timing (Modeling Input)		IVO	IVC	EVO	EVC	
(Valve)	Primary (0.006-in) Timing:	54.0	84.0	102.0	52.0	
(Valve)	Primary 720-Based Timing:	306.0	624.0	78.0	412.0	
(LOBE)	Secondary (0.050-in) Timing:	37.5	67.5	83.0	33.0	->STS
(LOBE)	Secondary 720-Based Timing:	322.5	607.5	97.0	393.0	
(Valve)	Rocker-Math Timing Changes:	0.0	0.0	0.0	0.0	Reset

The simulation requires both Primary (0.006-inch Valve Lift) and Secondary (0.050-inch Lobe Lift) cam-timing points. These data entry fields are shown cyan, above. Both groups of valve-timing events (and lift, rocker ratio and valve lash) are needed to perform a simulation and calculate the *Measured Timing* values displayed in the lower portion of the Camshaft Category (shown earlier in this section).

Camshaft Category

Extrapolating Seat-To-Seat Event Timing

Pressing the ->STS button in the Camshaft category will extrapolate new/updated Seat-To-Seat valve-event timing from the current 0.050-in Secondary values. Use this feature if you only have access to 0.050-in lobe-lift timing data for a particular camshaft.

curate results when using 10-Point cam timing values (discussed below).

0.050-Inch (LOBE) Timing

This timing method is widely used and has been standardized by cam manu-facturers. 0.050-inch cam timing points are always measured at: **0.050-inch LIFTER Rise** for both the intake and exhaust valves.

This measurement technique was not originally introduced to help evaluate valve opening-and-closing events, valvetrain acceleration, or cam suitability for a particular application. Rather it was intended to be used as an accurate *index* of cam-to-crank position during cam installation. Because of this, it is based on the movement of the cam follower (lifter) rather than the valve, eliminating valve lash and rocker ratio when measuring cam-to-crank timing. Since the follower will be positioned within the cam *acceleration ramp* at 0.050-inch lobe lift, and will have begun rapid movement up the lobe flank, using this point provides good way to verify the precise rotational index of the camshaft relative to the crankshaft. Although 0.050-inch valve timing events do not pinpoint when the intake and exhaust valves open or close, they provide valuable data for the simulation to mathematically model acceleration rates and to generate an accurate valve lift curve.

> **Important Note**: Keep in mind that 0.050 Lobe Lift points (also called Secondary Timing) do NOT change when the rocker ratio or valve lash are altered. These timing points are measured on the cam follower, and only the lobe shape, lifter design, and the rotational-index between the cam and the crankshaft (ICA or ECA) will change Secondary Timing.

If you only have access to 0.050-inch timing data for a particular camshaft, you can still model the valve timing (albeit with somewhat reduced accuracy). Use the **Convert To STS Timing (->STS)** button to have the simulation extrapolate a "best guess" of seat-to-seat timing from the current 0.050-in Secondary values.

> **Note-1**: To optimize accuracy, *whenever possible, use the cam manufacturer's*

Camshaft Category

"10-Point" Cam Data consists of valve-event timing for both the intake and exhaust lobes at both seat-to-seat and 0.050-inch timing points. These eight values, plus the maximum lift of each lobe, produce 10-data points for each camshaft. The simulation uses 10-Point data to generate a valve-motion curve.
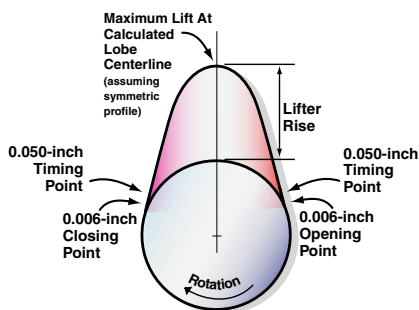
Maximum Lift At Calculated Lobe Centerline (assuming symmetric profile)

Lifter Rise

0.050-inch Timing Point

0.050-inch Timing Point

0.006-inch Closing Point

0.006-inch Opening Point

Rotation

10-Point Lifter Acceleration Is Derived From Seat-To-Seat and 0.050-Inch Opening and Closing Points, And The Maximum Lift Of Each Lobe

published Seat-To-Seat and 0.050-in Timing values as the basis for 10-Point cam simulations.

Note-2: Using the ***->STS*** feature is never needed if you are importing *Lobe Profile* files, since both Seat-To-Seat and 0.050-in Lift points are obtained by direct measurement of the imported profile data. More information about *10-Point* cam modeling and how it differs from *Lobe-Profile* modeling is presented next.

10-Point And Lobe-Profile Cam Data

The simulation can use two distinct methods of modeling cam events and generating a valve-motion curve. They are based on: **1)** *10-Point Cam Data*, or **2)** *Lobe-Profile Data*. Here is a quick overview of these methods and how they are used to complete a simulation:

10-Point Cam-Data (CamFiles) consists of *Valve-Event and Lobe-Event Timing* for both the intake and exhaust lobes. These eight values, plus the maximum lift of each lobe, produce 10-data-points for each camshaft. The simulation uses this 10-Point data to accurately extrapolate the entire valve-motion curve. **Profile Lobe Data (Profile Files)** contain a virtual "blueprint" of cam lobes. While *Profile Files* may differ in their content depending on what measuring apparatus was used to analyze cam lobe shapes, they all contain lobe-lift data recorded at discrete rotational increments, typically at each degree of cam rotation (360 points). This profile data can be used as the basis for a valve-motion curve (only interpolation is needed to determine the exact valve lift for any crank positions "between" recorded values in the profile file). Additional information on using and importing *Lobe Profile Files* is presented later in this chapter.

Camshaft Category

Lobe-ProFile Data

Maximum Lift Point Is Defined To Be Cam Lobe Centerline

Lobe Lift Is Measured At Each Degree Of Cam Rotation

Lifter Fall Determines Valve Duration Closing Point (0.020-inch is common on many solid-lifter cams)

Lifter Rise Determines Valve Duration Opening Point (0.020-inch is common on many solid-lifter cams)

Lobe Profiles Are Directly Measured At Each Degree Of Camshaft Rotation
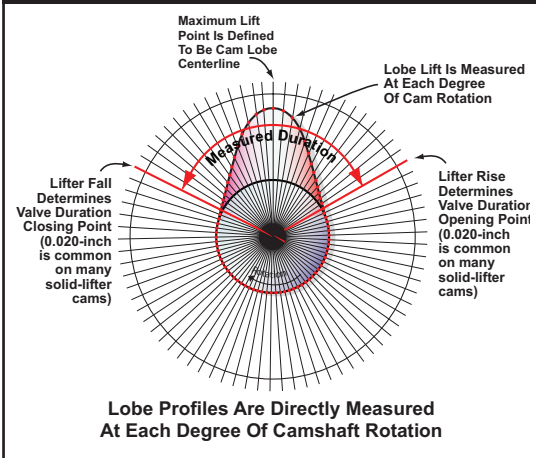
Profile data is made up of lobe-lift data recorded at, usually, each degree of camshaft rotation (by installing the cam in a special fixture designed for this recording process). The simulation can use this data to determine valve positions at any point during the simulation.

You may be asking, "Since the simulation can read Lobe Profile files, why use any other "less precise" cam-data in a simulation"? The answer is: While cam profiles do contain the most accurate valve-motion data for a specific camshaft, using them in an engine simulation has some limitations. When a profile data set is used, modifying individual valve-opening or closing points is next to impossible. For example—just as in the real world—you can advance and retard profile data (for each lobe), change lash, rocker ratio, and that's about the end of your tuning options. If you would like to test different intake timing (IVO, or IVC independently), your only option is to import a completely different profile using new timing values. Since individual valve-event timing is "baked into" profile data, it can't be easily

Profile Data Conversion To 10-Point

When Cam-Profile data is being used on the intake and/or exhaust valve, it is indicated in the *Modeled From* field. If you would like to use modified valve-event timing, profile data must be converted to 10-Point timing using the *Convert To 10-Point (->10-pnt)* button. When the conversion is complete (engine power should be very similar if the conversion process was successful), you can modify any timing point and test its effect on engine output.

changed without "corrupting" the original profile shape. When it comes to profile files, testing various cam-timing combinations requires loading completely different profiles, much the same way you would install a different cam in a real-world dyno test.

The Best Of Both Worlds

10-Point cam data can make cam simulation testing a lot more versatile, especially when you are searching for optimum cam timing for a new application; one for which you may not have done extensive testing or research. 10-Point data sets are just that, 10 data points that the simulation uses to extrapolate a full valve-motion curve (in effect, a new "profile" data set) that establishes valve lift at each crank degree during the simulation process. The 10 points used are: *IVO*, *IVC*, *EVO*, and *EVC*, at both (SAE standard) 0.006-in valve lift and 0.050-in lobe lift points...that's 8 data points...plus the maximum lift for each lobe; that's 10 data points. This data, readily available on manufacturer's "cam cards," cam catalogs and websites, is analyzed by the *Fitter* routines in the simulation to derive lobe and valve lift curves using a sinusoidal dynamics model that exactly passes through all 10 timing points.

Note: The simulation can also model any cam from any manufacturer using any Primary and Secondary lift heights. The **Calculate/Analyze Cam Timing Using Any Lift Height** group at the bottom of the Camshaft Category can convert any duration measured at any lift height to the SAE standard lobe and valve events used in the simulation. Details of this feature are provided at the end of this section of the User Manual (just before the section *The CamManager™*).

Since both Dynomation6 and DynoSim6 use the *Fitter* algorithm "on-the-fly" to derive valve-lift curves from event timing, you are free to alter any timing point and instantly see how it affects engine performance. You can quickly change any valve event, lift, or advance/retard timing and test the effects on engine performance.

CamDisk™ for Dynomation6, included on the Dynomation CD, includes over 6000, 10-Point CamFiles. Using the CamManager™ and the Quick and ProIterator™, Dynomation6 can automatically locate optimum profiles for virtually any application. Install the CamDisk library *AFTER* Dynomation6 has been installed on your computer.

Camshaft Category

Camshaft Type Menu
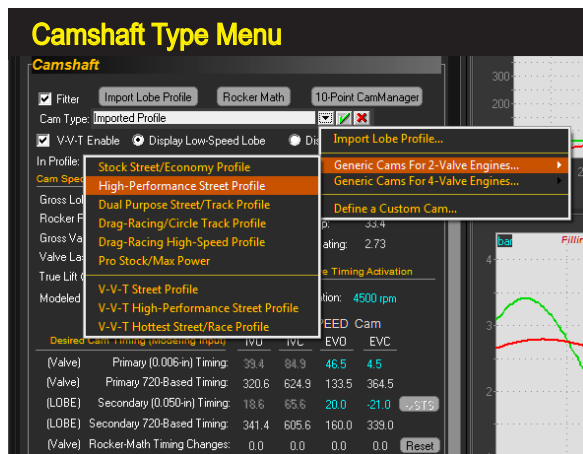
Several "generic" 10-Point cam data-sets are included in the *Cam Type* drop-down menu. They are separated into two groups, cams for 2-valve engines and cams for 4-valve engines. In addition, you can easily input any custom timing and valve-lift specifications.

When you have completed your testing, you can: **1)** Use the *CamManager™* to automatically search through a 10-Point camfile library to locate "real" camshafts with published data that closely matches your test timing, or you can **2)** Find cam-profile data sets that have *event timing* close to the discovered optimum. In both cases, using 10-Point cam timing facilitated finding best cam for your application.

The following several sections discuss how 10-Point cam files are loaded, tested, and manipulated in the *Camshaft Category* and in the *CamManager™*. Importing Lobe Profile Files is also discussed in detail. Even if you were planning on using Lobe Profile data almost exclusively, we suggest that you read the following sections on 10-Point timing. The information may help you improve your ability to test and locate optimum cam combinations for your test engine.

10-Point *Generic Camshaft* Menu Selections

The *Cam Type* menu in the *Camshaft* category contains 18 "generic," camshafts (modeled using 10-Point timing data; see the previous section for an introduction to 10-Point cam timing) separated into two groups: Cams for 2-valve engines and cams for 4-valve engines. Since 2- and 4-valve applications can differ significantly in valve lift, and to some extent duration, the menu arrangement will help you quickly select a starting-point cam on which to base further testing.

When using any of these camshafts, the lift and timing data is loaded into the appropriate fields in the *Camshaft* category. In addition, the **Intake** and **Exhaust Centerlines**, the **Lobe Center Angle**, the **Intake** and **Exhaust Valve Duration**, **Valve Overlap** and other cam-related data is measured by the *Fitter* (after the simulation is has been completed) and displayed in the lower portion of the *Camshaft* category.

A general description of each cam in the **Cam Type** menu is provided in the next section.

Camshaft Category

2- And 4-Valve (Non-VVT) Cam Selections

2- And 4-Valve, Stock Street/Economy Cams

These first cam selections are OEM-replacement street cams for 2- and 4-valve engines. The lifter acceleration rate falls between 2.51 and 2.83 (on a 1.00-to-7.00 scale), which indicates that valve motion for these cams fall in the range of stock OEM to mild street performance applications. Rated valve lift is 0.510/0.510-inch for 2-valve engines and 0.305/0.305-inch for 4-valve engines (intake/exhaust).

The EVO timing maintains combustion pressure late into the power stroke and early IVC minimizes intake flow reversion. Late IVO and early EVC produce 38 degrees of overlap for 2-valve engines and 11-degress for 4-valve engines, enough to harness some scavenging effects but limited enough to prevent severe exhaust gas reversion into the induction system. The characteristics of these cams are smooth idle, good power from 600- to 5000-rpm for 2-valve engines and typically between 1500- and 6500-rpm for 4-valve engines. These generic cams produce good fuel economy and work well in high-torque-demand applications.

2-Valve And 4-Valve High-Performance Street Cams

These profiles are designed to simulate high-performance "street" camshafts for 2- and 4-valve engines. These cams produce lifter acceleration rates between 3.10 and 3.32, indicating that valve motion falls in the range of a typical, mild, street-performance camshaft. Rated valve lift is 0.552 / 0.552-inch for 2-valve

Cam timing events, lift, and more are displayed in the *Camshaft Category* on the Main Program Screen. Cam specifications shown in Light Blue can be changed with a mouse click. Since the engine being modeled in this photo uses a 10-Point cam model, all remaining cam specifications can be modified both in the Cam-Shaft Category and in the CamManager™. Note this cam uses non-variable timing as shown by an unchecked *V-V-T Enable* box.

engines and 0.432 / 0.435-inch for 4-valve engines (intake/exhaust).

This camshaft uses relatively-late EVO to fully utilize combustion pressure and early IVC minimizes intake flow reversion (can be a serious problem on 4-valve engines, since low-lift flow can be substantial when both valves are open and expose large curtain areas). IVO and EVC produces between 54- and 60-degrees of overlap; profiles clearly intended to harness exhaust scavenging effects. The modestly-aggressive overlap allows some exhaust gas reversion into the induction system at lower engine speeds, affecting idle quality and low-speed torque. The characteristics of these cams are fair idle, good power from 1800- to 6000-rpm in 2-valve engines and between 2200- and 7500-rpm in typical 4-valve engines. Good fuel economy is still a by-product of the relatively "mild" cam timing. Both *High Performance Street* cams can be used with higher acceleration rates (typically up to about 4.00) to model more aggressive profiles.

2-Valve And 4-Valve Dual Purpose Street/Track Cams

These profiles simulates high-performance aftermarket camshafts designed for street and mild track applications in 2- and 4-valve valve engines. These cams produce lifter acceleration rates of approximately 3.90, indicating that valve motion falls in the range of a high-performance street and mild racing. Rated valve lift is 0.608 / 0.614-inch for 2-valve engines and 0.472 / 0.457-inch for 4-valve engines (intake/exhaust).

EVO timing on this camshaft is beginning to move away from specs that would be expected for optimum combustion pressure utilization, with more of an emphasis on blowdown and minimizing exhaust-pumping losses. The later IVC attempts to strike a balance between harnessing the ram effects of the

Drag-Racing, High-Speed Cam Modeling
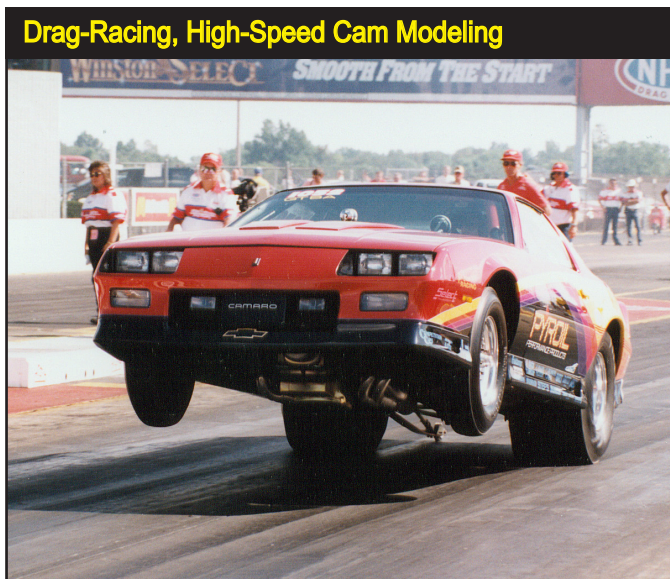
All-out drag-racing cams for 2- and 4-valve engines are designed to optimize power on larger displacement engines at very high engine speeds with large-tube, open headers, and high compression ratios. These camshafts are often not effective in small displacement engines.

induction system while minimizing intake flow reversion. IVO and EVC produce 63- to 65-degrees of overlap, profiles designed to harness exhaust scavenging. The modestly aggressive overlap allows some exhaust gas reversion into the induction system at lower engine speeds, affecting idle quality and low-speed torque (can be a more substantial issue on 4-valve engines, since low-lift flow can be substantial when both valves are off their seats and expose large curtain areas). The characteristics of this cam are lopey idle, good power from 2500- to 6500-rpm in 2-valve engines and between 2500- and 7800-rpm in typical 4-valve engines. These cams develop considerable power at higher engine speeds and are especially effective in lightweight vehicles.

2-Valve And 4-Valve Drag-Race/Circle-Track (And Road-Racing) Cams

These profiles are designed to simulate competition, aftermarket camshafts for 2- and 4-valve engines. These cams produce lifter-acceleration rates of approximately 4.30, indicating that valve motion lies in the range normally used in competition-only engines. Expect somewhat reduced valvetrain life. Rated valve lift is 0.656 / 0.640-inch for 2-valve engines and 0.472 / 0.457-inch for 4-valve engines (intake/exhaust).

EVO timing places less emphasis on utilizing combustion pressure and more emphasis on beginning early blowdown to minimize exhaust-pumping losses. The later IVC attempts to strike a balance between harnessing the ram effects of the induction system while minimizing intake flow reversion. IVO and EVC produce 91-degrees of overlap (60-degrees for 4-valve engines) to harness exhaust scavenging. This aggressive overlap is designed for higher engine speeds with open headers and allows exhaust gas reversion into the induction system at lower rpm, affecting idle quality and torque below 3000- to 4000-rpm. The characteristics of these cams are very lopey idle, good power from 6600 to 8600rpm in 2-valve engines and between 3000- and 8000-rpm

2-Valve ProStock Camshafts

Optimizing power on very-large displacement engines at high engine speeds and with high compression ratios, requires EVO timing that focuses on early blowdown to minimize pumping losses. Late IVC helps harness the full ram effects of the induction system while relying on intake pressure wave tuning to minimize intake-flow reversion.

in typical 4-valve engines, with little consideration for fuel economy. These cams develop substantial power at higher engine speeds and are especially effective in lightweight vehicles.

2-Valve And 4-Valve Drag-Race/High-Speed Cams

These cam profiles simulate aftermarket high-performance competition cam-shafts for 2- and 4-valve engines. These cams produce lifter-acceleration rates between 3.94 to 4.60, valve motion that falls in the range normally used in competition-only engines. These camshaft produce good power while sacrific-ing some valvetrain life (especially considering the 2-valve cam acceleration rates of 4.60). Rated valve lift is 0.692 / 0.692-inch for 2-valve engines and 0.488 / 0.472-inch for 4-valve engines (intake/exhaust).

All timing events on this camshaft are designed to optimize power on larger displacement engines at very high engine speeds with large-tube, open head-ers, and high compression ratios. This camshaft may not be effective in small displacement engines. EVO timing places the utilization of combustion pressure on the "back burner" and focuses emphasis on beginning early blowdown to minimize pumping losses during the exhaust stroke. This technique will help power at very high engine speeds, especially on 2-valve, large-displacement engines that do not easily discharge the high volume of exhaust gasses they produce. The late IVC attempts to harness the full ram effects of the induction system while relying on intake pressure wave tuning to minimize intake-flow reversion. IVO and EVC produce 104-degrees of overlap (71-degrees for 4-valve engines), a profile that is clearly intended to utilize exhaust scavenging effects. This very aggressive overlap seriously affects idle quality and torque below 4500rpm. A 5000rpm stall torque converter is recommended for auto-matic transmission applications. The characteristics of this cam are extremely lopey idle, good power from 4500 to 7200rpm in 2-valve engines and between 5000- and 9500-rpm in typical 4-valve engines, with no consideration for fuel consumption.

2-Valve ProStock/Max-Power Cam (modeled after CompCams 11-728-9)

This profile is designed to simulate an all-out, maximum-power competition camshaft for 2-valve engines (because of the inherent better breathing in a 4-valve design, a cam of this design—an all-out attempt to generate maximum blowdown at the very highest speeds—is not necessary). This cam uses roller-solid lifters and produces an acceleration rate of 5.40, indicating that valve motion falls in the high range normally used in competition-only engines. Valvetrain loads will be substantial, and frequent replacement of valvesprings and other components may be required. Rated valve lift is 0.867-inch for the intake and 0.816-inch for the exhaust.

This ProStock cam is designed for one thing: maximum power at all costs. It is designed to optimize power on very-large displacement engines at very high engine speeds with large-tube, open headers, and very-high compression

Camshaft Category

V-V-T (VTEC) Camshaft Modeling
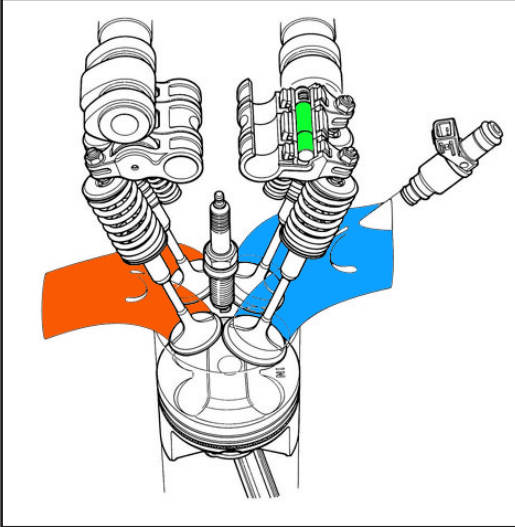
You can model variable valve timing (V-V-T) as used by Honda (in their VTEC system) and engines with similar valvetrain designs. This mechanism uses a discrete low-speed profile, and then at a particular engine speed switches to a high-speed cam profile. Honda uses a simple pin (shown in green) driven with oil pressure (timed with an electronic controller) that locks the outer two low-speed rockers (actuating the valves) to the center rocker that rides on the high-speed lobe. This allows the engine to maintain excellent driveability and fuel economy at low speeds, yet produce power similar to a "race" engine at higher speeds.

ratios. EVO timing focuses on beginning early blowdown to minimize pumping losses, a technique that helps large-displacement engines discharge the high volume of exhaust gasses they produce. Late IVC attempts to harness the full ram effects of the induction system while relying on intake pressure wave tuning to minimize intake-flow reversion. IVO and EVC produce 110 degrees of overlap. This very aggressive overlap basically has no idle quality or torque

Variable Valve Timing, like the VTEC system used on this B16 (1600cc) Honda engine, lets the engine build optimum power yet obtain good driveability and fuel economy (and much lower emissions) at lower engine speeds.

V-V-T Best Of Both Worlds

Camshaft Category

Porsche Boxter VarioCam™
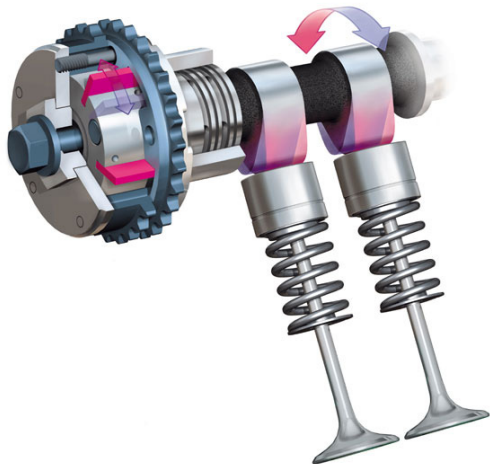
The Porsche Boxter uses variable valve timing technology called VarioCam™. It alters the timing on the intake valves based on engine speed and load. As well as enhancing power and torque, the system offers smoother performance, improved fuel economy and lower emissions.

below 6000rpm. The characteristics of this cam are extremely lopey idle, and awesome power potential from 7000 to 9000rpm.

2- And 4-Valve Variable-Valve-Timing (V-V-T) Cam Modeling

There have been two major engineering "enhancements" incorporated in modern automobile engines that have contributed to new levels of performance and the potential for even more. Getting more air and fuel in the engine has always been the bottom-line for performance, and 4-valve cylinderheads accomplish that goal with aplomb. While certainly not a "new" invention, 4-valve cylinder heads have become commonplace in showroom floors. But that's not the whole picture, for it's the combination of better breathing cylinder heads and Variable-Valve-Timing schemes that have brought power levels to such heights. These technologies have allowed the auto manufacturers to build a "one-two punch" into modern engines: 1) High flow potential with "aggressive," high-speed cam profiles combined with low-restriction ports and valves, and 2) Maintain low emissions and excellent low-rpm power with mild, low-speed cam timing. Variable valve timing lets car manufacturers have what they need—a "mild" engine that meets emissions and driveability requirements—while giving performance enthusiasts what they want—no-compromise valve timing and optimum power.

The Variable-Valve-Timing (V-V-T) model in the simulation functions like the basic cam-control system used on many Honda automobile engines (called *VTEC™*, for *Variable Timing Electronic Control*). At low speeds, the engine operates on *Low-*

Camshaft Category

Speed Lobe profiles designed to minimize emissions and optimize driveability. Then, at some higher engine speed (usually between 4000 to 6000rpm), the valvetrain "switches over" to *High-Speed Lobe* profiles that, primarily, are designed for performance. Without the V-V-T system, Honda and other car manufacturers wouldn't even consider using a cam with the timing specs of the high-speed lobes, since the engine would barely idle, have poor low-speed throttle response and torque, and generate plenty of unwanted emissions. But when that same cam is limited to high-rpm, wide-open-throttle use, power and driveability coexist.

> **Note-1:** With more sophisticated electronic valvetrain systems, the timing of individual intake valves is staggered at low rpm and their lift is asymmetric, which creates charge swirl within the combustion chambers, reducing emissions and improving fuel economy. At higher rpm, when more performance is desired—but before the valvetrain transitions to the High-Speed Lobes—both valves open in unison. Finally, at still higher speeds, the variable-valve-timing computer switches, as described previously, into high-lift, long-duration cam profile program.

> **Note-2:** There are many variations of variable cam timing technologies. The most advanced of these designs can provide virtually "ideal" timing for every driving situation, from idle through full-throttle performance and every step in between.

Whenever you select any of the V-V-T cams from the *Cam Type* menu, V-V-T modeling will be activated, as indicated by a checkmark in the box next to **V-V-T Enable** in the **CAMSHAFT** category. Also, when V-V-T is activated, the box next to the **Variable Valve Timing** field in the *10-Point CamManager™* (discussed later in this chapter) will display a checkmark.

4-Valve V-V-T Mild Street Cam

This profile is designed to simulate conservative OEM camshafts that are used in some 4-valve, V-V-T engines. This cam uses direct-rocker valve actuation (Overhead Cam) and produces an acceleration rate of 2.55 for the Low-Speed Lobe and 2.85 for the High-Speed Lobe, indicating that valve acceleration for these profiles fall well within the range of stock to mild-performance rates. Rated valve lift is 0.325-inch (8.25mm) on the Low-Speed Lobes and 0.420 / 0.416-inch (10.67 / 10.57mm) for the intake and exhaust on the High-Speed Lobes.

This camshaft uses very mild cam timing on the low-speed lobes, with only 15-degrees overlap. This profile is designed, primarily, to minimize emissions and maximize fuel economy up to the activation rpm of the high-speed lobes (typically 5000-to-5500rpm). The high speed lobe is of moderate performance design, with 30% more lift and a 6% increase in valve durations. Valve overlap is increased to 25-degrees, so exhaust scavenging effects will be minimal.

The overall characteristics of this cam are smooth idle, good power from 1000 to 7500rpm, and good part-throttle fuel economy.

Camshaft Category

2- And 4-Valve V-V-T Performance Street Cams

These profiles are designed to simulate mild-performance street camshafts commonly for 2- and 4-valve, V-V-T (VTEC) engines. These cams produce an acceleration rate of 2.83 to 2.41 for the Low-Speed Lobes about 3.15 for the High-Speed Lobes, indicating that valve acceleration for these profiles fall in the range of mild to performance rates. Rated valve lift is 0.510 (2-valve) / 0.335-inch (4-valve) on the Low-Speed Lobes and 0.552 (2-valve) / 0.459 & 0.416-inch (4-valve, intake/exhaust) for the High-Speed Lobes.

This camshaft uses mild cam timing on the low-speed lobes, with only 38-degrees overlap (2-valve cams) and 11-degrees (4-valve cams). These profiles are designed, primarily, to reduce emissions and optimize fuel economy up to the activation rpm of the high-speed lobes (typically 5000-to-5500rpm). The high speed lobe is of moderate performance design. Valve overlap is increased to 60-degrees (2-valve) and 46-degrees (4-valve), a sufficient amount to begin harnessing some exhaust scavenging effects. If the high-speed lobe was used below about 4000rpm, the modestly-aggressive overlap would allow exhaust-gas reversion into the induction system at lower engine speeds, affecting idle quality and low-speed torque. The overall characteristics of this cam are smooth idle, good power from 1000 to 7500rpm, and good part-throttle fuel economy.

2- And 4-Valve V-V-T High-Performance Street Cams

These profiles are designed to simulate high-performance street camshafts for 2- and 4-valve, V-V-T (VTEC) engines. These cams produce acceleration rates of 2.93 to 3.12 for the Low-Speed Lobes and 3.84 to 3.93 for the High-Speed Lobes, indicating that valve acceleration for this profile falls in the range commonly used for high-performance applications. Rated valve lift is 0.552 (2-valve) / 0.354-inch (4-valve) on the Low-Speed Lobes and 0.608

VANOS™ BMW Technology
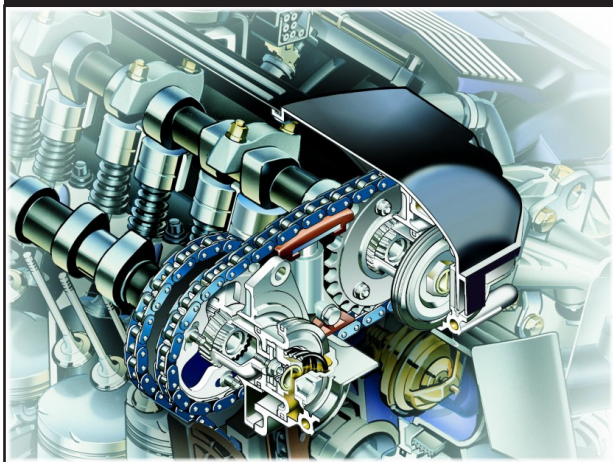
Variable Valve Timing has become a "standard" on most engines. This Double-VANOS system, used on the BMW M3, changes the relationship between cams (alters the cam-centerline angle) to modify valve overlap as engine rpm and engine load change.

Camshaft Category

Activating V-V-T And Displaying High-Speed Lobe

When any of the V-V-T cams have been selected or a V-V-T CamFile has been loaded in the CamManager™, V-V-T modeling will be activated, indicated by a checkmark in the *V-V-T Enable* box in the CAMSHAFT category and in the *Variable Valve Timing* box in the CamManager™. To display High-Speed lobe timing, click the *Display High-Speed Lobe* radio button.

& 0.614-inch (2-valve, intake/exhaust) / 0.472 & 0.457-inch (4-valve, intake/exhaust) for the High-Speed Lobes.

These camshafts use somewhat more aggressive cam timing on the low-speed lobes, with 60-degrees of overlap (2-valve cams) and 29-degrees (4-valve cams). These profiles will begin to harness exhaust scavenging effects even while running the low-speed cam timing. The increased overlap allows some exhaust gas reversion into the induction system at lower engine speeds, slightly affecting idle quality and low-speed torque. The high speed lobes are designed for performance, with 20% to 33% more lift and a 10% to 18% increase in valve durations. Valve overlap is increased to 63-degrees (2-valve cams) and 65-degrees (4-valve cams), modestly-aggressive values that take advantage of free-flowing headers and mufflers (recommended for this cam). The narrow lobe centerline angle of 102-degrees on the 4-valve cam applications should be widened (with adjustable cam sprockets on dual-overhead cam engines) for turbocharged applications and even on some naturally-aspirated engines with low exhaust restriction. The overall characteristics of this cam are a slightly rough idle, good power from 2500 to 8500rpm, and moderate -to-good part-throttle fuel economy.

2- And 4-Valve Max Street/Race Cams

These profiles are designed to simulate maximum performance street camshafts for 2- and 4-valve, V-V-T (VTEC) engines. These cams produce acceleration rates of 2.69 to 3.12 for the Low-Speed Lobes and 4.09 to 4.27 for the High-Speed Lobes, indicating that valve acceleration for this profile falls in the range for high-performance applications. Rated valve lift is 0.552 (2-valve) / 0.380 & 0.370-inch (4-valve, intake/exhaust) on the Low-Speed

Camshaft Category

Lobes and 0.656 & 0.640-inch (2-valve, intake/exhaust) / 0.500 & 0.480-inch (4-valve, intake/exhaust) for the High-Speed Lobes.

These low-speed lobe profiles for 2-valve engines are the same as the previous *High-Performance Street Cam*. The 4-valve camshaft uses relatively mild cam timing on the low-speed lobes, with only 16-degrees overlap. This profile is designed to maintain a good idle and torque at low engine speeds. The high speed lobes are a high-performance design, with an 18% to 31% more lift and 9% to 20% increase in valve durations. Valve overlap is increased to an aggressive 91-degrees (2-valve cams) and 76-degrees (4-valve cams) that takes advantage of free-flowing headers and mufflers (headers, while not required, will significantly improve performance with this cam). The lobe centerline angle (106- to 109-degrees) of the high-speed lobes can be widened somewhat (with adjustable cam sprockets) for turbocharged applications and even on naturally-aspirated engines with very low exhaust restriction. The overall characteristics of this cam are good to slightly-rough idle and excellent power from 1500 to 9000rpm, with good to moderate part-throttle fuel economy.

Note: Each of the previous application-specific cams can be modified by directly entering custom valve-event timing or other cam specifications.

Variable Valve Timing (V-V-T) Activation
And Custom V-V-T Modeling

When you select a V-V-T cam from the **Cam Type** menu in the **CAMSHAFT** category, or you import/load a V-V-T cam, *Variable Valve Timing* modeling will be activated, however, you can activate or deactivate *Variable Valve Timing* at any time. When activated, *Low-Speed Lobe* timing will be used in the simulation until the engine reaches the *HS Lobe Activation* rpm, after which *High-Speed Lobe* timing will be used.

Note: When V-V-T is deactivated, by un-checking the **V-V-T Enable** checkbox, only *"Low-Speed Lobe"* timing will be used in the simulation.

You can also create a V-V-T cam from "scratch" to meet any requirement. For example, if you have found a non-V-V-T camshaft that produces good low-speed performance and another cam that produces good high-speed power, you can "combine" the two cams in the simulation. Here's how: First, import or directly enter the cam timing for the low-speed lobe. When low-speed data has been entered (and all other engine component specs have been entered), a simulation will be performed. Next, activate V-V-T camshaft modeling (by clicking the **V-V-T Enable** checkbox). The *Camshaft* Category indicator may switch to RED, indicating that the category is no longer complete. Now, click on the **Display High-Speed Lobe** radio button and enter the cam timing for the High-Speed Lobe. When complete, the simulation will model the original Low-Speed camshaft up to the **HS Lobe Activation** speed, after which High-Speed timing will be used.

Camshaft Category

True Valve Lift Menu

Use the *True Lift* data field to quickly enter after-lash and after-rocker-ratio maximum valve lift values for the intake and/or exhaust valves. Note: If you perform an entry in this field, the Gross Lash will be re-calculated to maintain the mathematical relationships between the lift, lash, and ratio fields. Use *Auto Calculate Valve Lift* to have the simulation automatically calculate BOTH intake and exhaust valve-lift values when valve lift data is unknown (see the text for more information on using "auto calc" features in the simulation).

Lobe-Lift, Rocker-Ratio, Valve-Lash And Net Valve Lift

The *Intake* and *Exhaust* **True Lift @ Valve** fields display the maximum lift (at the valves) for the currently-selected camshaft, regardless of whether the cam was chosen from the *Cam Type* menu, loaded as a Cam Profile, manually entered as valve-event specs, or loaded from a CamFile using the *10-Point CamManager™*.

The values displayed in the **True Lift @ Valve** fields are mathematically linked with the current *Lash* values and the *Rocker Ratios*. Normally, the *True Lift @ Valve* values are computed from data in the fields directly above: **Gross Lobe Lift**,

The simulation maintains the mathematical relationship between all the Lift fields in the Camshaft Category. If you change the *Lash*, the *True Lift* will be recalculated. If you change the final *True Lift*, the program will assume that the *Gross Lobe Lift* was changed and it will recalculate that value to keep the mathematical relationships "in sync."

Valve Lift, Lash, And Rocker Ratio

Gross Lobe Lift
x Rocker Ratio
Gross Valve Lift
Valve Lash
Net Valve Lift

Camshaft Category

Rocker Ratio, and *Valve Lash*. The calculation begins at **Gross Lobe Lift**. This value is multiplied by the **Rocker Ratio** and the result is displayed in the **Gross Valve Lift** field. This value is then reduced by the **Valve Lash** to yield the final **True Lift @ Valve** for both the intake and exhaust valves. The simulation maintains this mathematical relationship. If you change the *Lash*, the *True Lift* will be recalculated. In fact, if you change the *True Lift* directly, the program will assume that the *Gross Lobe Lift* must have changed, and it will recalculate that value to keep the mathematical relationship "in sync."

> **Rocker-Ratio, Lash, And The *Rocker-Math Calculator*:** *DO NOT* use the Rocker-Math Calculator (discussed in more detail later in this chapter) to enter the cam manufacturer's specifications for Rocker Ratio and Valve Lash. Always enter these specs in the Camshaft Category directly. Then, after the recommended specifications have been entered, *use the Rocker Math Calculator to determine how CHANGES to these stock specifications will affect cam timing, valve lift, and simulation results*. For more information on using the *Rocker-Math Calculator*, see the chapter *Version-6 Built-In Calculators*).

Using the *Net (Sim) True Valve-Lift* Menu

As mentioned previously, the *Direct-Click™* menus for the **True Lift @ Valve** fields allow you to quickly change the maximum valve lift, useful in those cases where you wish to perform "free-form" testing of lift vs. engine output. If you have imported a specific cam, be aware that the original Gross Lift values from your CamFile will be overwritten to maintain the mathematical relationship between the newly entered Net Lift and the existing Rocker Ratio and valve Lash. The **True Lift @ Valve** menus are intended for those times when you aren't concerned about maintaining existing cam specs; you simply wish to enter a specific lift without concern for rocker ratio, lash, or other considerations.

This also applies to the **Auto Calculate Valve Lift** option at the bottom of the **True Lift @ Valve** menu. This function (see photo, below) is useful when you
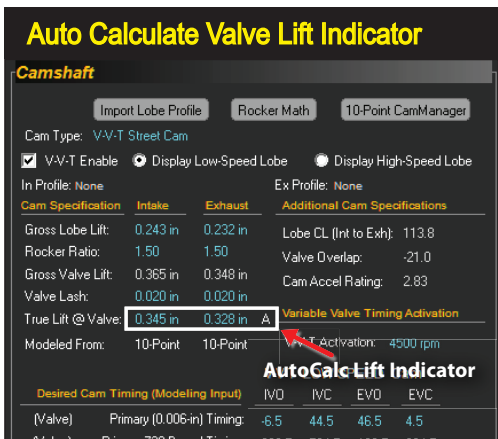
When **Auto Calculate Valve Lift** is enabled, the simulation will automatically calculate BOTH intake and exhaust valve-lift values. The Auto-Calculation feature displays an "A" next to the calculated valve lift values when it is activated. Selecting *Auto Calculate* from the menu, instructs the simulation to automatically calculate appropriate valve lifts values for both the intake and exhaust valves. This is useful to quickly estimate typical cam manufacturer's valve lift for the selected cam timing and application.

Camshaft Category

Rocker-Math Calculator

Follow a simple rule when using Rocker Math: *Enter the original cam manufacturer's specifica-tions for Rocker Ratio in the Camshaft Category FIRST!* **Then use the Rocker Math Calculator to determine how CHANGES to these stock specifications affect cam timing, valve lift, and engine power.**

want to use an appropriate valve-lift based on the current valve diameters. When *Auto Calculate* is enabled, the simulation will automatically calculate the lift for BOTH intake and exhaust valves (*Gross Lobe Lift* may be recalculated to maintain the mathematical relationships with the *Rocker Ratio*, and *Valve Lash*. The *Auto-Calculation* feature can be turned off by re-selecting *Auto Calculate Valve Lifts* ("**A**" indicator and the checkmark in the menu will be removed).

> **Note 1:** *Auto Calculate Valve Lifts* is turned off by default when any CamFile is loaded, since each CamFile represents a "real-world" cam that has specific valve-lift values associated with it. However, you can reactivate *Auto-Calculation* at any time by selecting it from the menus.

> **Note 2:** If **Valve Diameters** are also being automatically calculated in the *Cylinder Head* component category—cylinder-bore diameter and a cylinder-head selection must be made before the program can complete the calculation of valve diameters and, consequently, valve lifts.

An Introduction To The Rocker-Math Calculator

The Camshaft Category includes a **Rocker-Math Calculator** that will help you analyze how and why changes in rocker ratio affect engine output. To use the Calculator appropriately, follow this simple rule: *Enter original cam manufacturer's specifications for Rocker Ratio in the Camshaft Category* **FIRST!** Then use the *Rocker Math Calculator to determine how CHANGES to these specifications will affect cam timing, valve lift, and engine power*. For more information on using the *Rocker-Math Calculator*, see the chapter *Version-6 Built-In Calculators* later in this manual.

Lifter/Valve Acceleration Rates

Camshaft Category

Valve Motion Curves & Acceleration
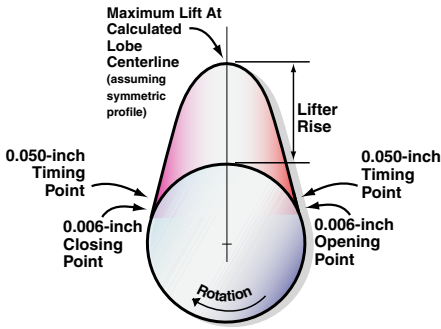
10-Point Lifter Acceleration Is Derived From Seat-To-Seat and 0.050-Inch Opening and Closing Points, And The Maximum Lift Of Each Lobe

The simulation calculates valve acceleration rates based on current valve timing specifications. Valvetrain acceleration values range from 1.00 (very-low acceleration) to 9.00+ (extremely-high acceleration). This data is provided for any Lobe Profile or any 10-Point cam model.

The simulation performs an analysis of the valve motion generated by cam lobe profiles and displays an overall valvetrain acceleration rate. The acceleration values are calculated from an analysis of intake and exhaust opening and closing ramps, flank and nose radii, all timing specifications, and maximum lift data. The values (from 1.0 to 9.0+), displayed in the *Additional Cam Specs* group in the **Camshaft Category**. *Individual* intake and exhaust valve acceleration rates are displayed in

Lifter/Valve Acceleration Display

The simulation automatically calculates values that represent the *Overall Cam Acceleration* and individual *Valve Acceleration Rates* by performing an analysis of valve timing and lobe dimensional specifications. The Intake **and** *Exhaust Valve Acceleration* values are displayed in the *Measured Values* group. The overall *Cam Acceleration Rate* is shown in the *Additional Cam Specifications* fields. The acceleration rates vary from 1.0 to as high as 9.0+ (see the text for more information on these values).

the *Measured Values* group.

The range of acceleration values form a relative scale that offers 900-steps of granularity. The values include acceleration rates used in modern production/street cams (with values ranging from 1 to 3) to racing-only applications (with values from 4 or 5 to as high as 6 to 8+). Using this simple designation, you can quickly determine the general operating characteristics of any camshaft.

Use the following guidelines to evaluate *Acceleration Rates*:

- **1.00   Very Low Acceleration**
- **2.00   Stock Production, OEM Camshafts, Long Valvetrain Life**
- **3.00   Performance OEM and Aftermarket Performance, Good Life**
- **4.00   Mild Racing & Performance, Limited Street Application**
- **5.00   Racing, Valvetrain Loads Require Use Of Optimum Components**
- **6.00   Racing, Reduced Valvetrain Life With Best Components**
- **7.00   High Valvetrain Loads, Requires Regular Valvetrain Maintenance**
- **8.00+ Extreme Acceleration And Loads, Limited Valvetrain Life**

If you wish to evaluate *changes* in valve acceleration, you must alter the shape of the lobes; defined, in part, by the location of Seat-to-Seat and 0.050-inch timing points and the maximum lift of the lobes. Moving 0.050-timing closer to Seat-To-Seat (and/or increasing lift) will mathematically reshape the lobe and increase the acceleration (this requires 10-point cam timing as described earlier). Moving 0.050-timing farther from Seat-To-Seat values will decrease acceleration. All modified timing values are analyzed for stability by the *Fitter* algorithms in the simulation.

The simulation analyzes valvetrain dynamics for any 10-point cam used in the simulation. If the *Fitter* algorithms cannot generate a stable sinusoidal curve that passes through the opening, closing and max-lift points, one or more of the Dynamics Indicators in the Camshaft Category will switch from green to yellow or red (depending on the severity of the mismatch).

Camshaft Category—Other Lift Timing

Values that generate lobe shapes that cannot be properly modeled will trigger one of the **Valvetrain Dynamics Indicators** in the Camshaft Category to switch from green to yellow or red.

> **Note-1:** Select **Show Simulation Log** from the Simulation Menu at the top of the program screen to obtain additional information about any displayed yellow or red valvetrain indicators.

> **Note-2:** In Dynomation-5 & DynoSim5, the acceleration rate could be directly modified. Now, cam/valve timing points directly control the acceleration values for both the intake and exhaust valves. You can still modify the acceleration rates, however, it requires changing valve timing, rocker ratios, and/or maximum lift values.

> **Note-3:** The valvetrain acceleration rates for 4-valve engines often are not as high as those for 2-valve engines designed for similar applications. 4-valve engines benefit from higher low-lift flow (due to the increased curtain area exposed as both valves open), and consequently, cams for these engines do not normally accelerate the valves as rapidly off of the seats.

Modeling 10-Point Cams With Non-SAE Lift-Height Standards

If the cam you would like to model does not use the SAE standard 0.006-inch (Valve) lift height for Primary cam specification and/or the 0.050-inch (LOBE) lift height for Secondary cam specifications, a "standards converter" is provided in the **Multi-Standard Lift Calculator** group (at the bottom of the *Camshaft Category*). It can be used to convert almost any cam timing measured at any lift height to the SAE standard used by the simulation.

This **Multi-Standard Lift Calculator** generates intake and exhaust valve-lift curves that accurately model the original, non-standard camshaft timing. Start off by entering the Primary and Secondary lift-heights (i.e., the Valve and LOBE lift heights used by the cam manufacturer to determine the primary and secondary duration

The Alternate-Lift Height Calculator

[LOBE]	Timing @ 0.020-in Lobe Lift:	33.0	76.0	76.0	30.0
[LOBE]	Timing @ 0.050-in Lobe Lift:	24.0	64.0	64.8	18.2
[LOBE]	Camshaft Ramp Dynamics:				
[Valve]	Acceleration Rates:	3.55		3.32	
[Valve]	Centerline [0.006-in Timing]:	109.5		113.0	
[LOBE]	Centerline [0.050-in Timing]:	110.0		113.3	
	Other True Timing Measurements				
[Valve]	Duration @ 0.006-in Valve Lift:	305.0		304.0	
[LOBE]	Duration @ 0.050-in Lobe Lift:	268.0		263.0	
[Valve]	Duration @ 0.050-in Valve Lift:	281.0		276.0	

Multi-Standard Lift Calculator — Calculate SAE Timing From Other Lobe-Lift Specs

[LOBE]	Pri. Duration @ 0.100 mm	308.0	304.0	
[LOBE]	Sec. Duratoin @ 1.000 mm	274.0	270.0	Calc OK
[Valve]	Timing @ Alternate PRI Lift:	44.0	84.0	85.3	38.7
[LOBE]	Timing @ Alternate SEC Lift:	27.0	67.0	68.3	21.7
[Valve]	Use -> PRI @ 0.006-in Lift:	42.5	82.5	83.8	37.2
[LOBE]	Use -> SEC @ 0.050-in Lift:	24.0	64.0	65.3	18.7

-> Use

The simulation includes a cam timing converter that can model any cam designed at any Primary or Secondary lift height. The *Alternate-Lift Calculator* is located at the bottom of the Camshaft Component Category. This photo illustrates using a cam designed with 0.1-mm primary and 1.0-mm secondary lift heights. The duration values were supplied by the cam manufacturer. The calculator determined the corresponding event timing for SAE lift heights used in the simulation.

Camshaft Category—Other Lift Timing

Use the *Alternate-Lift Calculator* by first entering the *Alternate Primary* and *Secondary Lift Heights* used by the cam manufacturer. Then enter the *Primary* and *Secondary Duration* values. The calculator will determine the valve-event timing for the alternate lift heights and the equivalent 0.006-lift Primary and 0.050-lift Secondary timing used by the engine simulation.

Converting Alternate-Lift Heights to SAE Standards

	Other True Timing Measurements		
(Valve)		**Pri and Sec Lift Height**	
(LOBE)	Duration @ 0.050-in Lobe Lift:	268.0	263.0
(Valve)	Duration @ 0.050-in Valve Lift:	281.0	276.0

Pri and Sec Duration At Alternate Lift Height

Multi-Standard Lift Calculator	Calculate SAE Timing From Other Lobe-Lift Specs				
(LOBE)	Pri. Duration @ 0.100 mm	308.0		304.0	
(LOBE)	Sec. Duratoin @ 1.000 mm	274.0		270.0	Calc OK
(Valve)	Timing @ Alternate PRI Lift:	44.0	84.0	85.3	38.7
(LOBE)	Timing @ Alternate SEC Lift:	27.0	67.0	68.3	21.7
(Valve)	Use -> PRI @ 0.006-in Lift:	42.5	82.5	83.8	37.2
(LOBE)	Use -> SEC @ 0.050-in Lift:	24.0	64.0	65.3	18.7

values). Now enter the specified primary (Valve) and secondary (LOBE) duration values into the calculator. Also enter the intake and exhaust centerline values in the *Camshaft Category*, as these specs are used by the calculator to determine SAE valve-event timing.

The calculator can now generate SAE lift curves using the non-standard lift heights. The net result is a pair of lift curves (intake and exhaust) that model the original cam but use SAE standards for lift heights as required by the engine simulation.

When you are satisfied that the calculator has determined the correct timing specifications (the cam-card specs match those displayed in the calculator), click the *USE* button to transfer the Primary and Secondary timing events to the simulation.

> **Note-1:** The cam specs shown in the *Multi-Standard Lift Calculator* group are only used in the actual simulation *after* you press the *USE* button, transferring them to the *Input Cam Timing (Zero Lash)* fields in the *CamShaft Category*. If any values in the calculator group are changed, they will not affect the simulation until you once again press *USE* to update the engine simulation values.

The following step-by-step guide details how to use the *Multi-Standard Lift Calculator* group:

1) Validate Cam Duration And Timing Values

Find out if the cam manufacturer included lash or rocker ratio in the determination of duration and individual timing events. This practice can vary from manufacturer to manufacturer. Carefully inspect the cam card (and/or contact the manufacturer) to fully understand their measuring techniques.

IMPORTANT: You need to obtain and enter <u>LOBE Duration</u> values; that's equivalent to *valve* duration based on *valve-event timing* without lash or rocker ratios applied.

2) First, Enter Lift and Centerline Values In The Camshaft Category

Enter the *Lobe Lifts* and *Intake and Exhaust Lobe Centerlines* in the appropriate fields of the *Camshaft Category*.

Camshaft Category—Other Lift Timing

Calculated SAE Timing Values

Entered Alt-Lift Timing

Calculated Equivalent SAE Timing

The Primary and Secondary (Valve and LOBE) *Alternate Primary* and *Secondary Lift Timing* values should match the cam manufacturer's specs. The *Calculated SAE Equivalent* timing can be transferred to the simulation by pressing the USE button.

3) Enter the *Primary* and *Secondary* (LOBE) Duration values into the Calculator
Start by entering the *Primary* and *Secondary lift-height specs* from the cam card (or from other data published by the manufacturer) into the "*Primary Duration @*" and "*Secondary Duration @*" fields in the *Multi-Standard Lift Calculator*. Note that these values must be LOBE timing, which does not vary with rocker ratio or valve lash (they are equivalent to VALVE timing with the rocker ratios set to 1.0-to-1.0, and the valve lash set to zero).

5) Verify The Predicted Alternate Timing Specs
Verify that the predicted *(Valve) Primary @ Alternate PRI Lift* and *(LOBE) Secondary @ Alternate SEC Lift* values match the opening and closing timing on your cam card. If the data does not match, make sure you have entered the correct lift-heights, duration values, and lobe-centerline angles.

6) Transfer The Calculated Valve-Lift Curves To The Simulation
If the intake and exhaust event timing at the *alternate lift* heights match the cam manufacturer's data, press the *USE* button to transfer the equivalent SAE lift curves and timing values to the simulation.

Note: If you encounter issues using the *Alternate-Lift Converter*, please contact support@motionsoftware.com and describe the problem in detail. Please attach your engine file (your_engine_filename.dxml) to the email to help our simulation designers duplicate your issue and track down the cause.

Camshaft Category—The CamManager™

The 10-Point Cam Manager™

The **10-Point CamManager™** is a cam manipulation tool for 10-Point cam specs built into the simulation. It offers complete control and visualization of all 10-Point timing and lift values. Use it to enter and modify any cam-related information and search for "real-world" camshafts from your cam libraries that matches any desired range of timing values.

Using The *10-Point CamManager™*

The *CamManager™* is a comprehensive tool built into the simulation that will help you display, test, modify, and even search your cam libraries for optimum 10-Point cam modeling data for any engine application.

Note About Lobe Profile Files: When a *Lobe Profile* file is loaded into the simulation, the *CamManager* is no longer directly accessible. If you attempt to open the *CamManager*, a dialog will let you know that if you proceed, all Profile data used by the current simulation will be converted to 10-Point timing data. The reason for this is that <u>the CamManager is specifically designed as a 10-Point valve-event manipulation tool</u>. Since individual valve-events cannot be changed when Lobe Profile files are used (only centerlines, cam advance and retard, rocker ratio, and lash can be altered; these specifications can all be modified directly from within the *Camshaft* Component category and in the ProIterator) the *CamManager* features are not available when Lobe Profile data is used in the simulation.

Camshaft Category—The CamManager™

The CamManager™ Features And Functional Groups

Tabbed Data-Entry Pages

Variable Valve Timing Lobe Selection/ Activation

Functional Group Boxes

OK/Apply/ Cancel

Open/Save CamFiles

Cam-Timing Diagram

Lifter Acceleration Display

The *CamManager™* incorporates a wide range of functionality and tools that are intuitive and easy-to-use. The *CamManager* is divided into groups that separate functionality into several "boxes" (see text for more information).

The *CamManager* has many features and capabilities. To make this comprehensive tool as easy-to-use as possible, the interface is divided into functional "groups." Each group of controls is positioned within a "titled box" or on a "tabbed data page." For example, when the *CamManager* is first opened (click the **CamManager button** in the **Camshaft** category), you will see the following groups: **1)** *Variable Valve Timing* in the top-left corner, **2)** The *Cam Description* group is located just below it,

When a *Cam Profile* files are used, the CamManager™ is no longer available (works only with 10-point cam files data). If you attempt to open the CamManager, a dialog will let you know that if you proceed, all current Lobe-Profile data will be converted to 10-Point data.

Lobe Profile Data And The CamManager

Camshaft Category—The CamManager™

V-V-T And Description Groups

10-Point CamManager™ - Low Speed Lobe

☑ Variable Valve Timing
  ○ Low Speed Lobe   Activation RPM:
  ○ High Speed Lobe   [5000 ▼]

Cam Description
  File name: [test1.dcm]
  Cam Name: [2-Valve Stock Street/Economy]
  Engine: [Chevy ▼]
  Useage: [Street, Towing & RV ▼]
  Lifter: [Roller Hydraulic ▼]
  Manuf.:   [CompCams]
  Email/Web: [www.compcams.cojm]

The CamManager *Variable-Valve-Timing* and *Cam Description* groups are located in the upper-left of the main dialog box. Use these features to activate V-V-T and set the switch-over rpm.

Extended App Info Group

Extended Application Info
  Recommended Engine Type:
  [350 V8   ▲
                             ▼]
  Cam Application:
  [Excellent Fuel Economy   ▲
                             ▼]

The *Cam Description* and *Extended Info* groups allow you to enter basic information about the current cam. All information in these groups is optional and has no affect on the simulation.

3) Below that is displayed *Extended Application Info* and *Estimated Lifter Acceleration*, **4)** There are three **Tabs** in the top-center of the dialog box that access the *Camshaft Specs*, *Valve-Event Timing*, and *Search For CamFiles* pages, **5)** In the lower portion of the dialog is the *Camshaft Timing Diagram*, that shows all critical valve-timing specifications at a glance, **6)** On the right is a group of buttons that load and save 10-Point CamFiles.

Each of these groups of controls has a specific functionality that is detailed here:

Variable-Valve-Timing Activation/Selection Group (top-left corner in *CamManager*)
The *Variable Valve Timing* (V-V-T) model in the simulation, as described earlier, functions like the (dual lobe) VTEC system used in some Honda engines. At low speeds, the engine operates on the *Low-Speed Lobe*. Then, at some higher engine speed (selectable from 2500 to 8000rpm), the valvetrain switches over to the *High-Speed Lobe*. Activate V-V-T by placing a checkmark in the box next to the **Variable Valve Timing** field; this group also contains the high-speed lobe **Activation RPM** field (both of these controls are duplicated in the **CAMSHAFT** category).

Cam Description Group (upper-left in *CamManager*)
Basic information about the current cam is displayed in this group (most of this information is obtained from cam manufacturer's catalog listings; you can add/change or modify it at any time). *The information in this group is stored with the cam, but is not used in the simulation.* The **Filename** field displays the name of the currently loaded 10-Point CamFile. The **Cam Name** provides a short description of the cam. The **Engine** field indicates the engine family for which the camshaft was designed. The **Usage** field indicates the intended

application. The **Lifter** field displays the lifter technology used with this cam. The **Manufacturer** field displays the manufacturer/designer of the cam. And the **Email/Web** field provides contact info. The **Engine**, **Usage**, and **Lifter** fields have suggested entries available in drop-down menus. And although you can skip entering any information into these fields, we recommend that you complete them since this data can help locate the camshaft when performing CamFile searches, described later.

About Entered Text And Special Characters: If you paste data into any of the data fields in the Cam Manager (e.g., using the Ctrl-C and Ctrl-V cut-and-paste functions), the simulation will examine the text for special (non-standard-text) characters. If any are found, they will be replaced with spaces. If this is not done, Camfiles or Engine files may not open properly.

Extended Application Info (left-center in CamManager dialog)

Additional operational characteristics of the cam and engine can be included in this group. The **Recommended Engine Type** field typically displays the engines for which the cam was designed. The **Cam Application** field can be used to describe cam specs, operation, related requirements, and performance characteristics. The information in this group was usually obtained from cam manufacturer's catalogs and other published data sources. You can edit, modify, or add information to either of these fields.

Note: All data in the *Cam Description* and *Extended Application Info* groups is optional and has no affect simulation output; it is provided for information and documentation only.

Camshaft Valve-Lift Timing Diagram (lower-center in *CamManager* dialog)

This graph, sometimes called a "twin-hump" diagram, shows the motion curves for the intake and exhaust valves (Note: Only the portion of crank rotation during which the intake and exhaust valves are open is shown, 120-degrees on this diagram). Exhaust valve lift is on the left, in red; intake lift is on the right, in blue. The lobe-timing points (IVO, IVC, EVO, and EVC), Overlap, Duration,

The "Twin-Hump" valve motion diagram in the *CamManager* is dynamically updated anytime timing specifications are changed (see text).

Dynamic "Twin-Hump" Camshaft Timing Diagram
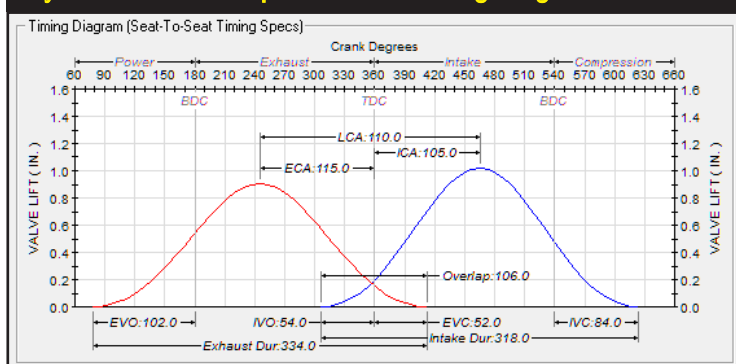
Camshaft Category—The CamManager™

Estimated Lifter Acceleration Rate
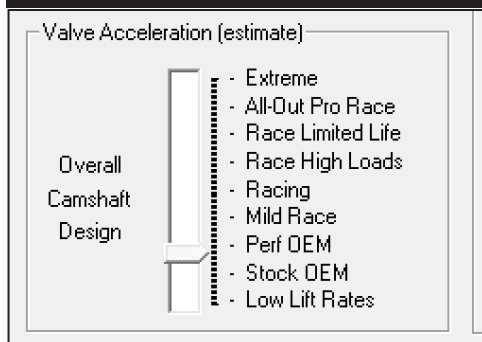
When all 10-Point timing values have been entered, the Valve *Acceleration Rate* will be calculated and displayed. Changes in the shape of valve-motion curves (by changing the valve timing points or the maximum lift for both the intake and exhaust valves) will be reflected in the displayed acceleration value. A more precise numerical value for Valve Acceleration is displayed in the CamShaft Category on the Main Program Screen.

Lobe Centerlines (ECA and ICA), and the Lobe-Center Angle (LCA) are all dynamically drawn on the graph. In addition, maximum valve lift is illustrated by the height of the curves. The graph is always up to date; reflecting any changes in cam timing specifications.

Estimated Valve Acceleration (lower-left in CamManager dialog)

The simulation computes an overall valve acceleration rate and displays a typical application for that level of acceleration. The acceleration is directly derived from the "shape" of the valve-motion curves. In order to calculate Valve Acceleration, the simulation requires both the *Seat-To-Seat* and *0.050-inch Valve-Event Timing* points, plus the *Maximum Valve Lift* for the intake and exhaust valves. If all of this (10-Point) valve-event data is available, the overall **Valve Acceleration Rate** will be calculated and displayed. If the simulation does not have sufficient data to perform the calculation, "Undefined" will be displayed in this group.

Note: A more precise numerical value for Valve Acceleration (both combined and for individual valves) is displayed in the CamShaft Category on the Main Program Screen.

Camshaft Specs Tabbed Data Page

This tabbed page displays Intake and Exhaust duration and related cam/lobe values. Shown are *Seat-To-Seat and 0.050-inch Duration* and *Centerlines*. Valve Lift data (lash and rocker ratio) are entered in the Camshaft Category on the main program screen (they are not pure cam specs, but are modified by the valvetrain).

Camshaft Category—The CamManager™

Tabbed Data Pages (upper-center of CamManager dialog)

The three following data-entry-and-display pages are available as tabbed windows in the *CamManager*.

Camshaft Specs Tabbed Page

The first tabbed data page shows "published" cam specs that are typically provided by cam manufacturer's in their catalogs or web listings. Included are *Duration*, *Intake & Exhaust Centerline, Lobe CenterAngle, Overlap*, and *Valve Lift*. When this data has been entered (except for Valve Lift, *Overlap* and *Exhaust Centerline*, which are either automatically calculated or must be entered in the CamShaft Category, see the Note, below), the CamManager will calculate and update valve motion data shown on the *Camshaft Timing Diagram*. The motion curves illustrate seat-to-seat valve motion curves, however, you can click the *0.050-inch Timing* radio button located just above the graph, and the valve motion curves will be redrawn to display <u>Lobe/Tappet</u> timing specs.

Note: The *Exhaust Centerline* field in the CamManager is a calculated, display-only field. If this seems odd, consider that it is a directly calculated from the Intake Centerline and Lobe CenterAngle. If you *could* directly change the *Exhaust Centerline*, the program would have to change either the Lobe CenterAngle and/or the Intake Centerline to keep those values in mathematical sync. The question is: *what field should be changed? Which one should the program override?* Because of this dilemma, only the Intake Centerline and Lobe CenterAngle are directly changeable. As you view the lift curve graph at the bottom of the CamManager screen, you'll notice that when you change the *Lobe CenterAngle* or the *Intake Centerline*, the *Exhaust Centerline* changes to make the math add up. These three variables are all mathematically linked; changing one must change at least one of the other two.

Valve-Event Timing Tabbed Page

The second tabbed data-entry page displays valve-event timing for both seat-to-seat and 0.050-inch timing specs. Included are *IVO, IVC, EVO, EVC*

The center tabbed page shows *Lobe (Cam) Event* **timing in two columns: on the left are IVO, IVC, EVO, and EVC for Seat-to-Seat timing and 0.050-inch event timing is shown on the right.**

Valve-Event Timing Tabbed Data Page

Camshaft Specs	Valve Event Timing	Search for Cam File

STS Valve Timing (@ 0.006-in lift, zero lash)

Intake Opens (IVO, BTDC):	30.8
Intake Closes (IVC, ABDC):	70.9
Exhaust Opens (EVO, BBDC):	77.8
Exhaust Closes (EVC, ATDC):	31.2

0.050-Inch Lobe-Lift Timing

Intake Opens (IVO, BTDC):	4.0
Intake Closes (IVC, ABDC):	40.0
Exhaust Opens (EVO, BBDC):	51.8
Exhaust Closes (EVC, ATDC):	0.1

Gross Lift At Valve (Including Rocker Ratio And Lash)

Int True Lift (Set In Cam Category)	0.5568	Int True Lift (Set In Cam Category)	0.5664

and ***Maximum Valve Lifts (which must be entered in the main CamShaft Category)***. When this data is complete, the simulation will (re)calculate the Camshaft Specs and use the new data in the simulation. <u>Both</u> seat-to-seat <u>and</u> 0.050-inch *valve-event* values must be entered (completing all data fields on this tabbed page), before a simulation can be performed.

Note: The fields shown in the *Valve-Event Tabbed Page* constitute the 10 data values defined in 10-Point cam data.

Search For CamFiles Tabbed Page

The third tabbed page provides considerable versatility in locating 10-Point CamFiles for your test engine (this **Search** function works only with 10-point camfiles; loading and using Lobe-Profile Files is discussed in the next section). The *Search Tab* lets you automatically look through thousands of CamFiles and locate only those that meet your criterion. For example, you can find all the cams designed for a Honda, or locate all cams that closely match the specifications discovered in an ***Iterator™*** test series (more on the *Iterators* later). If you would like to search for specific filenames or cam descriptions, first enter any search terms into the ***Criteria*** fields. If you would like to locate CamFiles that fall within a range of timing values centered around the current camshaft timing (the current cam is the cam currently used in the simulated engine), check the box labeled ***Find The Following Specs***. Finally, click the **Search** button to perform a search of all CamFiles found in the folder listed in the ***Look In*** field *and in any folders that are nested below that folder* (a full *recursive* search for all .DCM, .CAM, and .SCM files is performed). When a list of matching CamFiles is presented, simply click on any file to view its characteristics (use the up-and-down arrow keys to quickly move through the results list). Transfer any CamFile into the CAMSHAFT component category and into the simulated engine by clicking **Apply** ("installs" the cam and leaves the *CamManager* open) or **OK** ("installs" the cam and closes the *CamManager).*

Search For CamFiles Tabbed Data Page

The *Search For Cam-Files* tabbed page provides a way to rapidly locate 10-Point Cam-Files. You can search through thousands of CamFiles and locate only those that meet your criterion.

Camshaft Category—The CamManager™

Note: To extend the search capabilities of the CamManager, a *10-Point Cam Data Library* is included with all Version 6 simulations. You can install the additional library files into the normal directory structure of the simulation (the installer is provided in the program installer package), adding more than 6000 CamFiles to those already available to the *CamManager*.

CamManager Button Functions

The standard **OK**, **Cancel**, and **Apply** buttons transfer (or Cancel) any changes made in the CamManager. The **Clear** button erases all data in the CamManager in preparation for entering all specs from "scratch." Finally, the **Open** and **Save** buttons are used to load and save cam files (using **Apply** updates the simulation with the current cam specs and leaves the *CamManager* open).

Important Note: Keep in mind that if you enter or change cam specifications within the *CamManager after* saving or retrieving a CamFile, the *changed cam data* WILL be saved in the engine (.DXML) file *but NOT in a separate CamFile* unless you specifically *Save* the modified cam data to a CamFile on your computer.

The CamManager uses the standard *OK, Cancel, & Clear* buttons. *Open* and *Save* loads and saves 10-Point Camfiles.

Loading 10-Point CamFiles

The *CamManager* is the "central clearing house" through which you can load, import, save, and search for 10-Point CamFiles. Here is a quick overview of how to load and save various CamFiles:

Opening .DCM, SCM, and .CAM Camfiles (Motion Software CamFiles)

The CamManager will *Open* camfiles that were either saved by Dynomation6 or DynoSim6 (.**DCM** file) or other Motion Software engine simulations. To open any of these files, simply click the **Open** button in the CamManager and select the desired file type from the *File Open* dialog box. Here are the 10-point camfiles compatible with this simulation:

10-Point Cam-Timing Files	File Extension
Dynomation-5 & -6 CamFiles	.DCM
DynoSim6 CamFiles	.DCM
DynoSim-4 & -5 CamFiles	.CAM
SC-DynoSim CamFiles	.SCM

Camshaft Category—The CamManager™

To import any of these camfiles, open the *CamManager*, click the **Open** button, select the desired file extension from the *Files Of Type* drop-down list, then locate the file you wish to import and click **OK**. When the dialog box closes, press the **Apply** or **OK** button in the *CamManager* to transfer the cam data into the simulation.

Saving CamFiles

Open the *CamManager*, click the **Save** button, locate the **CamFile (.DCM)** folder (or one of its subfolders), then save the current CamFile to your computer.

Important Note: Keep in mind that if you enter or change any cam specifications either inside or outside the *CamManager*, the changed cam **data** WILL be saved with the engine (.DXML) file *but WILL NOT be saved in a separate CamFile* unless you specifically **Save** the modified cam data from within the *CamManager*.

Camshaft Category—The CamManager™

Camshaft Category—Using Lobe Profiles

Importing And Using Lobe-Profile Files

Version6 engine simulations can Open/Import a variety of cam-data files. These files are divided into two main groups: **1)** *10-Point CamFiles*, and **2)** *Lobe-Profile Data Files*. 10-Point cam data is primarily handled through the *CamManager* (see previous section). This section discusses importing and using Lobe-Profile data files in your engine testing program.

> **Note:** For more information on the differences between 10-Point and Profile data, refer to **10-Point And Lobe-Profile Cam Data** earlier in this chapter.

Lobe-Profile data is virtual "snapshot" of a single cam lobe (separate data-sets commonly are used for intake and exhaust lobes; exceptions are discussed below. Lobe-lift data is recorded at each degree of camshaft rotation. This is accomplished by installing the cam in a special lathe-type fixture designed to measure lifter rise as the cam is rotated on centers. Audie Technology, and several other companies, manufacture these devices. Profile files vary in format, but most contain 360 data points. Some profile files also contain data-sets for more than one lobe, centerline data, application info, etc.

This simulation can *Import* several common Lobe-Profile file formats.

Supported Cam-Profile Files	Common File Extension
CamPro	.P1 and .P
CamProPlus	.CPP
Cam Doctor	.CXX (XX = lobe ID numbers)
S96	.S96

CamPro and CamProPlus (CPP) Profile Files

CamPro files were developed by Audie Technology, Inc. The file extensions for *CamPro* files are **.Px** or **.Pxx**, where **x** or **xx** is usually the cylinder num-

Lobe-ProFile Data

Maximum Lift Point Is Defined To Be Cam Lobe Centerline

Lobe Lift Is Measured At Each Degree Of Cam Rotation

Measured Duration

Lifter Fall Determines Valve Duration Closing Point (0.020-inch is common on many solid-lifter cams)

Lifter Rise Determines Valve Duration Opening Point (0.020-inch is common on many solid-lifter cams)

Lobe Profiles Are Directly Measured At Each Degree Of Camshaft Rotation

Lobe-Profile data is made up of lobe-lift data recorded for the intake and/or exhaust lobes (by installing the cam in a special fixture designed to measure and record this data). The simulation can individually import intake and exhaust data in several common lobe-profile formats.

ber of the cam lobes. *CamProPlus* files have a **.CPP** extension. These files contain both numeric and binary (boolean) data. All text-based data is ASCII format. Each CamProPlus file can contain data for one or more lobes. The first few bytes in the file hold a special identifying code and a revision letter. This is followed by a header section with information on the test setup used to measure the lobes and other details. The file next contains a table with text-based, descriptive entries for each lobe (the table may contain "extra" unused entries). The actual lobe measurements follow the table.

Cam Doctor Profile Files

Profile files are ASCII format (text-based). The file extension is **.Cx** or **.Cxx**, where **x** or **xx** is the cylinder number of the cam lobes contained in that file. Each file typically encapsulates data for one intake and one exhaust lobe. After several lines of header information, the lobe-lift data encompasses the next 360 lines (lift values are recorded for each degree of cam rotation). Each data line provides two values: the intake and exhaust valve lift, just as they were read from the sensors used on the measurement fixture. The centerline of the cam lobe can fall at any location in the data set, since data recording begins when the rotary-encoder transmits an index pulse and the position of this timing pulse depends upon how the encoder was attached to the end of the camshaft. The data (numbers) found in the header lines can be used to locate the centerline, but it is a complicated process. Instead, most users simply set the centerline after the data file is processed.

The S96 Profile Files

This file format was invented by Harvey Crane (it is similar to an earlier **.396** file format, but with "smoothed" data). It is ASCII encoded (text based) with one lift value per line, and data is spaced in 1-cam-degree increments. It contains 396 lines of data; 360 points plus 36 additional points of overlapping

CamProPlus Measuring Tool From Audie Technology
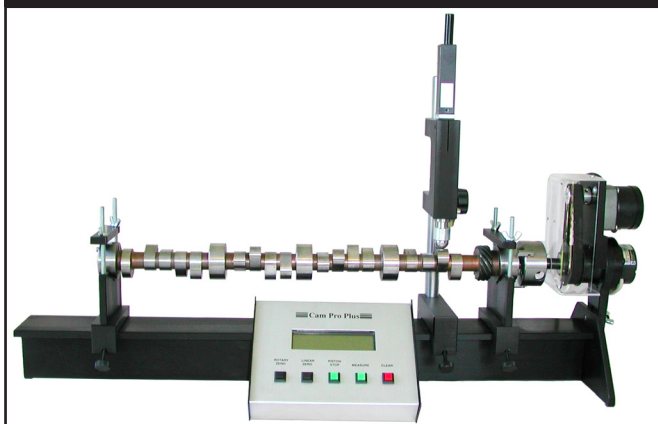
Cam Pro Plus is a software and hardware package for measuring and analyzing lobe- and valve-motion profiles. Developed by Audie Technology (www.audietech.com). it can measure cams using this test stand or when they are installed in the engine. Several software packages are available for advanced cam-motion analysis.

Camshaft Category—Using Lobe Profiles
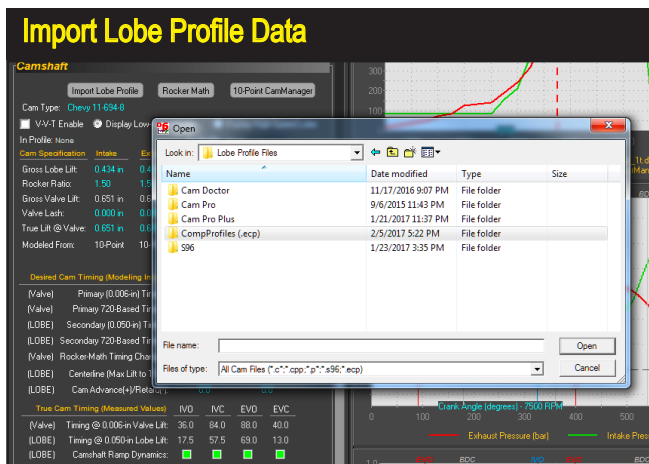
The simulation can Import many of common Profile file formats, including CamPro, CamProPlus, Cam Doctor, CompCams ECP, and S96. Press the *Import Lobe Profile* button in the Camshaft category to open the file-selection dialog shown here. This dialog lists several lobe profile groups that are supplied with Dynomation6 (included for testing and familiarization with profile-file usage).

data. Harvey included this redundant data as a quality check for his measuring process. Peak lift is always located in the middle of the data set. Two common variations of S96 exist: 1) With peak lift on line 180 (the overlapping points are located at the end of the file), and 2) With peak lift on line 198 (with the overlap data split between the start and the end of the file).

The .ECP Profile Files

CompCams lobe-profile format was developed by Motion Software, Inc. The lobe-profile data in these files was provided by CompCams, Inc, and (as of 2018) includes a large portion of the CompCams catalog. The raw profile data was processed and saved in .ECP files (Encrypted Comp Profile) in a protected format that prevents direct lobe copying and duplication. ECP files are only readable by Dynomation. The format protection does not affect file loading or accuracy when using these files in the simulation.

Note: The simulation automatically handles the subtitles of reading and interpreting Lobe-Profile file formats; refer to the following sections for help in using profile data in your simulations.

How To Import Lobe-Profile Data

Press the **Import Lobe Profile** button in the *Camshaft Category* to begin the import process. When the file-open dialog appears, select the desired sub-folder and file (review Lobe Profile Formats described in the previous section), then click *OK*. A *Lobe Profile Import* dialog will open to help you assign lobe data to a specific valve, set centerlines, and make other determinations about the data obtained from the file.

Here are the key features and data-entry groups in the *Cam Profile Import* dialog:

Camshaft Category—Using Lobe Profiles

Import Lobe Profile Dialog

After you have selected a lobe profile file, this dialog will open and help you verify the data and assign the lobe to any of four possible destinations: 1-Intake, 2-Exhaust, 3-Intake of the VVT High-Speed Cam and 4-Exhaust of the VVT High-Speed Cam.

Lobe Source-Data List

A list of the lobes in the selected file are displayed in the top-left *Source Data* list. Some file types (like .S96) will have one or two lobes, while other files (like CamProPlus files) may have several profiles included in the file. Click on any lobe described in the Source List to apply its data to the other fields in the import dialog.

Description

When any lobe is selected from the Source List, information about the profile data may be displayed in the Description box. Many profile files will only have simple lobe identifications, like "Intake," "Exhaust," or other brief descriptions of the selected lobe, while other files may offer more detailed information.

Lobe Lift and Duration

Since lobe-profile data only consists of lobe-lift values recorded at each cam degree, lobe duration must be calculated from lift data (keep in mind that we are referring to **Lobe Duration** not <u>Valve</u> *Duration*; to determine valve duration, multiply the lobe lift values by the rocker ratio and then subtract any valve lash). Lobe duration is determined by the lobe lift height selected for the beginning and ending points. Since the SAE standard starting-lift value is 0.006-inch, and since the most common rocker ratio is 1.5:1, the lobe lift for the same duration would be .006 / 1.5, or 0.004-inch. However, many cam manufacturer's use their own values to rate lobe duration. Common values are 0.006-inch, .0068-inch, 0.020-inch, 0.050-inch, 0.058-inch, and even 0.100

and 0.200-inch are used. The simulation displays the lobe duration for many of these lobe lift points. In addition, you can enter any lift value in the **Enter Custom Lift** field and the corresponding duration will be calculated.

Note: Once lobe-lift data is imported, the _valve_ seat-to-seat duration displayed in the Camshaft Component category is determined using 0.006-inch valve lift points (includes rocker ratio _and_ lash), On the other hand, 0.050-lift _lobe_ duration is determined from the 0.050-inch lobe/tappet lift points and does not include rocker ratio or lash.

Lobe-Profile Rendering And Base-Circle Diameter

The _Lobe Profile Import_ dialog also includes an illustration of the lobe selected in the Source List that is rendered from actual lobe-lift data. Use this illustration to make sure there are no missing points or interruptions in the lift curve; if you see any discontinuities, the profile data in the file is invalid.

Note-1: Since profile data only indicates how far the lifter rises _above_ the base circle of the cam (not the actual base-circle diameter), the simulation has to make a calculated "guess" of the base-circle size to render the profile. If the lobe-to-base-circle size relationship looks odd, you can enter a different base-circle diameter that may improve the "look." This display is only provided as a check for lift-data completeness; base-circle size does not affect the data used in the engine simulation.

Note-2: The arrow shown in the cam rendering indicates a normal clockwise rotation (cam rotation is always assumed to be clockwise); the opening flank is on the right side of the illustration and the closing ramp is on the left.

Lobe-Data Destination
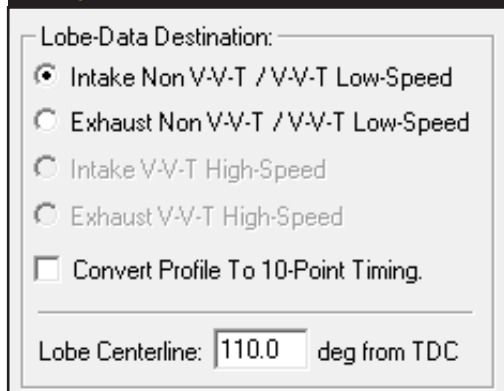
The profile data associated with lobes selected in the _Lobe Source-Data List_ are assigned to either an intake or exhaust valve (and in the case of V-V-T cam modeling, to the high-speed Intake or Exhaust valve) using the _Lobe-Data Destination_ group. This data-entry area contains either two or four radio but-

Invalid Lobe-Profile Data

Base Circle Diameter: 1.606 in
(Use To Verify Continuity Of Lift Data)
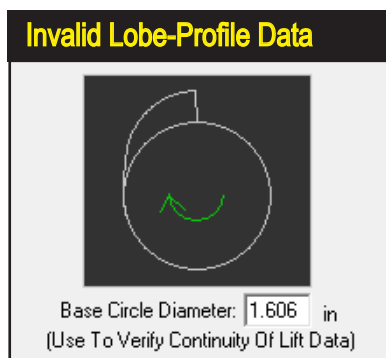
The cam-lobe image (rendered from lift data in the profile file) can confirm the validity of the data file and help you get a general "feel" for the shape of the lobe. The lift curve shown on the left is missing the opening ramp and is invalid.

Camshaft Category—Using Lobe Profiles

Lobe-Data Destination:
- ⦿ Intake Non V-V-T / V-V-T Low-Speed
- ○ Exhaust Non V-V-T / V-V-T Low-Speed
- ○ Intake V-V-T High-Speed
- ○ Exhaust V-V-T High-Speed
- ☐ Convert Profile To 10-Point Timing.

Lobe Centerline: | 110.0 | deg from TDC

The lobe profile selected in the *Lobe Source-Data List* can be assigned to either an intake or exhaust valve (and in the case of V-V-T cam modeling, to either of the high-speed cam lobes) using the *Lobe-Data Destination* group. The Centerline for the lobe data can also entered in this group. With a known centerline, the simulation is able to calculate Seat-To-Seat Valve- and 0.050 Lobe-Event timing (the simulation will set a default centerline value of 110.0 degrees whenever the *Lobe Profile Dialog* is opened).

tons, depending on whether *Variable Valve Timing* is currently active. Notice that the radio buttons allow the selection of only one destination.

The procedure here is to select a lobe profile, assign it to a specific intake or exhaust valve, then close the Import Dialog to complete the import for one lobe. Reopen the Import dialog to import additional lobe profiles.

Convert Wave-Action Profile Data To 10-Point Timing

Lobe-profile data can be "handed over" to the simulation in two ways: **1)** You can directly import the "raw" valve-lift data into the *Wave-Action* simulation (this is the default method). When this is done, *Profile* data will be used "as-is" to determine valve motion, or **2)** By checking the **Convert Profile Data To 10-Point Timing**, you can force the conversion of *Profile* lobe-lift data into *10-Point* timing values.

Note: You can still convert the lobe-profile lift data to 10-Point modeling at any time buy simply clicking the "**->10-pnt**" (Convert To 10-Point) buttons in the Camshaft category. Remember that profiles files are "fixed," while *10-Point* timing can be edited or modified in any way (you can modify valve-event timing, lift, etc.).

Lobe Centerline (in crank degrees)

Lobe-lift profile data must be "synchronized" with crankshaft and piston movement to be useful. *Lobe Centerline* is the data point that fixes the rotational index, in crank degrees, from the center of the lobe-profile data (point of maximum lift) to Top-Dead-Center piston position. You can enter *Lobe Centerline* data in the Import Dialog or you can enter (or modify) this value within the *Camshaft Category*.

Lobe-Lift And Acceleration Graph

To help you visualize the profile data selected from the Source List, lift and

Choosing Lobe Centerline

Lobe Profile And Centerline Measurement

Intake Centerline (106) Crank Degrees Between Top Dead Center And Lobe Centerline
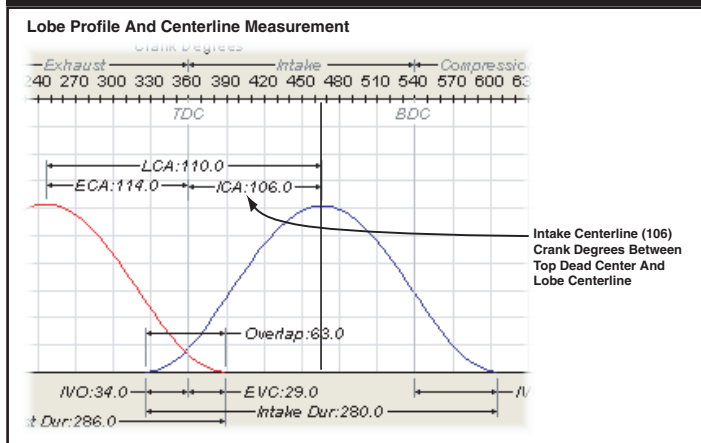
A lobe-lift profile must be "synchronized" with crankshaft and piston movement to be useful. The Lobe Centerline Angle (shown here as an LCA of 106 degrees) is the distance, in crank degrees, from the center of the lobe-profile data (in this case, the point of maximum intake-valve lift) to the Top-Dead-Center piston position.

acceleration curves are displayed in the *Lobe Lift and Acceleration Graph*. Keep an eye out for rough or bumpy curves that can indicate poor recorded data quality during cam lobe measurements.

Press the **OK** button in the *Import Dialog* to transfer the cam file data into the **CAMSHAFT** Category and update the simulation with the new cam specs..

Tuning And Modifying Profile Cam Data

Profile files, as mentioned, consist of hundreds of data points that define the position of the cam follower throughout its lift range. Because these points are

The Lobe-Lift and Acceleration Graph displays data for the profile selected in the *Lobe Source-Data List*. Use the graph to get an overall visualization of the lift curve and acceleration rates of the profile. A roughly shaped curve can signal erroneous data in the profile.

Lobe Lift And Acceleration Graph

Camshaft Category—Using Lobe Profiles

Convert Profile Data To 10-Point Modeling

"Profile" Indicates Lobe Profile Data Is Being Used.

"->10-pnt" Buttons will convert Profile data to 10-Point Modeling
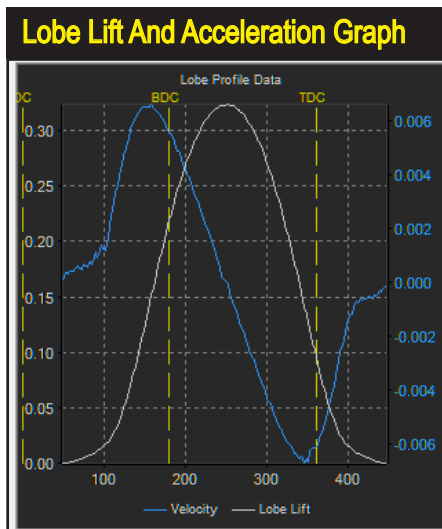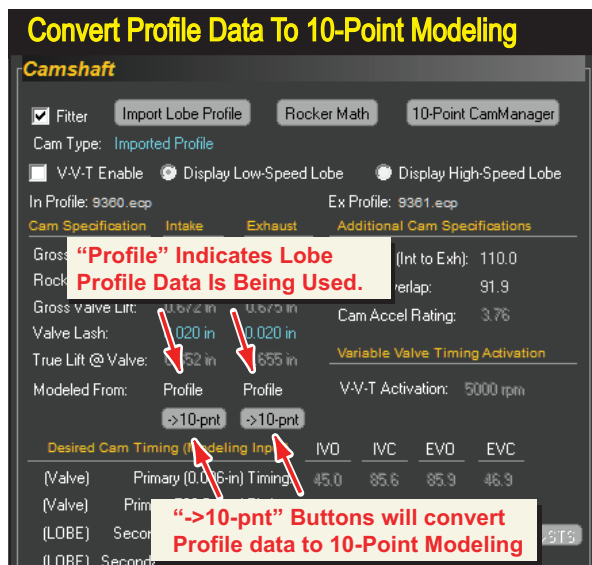
After a *Lobe Profile* has been imported into the simulation, the *Modeled From* field(s) in the Camshaft category will display "*Profile.*" You convert to *10-Point* modeling at any time by clicking a "->10-pnt" (Convert To 10-Point) button. Profile data for that lobe will be discarded and 10-Point valve-event timing will be used that matches the profile data as closely as possible.

fixed, they can not be individually changed without corrupting the lift curve. Basically, profile valve-event timing (IVO, IVC, etc.) is "built into" the data and cannot be altered without re-defining the entire profile. However, there are a few tuning "knobs" that you can turn when using Profile data without converting into 10-point data. Modifying the following cam specifications offer some options when Profile data is being used:

Timing Point/Specification	Comment
Cam Advance and Retard	Shifts entire lobe profile (see following text for details)
Intake Centerline	Shifts entire intake lobe timing
Exhaust Centerline	Shifts entire exhaust lobe timing
Rocker Ratio	Changes lobe-lift-to-valve-lift ratio and has some affect on duration
Valve Lash	Affects opening/closing timing and has some affect on duration

When Profile modeling is being used, you will notice that many of the valve-event specifications within the Camshaft category are dimmed and cannot be directly modified. To "unlock" all timing data fields, you must use the "*->10-pnt*" (Convert To 10-Point) buttons to convert profile data into 10-point timing; this allows you to modify all cam timing data, but keep in mind that converting to 10-point data will create...usually small...valve motion differences from the original cam Profile curve.

Camshaft Category—Cam Adv/Ret

Advance/Retard Cam Timing

The simulation allows direct entry of Intake and/or Exhaust camshaft lobe advance or retard values for both Profile and 10-Point cam modeling. Changing these specifications from zero (the default) to a positive value advances the cam lobe (in crank degrees) while negative values retard the lobe.

Why advance or retard the cam? It's just about the only valve-timing change that can be made <u>after the camshaft has been installed</u>. While it is sometimes possible to improve performance using this technique, let's investigate what happens when valve events are advanced or retarded in unison.

> **Note:** The **Prolterator** built into Version6 engine simulations can automatically evaluate cam centerline timing on single- and twin-cam engines. This powerful feature can be setup in seconds and runs a typical centerline analysis in just minutes (more details on this feature can be found in the **Pro-Iterator Chapter** later in this manual).

Tuning A Single-Cam Engine

It is generally accepted that advancing the cam improves low-speed power while retarding the cam can improve high-speed power. When the cam is advanced, IVC and EVC occur earlier and that can improve low-speed performance; however, EVO and IVO also occur earlier, and these changes tend to improve power at higher engine speeds, but to a lessor extent. The net result of these "conflicting" changes typically is a slight boost in low-speed power. The same goes for retarding the cam.

To ensure the most accurate cam timing and to facilitate tuning, a gear drive is often used in high-performance applications. This a CompCams drive was designed for GM LS engines (with three-bolt cams on all 24x and 58x LS engines).

DOHC Cam Sprockets

Camshaft Category—Cam Adv/Ret

Advance/Retard Menu

The simulation allows direct entry of camshaft advance or retard for each lobe (permits DOHC modeling). Changing this specification from zero (the default) to a positive value advances the cam lobes; negative values retard the lobes. See text for more information on how these changes can affect engine output.

Two events (later IVC and EVC) boost high-speed power and two (later EVO and IVO) boost low-speed performance, but again, to a lessor extent. The net result is often a slight boost in high-speed power.

Advancing or retarding a camshaft under these conditions has the overall affect of reducing valve-timing efficiency in exchange for slight gains in low- or high-speed

DOHC Cam Sprockets

In DOHC engines, like the Honda VTEC and many other engines, where separate cams are used for the intake and exhaust lobes, the cam advance/retard can be set independently for the intake and exhaust cams. In the real world, this tuning is easily accomplished with adjustable cam sprockets, like these from ZEX. Adjustable timing sprockets can also compensate for milled deck and head surfaces that can alter cam timing.

power. Consequently, many cam manufacturers recommend avoiding this tuning technique. If advancing or retarding allows the engine to perform better in a specific rpm range, the cam profile was probably not optimum for the engine in the first place. More power can be found at both ends of the rpm range by installing an optimized cam rather than advancing or retarding the wrong cam. However, if you already own a specific camshaft, advancing or retarding overall timing may offer some "fine tuning" capabilities.

Tuning DOHC Engines

Tuning DOHC engines with individual intake and exhaust cams is a straightforward process. While changing cam timing on an engine with a single cam forces you to change ALL lobes the same amount and in the same direction—often reducing any benefits you may have gained—on DOHC engines cam tuning takes on a much greater significance. When you install an optimum cam for a specific application, you can expect it to produce peak power (or torque) when installed "heads-up," or without any advance or retard. However, engine builders often find improvements by advancing one cam and retarding the other. This is especially true in turbocharged applications, where advancing the intake cam and retarding the exhaust cam increases the Lobe-Separation Angle (also called the Lobe-Centerline Angle, or LCA), reducing valve overlap, which sometimes improves forced-induction efficiency.

Dyno testing has shown that it is sometimes impossible to predict how the engine

DOHC Gear Train
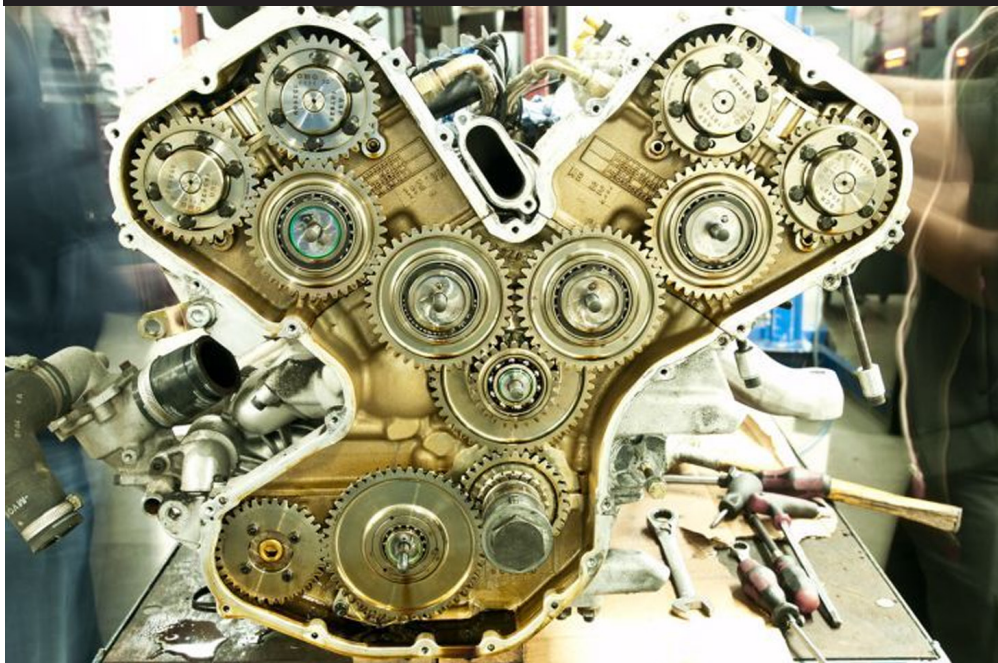
Camshaft Category—Cam Adv/Ret

will respond to intake- or exhaust-cam advance and retard. This is may be due to subtleties within, and interactions between, the induction and exhaust systems. While this simulation will give you valuable feedback, this fine-tuning process should be confirmed on a real-world dynamometer.

> **Note:** Make sure you keep in mind that changing cam advance or retard on a real-world engine will alter valve-to-piston clearance. Changing cam timing by 4-degrees or less will rarely cause interference problems, however, it is always the engine builder/tuner's responsibility to ensure mechanical contact between the pistons and values does not take place at any engine speed. Just starting and idling an engine with piston-to-valve interference can damage valves, break cams, and bend valvetrain components.

Combustion Category

The Combustion Category includes Compression Ratio, Thermal Coatings, Fuel Type, Air/Fuel Ratio (fixed or adjustable for each rpm point), Fuel Vaporization within the intake tract (before IVC), Combustion-Chamber geometry, and Ignition Timing. Each of these specifications directly affect how fuel is burned in the engine. This category also includes a Compression Ratio Calculator, and a Combustion-Chamber-Selection dialog. Clicking the *Estimate Ignition Timing* checkbox, will direct the simulation to determine the *MBT* for the engine at each rpm point (see text).

THE COMBUSTION CATEGORY
Applies To All Version6 Simulations, Except as Indicated

The Combustion Category contains combustion-related components and speci-fications. Included are *Compression Ratio*, *Thermal Coatings Modeling*, *Fuel Type (includes spark- and combustion-ignition fuels)*, *Air/Fuel Ratio (one value or a "map" of values)*, *Fuel Vaporization Percentage within the intake tract before IVC*, *Combustion-Chamber geometry*, and *Ignition Timing (for spark-ignition fuels only)*. Each of these specifications directly affect how fuel is burned and the overall combustion efficiency. This category also includes a *Compression Ratio Calculator* (see text),

Combustion Category

Compression ratio is calculated by dividing the total volume within the cylinder when the piston is located at Bottom Dead Center (BDC) by the volume that exists when the piston is positioned at Top Dead Center (TDC).

Basic Compression Ratio Equation

Compression Ratio =

$$\frac{\text{Swept Cylinder Volume} + \text{Combustion Space Volume}}{\text{Combustion Space Volume}}$$

and a *Chamber-Selection* dialog. Clicking the *Estimate Ignition Timing* checkbox will instruct the simulation to determine the **MBT** (**M**inimum **I**gnition **A**dvance **F**or **B**est **Torque**) for the engine at each rpm point.

> **Note:** When you select **Diesel** fuel, the following fields are dimmed, since they cannot be changed in compression-ignition applications:
>
> **Combustion Chamber Design:** An analysis of chamber shape is not included in this version of the simulation.
>
> **Ignition Timing:** Ignition point determined by the simulation

Compression Ratio and How It Is Calculated

The **Compression Ratio** menu, located in the *Combustion Component* category, establishes the compression ratio for the simulated engine (within the range of 3:1 to 30:1) for gasoline and other spark-ignition fuels or for Diesel fuel using compres-

The Compression Ratio field is located at the top of the Combustion Category. Compression Ratio is a comparison of the geometric volumes that exist in the cylinder (including the cylinderhead and gasket volumes) when the piston is located at BDC (bottom dead center) to the "compressed volume" when the piston reaches TDC (top dead center). The simulation will model any compression ratio from 3.00:1 to 30.00:1.
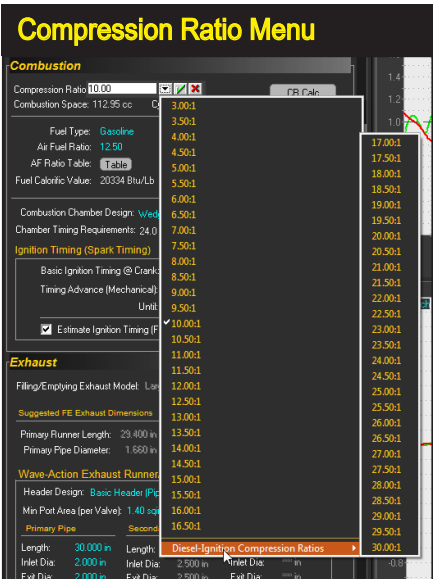
Compression Ratio Menu

sion ignition.

Compression-ratio is the comparison of two volumes in the engine, as illustrated by the "standard equation" shown nearby. The variables are: 1) *Swept-Cylinder Volume*, and 2) *Combustion-Space Volume*. While these are the only two volumes that affect compression ratio, each of these volumes is made up of multiple other volumes, so the first step in understanding compression ratio is to become familiar with these volumes.

Swept Cylinder Volume is the most straightforward: It is simply the three-dimensional space displaced by the piston as it "sweeps" from BDC to TDC, and is determined solely by the bore diameter and stroke length. The swept cylinder volume is calculated by the simulation—and displayed in the **Combustion** category—as soon as the bore and stroke have been selected for the test engine.

The other main variable in the compression-ratio equation is *Combustion-Space Volume*. This is the total volume that exists above the piston when it is positioned at TDC. This space includes the combustion-chamber volume, the volume taken up by the thickness of the head gasket, any volume added by the piston not rising fully to the top of the bore, plus any valve-pocket volume, less any volume displaced by the piston or piston dome protruding above the top of the cylinder bore (block deck). The complexity of these volumes is often a stumbling block in becoming comfortable with all aspects of compression ratio. However, the following explanation and illustrations should clarify these important concepts.

Imagine yourself as a "lilliputian," wandering around inside the engine. Take a stroll inside the combustion space. Picture in your mind what you would see in the cylinder with the piston at TDC. The combustion chamber would look like a ceiling above you. The floor would be the top of the piston. If the piston (at TDC) didn't rise completely to the top of the cylinder, you would see a bit of the cylinderwall around the edges of the floor, with the head gasket sandwiched between the head and block like trim molding around the room. There may be "steps" (valve pockets) in the top of the piston just under your feet (don't trip!). If the piston had a dome, it

A good way to visualize compression ratio volumes is to imagine yourself as a "lilliputian" wandering around inside the engine. You would see the combustion chamber above you like a ceiling. Your floor would be the top of the piston. *Swept Cylinder Volume* is the three-dimensional space displaced as the piston "sweeps" from BDC to TDC, and is determined solely by the bore diameter and stroke length.

Picturing Compression Ratio And Swept Volume

PISTON AT TDC

PISTON AT BDC
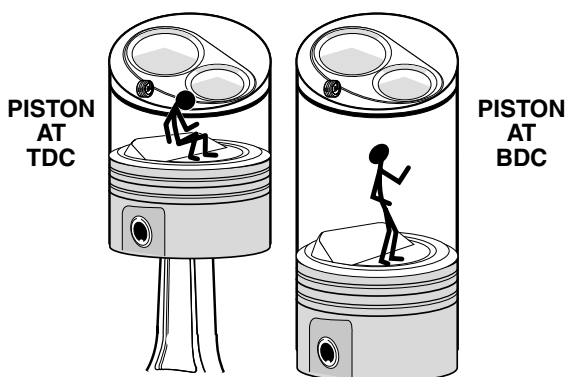
might act as a small room divider rising from the floor, to, perhaps, knee high. The space in this room (combustion space) would be larger if the piston was positioned lower down the bore or if the "steps" under your feet were deeper, and it would be tighter squeeze if the room divider (dome) volume was larger. This entire space is "home" for the compressed charge when the piston reaches TDC. This is the volume that makes up the combustion space, the denominator of the compression-ratio calculation equation. Now let's continue our tour of compression spaces, but this time we'll explore what we see inside the cylinder when the piston is located at BDC. The very same volumes that we just described (chamber, dome, notches, gasket, etc.) are still there, but are now located well above our head. It looks like the room has been stretched, like the elevator ride in the Haunted House at Disneyland. This "stretched" volume is described in the numerator of the compression-ratio equation. It's simply the original combustion volume plus the volume added by the "sweep" of the piston as it traveled from TDC to BDC. The ratio between these two volumes is the compression ratio.

Bore, Stroke, And Compression Ratio

Another quick look at the compression-ratio equation reveals that if engine displacement (swept volume) is increased, either by increasing the bore or stroke, the compression ratio will rise. In fact, with everything else being equal, a longer stroke will increase compression ratio much more quickly than increasing bore diameter. This is due to the fact that a longer stroke (without changing other engine dimensions) not only increases displacement, but it tends to <u>decrease</u> combustion space volume, since the piston moves higher the bore if the rod length and pin position are not changed (in our "lilliputian" example, raising the floor closer to the ceiling). This "double positive" results in rapid increases in compression ratio for small increases in stroke length. On the other hand, increasing cylinder-bore diameter also increases compression ratio but less significantly. This is due, in part, to the <u>increase</u> in *Combustion-Space Volume* that often accompanies a larger bore.

While *Combustion-Chamber Volume* is simply the volume of the chamber in the cylinder head, the *Combustion-<u>Space</u> Volume* is the <u>total enclosed volume</u> when the piston is located at TDC. This space includes the volume in the combustion chamber, plus any volume added by the piston not rising to the top of the bore, the volume within any valve pockets, and the head-gasket thickness, less any volume contained within the piston dome or displaced by the piston protruding above the top of the bore.

Top-Dead-Center Volumes

Combustion Chamber Volume

Combustion Space Volume

PISTON AT TDC
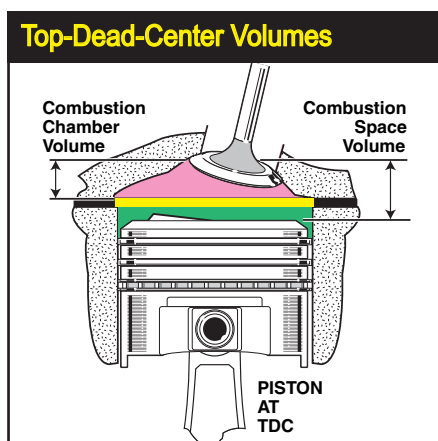
Combustion Category

Why does higher compression ratio produce more horsepower? Try to answer this question before you read the remainder of this caption. A combustion space containing twice as much volume as the cylinder produces a 1.5:1 compression ratio. Peak cylinder pressures after ignition will be about 250psi. With a combustion space about 1/10th of the volume of the cylinder, the compression ratio will be 10:1. Peak pressures reach about 1500psi. It is easy to see that the higher compression ratio generates much higher cylinder pressures. And these higher pressures exerted against the piston throughout the first half of piston travel from TDC to BDC on the power stroke increase torque and horsepower.

High Compression Boosts Power

COMPRESSION RATIO 1.5:1

COMPRESSION RATIO 10:1
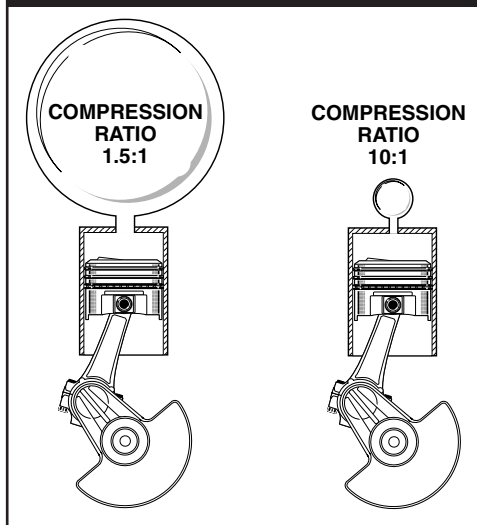
Changing combustion space—the other element in the compression-ratio equation—will also alter the compression ratio. Anything that reduces the combustion volume, while maintaining or increasing the swept volume of the cylinder, will increase the compression ratio. Some of the more common methods to accomplish this are decreasing the volume of the combustion chambers (by replacing or milling the heads), using thinner head gaskets, changing the location of the piston-pin or rod length to move the piston closer to the combustion chamber, installing pistons

Compression-Ratio Math Calculator

Activate the Compression-Ratio Calculator by clicking on the *CR Calc* button in the *Combustion* category. You can also open the calculator by selecting *Compression Ratio Math* from the Tools menu or by clicking on the *Compression Ratio Icon* in the Tool Bar. This tool lets you to directly enter combustion-chamber volumes, head-gasket thickness, etc., and determine their effects on engine compression. For detailed information on using the Compression-Ratio Calculator, see Built-In Calculators later in this manual.

Combustion Category

Thermal Coating Modeling

Combustion

Compression Ratio: 13.00   Thermal Coatings on AND Chamber
Combustion Space: 96.45 cc   Cylinder Volume: 1157.43 cc

No Thermal Coatings
Piston or Chamber Thermally Coated
✔ Piston AND Chamber Thermally Coated

Fuel Type: Gasoline   Nitrous Power:
Air Fuel Ratio: 12.50   Equivalence Ratio: 1.18
AF Ratio Table: Table   Fuel Cooling (LHV): 150.5 Btu/Lb
Fuel Calorific Value: 20334 Btu/Lb   Vaporization % (Before IVC): 100.0 %

Combustion Chamber Design: Pentroof with Squish
Chamber Timing Requirements: 15.0 Deg   Chambers...

Ignition Timing (Spark Timing)
    Basic Ignition Timing @ Crank: ** Deg   BTDC
    Timing Advance (Mechanical): 0.0 Deg   Per 1000 RPM

with larger domes, etc. These modifications and others can be explored using the built-in *Compression-Ratio Calculator*.

The Compression-Ratio Calculator

The simulation allows the direct selection and testing of a very wide range of compression ratios from 3:1 to 30:1. In addition, many engine builders need to use combustion-chamber volumes, head-gasket thickness, etc., to determine the compression ratio. The **Compression-Ratio Calculator**, quickly performs these functions. Detailed information on using the *Compression-Ratio Calculator* is included in the section **Built-In Calculators** later in this manual).

Piston/Chamber Thermal Coatings

You can model the use of thermal coatings on the piston top or combustion chamber, or both. Thermal coatings help keep heat in the cylinder that would otherwise dissipate into the water jackets of the engine. The more combustion heat that is held in the combustion space, the higher the cylinder pressure will be during the power stroke. This improves engine output and overall efficiency.

The menu selection lets you choose the following models: *No Thermal Coatings, Coating Applied To The Piston* OR *The Chamber, and* finally *Coatings Applied To The Piston* AND *The Chamber*.

Note: The model assumes that coatings applied to the chambers includes the valve heads (coating covers the entire chamber surface).

Fuel Type Menu Selection

You can also model several automotive fuels, including Diesel, plus Nitrous-Oxide injection with Gasoline or Methanol. Select any of these fuels options from the **FUEL** menu:

Combustion Category

Fuel Selection Menu

Model various fuels for dyno testing in Version 6 Engine Simulations. When any of these fuels have been selected, the commonly accepted peak-power air/fuel ratio is automatically inserted into the *Air/Fuel Ratio* field. Some fuels can be used with Nitrous-Oxide Injection.

- **Gasoline (Detonation Free)**
- **Methanol (Methyl Alcohol)**
- **Ethanol (Ethyl Alcohol)**
- **Propane (Gaseous fuel)**
- **Synthetic Gas (4 types)**
- **Nitromethane (Various %)**

- **Gasoline W/Nitrous Injection**
- **Methanol W/Nitrous Injection**
- **E85 (85% Ethanol, 15% Gasoline)**
- **LNG (Liquefied Natural Gas)**
- **Diesel (Compression Ignition)**

When any of these fuels have been selected, the simulation readjusts the air/fuel ratio for optimum power. This updated air-fuel ratio is displayed in the *Air/Fuel Ratio* field. Changes to the Air/Fuel ratio can be made at any time by making a selection from the Air/Fuel Ratio menu, by direct numeric entry, or by using the A/F
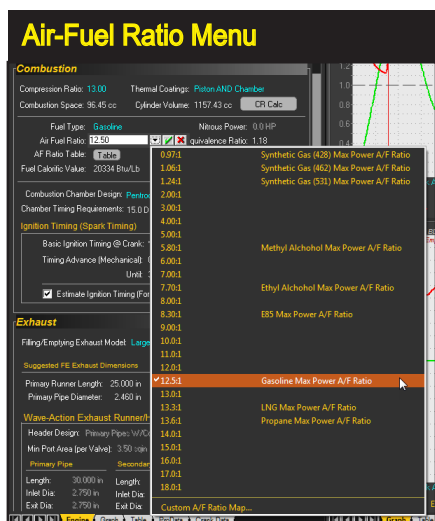
Air-Fuel Ratio Menu

When any fuel has been selected, the air/fuel ratio is adjusted for optimum power. The updated ratio is displayed in the *Air/Fuel Ratio* field. Changes to the Air/Fuel ratio can be made at any time by selecting the Air/Fuel Ratio menu or by direct numeric entry.

Combustion Category

Table to specify a unique air/fuel ratio for each rpm test point.

    To use the AF Ratio Table and create a "map" of air-fuel ratios for each rpm point, click on the **Table** button in the Combustion Category. You can fill all rpm points with the default ratio (to initially fill the table) using the **Fill With Default** button. If you would like to switch back to using a single A/F Ratio, open the **Table** dialog again and click on the **Use Single AF Ratio** button.

Nitrous-Oxide Injection

    There are many ways to boost engine power. However, nitrous-oxide injection is a particularly effective method. Developed during World War II for piston-engine fighter aircraft, nitrous-oxide gas—an oxygen-releasing substance—allows an engine to ingest more fuel

Air/Fuel Table

The simulation can also model a varying air/fuel ratio. The *Air/Fuel Ratio Table* dialog is available by selecting the *Table Button.* in the COMBUS-TION category. You can set individual AF Ratios for each rpm test point. Reset to a single AF ratio simply by pressing the *Use Single AF Ratio* button below the table.

while maintaining optimum air(oxygen)-to-fuel ratios. Injected into the cylinders with additional fuel, the effect is similar to instantaneous supercharging but without the losses from a belt- or exhaust-gas-driven device. Remarkable as it may seem, as much horsepower as desired can be added, with the limitations being dangerously high cylinder pressures, detonation, or component failure. There are no subtleties here: Add more nitrous and fuel; produce more horsepower!

    Most nitrous systems inject a fixed amount of nitrous and fuel (constant-flow), regardless of engine speed. In other words, when the nitrous switch is turned on,

Nitrous-Oxide Selection Menu

There are many ways to boost engine power, however, nitrous-oxide injection is a uniquely effective method. The simulation models constant-flow *Nitrous/Gasoline* and *Nitrous/Methanol* injection systems.

Combustion Category

the engine will immediately produce a fixed boost in power, solely determined by the amount of injected fuel and nitrous.

The simulation models a typical constant-flow **Nitrous/Gasoline** or **Nitrous/Methanol** injection system. Nitrous is "injected" by the simulation in precise amounts throughout specific increments of crank degrees, similarly to an actual nitrous system. You select the desired HP boost and the Rpm Range to apply nitrous, and the simulation will calculate the Nitrous Mass Flow required to produce that power (calculation of nitrous flow is performed before the actual simulation is run, so engine design and other factors...just like in the real world...can affect the actual power output reported by the simulation). The simulation will tell you how much of the injected nitrous mass actually entered the cylinder and how much was rejected back into the intake manifold (or lost to exhaust flow). The enhanced nitrous modeling included in this version-6 simulation includes these elements in a nitrous-augmented engine:

1) Nitrous Start/Stop Rpm Ranges can be selected
2) Nitrous Inflow restriction modeled at the intake valves
3) Cylinder filling (capacity vs. overfill backpressure) and gas reversion
4) Nitrous (and added fuel) effects on charge cooling (LHV)
5) Nitrous details are included in the Simulation Log and *ProTools* Printout
6) Enhanced Nitrous Injection is offered in all Version 6 engine simulations

A nitrous injection system designed to add 100 horsepower (flowing about 4 pounds per minute of nitrous oxide and additional fuel to maintain the correct air/fuel ratio), will produce a 100 horsepower boost instantly upon triggering the system.

Nitrous-Oxide Injection And BMEP
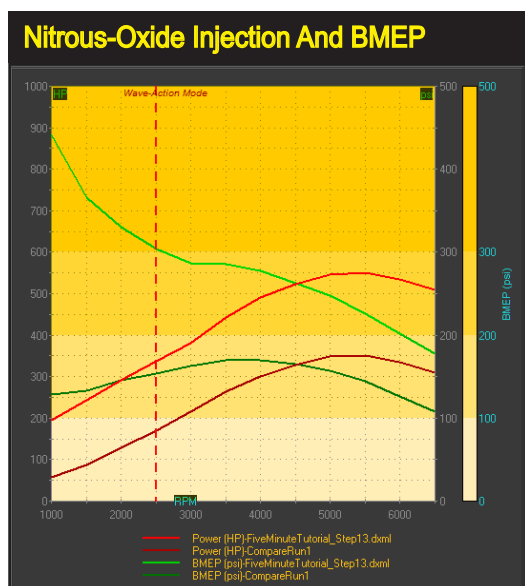
This graphic shows how cylinder pressures (in this case the BMEP shown by the green line) increase after a 200-horsepower nitrous system is activated (this pressure comparison includes colored DataZones™, discussed later in this manual). Since the simulation models a fixed-flow nitrous system, cylinder loads increase as engine speed decreases (the longer <u>time</u> the intake valve is open, the greater nitrous load is inducted into the cylinder). Below 3000rpm, BMEP exceeds 300psi (shown by the top, light green band). In this situation, nitrous-system activation should be delayed until 3000rpm to help maintain engine reliability.

Combustion Category

Nitrous Power Menu

To activate any Nitrous Boost Horsepower from zero to 800hp, first select one of the *Nitrous-Injection* selections from the Fuel Type menu (which activates the *Nitrous Power Menu*), then open the *Power Menu* to enter any power value. You can also use the Nitrous Start and Stop menus to the set the RPM range to apply Nitrous.

Remarkably, the engine will continue to produce an this additional 100 horsepower across the entire rpm range. That means a 100hp nitrous system activated at 2000rpm (when the engine may have been producing only about 70hp) can double or even triple power output!

These huge power boosts at low engine speeds (when each cylinder ingests a large "dose" per power cycle) can generate excessive cylinder pressures. As a result, fixed-flow-rate systems often include a delayed start trigger to allow the engine to reach sufficient speed to maintain safe cylinder pressures. Maintaining cylinder pressures below critical levels is essential to prevent detonation and mechanical failure. Version 6 engine simulations include Nitrous Start and Stop selections to limit the RPM range in which to apply Nitrous.

During engine simulations with nitrous augmentation, you can keep track of cylinder pressures (for example, a BMEP greater than 300psi is usually considered borderline dangerous). For example, the nearby test graph illustrates a 350 smallblock equipped with a 200hp nitrous system. Note that BMEP pressures below 3000rpm exceed 300psi.

> **Note-1:** BMEP (Brake Mean Effective Pressure) is an "average" cylinder pressure; when it exceeds 300psi, peak pressures often reach detonation levels and can quickly lead to mechanical failure—for a short description of BMEP, refer to the *Glossary* later in this manual.
>
> **Note-2:** There are other ways to help reduce low-speed cylinder pressure in a Nitrous engine including modified cam timing. It has long been known that increasing valve duration and overlap will lower cylinder pressures at lower engine speeds. This typically-unwanted phenomenon is a low-speed power

Combustion Category

killer, but combined with a nitrous-oxide injection system, it can permit earlier nitrous flow while optimizing power at higher engine speeds. Other ways to decrease low-speed cylinder pressures include reduced compression ratios, increased exhaust-system back pressure, reduced induction airflow, larger engine displacement, and of course, reducing the nitrous flow rate.

You can activate the nitrous-injection model by selecting *Gasoline/Nitrous Injection* or *Methanol/Nitrous Injection* from the **Fuel Type** menu (see previous page). You will see the following choices:

- **25 HP** (~1 lb/min N_2O flow)
- **100 HP** (~4 lb/min N_2O flow)
- **300 HP** (~12 lb/min N_2O flow)
- **500 HP** (~20 lb/min N_2O flow)
- **700 HP** (~28 lb/min N_2O flow)

- **50 HP** (~2 lb/min N_2O flow)
- **200 HP** (~8 lb/min N_2O flow)
- **400 HP** (~16 lb/min N_2O flow)
- **600 HP** (~24 lb/min N_2O flow)
- **800 HP** (~32 lb/min N_2O flow)

In addition to these 10 menu options, you can use the **Nitrous Power Menu** to manually enter any nitrous power value from zero to 800hp and use the **Nitrous Start** and **Nitrous Stop** menus to limit the boosted rpm range.

Combustion-Chamber Design

The *Combustion-Chamber Design* field in the **Combustion** category sets the fuel burn rate and adjusts the combustion efficiency based on the physical geometry of the combustion space.

Nine combustion-chamber geometries from the slowest-burning disc shape to the fast-burn pentroof designs model most high-performance and racing engines. Clicking the *Chambers Quick-Access™* button in the Combustion Category provides an overview of these chamber shapes, including the basic Chamber Timing Requirements of these designs (timing values in the dialog assume gasoline as the fuel). Note: This dialog does not open when using Diesel as a fuel.
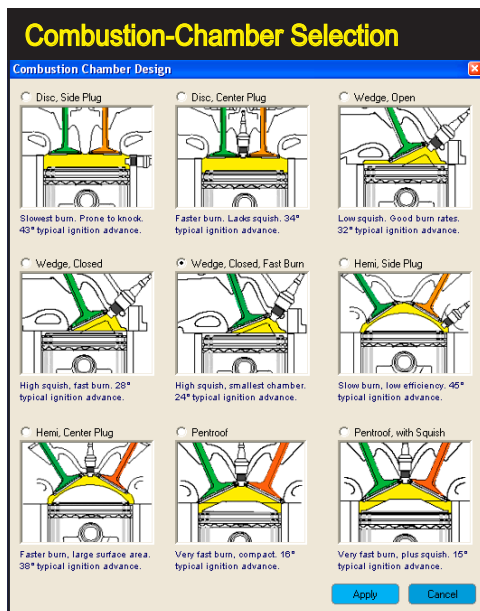
Combustion Category

In general, a "disc-shaped" chamber with a side-mounted sparkplug generates the lowest turbulence and flame speed. The burn-rate differences between this old technology chamber and a modern pentroof designs with squish-assisted turbulence can be as much as 30-degrees of reduced ignition timing "lead" to accommodate the faster-burn rates. A mathematical analysis of the burn process (can be thought of as simply a rapid pressure rise) reveals that peak power would be obtained if the fuel burned instantly at a crank angle slightly after TDC. This would produce the greatest push on the piston at the point at which this pressure is most efficiently converted into the highest force on the crankpin. While an instant fuel burn (and pressure rise) is not possible for several practical reasons, including knock prevention and maintaining mechanical integrity, typically a fast pressure rise in the cylinder will produce more power (reaching peak pressure at about 20- to 30-degrees ATC is typical for many high-output engines). To facilitate this, modern chamber designs are optimized for high turbulence and fame speeds while minimizing pre-ignition and detonation.

> **Note:** As always, there are exceptions. In a nitromethane-fueled engine, the slower burn-rate hemispherical chamber is often used. In this case, there are benefits to slowing down the burn rate and giving the flame front a symmetric volume in which to propagate. With nitromethane this results in a more uniform application of the huge cylinder pressures produced by this powerful racing fuel. The overriding consideration here is reliability not peak pressures!

The simulation provides nine combustion-chamber geometries from the slowest-burning disc shape, provided primarily for comparison purposes, to the fast-burn pentroof designs used in many modern high-performance and racing engines. Clicking
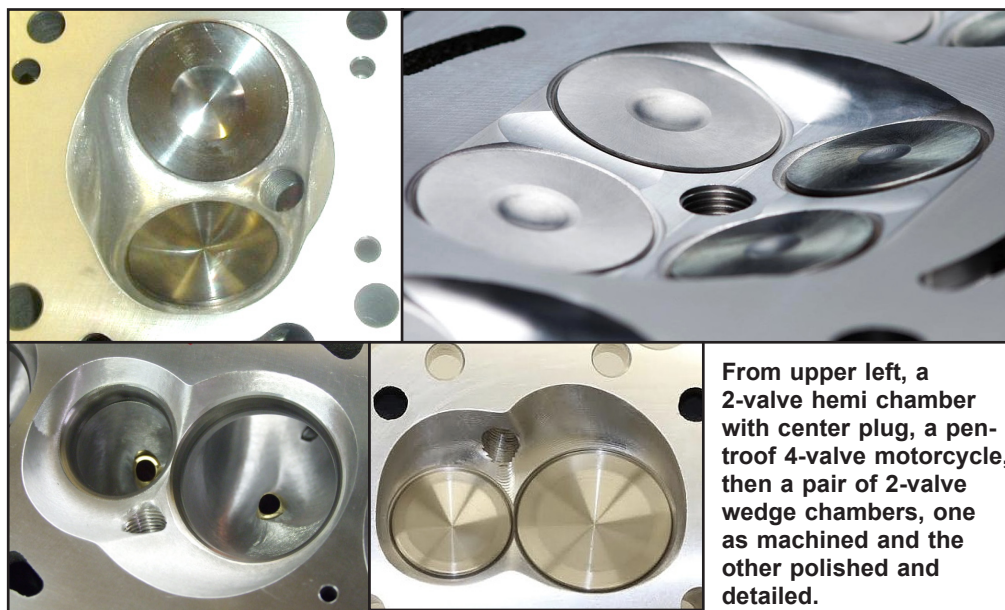
From upper left, a 2-valve hemi chamber with center plug, a pentroof 4-valve motorcycle, then a pair of 2-valve wedge chambers, one as machined and the other polished and detailed.

on the **Chambers** Quick-Access™ button (not active for Diesel fuel) provides an overview of these chamber shapes, including the basic *Chamber Timing Require-ment* (assumes gasoline as the baseline fuel).

The *Chamber Timing Requirement* is based on the *time lag* between the sparkplug discharge, referred to as the **SparkPoint**, and the beginning of cylinder pressure rise. This delay can be considered as the innate rate at which fuel burns in that chamber geometry. This is the *least possible timing advance* that the engine will need to produce optimum power with that chamber at low engine speeds. As speed increases, or if non-optimum air/fuel ratios are used, greater ignition lead (more time) will be required to efficiently burn the fuel and produce optimum power. This minimum timing requirement for optimum power at a specific operating condition is often called **MBT** (or **M**inimum ignition advance for **B**est **T**orque). An engine does not have single *MBT*, but rather many *MBTs* as engine speed, temperature, AF Ratio, fuel chemistry and other real-world factors change.

Ignition Timing

Optimum power production depends on the precise timing of the "ignition point," an event that, as described above, can vary substantially. There is a significant delay in the initiation of measurable combustion after the *SparkPoint*, and even beyond that, many factors affect lag and the rate of combustion. So, what shall we call the actual point of ignition? Should it be: **1)** when the spark occurs, or **2)** when the fuel starts burning, or **3)** when pressure begins to rise in the cylinder? Rather than debating this, we will use other terms that have more precise meanings.

To begin, let's establish the beginning of the *Combustion Cycle* at the **SparkPoint,** commonly know as the **Ignition Timing Point** of the engine. This will be the pre-cise point, in crank degrees before top-dead center (BTDC), when the initiation of

Optimum Ignition Point

Figure 20
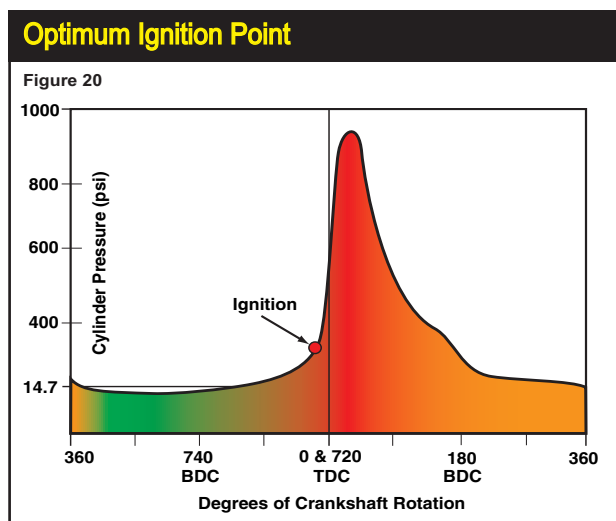
While an instantaneous fuel burn that produce peak pres-sures at the optimum crank angle is not possible, in general, the faster the pres-sure rises in the cylinder the more power the engine will produce. Modern combus-tion chamber designs are optimized for high turbulence and fame speeds within the constraints of preventing pre-ignition and detonation.

Combustion Category

Activate a manual, rpm-based advance curve by un-checking "Estimate Timing..." and specifying the initial-timing advance and the number of degrees for each 1000-rpm increase in engine speed. You can cap the advance curve by specifying a limit rpm.

spark occurs. After the *SparkPoint* there is a delay before the pressure within the cylinder actually increases. This delay can be exaggerated when the piston begins to move down the bore after TDC, increasing combustion-space volume. At some point, however, the increasing volume is overcome by the rapid rise in temperature from expanding combustion. The first indication of a pressure increase is usually defined as the *Start Of Combustion*, even though it's obvious that the combustion process was already underway, but remained in a low-burn-rate initiation phase.

We mentioned that the *Ignition-Timing* requirement for optimum power is commonly called the *MBT* (Minimum advance for Best Torque). But this timing requirement changes as turbulence in the combustion chamber changes, since the burn rate is substantially linked to, among other things, the turbulence of the gasses in the combustion chamber. Furthermore, turbulence also changes when: **1)** Engine speed changes, **2)** Air/Fuel ratio changes, and **3)** When engine load (throttle position) changes. And, as mentioned, there are additional subtle changes to burn rate from charge temperature changes, differences in compression ratio, and even bore size and stroke length.

How can we take all these factors into consideration and determine the *MBT* for all engine speeds? The simulation performs these complex calculations and derives an estimate of an *Optimum Ignition* advance curve. To activate this feature, simply check the **Estimate Ignition Timing** checkbox in the Combustion Category (selected

Manual and MBT Advance Curves
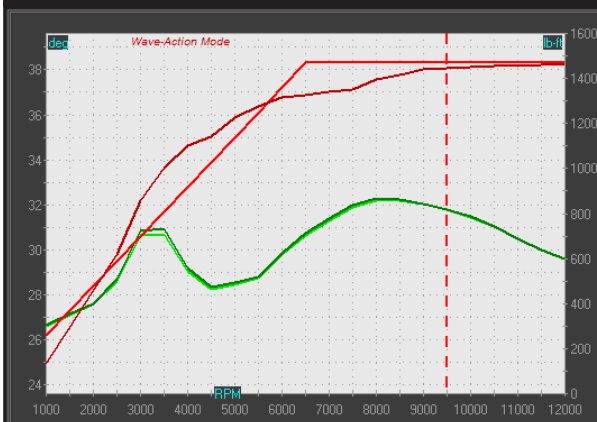
The advance curve (set in the Manual Advance Curve dialog shown above and illustrated in this graph by the light red line) started at 24-degrees advance, increased 2.2 degrees up to a limit at 6500rpm. The dark-red curve is the MBT advance curve calculated by the simulation.

Combustion Category

by default). You can view the calculated ignition timing values in the *ProData Table*, and on the top-right RPM graph.

If you would like to enter a specific ignition timing value and/or an advance curve based on engine speed, that option is also available. Un-check the **Estimate Ignition Timing** box, and enter values for **Basic Ignition Timing (BTDC).** This is the SparkPoint relative to TDC measured in crank degrees. In addition, you can direct the simulation to calculate an advance curve by specifying the number of degrees of advance for each 1000-rpm increase in engine speed. You can "flatten" the advance curve by specifying a limit rpm, after which no further advance is applied. After defining your own ignition values, you can quickly compare your curve with the *MBT* determined by the simulation; simply re-check the *Estimate Ignition Timing* box and note the changes in the advance curve and engine power.

> **Note:** Overall engine design, including chamber geometry, fuel type and quality, and other factors, can produce variations in optimum ignition timing and advance curve requirements. In addition, a 3D model of the combustion space and a full analysis of flame travel (beyond the capabilities of this simulation) are required to fully simulate the ignition timing requirements, especially a high-performance or race engine. Use the ignition timing features available in Version 6 simulations to help you establish baseline timing requirements. Actual dyno testing under real-world conditions is the only way to precisely determine an optimum ignition timing curve.

Combustion Category

Exhaust Category

The **EXHAUST** category establishes a mathematical model for the exhaust system of the simulated engine. Dynomation6 incorporates two distinct simulation methods, and each of these methods uses its own exhaust modeling technique. The *Filling-And-Emptying* simulation (also used in DynoSim6) is a simplified exhaust model, with limits on length and diameter resolution. The *Wave-Action* method performs a much more rigorous mathematical analysis of exhaust pressure waves and, as a result, requires more detailed data inputs to complete a simulation.

An overview of *Filling-And-Emptying* menu selections is provided below. *Wave-Action* exhaust data entry is covered in the next section.

IMPORTANT NOTE ABOUT ALL HEADER CHOICES: Some engines can de-rive scavenging benefits from suction waves created in the collector by adjacent cylinders (although studies have revealed that the benefits, when they exist, are relatively small). The "one-cylinder-scavenges-another" is not modeled in this simulation. Instead, the headers are assumed to deliver a scavenging wave only to the cylinder that generated the initial, and any subsequent, pressure pulses.

DynoSim6 and Filling-And-Emptying (FE) Exhaust Selections

While the simplified *Filling-And-Emptying* exhaust method cannot not resolve exact header dimensions (approximations of lengths and sizes are shown in the

The Exhaust System Category establishes the exhaust modeling for the FE Simulation (on the top, within the red box) while the Wave-Action Simulation choices take up the lower part of the Exhaust category. The FE selections are made from a two menus, consisting of 15 Model choices and one Tuning Speed selection. The WA exhaust selections are more detailed and allow hundreds of pipe, junction, and taper possibilities. When the WA simulation is used, the FE Exhaust choices are dimmed and vice-versa.

Exhaust Category—FE (DynoSim6) Model

Filling-And-Emptying Exhaust System Menu

Stock Manifolds	Mufflers With/Cat
H.P. Manifolds	Mufflers With/Cat
H.P. Manifolds	Mufflers Without/Cat
Small-Tube Headers	Mufflers With/Cat
Small-Tube Headers	Mufflers Without/Cat
Small Headers	Open Collector
Small Headers, Flow Optimized, Race Design	Open Collector
Small Tri-Y Headers	Open Collector
Large-Tube Headers	Mufflers With/Cat
Large-Tube Headers	Mufflers Without/Cat
Large Headers	Open Collector
Large Headers, Flow Optimized, Race Design	Open Collector
Large Tri-Y Headers	Open Collector
Large Stepped-Tube Headers	Open Collector
✓ Large Stepped-Tube, Flow Optimized, Race Headers	Open Collector

The simulation contains different exhaust-system models for the *Filling-And-Emptying* and *Wave-Action* simulations. The *Filling-And-Emptying* selections (shown here) are "generic" models of various exhaust manifolds and headers, both with and without mufflers and cata-lytic converters.

FE exhaust group and more accurate values are displayed in the Simulation Log), it can predict engine power changes from various discrete selections of exhaust manifolds and headers provided in the ***Filling/Emptying Exhaust Model*** menu and from the engine speeds indicated in the ***FE Exhaust Tuning RPM*** menu.

Each of the exhaust system selections apply a unique tuning model to the simulation. The menu includes fifteen selections, seven of which include mufflers, four include mufflers and catalytic converters, the remaining selections use "open collector" systems. When any of these exhaust configurations are used, the simulation will calculate and display an estimate of tubing diameters and lengths:

Stock Manifolds And Mufflers W/Cat

The first choice in the *Exhaust Model* menu simulates the most restrictive exhaust system; a "log-type" design, where all ports connect at nearly right

The first choice in the F*illing-And-Emptying Exhaust Model* menu simulates typical, pro-duction, cast-iron, "log-type" exhaust manifolds, where all ports connect at nearly right angles to a common "log" passage. These manifolds are designed to provide clearance for various chassis and engine components and provide less than optimum exhaust flow.

"Log-Type" Exhaust Manifold
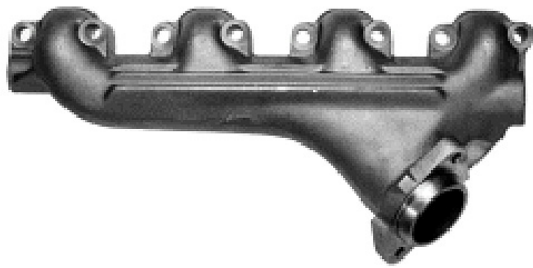
Exhaust Category—FE (DynoSim6) Model

HP Manifolds And Mufflers

The *HP Manifolds And Mufflers* exhaust-system choices offer a measurable improvement over the stock-exhaust selection. High-performance exhaust manifolds are designed to improve exhaust gas flow and reduce system restriction. They are usually a "ram-horn" or other "sweeping" design with fewer sharp turns and larger internal passages, like this manifold for the Eclipse/Laser. The connecting pipes to the mufflers are large diameter and the mufflers generate less back pressure.

angles to a common passage. This exhaust-modeling selection assumes that the manifolds are connected to typical OEM mufflers and catalytic converters with short sections of pipe. The exhaust manifolds and mufflers cancel all scavenging effects, and back pressure levels in the exhaust system effectively cancel blowdown effects of early EVO timing and increase pumping work losses during the exhaust cycle.

H.P. Manifolds And Mufflers W/Cat and WO/Cat

These choices offers significant improvements over the stock exhaust system discussed previously. The high-performance exhaust manifolds modeled are designed to improve exhaust gas flow and reduce system restriction. They are usually a "ram-horn" or other low-restrictive designs. While this system is considered a "high-performance" design, it offers little tuning effects and virtually all suction waves are fully damped. This exhaust system may allow some benefits from early-EVO timing blowdown effects.

Custom HP "Manifolds"

Here is an excellent example of a high-performance "manifold" replacement from Edelbrock for the Ford Focus. When used with mufflers, model this system using the *Filling-And-Emptying H.P. Manifolds And Mufflers* Model selection. When used without mufflers, select *Small Headers Open Exhaust*.

Exhaust Category—FE (DynoSim6) Model

Original Max-Wedge Exhaust

Here is another example of factory high-performance cast-iron exhaust manifolds. These were developed by Chrysler in the 1960's for their Max-Wedge Program. You would not believe how heavy these manifolds were! It's also hard to believe that today they sell to collectors for more than $2000 a set!

Small Tube Headers, Mufflers W/Cat and WO/Cat

This is the first exhaust-system model that begins to harness the tuning potential of wave dynamics in the exhaust system. Suction waves are created in the collector, but are somewhat damped by the attached muffler and catalytic converter, if used.

Small- And Large-Tube Headers, Open Exhaust

These choices simulate headers with primary tubes individually connecting each exhaust port to a common collector or tube. Strong suction waves, undamped by mufflers or a tailpipe system, provide a substantial boost to cylinder filling

Small-Tube tri-Y Headers

Large-Tube Headers

This is Tri-Y header harness the tuning potential of wave dynamics in the exhaust system. This tubular exhaust system from S&S for Honda engines offers good wave-dynamic scavenging with the potential of both power and torque boosts.

Large-tube headers mirror the capabilities of their small-tube counterparts, but their reduced restriction boosts performance on larger displacement engines.

and exhaust gas outflow. The small version of these headers often provide the best improvements on smaller displacement engines.

Small- and Large-Tube Headers, Flow Optimized, Race Designs

These choices simulate headers with large, sweeping curves, top-quality welding and construction. This professional design produces strong suction waves provide a substantial boost to cylinder filling and exhaust gas outflow. Backpressure is minimized. The small version of these headers show the greatest benefits on higher-rpm, smaller displacement engines.

Small and Large Tri-Y Headers, Open Exhaust

These choices simulate open headers with separate primary and secondary tubes that converge to a common collector. This design creates strong suction waves over a wide rpm range that aids cylinder filling and exhaust gas outflow. The small version of these headers show the greatest benefits on smaller displacement engines.

Large-Tube "Stepped" Selections

The "stepped" design of the primary tubes reduces pumping work on some engines. As high-pressure compression waves leave the port and encounter a step in the tube diameter, they return short-duration rarefaction waves. These low-pressure "pulses" assist the outflow of exhaust gasses by returning to the open exhaust valve(s) and assisting cylinder depressurization and reducing pumping work. This can generate a measurable increase in horsepower on large displacement and/or high-rpm engines.

Setting The *FE Exhaust Tune RPM*

The FE Exhaust Category includes an input for **FE Exhaust Tune RPM** that ap-

Large, Stepped-Tube Racing Headers
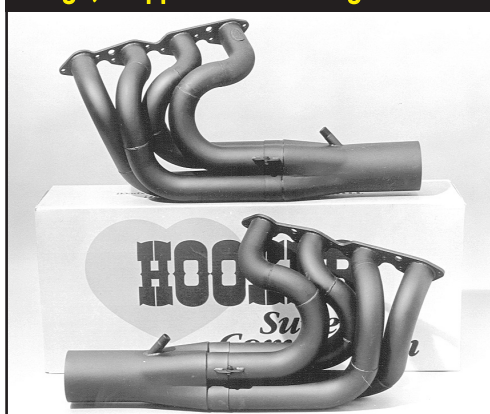
Large-tube Stepped headers have large-diameter primary tubes with one or more transitions to larger tubing diameters. These "steps" can reduce pumping work and improve horsepower on high-rpm engines.

Exhaust Category—FE (DynoSim6) Model

FE Exhaust Tune RPM & Pipe Lengths

Exhaust

Filling/Emptying Exhaust Model: Large Optimized Stepped Headers, Open

FE Exhaust Tune RPM: 9000

4500	Typical Speeds, Wider Header Tuning Range
5000	Typical Speeds, Wider Header Tuning Range
5500	Typical Speeds, Wider Header Tuning Range
6000	Typical Speeds, Wider Header Tuning Range
6500	Typical Speeds, Wider Header Tuning Range
7000	High Speeds, Narrower Tuning Band
7500	High Speeds, Narrower Tuning Band
8000	High Speeds, Narrower Tuning Band
8500	High Speeds, Narrower Tuning Band
9000	High Speeds, Narrower Tuning Band
9500	High Speeds, Limited Applications, Narrow Tuning Band
10000	High Speeds, Limited Applications, Narrow Tuning Band
10500	High Speeds, Limited Applications, Narrow Tuning Band
11000	High Speeds, Limited Applications, Narrow Tuning Band
11500	High Speeds, Limited Applications, Narrow Tuning Band
12000	High Speeds, Limited Applications, Narrow Tuning Band
AUTO Estimate Optimum Exhaust Tune RPM	

The FE Exhaust Category includes an input for *Tune RPM* that establishes the approximate engine speed at which the exhaust system reaches peak tune (the rpm value is also used to estimate pipe lengths and diameters). You can also select *AUTO Estimate...*at the bottom of the menu... that will automatically run a series of simulations and pick a (near) optimum selection for *Tune RPM*.

plies to (FE only) header-based exhaust systems (excluding the first three manifold choices in the Exhaust menu). The rpm value you enter into this field establishes the target engine speed at which the exhaust pressure waves return the strongest reflection (suction) wave back to the cylinder to assist exhaust scavenging. This scavenging process improves exhaust gas outflow and intake inflow by lowering cylinder pressures.

The FE also provides an *Auto Estimate FE Exhaust Tune RPM* feature included in the *Tune RPM* menu. It will find the tune rpm that produces the highest engine output, +/-1000rpm. This feature can be useful in determining the best pipe dimensions. Note that the **Auto** feature may sometimes return unexpected (too high) rpm values. This is due to the simulation searching for a balance between which pulse returns to the exhaust valve and how much frictional loss occurs in the pipes. Short pipes have low losses and can produce good power at the engine tune point, but the power curve may be too peaky, dropping off quickly on either side of the rpm peak.

> **Note:** To run the *FE Auto Estimate* feature, the engine must be complete and have successfully completed at least one simulation.

Dynomation6 Wave-Action (WA) Exhaust Menu Selections

The following section details extended exhaust simulation features available in the Wave-Action simulation in Dynomation6 (does not apply to the FE simulation used in DynoSim6).

The exhaust system—perhaps more than any other single part of the IC engine—is a virtual "playground" for high-pressure wave dynamics. The interactions

of these complex waveforms require sophisticated techniques to fully describe the physics of these high energy waves (called finite pressure waves). The WA model constructs mathematical representations of the pressure waves as they are created and continues to mirror their changes (updating wave shapes, intensity, and inter-actions) in all exhaust passages and junctions throughout the four-cycle process. This analysis is repeated at each rpm point (requires billions of calculations!). The *Wave-Action* model, and its ability to precisely recreate, analyze and display port pressures and velocities, will help you pinpoint optimum exhaust-system designs and dimensions.

>**Note:** The *Wave-Action* model is designed to simulate open-exhaust systems without mufflers; however, you can introduce a restriction into the exhaust system by reducing the size of the *final pipe diameter that exits at the atmo-sphere.* Also, consider using the *Filling-And-Emptying* model to simulate the effects of mufflers, catalytic converters, etc. It will use pipe dimensions similar to the WA (precalculated pipe dimensions are displayed in the Simulation Log) and can model several restrictive exhaust systems with very good accuracy.

Header Design

>The **Wave-Action** *Header Design* menu established the basic pipe-junction model for the exhaust system. Eighteen model selections are available in from the menu:

Basic Exhaust/Header Configuration	Active Elements Used
Individual--Primary Pipes Only	PRI
Individual--With Megaphone	PRI, MEG
Individual--W/One Step	PRI, SEC
Individual--W/One Step And Megaphone	PRI, SEC, MEG
Individual--W/Two Steps	PRI, SEC, TER
Individual--W/Two Steps And Megaphone	PRI, SEC, TER, MEG
Basic Header--(Pipes Merge At Collector)	PRI, COL1
Basic Header--With Megaphone	PRI, COL1, MEG
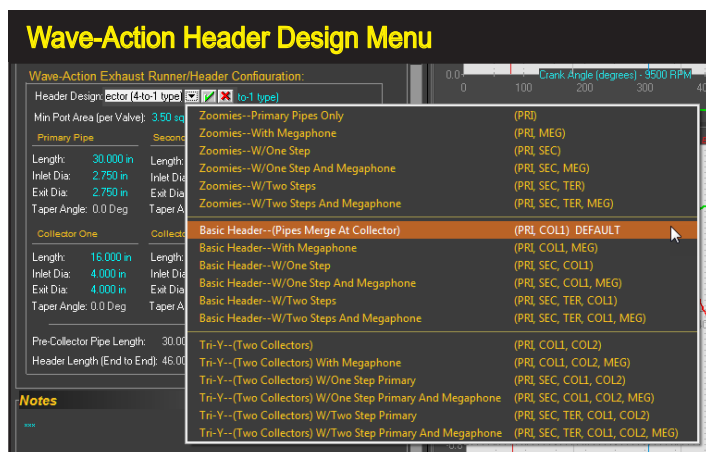
The Wave-Action *Header Design* menu provides 18 "basic" system configura-tions of pipes and junctions. Each menu choice activates the specific elements used to build that header configuration. From these basic types, you can construct almost about any possible exhaust header.

Exhaust Category—Wave-Action Model

Wave-Action Exhaust Elements

Dynomation6 Exhaust Elements
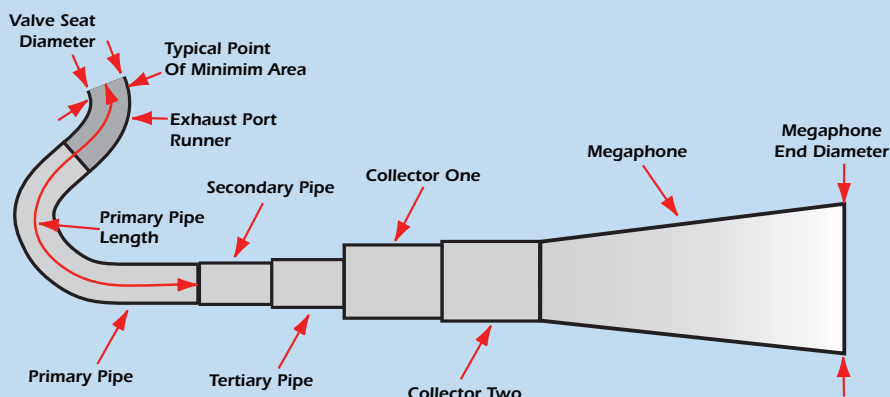
The Dynomation6 *Header Design* menu offers a "palette" of components that you can use to build virtually any header design. This drawing shows all elements arranged in order. Dynomation will only activate the elements necessary to build the header configuration you select. Choose from single pipe (Zoomies—as used on funny-cars and top-fuel dragsters) to complex headers with multiple collectors that merge pipe segments of increasing diameters (Tri-Y designs). Each element is actually modeled as an individual pipe segment, so except for name and dimensions, *Pipe, Collector,* and *Megaphone* are handled in a very similar ways by the WA simulation.

Basic Header--W/One Step	PRI, SEC, COL1
Basic Header--W/One Step And Megaphone	PRI, SEC, COL1, MEG
Basic Header--W/Two Steps	PRI, SEC, TER, COL1
Basic Header--W/Two Steps And Megaphone	PRI, SEC, TER, COL1, MEG
Tri-Y--(Two Collectors)	PRI, COL1, COL2
Tri-Y--(Two Collectors) With Megaphone	PRI, COL1, COL2, MEG
Tri-Y--(Two Collectors) W/One Step Primary	PRI, SEC, COL1, COL2
Tri-Y--(Two Collectors) W/One Step Primary & Megaphone	PRI, SEC, COL1, COL2, MEG
Tri-Y--(Two Collectors) W/Two Step Primary	PRI, SEC, TER, COL1, COL2
Tri-Y--(Two Collectors) W/Two Step Primary & Megaphone	PRI, SEC, TER, COL1, COL2, MEG

PRI: **Primary Pipe** always begins at minimum area in the exhaust port

SEC: **Secondary Pipe** always begins at end of the PRI pipe

TER: **Tertiary Pipe** always begins at end of the SEC pipe

COL1: **First Collector** always begins at end of the PRI, SEC, or TER pipe and can join/merge multiple pipes

COL2: **Second Collector** always begins at end of COL1 and can join/merge multiple collectors

MEG: **Megaphone** is always last element in the header system

Exhaust Category—Wave-Action Model

Minimum Exhaust Port Area

This is the exhaust port area at the point of greatest restriction. It is usually located near the valve seat and valve guide (similar to the *Intake Port Minimum Area*). If the engine being modeled has multiple-valves per port, enter the area for a single (each valve) port branch; don't sum minimum areas of multiple port branches.

Pipes Per Cylinder

The *Wave-Action* model can simulate exhaust systems that use two exhaust pipes per cylinder, a design sometimes used on high-output engines that usually have with two exhaust valves per cylinder (Rotax, Honda, and other specialized high-output engines). In all other cases where one exhaust pipe is attached to a single port outlet for each cylinder, this data field should be set to *1 (one)*.

Primary, Secondary, and Tertiary Pipe Lengths, Inlet Diameters, And Exit Diameters

*The **Primary Pipe Length** is the distance from the exhaust port minimum-area location to the end of that pipe segment*. The **Secondary** and **Tertiary** pipe lengths begin at the end of the previous pipe and terminate at the next junction or at the atmosphere. The **Pipe *Inlet Diameters*** are the inside diameter of the pipe at the point of connection to the cylinder head or a previous pipe. When the inlet diameter is larger than the previous pipe exit diameter, a *Step* is created that will generate a rarefaction wave. **Pipe *Exit Diameters*** are either the same size as the *Inlet Diameter* (for a simple tube) or is larger when modeling a *tapered pipe segment*. If the inlet and exit diameters are different, the simulation models the pipe segment using a perfect taper and displays the **Taper Angle** used in the calculation.

Primary and Secondary Collector Lengths, Inlet Diameters, And Exit Diameters

The *Collector Length* begins at the end of the previous pipe or collector and terminates at the next junction or at the atmosphere. The **Collector *Inlet Diameters*** are the inside diameter of the pipe at the point of connection to the previous pipe(s). When the inlet diameter is larger than each of the previous pipe(s) exit diameter, a *Step* is created that will generate a rarefaction wave. **Collector *Exit Diameters*** are either the same size as the *Inlet Diameter* (for a straight collector) or is larger or smaller when modeling a *tapered collector*. If the inlet and exit diameters are different, the simulation models the pipe segment using a perfect taper and displays the **Taper Angle** used in the calculation.

Megaphone Length And Exit Diameter

This is the **Length** and **Exit Diameter** of the Megaphone. *Note that the inlet of the Megaphone is always set to the exit diameter of the previous pipe segment (you cannot create a Step at the entrance of a megaphone).* By entering an *Exit Diameter* larger than the *Inlet Diameter,* the simulation will model the

Exhaust Category—Wave-Action Model
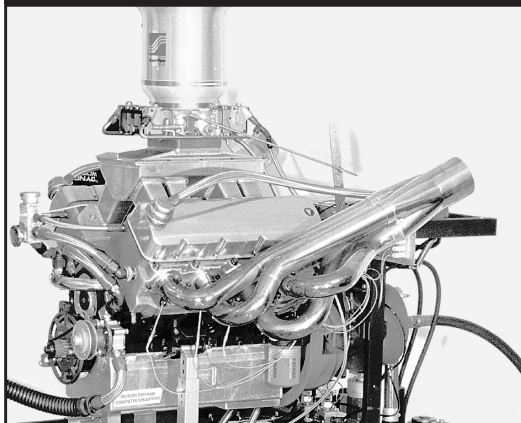
The Dynomation6 *Wave-Action* exhaust simulation can model virtually any headers design, including the ubiquitous 4-to-1 header. Select the header type from the *Header Design* menu: in this case *Basic Header—(Pipes Merge At Collector)*. This activates the appropriate *Primary Pipe* and *Collector One* elements. The model will join the exit of the primary pipe(s) to the inlet of the collector. The end of *Collector One* is assumed to terminate at the atmosphere.

megaphone using a perfect taper and display the **Taper Angle** used in the calculation.

Note: Only the Wave-Action exhaust elements currently in use (not dimmed) are active, based on the current selection from the **Header Design** menu. When you switch to a different Header Design, Dynomation will remember current pipe specifications (even those not used in the new model just selected), so if you switch back to a previous model, the values you entered remain available.

Real-World Examples Of WA Exhaust Header Modeling

Dynomation6 Wave-Action exhaust modeling offers a "palette" of components that you can use to build virtually any header design in just minutes. Start by selecting the basic configuration that matches your overall design from the **Header Design** menu in the Exhaust Category. Next, enter the dimensions for the elements that are active in your design (not dimmed). When complete, the simulation will run and use the system model you have defined. Let's build a couple of real-world exhaust models to illustrate this process.

"Standard" 4-into-1 Header

We'll start out with a typical 4-to-1 header, where four cylinders (or four of eight cylinders in a V8 engine) are merged together using a collector.

Note: The term "collector" and "collector extension" are sometimes confused or have unique meanings in different parts of the world. To be consistent in Dynomation, the term **collector** will refer to the pipe that "collects" the outflow of other pipes and merges them into a single flow stream (usually of larger cross-sectional area). The length of the collector is the length of this merge area PLUS the length of remainder of the collector pipe that, at its end, opens

Exhaust Category—Wave-Action Model

The key to determining where the Primary Pipe ends and the Collector(s) begin is to locate the point of greatest area change within the header. This is where the strongest rarefaction waves are generated. On this header, the area change is the greatest at approximately the middle of the merge sections of the collectors.

Measuring Exhaust Tubing Lengths

Collector Two Length

Collector One Length

Point of Greatest Area Change

Point of Greatest Area Change

Point of Greatest Area Change
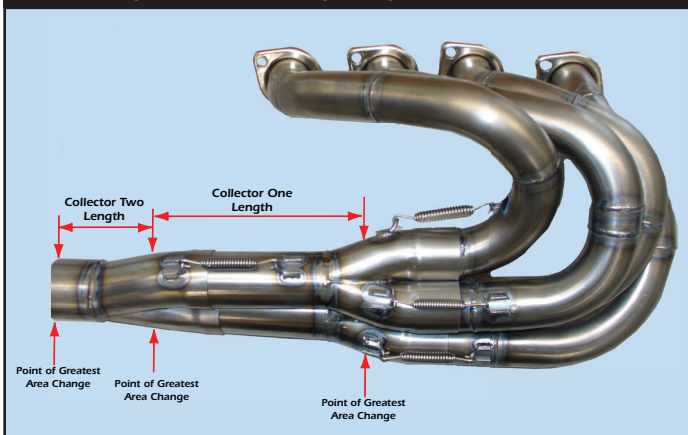
to the atmosphere. The term "extension," if used in this context, will mean any slip-on/removable pipe used to extend the overall length of the collector (modeled by adding a 2nd collector with the same entrance and exit diameter).

To activate the elements needed for the "standard" header, first make sure that the WA Simulation is active by clicking the **Wave Action Model** radio button in the Simulation Category. Then select *Basic Header—(Pipes Merge At Collector)* from the **WA Header Design** menu in the Exhaust Category.

The exhaust component menus will activate the **PRI** (Primary Pipe) and the **COL1** (Collector One) groups. Enter the primary tube length by clicking on the "*******" or the current value displayed in the *Primary Tube Length:* field. You can select a value from the menu or you can directly enter any value within the limits of the program (shown in the Status Bar at the bottom of the program screen). Move to the *Inlet Dia:* field by either clicking on the field or (without using the mouse) press *Enter* to accept the *Length:* and automatically jump to *Inlet Dia:*. Press *Enter* again to open that field**,** enter a value, then continue pressing *Enter* to move through the remaining fields. When all data for the Primary Pipe and Collector One groups has been entered, the Exhaust Category will be complete (providing you have also entered a value in the *Minimum Exhaust Port Area* and *Pipes Per Cylinder* fields, as described earlier in this section).

NOTE ABOUT EXHAUST TUBING LENGTHS: As far as simulation is concerned, the lengths of the various elements of the exhaust system are determined by the location of the greatest area transition between one element and the next. This is where the strongest rarefaction wave is returned to the exhaust valve to assist exhaust-gas outflow.

When measuring the ending point between a primary and secondary pipe, the point of transition is easy to see; it's the junction of the two elements.

Exhaust Category—Wave-Action Model

However, the functional transition between merging pipes and a collector is not as clear. When a pressure pulse reaches a collector junction, the simulation "sees" the strongest evidence of the junction when the outgoing pressure pulse reaches the area of greatest change between the pipe and collector. However, depending on the interior shape of the collector (where the primary pipes end within the collector, whether the pipes have transition "cones," etc.), it may occur closer to the multiple-pipe merge entrance or closer to the straight portion of the collector. If you are in doubt about the exact location of a transition, use the center of the merge span of the pipes into the collector.

Add A "Step" To The Primary Pipe

If you would like to model a step in the Primary Pipe before the Collector, select **Basic Header—W/One Step** from the *WA Header Design* menu. This will activate the *Secondary Pipe* group. Since the Step is a transition from one pipe size to a larger diameter, the Step acts as the beginning of a Secondary element in the exhaust system.

Enter the *Length:*, *Inlet Dia:* and *Exit Dia:* values for the Secondary Pipe. Note that you may have adjust the Primary Pipe length if you wish to keep the overall length of tubing at an optimum length for the application (the *Pre-Collector Pipe Length:* is displayed at the bottom of the Exhaust Category).

A Tri-Y Header With Two Steps

Finally, let's consider a more complex model: a Tri-Y header (with two collectors) and a primary section incorporating two steps (modeling Primary, Secondary, and Tertiary pipe sections). Select *Tri-Y—(Two Collectors) W/Two Step Primary* from the **WA Header Design** menu.

This menu choice will activate the **PRI, SEC, TER** pipes (Primary, Secondary and Tertiary Pipes) and the **COL1, COL2** (Collector One and Two) groups. Enter in the Primary, Secondary, and Tertiary pipe data as before. Keep in mind that all of these lengths combine to make up the total pipe length before the

Wave-Action Stepped-Header Modeling

These custom Tri-Y headers have stepped Primary pipes that increase in diameter about 8-inches from the port flange. The larger tubes them merge into two first collectors (*Collector One*) which then merge into a second collector (*Collector Two*) which finally terminates at the atmosphere.

Exhaust Category

Custom Header Pipe Segments

If you build your own headers, pre-formed pipe segments, like these tapered collectors from _Cone Engineering_ makes the job much easier.

first collector. Then enter Collector One and Collector Two data, with the first Collector **Length:** based on the information in the note **ABOUT EXHAUST TUBING LENGTHS** presented earlier in this section. Typically, these distances are measured from the <u>middle of the first merge section</u> to the <u>middle of the second merge section</u> and the Second Collector length measured from the <u>middle of the second merge section</u> to the <u>end of the open pipe</u> at the atmosphere.

Exhaust Category

SIMULATION RESULTS

(4) Graph Options

(2) DataSim™ Window

(7) 3D Engine Rendering

(1) Main Results Graphs

(3) Horizontal Dividers

(4) Graph Options Axis Data

(5) Graph Properties And Data Zones™

(1) Display Left Graph

(3) Vertical Divider

(6) Display Data Tables
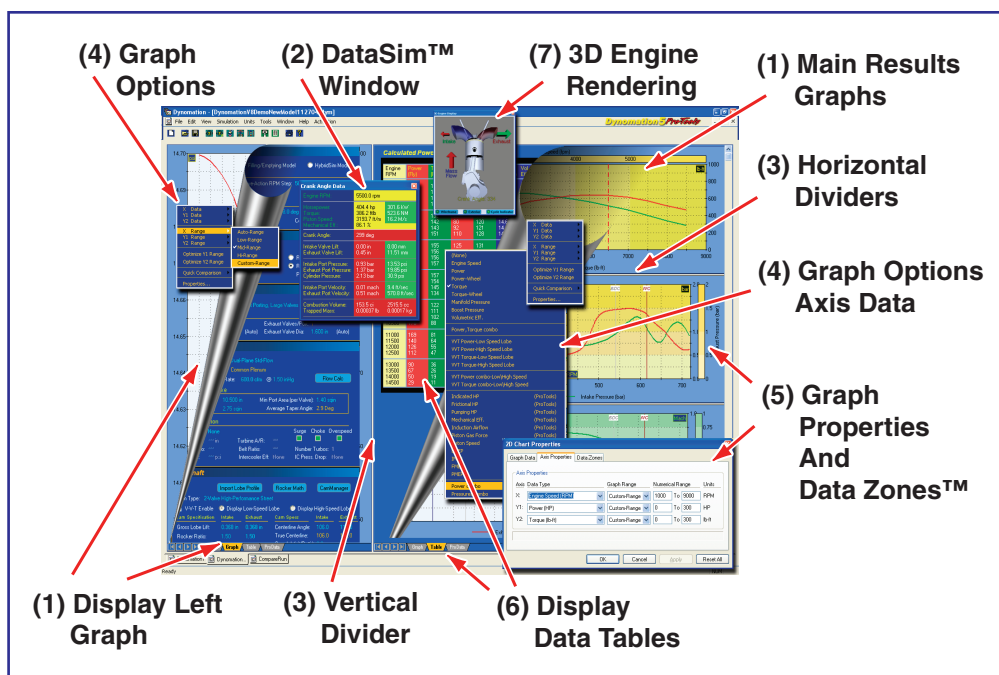
Applies To All Simulations
Except as Indicated

The **Simulation Results** graph, table and other data displays are designed to help you retrieve the most information from any simulation. Each of the numbered areas shown above are described below:

1) **Main Results Graphs—**The main program screen is divided into two main sections (the left and right panes), with the component-selection categories on the left and the main results graphs/tables on the right. The *Wave-Action* and *Filling-And-Emptying* simulations display engine performance (horsepower and torque, by default) in the top-right results graph. The simulation also analyzes and displays intake, exhaust, and cylinder pressures and port-flow velocities

Simulation Results Displays

Graph Reticule Lines

All results graphs can display a red-dotted reticule line to indicate specific points within the data being displayed. For example, the top horsepower graph reticule line indicates a specific engine speed, while the center and lower graphs point to a crank-angle (for Wave-Action simulations only). To "activate" any of these reticule lines, simply click on the graph.

Graph Reticule Lines

which are presented in the in the center and lower crank-angle-based graphs (and in the graph located under the Component Categories activated using the **Display Data Tabs** [#6] in the photo on the left). For more information on how to interpret port pressure and flow graphs, refer to **Wave-Dynamics Analysis** (later in this manual).

Graph Reticule Lines

All results graphs can display a red-dotted reticule line to indicate specific points in the data being displayed. For example, the top horsepower graph reticule line indicates a specific *engine speed*, while the center and lower graphs point to a *crank-angle* (for Wave-Action simulations only). To "activate" any of these reticule lines, simply click on the graph. To use the reticule lines, first drag the reticule on the top horsepower graph to set an rpm point. Next, click on either

Left Underlying Graph

In addition to the three graphs displayed on the right side of the main program screen, a fourth graph is available by clicking on the *Graph* tab located at the bottom of the Component Category pane (left Pane). This graph shows Cylinder-Pressure vs. Cylinder-Volume by default (commonly known as a P-V diagram), however, you can re-assign any crank-angle based data to either the Y1 or Y2 graph axis and either crank-angle or cylinder-volume to the X axis.

the center or lower graph to set a specific crank angle. The exact values "un-der" these reticules is displayed in the *Crank-Angle DataSim™* window (see #2, below).

Graph Underlying Component Categories

In addition to the three graphs on the right pane of the Main Program Screen, an additional graph is available by clicking on the ***Graph*** tab located at the bottom of the left pane. This graph, like the center and lower graphs on the right pane, is based a "crank-angle" display. The graph typically displays *Cylin-der Pressure* in bar (1bar = atmospheric pressure) vs. *Cylinder Volume*, often called a **P-V**, or pressure-volume graph.

Note-1: For more information about interpreting P-V diagrams, see ***Pressure-Wave Dynamics***, later in this manual.

Note-2: The center, lower, and graph under the component categories dis-play data curves when: **1)** A simulation has been completed, and **2)** The top *Horsepower/Torque* graph reticule line has been activated (by clicking on the graph) to set a specific rpm point for which the crank-angle data is displayed.

2) ***DataSim™ Window***—The exact values "under" the reticule lines on all graphs are displayed in the *Crank-Angle DataSim™* window. Open this floating display by clicking the ***DataSim™ Icon*** on the Main Program **Toolbar** or by selecting ***Crank-Angle DataSim™ Window*** from the **Tools** drop-down menu. Moving the reticule line on the (top-right) rpm-based graph and/or any crank-angle based graph will update all *DataSim™* fields. You can step forward and backwards through crank-angle data one-degree at a time by first clicking on any crank-

The exact values "under" reticule lines are dis-played in the *DataSim™* window (also, see illustra-tion, next page). Open this helpful tool by clicking the *DataSim™* Icon on the Main Program *Toolbar* or by selecting *Crank-Angle DataSim™ Window* from the *Tools* drop-down menu. You can step forward or backwards through crank-angle data one-degree at a time by first selecting a graph (just click on it), then press either the right or left arrow keys. Hold the SHIFT key to increase steps to 10-degrees per keypress. This photo shows the latest DataSim Window values, that now consists of twice as many data points as before. Make sure to upgrade to the latest version of your simulation; it's free for the life of the program.

DataSim™ Window

Engine RPM:	6400.0 rpm	
Horsepower:	467.6 hp	348.7 kW
Torque:	383.7 ftlb	520.2 NM
Piston Speed:	3200.0 ft/m	16.3 M/s
Mechanical Eff.:	89.0 %	
Crank Angle:	0 deg	
Intake Valve Lift:	0.00000 in	0.0000 mm
Exhaust Valve Lift:	0.00000 in	0.0000 mm
Intake Lobe Lift:	0.00000 in	0.0000 mm
Exhaust Lobe Lift:	0.00000 in	0.0000 mm
Intake Port Pressure:	0.900 bar	13.06 psi
Exhaust Port Pressure:	0.974 bar	14.13 psi
Cylinder Pressure:	39.178 bar	568.2 psi
Intake Port Velocity:	-0.023 mach	-26.1 ft/sec
Exhaust Port Velocity:	-0.059 mach	-66.2 ft/sec
Piston POS (from TDC):	0.000 in	0.00 mm
Combustion Volume:	3.973 ci	65.10 cc
Trapped Mass:	0.001593 lb	0.7227 g
Int Discharge Coef:	0.695	0.695
Exh Discharge Coef:	0.680	0.680
Intake Flow Area:	0.000 sqin	0.0 sqmm
Exhaust Flow Area:	0.000 sqin	0.0 sqmm

Simulation Results Displays

Simulation Crank-Angle Data Displays

Wave-Action Simulation Port Pressures And Velocities

(2) Crank-Angle Data-Sim™ Window

(1) Crank-Angle Data Displays

Center Graph

(3) Port Pressures

(4) TDC/BDC Piston Position Indicators

(5) Port Velocities

(6) Valve-Event Timing Indicator Lines

Lower Graph

Data Values "Under" Dotted Reticule Lines
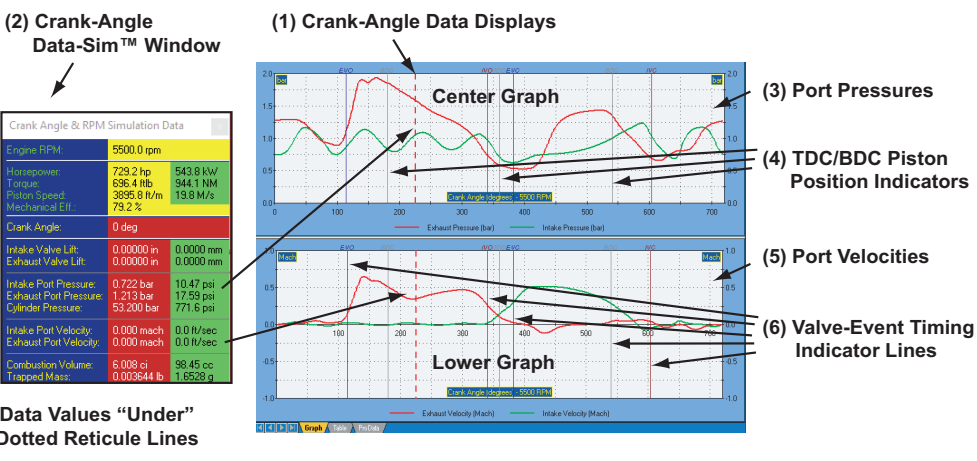
The simulation Crank-Angle Data Displays (1) reveal an entire spectrum of pressures and flow velocities for the simulated engine. The crank-angle *Port Pressures* graph (3) displays intake, exhaust, and (optionally) cylinder pressures and other data. *Port Gas Velocities* for intake and exhaust flow are shown by default in the lower graph (5). Exact values of displayed data are shown in the *Crank-Angle DataSim™* window (2). Simply position the *Dotted Reticule Line* over the data you wish to examine. The *Crank-Angle* graphs also have indicator lines for valve-event timing (6) and piston position (4). Studying these pressure curves can help locate inefficiencies and diagnose tuning problems (for more information about how to interpret the port pressure and flow graphs, refer to *Wave-Dynamics Analysis* later in this manual).

Any of the four screen-pane dividers can be moved to allow specific data displays more screen "real estate." You can drag the vertical divider to the left or right and move any horizontal graph dividers to the top and bottom to show graphs at any size you wish.

Moveable Screen Pane Dividers
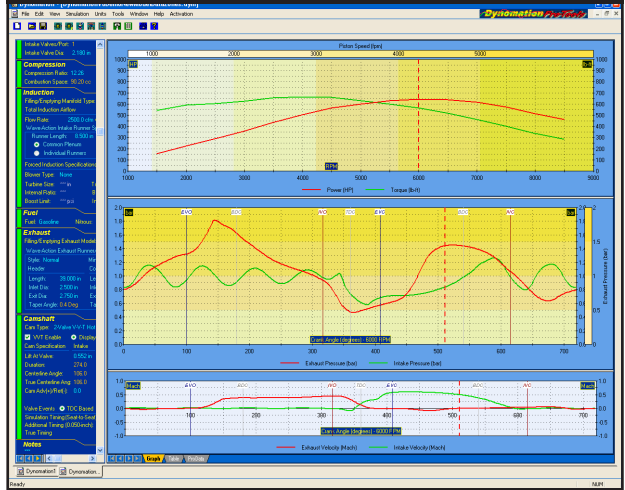
angle graph (to select it), then pressing either the right or left arrow keys. Holding **SHIFT** key will increase the cursor key steps to 10-degrees.

3) **Vertical And Horizontal Screen Dividers—**The center divider between the left and right panes and the horizontal dividers between the right-pane graphs can be moved (click and drag) to re-size any results display to suit your requirements. When any screen area has been adjusted, the graphs will redraw and re-scale to take advantage of changes in display area. Divider positions are not "remembered" from session to session (however, all other graph and display defaults are stored in the **.dxml** engine simulation file and will be reapplied the next time the engine file is opened).

4) **Graph Options Menu—**Right click on any graph to open the *Graph Options Menu.* This menu provides fast access to custom graph settings, including data displayed, axis scaling, establishing side-by-side comparisons, and more:

Axis Data

The first three choices from the *Graph Options Menu* allow you to customise the data displayed on each of the three graph axis (Horizontal **X** Axis, Left Vertical **Y1**, and Right Vertical **Y2**; see photo on next page). Each of these axis can be assigned a variety of simulation results (the exact selection varies depending on whether the **X** Axis is *rpm*, *crank-angle* or *cylinder-volume*). Some the data-set choices are: Indicated, Frictional, Pumping, and Brake Horsepower, Wheel Horsepower, Torque, Volumetric Efficiency, a variety of Pressures (Imep, Fmep, Pmep, Bmep, Manifold, Boost), velocities (Intake and Exhaust Port Flow), Intake and Exhaust Valve Lifts, CD's and Curtain Areas, Cylinder Volume, Mechanical Efficiency, Induction Airflow, Gas Force On The Piston and more.

Axis Scaling

The second group in the *Graph Options Menu* allows you to select several methods of axis scaling. You will notice that each axis can be scaled to a

Right-click any graph to show the *Graph Options* menu. Use the *Data* choices to assign any simulation data set to any axis you desire (within the graph family of either *rpm-based*, or *crank-angle* based data). The *Range* choices provide a variety of fixed and auto-scaling options for each axis. Use the *Optimize Y(x) Range* selections to quickly set optimum (fixed) scaling for any currently displayed data. For even more options, choose *Properties* to open the *Graph Properties Dialog* box.

Simulation Results Displays

Optimize Y1/Y2 With Fixed Scaling
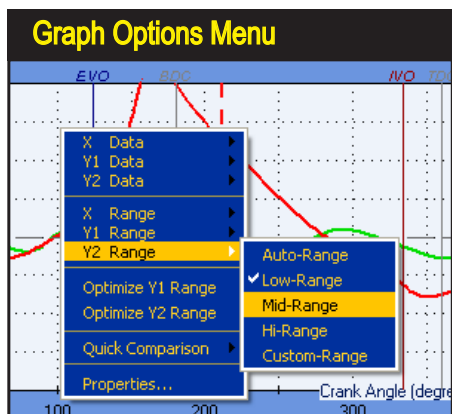
Results graph consists of two vertical axis, a left and right. Either of these axis can be quickly set for optimum viewing using the *Optimize Y1* (left axis) or *Optimize Y2* (right axis) choices (lower graph on right is *Optimized* display of exactly the same data). Optimize choices apply Fixed-Axis scaling (axis will not re-scale automatically). To reset the graph back to normal scaling, select one of the *Range* options, or select *Properties* from the bottom of the Options Menu and click *Reset All*.
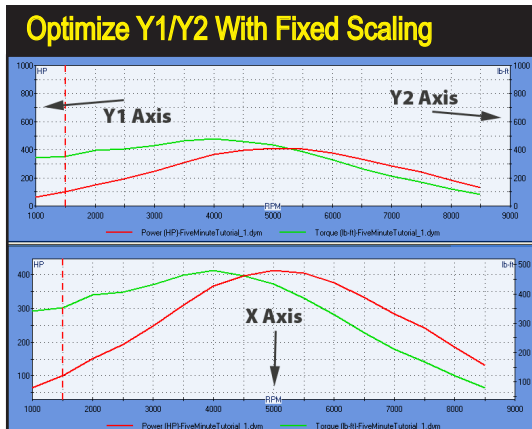
fixed *low*, *medium*, and *high* value. In addition, an auto-scaling feature can be enabled for any axis (turned off, by default). Use any fixed-axis scaling to maintain constant axis values. This establishes a "fixed visual baseline," making it easy to see changes in power or torque as engine specifications are modified (graph won't rescale). However, when component changes dramatically alter data, use the auto-scaling feature will ensure that the entire data set is always visible and shown at 80 to 90% of full graph height for good resolution.

Optimize Y1(Left) / Y2(Right) Ranges

The *Graph Options Menu* includes two **Optimize Y1/Y2 Range** selections that quickly establish optimum viewing using *Fixed Axis Values* (axis will not re-scale automatically after it has been set). To reset the graph back to normal range scaling, select one of the *Range* options described in the previous paragraph, or select **Properties** from the bottom of the *Options* menu and click **Reset All** to restore the display (only on the selected graph) to default values.

The Graph Options Menu also contains a *QuickCompare™* feature that instantly establishes any engine as a "baseline," allowing easy-to-see changes in engine performance from that point onward.

QuickCompare™ Graphic Display

Simulation Results Displays

QuickCompare™ Baseline Testing
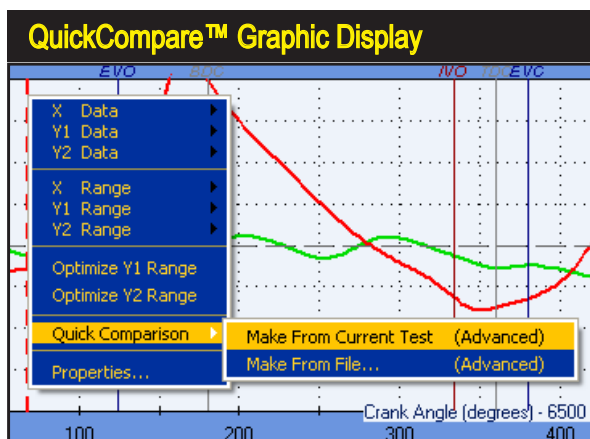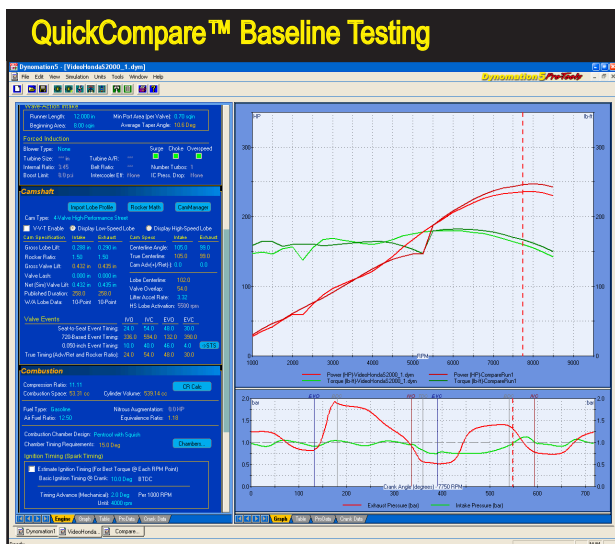
To apply *QuickCompare™* simply right-click on a graph on which you wish to establish the comparison, then select *QuickCompare™,* finally select *Make From Current Test.* Changing engine components or specifications from that point forward will instantly show "side-by-side" performance differences.

QuickCompare™

The *Graph Options Menu* also contains a *QuickCompare™* feature that instantly sets any engine as a comparison baseline, allowing easy-to-see changes in engine performance from that point onward. To setup a comparison using the currently simulated engine, right click on a graph for which you wish to establish the comparison, select **Quick Compare™**, then select **Make From Current Test**. This will "copy" the current engine to an additional engine instance, name the new simulation *CompareRunX*, and setup a back-to-back comparison with the current engine on your display. Changing any component on the current engine will instantly show any simulation differences. The *QuickCompare™* feature also allows similar comparisons with any saved (.dxml) engine file (think of the saved engine as the comparison baseline). Choose the **Quick Compare™**, then select **Make From File**. This will open the saved engine, name the simulation *CompareRunX*, and setup a back-to-back comparison with the current engine on the selected graph.

Graph Properties—Graph Data
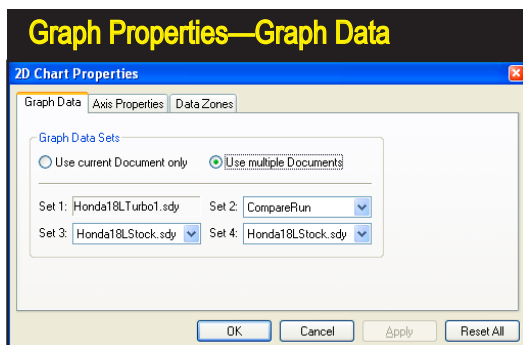
Use the *Graph Options Properties Dialog* box to establish on-graph comparison of up to four engines using the *Graph Data* tab. Make sure "Use Multiple Documents" is enabled, then select the comparison engines from the *Graph Data Sets* drop-down menus. To cancel any of these comparisons, select "Use Current Document Only."

Simulation Results Displays

Compare Up To Four "Open" Engines
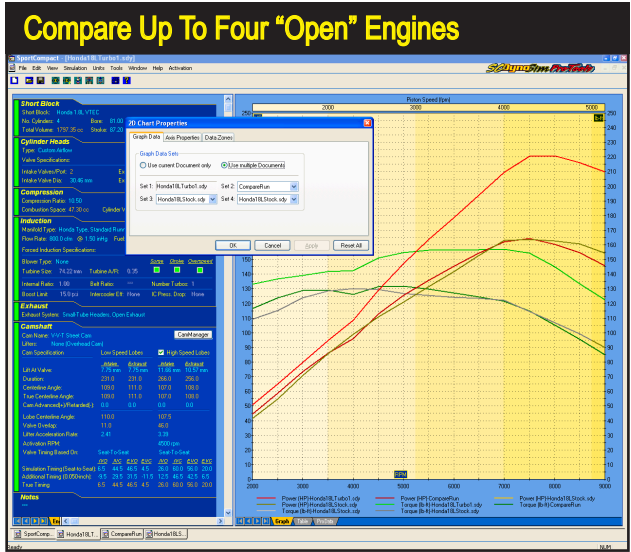
This comparison of three engines was setup using the *Graph Options Properties Dialog*. Up to four "open" engines can be compared on any graph. This graph shows how horsepower and torque varied on all three test engines.

5) **Graph Properties Dialog**—This dialog contains three tabbed pages that provide detailed control of the current graph. Here is an overview of each tab:

Graph Data: Using the drop-down lists on this page you can quickly establish on-graph comparisons with up to three additional different engines. The engines you wish to include on the current graph must be "open," with active **Selection Tabs** at the bottom of the **Main Program Screen**. First, make sure the **Use Multiple Documents** radio button is selected (allows multiple data sets). Next, select the specific engine you wish to add using the **Graph Data Sets** drop-down menus. Up to three additional engines can be selected using **Set 2**, **Set 3** and **Set 4** menus. When you click **Apply** or **OK**, the current graph will redraw and include the desired engine-data comparisons. Legends at the bottom of the graphs will provide keys to all graphic curves.

Graph Properties—Axis Properties

Change the data and display characteristics of the current graph by modifying the *Data Type*, *Graph Range*, and *Numerical Range* using the Axis Properties tabbed page

Simulation Results Displays

Graph Properties—Data Zones™
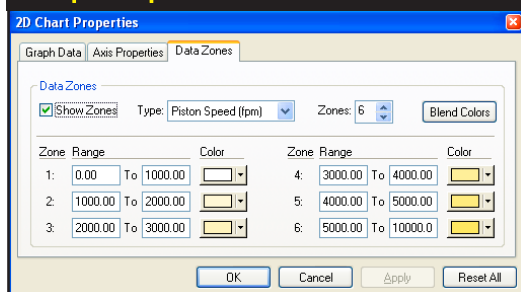
DataZones™ extend the graphic-display and data-analysis capabilities of the simulation. Using this feature, you can display additional engine data, show ranges, or clearly label dangerously high pressures, engine speeds, and more.

Axis Properties shows the current **Data Types** and **Ranges** for the current graph. Modify any characteristic by changing properties. Create custom numeric ranges by selecting **Custom-Range** from the bottom of any of the drop-down **Range** menus).

DataZones™ Displays: **DataZones™** extend the graphic-display and data-analysis capabilities of the simulation. Using this feature, you can display additional engine data and/or show ranges for target power values or label dangerously high pressures, engine speeds, and more. To setup a **DataZone**, first select the data that you would like to display from the **Type** drop-down menu. Next, select the number of **DataZones** you would like to display by clicking on the "up" or "down" arrows next to the **Zones** field. You can modify the **Range** values and **Colors** for each zone (to have the simulation build a uniform transition between these colors for intermediate zones, set a starting and ending colors then press **Blend Colors**). Click

Piston-Speed DataZone™
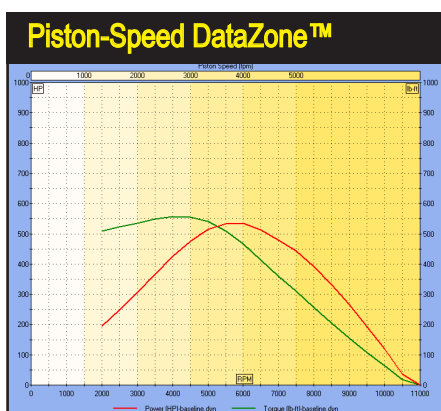
This **DataZone** display shows Piston-Speed bands drawn on the standard HP/Torque graph. The range values are indicated at the top of the graph.

Multiple DataZones™ Defined
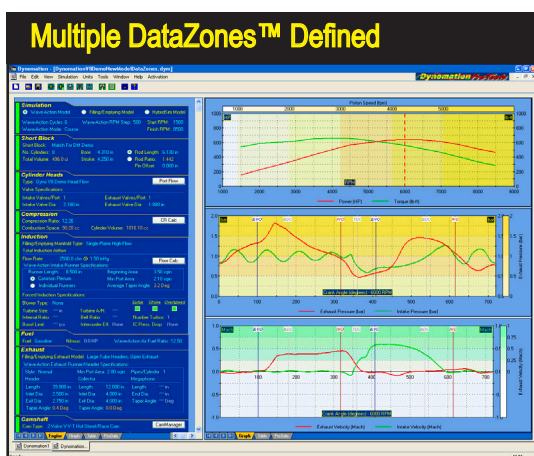
DataZones™ can be defined for any of the four graphs available in the simulation. DataZone setups are saved with .dxml engine files.

Simulation Results Displays

Table Shows Exact Values

In addition to extensive graphing capability, the simulation offers several chart displays that can be viewed by clicking any *Table, ProData, or Crank Data* **tab located at the bottom of either the left or right pane. The tables list engine variables recorded during the simulated dyno run. In this illustration, exact simulation results are displayed in 500rpm increments from 2000 to 11,000rpm.**
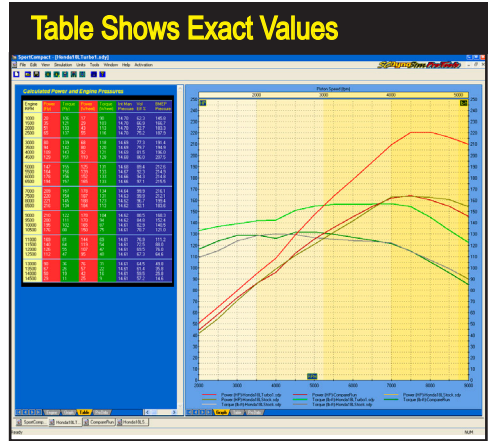
on *Apply* or *OK* to draw the specified zones on the main graphic display. **Note:** *DataZone* variables can only be selected if they are the <u>same as</u>, or <u>directly derived from</u>, one of the variables currently displayed on the graph. In other words, it has to "make sense" to display that data set on the current axis. For example, you can display *Piston Speed DataZones* on standard Horsepower/Torque graphs since *Piston Speed* is directly calculated from engine rpm, but *Manifold Pressure* cannot be displayed, since it is not <u>directly calculated</u> from (and its units are not related to) engine rpm, power, or torque, the three main data sets displayed on the standard Horsepower/ Torque graph.

Reset All is available at the bottom of the *Graph Properties Dialog* that lets you reset the current graph back to default settings, including the default axis data, scaling, etc. (this full reset applies only to the selected graph).

Engine-Pressures, Rpm-Based Data Table

PROTOOLS CALCULATED POWER AND ENGINE PRESSURES

Engine RPM	Power (Fly)	Indicated Power	Frictional Power	Pumping Power	Mech. Eff %	Induction Airflow	Piston Force	Piston Speed	IMEP Pressure	FMEP Pressure	PMEP Pressure
2000	86	124	21	16	70.5	125.7	1535	1333	108.2	18.2	13.7
2500	130	176	28	16	74.9	180.4	1745	1667	122.9	19.9	11.0
3000	169	225	37	16	76.2	229.9	1853	2000	130.5	21.7	9.4
3500	229	295	47	16	78.4	308.4	2089	2333	147.2	23.6	8.1
4000	301	383	59	18	79.8	403.8	2370	2667	167.0	25.7	8.0
4500	377	477	72	23	80.1	502.3	2626	3000	185.0	28.0	8.9
5000	441	561	87	27	79.7	596.1	2776	3333	195.6	30.4	9.4
5500	486	626	104	30	78.7	680.3	2815	3667	198.4	32.9	9.4
6000	510	671	123	31	77.1	747.4	2768	4000	195.1	35.6	9.0
6500	521	704	143	32	75.1	810.0	2680	4333	188.8	38.5	8.5
7000	507	711	166	31	72.3	854.0	2513	4667	177.1	41.4	7.7
7500	490	719	192	30	69.2	888.6	2371	5000	167.1	44.6	6.9
8000	443	696	219	27	64.6	912.1	2152	5333	151.7	47.8	5.9
8500	405	685	250	25	59.9	932.1	1994	5667	140.5	51.3	5.1
9000	346	655	283	21	53.6	945.4	1802	6000	127.0	54.8	4.1
9500	271	611	319	17	45.1	938.7	1591	6333	112.1	58.5	3.0
10000	224	599	358	14	38.0	961.0	1482	6667	104.4	62.4	2.4
10500	131	541	400	8	24.6	951.9	1275	7000	89.8	66.4	1.3
11000	55	504	445	3	11.1	953.0	1134	7333	79.9	70.5	0.5

This is the *ProData™* **table (display by clicking the** *ProData* **tab at the bottom of the left and right display panes). The table provides a detailed listing of engine pressures, piston speeds, gas forces, induction airflow, mass flow and more.**

Simulation Results Displays

Crank-Angle Data Table

Calculated Crank Angle Data

Crank Angle	Intake Pressure (bar)	Exhaust Pressure (bar)	Cylinder Pressure bar	Trapped Mass (LB)	Intake Velocity Mach	Exhaust Velocity Mach	Cylinder Volume (CI)	In Valve Lift (IN)	Ex Valve Lift (IN)
0	0.99	0.98	30.14	0.00121	-0.00	0.02	13.02	0.000	0.000
5	0.99	0.96	52.23	0.00121	-0.00	0.02	13.23	0.000	0.000
10	0.99	0.94	77.51	0.00121	0.00	0.01	14.27	0.000	0.000
15	0.99	0.92	81.08	0.00121	0.00	0.01	15.46	0.000	0.000
20	1.00	0.91	72.03	0.00121	0.00	0.01	18.00	0.000	0.000
25	1.01	0.90	60.31	0.00121	0.00	0.00	20.83	0.000	0.000
30	1.01	0.90	49.08	0.00121	-0.00	-0.00	24.26	0.000	0.000
35	1.00	0.91	41.74	0.00121	-0.00	-0.01	27.35	0.000	0.000
40	0.99	0.92	33.09	0.00121	-0.00	-0.01	32.48	0.000	0.000
45	0.99	0.94	28.60	0.00121	-0.00	-0.01	36.19	0.000	0.000
50	0.99	0.97	23.30	0.00121	-0.00	-0.02	42.13	0.000	0.000
55	0.99	0.99	20.51	0.00121	0.00	-0.02	46.30	0.000	0.000
60	0.99	1.01	17.17	0.00121	0.00	-0.01	52.80	0.000	0.000
65	1.00	1.04	14.60	0.00121	0.00	-0.01	59.56	0.000	0.000
70	1.01	1.05	13.20	0.00121	0.00	-0.01	64.18	0.000	0.000
75	1.01	1.07	11.47	0.00121	-0.00	-0.01	71.22	0.000	0.000
80	1.00	1.08	10.51	0.00121	-0.00	-0.00	75.97	0.000	0.000
85	1.00	1.08	9.31	0.00121	-0.00	-0.00	83.13	0.000	0.000
90	0.99	1.08	8.48	0.00121	-0.00	0.00	89.02	0.000	0.000
95	0.99	1.08	7.81	0.00121	-0.00	0.00	94.70	0.000	0.000
100	0.99	1.07	7.25	0.00121	0.00	0.01	99.05	0.000	0.000

The simulation can also display a table of crank-angle-based pressures and other data derived at each 5-degrees of crank rotation throughout the 720-degrees of the four-cycle process.

6) **Exact-Value Table Displays—**The simulation also offers several tables of data generated during the simulation process. These exact-value resources are displayed by clicking the **Table**, **ProTable**, or **Crank Data** tabs located at the bottom of the left or right display panes. The **Table** and **ProTable** charts display engine variables recorded at the same step-rpm and range that was used during the simulation (set in the *Simulation Category* at the top of the main program screen). The **Crank Data** table shows a "snapshot" of crank-angle based data taken at each 5-degrees throughout the 4-stroke process. Scroll down the table to view the changes in flow at IVO; see reversion just before IVC, examine valve lift, cylinder pressures, and more.

7) **3D-Engine Pressure/Flow Display—**A real-time, 3D-Engine rendering (available

The simulation includes a 3D-Engine rendered in real time that shows valve and piston positions synchronized with the reticules on the crank-angle graphs. Port pressures and velocities are indicated with shades of color and directional arrows. *Mass Flow* (into and out of the cylinder) is also indicated. This can be a very helpful tool to visualize gas dynamics and the otherwise-hidden characteristics of a running engine.

3D Transparent Engine
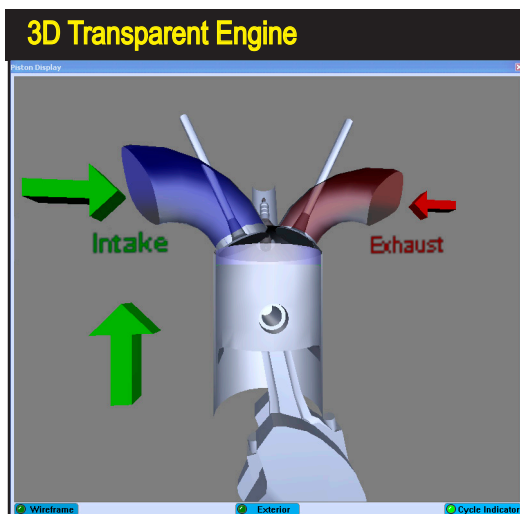
Simulation Results Displays

To activate the 3D Engine Display either click on the *3D Engine* icon in the Toolbar or select *3D Engine Pressure/ Flow Display...* from the *Tools* menu.
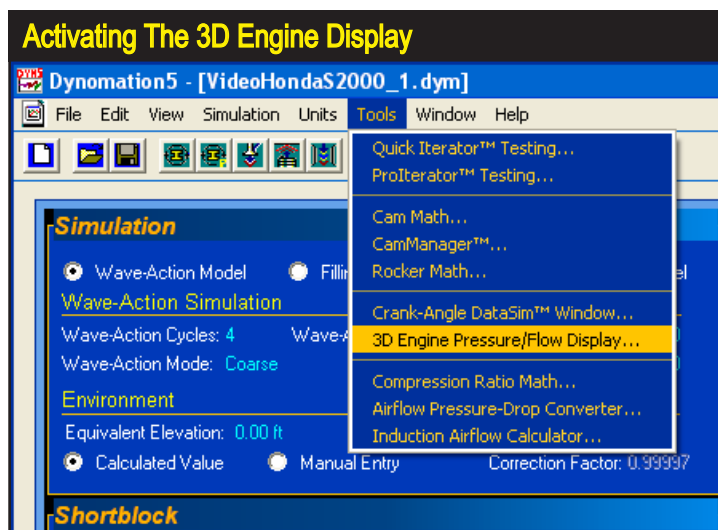
for both simulation modes: *Filling & Emptying and Wave-Action* shows valve and piston positions synchronized with the reticule locations on the crank-angle graphs and with the data displayed in the ***DataSim™*** window. Port pressures and velocities are indicated with shades of color and directional arrows. You can also see a *Mass Flow* indicator (flow into and out of the cylinder). This display can be helpful in visualizing gas flow and otherwise-hidden characteristics of a running engine. For additional information about interpreting the data shown in the *3D Engine* display and the crank-angle graphs to which it is synchronized, refer to the *Wave Dynamics Analysis* chapter later in this manual.

Display the 3D Engine view by clicking the **3D Engine** icon in the **Toolbar**, or by selecting **3D Engine Pressure/Flow Display...** from the **Tools** menu.

The main features of the 3D Engine display are shown in a nearby photo and described below:

1) **Intake Port Color Intensity**—The intake port wall color varies from gray to a saturated blue. This color is directly linked to the measured pressures (wave intensities) within the intake port. Low pressures (expansion waves) produce a colorless gray, while a strong blue is generated when a high pressures (compression waves) reach the valve.

2) **Exhaust Port Color Intensity**—The exhaust port wall color varies from gray to a saturated red. This color, similar to the intake port, is directly linked to the measured pressures (wave intensities) within the exhaust port. Low pressures (expansion waves) produce a colorless gray, while a red is generated when a high pressures (compression waves) reach the valve.

3) **Intake Port Flow Arrow**—Flow velocity and direction within the intake port are indicated by this arrow. When the arrow is green and points inward, toward the port, overall induction flow is moving toward the valve.

Simulation Results Displays

3D Engine Display Features

Intake Port Blue Intensity (1)
Indicates Port Pressure Level

(2) Exhaust Port Red Intensity
Indicates Port Pressure Level

Intake Arrow (3)
Indicates Flow
Rate And Direction

(4) Exhaust Arrow
Indicates Flow
Rate And Direction

Gray Arrows (5)
Indicates Net
Zero Flow
(Valve Closed)

Mass Flow (6)
Indicator
In/Out Of Cylinder
(*ProTools™*)

(7) Crank Angle
Position Indicator
(720° Based)

Display Only (8)
Wireframe

(9) Display Exterior
Engine Outline

(10) Display Color
Engine-Cycle "Fill"

The size of the arrow indicates the rate of flow. A red arrow pointing outward indicates reverse flow (reversion of charge back into the intake port). *Also see 5, Gray Arrows*, below, for more information about arrow displays.

4) **Exhaust Port Flow Arrow—**Flow velocity and direction within the exhaust ports is indicated by this arrow. Green arrows pointing outward indicates normal exhaust flow moving away from the valve. A Red arrow pointing inward, toward the valve, indicates reverse flow (reversion of exhaust gasses moving back into the cylinder). The size of the arrow indicates the flow rate. *Also see 5, Gray Arrows*, below, for more information about arrow displays.

5) **Gray Arrows—**When either the intake-port or exhaust-port arrow turns gray, this indicates that the valve within that port has closed, and net flow is zero. The arrow, however, may oscillate inward and outward as pressure pluses and particle velocity oscillates back and fourth in the port.

6) **Cylinder Mass Flow Arrow—**This arrow indicates the changes in "trapped mass" within the cylinder; the size of the arrow indicates the mass rate of change. A green downward arrow indicates that mass is flowing into the cylinder (e.g., during the intake stroke), and a red up-ward arrow indicates that cylinder mass is decreasing (e.g., during the

exhaust stroke). The *Mass Flow Arrow* also can also identify induction reversion during overlap and just before IVC. When there is no change in cylinder mass (when both valves are closed), the *Mass Flow Arrow* is no longer displayed.

7) **Crank Position Indicator**—The rotational position of the crank is displayed using 720-degree notation. The zero position starts at TDC on the beginning of the power stroke. This corresponds to the crank-angle graphs on the right of the Main Program Screen.

8) **Display Only Wireframe**—If you wish to display the 3D Engine using only the underlying wireframe model, click this control (faster screen draw).

9) **Display Exterior Engine Outline**—This control toggles on and off a transparent image of the exterior dimensions of the engine.

10) **Display Color Engine Cycle**—This control toggles on and off the color fill for the cylinder that indicates the current engine cycle: Red indicates the power stroke; Yellow for the exhaust stroke; Green for the intake stroke; and Blue for the compression stroke.

Note-1: You can control the size, view, and position of the 3D Engine. To "grab" the engine, rotate it, and position it within the window, hold down the left mouse and drag. To enlarge and reduce the size of the 3D Engine, hold down the right mouse button and drag inward and outward.

Note-2: You can step forward and backwards through crank-angle data one-degree at a time by first clicking on any crank-angle graph (to select it), then either the right or left arrow keys. Holding **SHIFT** key will increase the cursor key steps to 10-degrees. The *3D Engine* display will remain synchronized to the current crank-angle reticule line as it moves with each keypress.

Wave-Action Filling & Emptying Simulations 6

Version 6

QUICK-ITERATOR™

Iterator Running Status

Quick Iterator™ Dual-Phase Status

Find Optimum Cam Timing Group

Simulation Tests In Current Phase

Current Results

Best Results

Best Power/Torque Increase

Windows Close Button

Save Iterator Result

Find Optimum Bore/Stroke Group

With the power and speed of simulation makes is possible to fill file cabinets with *simulated* dyno-test data. In fact, it's easy to become "bogged down" in an overabundance of data. Sorting through simulation results, analyzing the best power curves, and selecting promising component combinations can turn into a job nearly as difficult as the old trial-and-error dyno testing method!

One solution to this problem was introduced by Motion Software's as *Iterative Testing™ (pronounced "it-er-ah-tive" and "it-ter-a-tor")*, a powerful feature available in both the FE and WA simulations. Iterative testing performs series of simulation calculations automatically until the final, optimum answer is obtained.

The simulation incorporates two versions of Iterative testing, *The Quick Iterator™* is extremely simple to use (refer to the next chapter for information on the advanced *ProTools™ ProIterator™*). By clicking a single button, the *Quick Iterator* will perform a test series to find optimum horsepower or torque. The *Quick Iterator*

Using The *Quick Iterator*™

When the *Quick Iterator* has completed its analysis, you can save the results (by clicking the *Save* button). The program will generate a new simulated engine with the component combination that produced optimum power or torque. The new engine will be added to the "open" engines included in the *Engine Selection Tabs* at the bottom of the *Main Program Screen*. *Quick-Iterator* engines can be analyzed, compared with other open engines, and modified in any way, just like any other engine in the simulation.

applies an optimization process called ***Dual-Phase™ Testing*** to find the best combination in short period of time. The first test phase uses a wide range of testing values. After the best result has been found from this wide-range search, a second phase is performed using a much narrower range of test values. The *Dual-Phase* approach allows the *Quick Iterator* to perform, for example, a search for optimum cam timing in less than 2500 simulation runs; typically, less than two minutes of processing time for the *Filling-And-Emptying* simulation and about 10 minutes for a *Wave-Action* simulation (on 4.0 Ghz or faster computer systems).

Using The *Quick Iterator*™

To perform a *Quick Iterator* analysis, select all the components for the baseline engine. Make sure all *Title Bars* in each Component Category are **dark-tone,** indicating that all engine components have been selected. There are two main testing groups in the **Quick Iterator** and two buttons in each group. The upper group searches for optimum cam timing for either peak horsepower or peak torque. The lower group determines the best bore and stroke combination (maintaining current engine displacement) for either optimum horsepower or peak torque.

Press either the ***Best HP*** or ***Best Torque*** button in the ***Optimize Cam Timing*** group to begin an analysis of valve-event timing that will optimize horsepower or torque within ±500rpm of the current power or torque peak. The **Quick Iterator** assumes that the current cam used the simulated engine is a "roughly" appropriate for the intended application and uses current cam timing as a starting point. You can follow the progress of the **Quick Iterator** by viewing the indicators in the ***Iterator Status*** group.

To perform an analysis of cylinder-bore and crankshaft-stroke dimensions, press

Using The *Quick Iterator*™

To pinpoint improvements located by the Iterator, you can setup back-to-back comparisons with the original, baseline engine. Right-click the graph, select *Properties*, then include the baseline engine as one of the *Data Sets*. The baseline engine curves will be drawn on the current graph, and the key-legend at the bottom of the graph will be updated to include the new comparison.
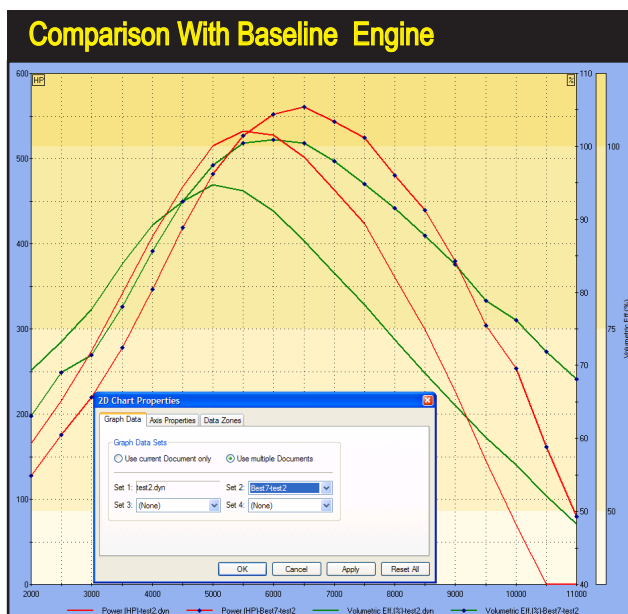
either the **Best HP** or **Best Torque** button in the lower group. The **Quick Iterator** will determine the best bore-and-stroke combination for optimum horsepower or torque within ±500rpm of the current horsepower or torque peak (current displacement will be maintained).

When *Iterative* testing is complete (you can stop testing at any time by pressing the **Stop** button; press **Resume** to continue testing), the **Iterator Best Result** graph will show the improvement in horsepower or torque found with the new component specifications. You can keep the results by clicking **Save**. In a few seconds, the simulation will create a new, simulated engine incorporating the component combination that produced optimum power or torque. Switch between the new engine and the baseline engine by using the **Engine Selection Tabs** at the bottom of the *Main Program Screen*. *Quick-Iterator* created engines can be analyzed, tested, and modified in any way, just like any other engine in the simulation. In fact, it is possible to use the result to begin a **new** *Iterator* test to further "home in" on the desired results.

The *Quick Iterator* will almost always find more power or torque. To pinpoint the improvements, setup a back-to-back comparison with the original, baseline engine. Simply right-click on the power/torque graph of the Iterator engine, select **Properties**, then include the baseline engine in one of the four **Data Sets** shown on the *Graph Data* page. The baseline engine curves will be included on the current graph, and the key-legend at the bottom of the graph will be updated.

Note: Whenever the *Quick Iterator* is used with **Variable Valve Timing** to find the best Horsepower or Torque, tests are only run on the High-Speed Lobe profiles. The low-speed lobes are not tested and remain unchanged. If peak

Using The *Quick Iterator*™

torque is generated by the low-speed cam and you wish to optimize torque using the low-speed profile, build a separate engine—without V-V-T—using the low-speed cam data. Then run a *Quick Iterator* on that engine model to find the best-torque, valve-event timing.

Wave-Action Filling & Emptying Simulations 6
Version 6

PRO-ITERATOR™

ProIterator™ Title Bar — QuickStart Presets — Tabbed Data Pages

Functional Group Boxes

Windows Close Button

Power/Torque Found By Iterator — Rest All — Close Quit — Save — Close Save State
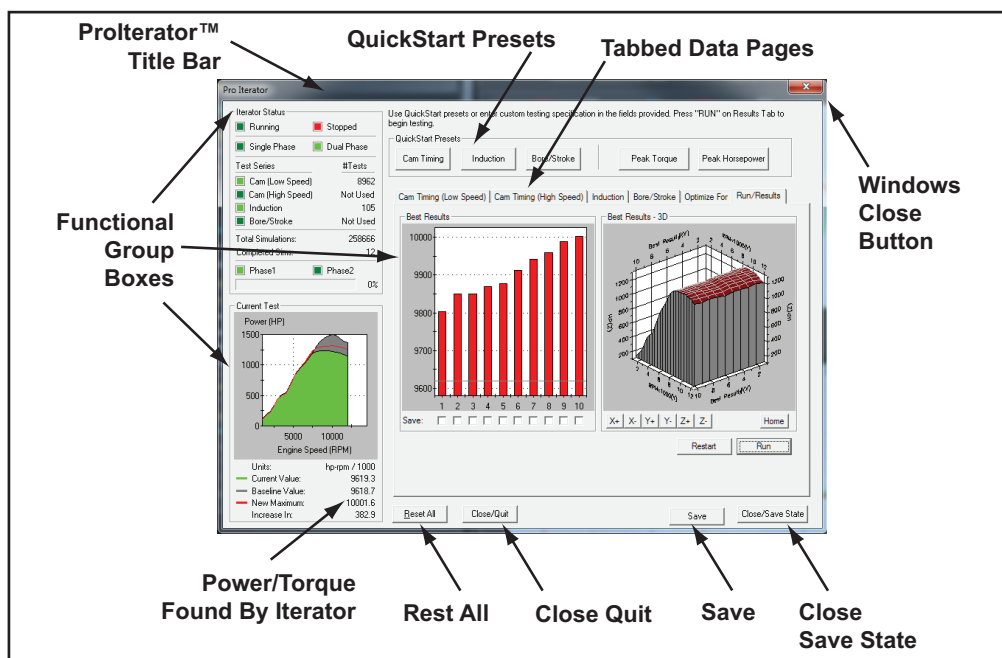
The **ProIterator™** (*pronounced "it-ter-a-tor" and "it-er-ah-tive testing"*) provides a powerful and easy-to-use tool for optimizing engine components. The serious engine designer needs the ability to perform Iterative testing on multiple engine systems, such as *Induction*, in the testing criterion. This, combined with greater flexibility in setting testing parameters, area-under-the-curve results analysis, variable power-band ranges, and more, is offered in the **ProIterator™**.

Using The *ProIterator™*

Open the *ProIterator™* by choosing **ProIterator™ Testing...** from the **Tools** menu or by clicking the **ProIterator™ Icon** in the **Toolbar.** Tips on setting up the *ProIterator* are provided in the **Walkthrough** later in this section.

The *ProIterator* main screen consists of the following Groups:

Using The *ProIterator™*

Iterator Status

■ Running	■ Stopped
■ Single Phase	■ Dual Phase

Test Series	#Tests
■ Cam (Low Speed)	7186
■ Cam (High Speed)	Not Used
■ Induction	Not Used
■ Bore/Stroke	Not Used

Total Simulations:	7186
Completed Sims:	138

■ Phase1	■ Phase2

22%
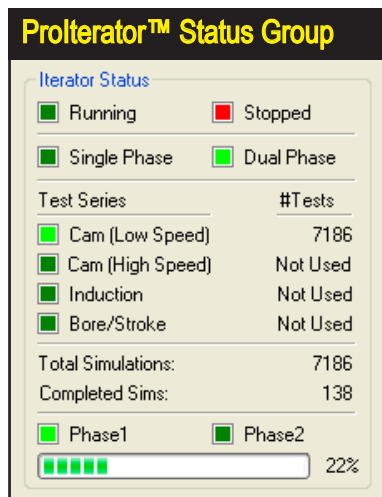
Indicators within the status group clearly show current testing progress. *Running* and *Stopped* are located directly above the *Single-* and *Dual-Phase™* indicators; the *ProIterator™* can use *Dual-Phase™* optimization to find the best combinations in the shortest time. The *Test-Series* indicators show whether a particular engine component category will be included and modified in the test series.

Iterator Status Group (upper-left corner)

Indicators within this group show current testing status. The *Running* and *Stopped* "lights" are located directly above the **Single-** and **Dual-Phase™** indicators. Like the *Quick Iterator* (discussed in the previous chapter), the **ProIterator™** can use *Dual-Phase™* optimization to find the best combinations in reduced time. The first phase tests over a wide range of values. After the best result is found, a second testing phase is performed using a narrower testing range. The **ProIterator™** extends the dual-phase capability by allowing all variable "steps" to be fully customizable. *Dual-Phase™* and standard *Single-Phase™* testing can be toggled on and off as desired.

Below the phase indicators are **Test-Series** fields that show whether a particular engine component category will be included in the Iterator series. The three test groups, **Cam Timing** *(*for V-V-T cams: *High-Speed* and *Low-Speed* are available*)*, **Induction**, and **Bore/Stroke** correspond to the similarly named tabbed pages in the center of the dialog box. A light-green indicator shows that this group will be included in the test series, and **# Tests** displays how many simulation cycles will be performed within this category. The total number of simulations is shown below, along with the number of completed test runs. At the bottom of the *Iterator Status* group are the **Phase 1** and **Phase 2** indicators (only visible when *Dual-Phase™* testing has been enabled) and a progress bar that shows the progress of each phase in multi-phase testing.

Current Test Group (bottom-left corner)

The **Current Test** graph displays three horsepower or torque curves (and area-under-the-curve, if selected). The gray curve represents the initial, base-line power/torque; the green curve is the current Iterator cycle test result, and the red curve is the highest power discovered up to that point in the testing series. A key-legend is provided below the graph along with the exact values

Using The *ProIterator™*

The *Current Test* group displays three horsepower or torque curves: The gray curve represents the baseline, the green curve is the current test, and the red curve is the highest power/torque found. Exact values for the baseline, current, maximum, and gain-or-loss in power/torque are provided at the bottom of the *Current Test* box.
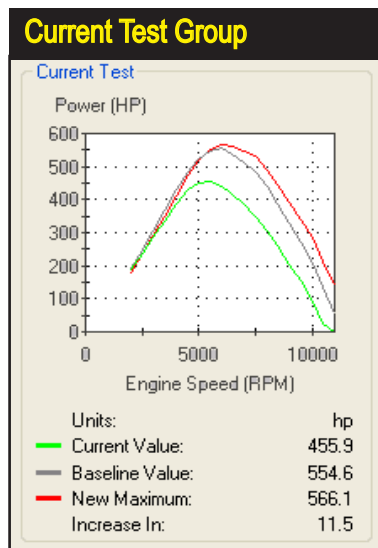
for the baseline, current, maximum, and gain-or-loss in power/torque.

QuickStart Presets Group (top center)

The convenience of one-button testing incorporated in the *Quick Iterator™* is also a part of the ***ProIterator™***. The ***ProIterator™*** does not begin testing when a *QuickStart* button pressed. Instead, it loads a "typical" set of testing parameters into the appropriate tabbed data page (each tabbed data page is discussed next). For example, if you click the **Cam Timing** and **Peak Horsepower** presets, the **Cam-Timing Page** establishes a *Dual-Phase™* cam-timing testing series based around the current camshaft and the **Optimize-For Page** selects *Peak Horsepower* as the principal search criterion. Use the **Reset All** button located at the bottom of the screen to clear all Presets and return the tabbed-pages to their default setup.

Tabbed Data Page Group (center of screen)

Six data-entry-and-display pages are available as tabbed screens at the center

QuickStart Presets Group
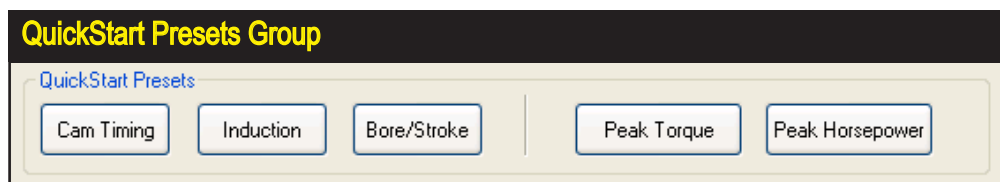
The convenience of one-button quick testing has been incorporated in the *ProIterator™*. Click any preset button (e.g., *Cam Timing* and *Peak Horsepower)* to establish a testing series on the appropriate tabbed data page. Use the *Reset All* button at the bottom of the dialog box to clear all Presets and return the tabbed-pages to their default setup.

Using The *ProIterator*™

Camshaft-Timing "Tabbed" Data Pages
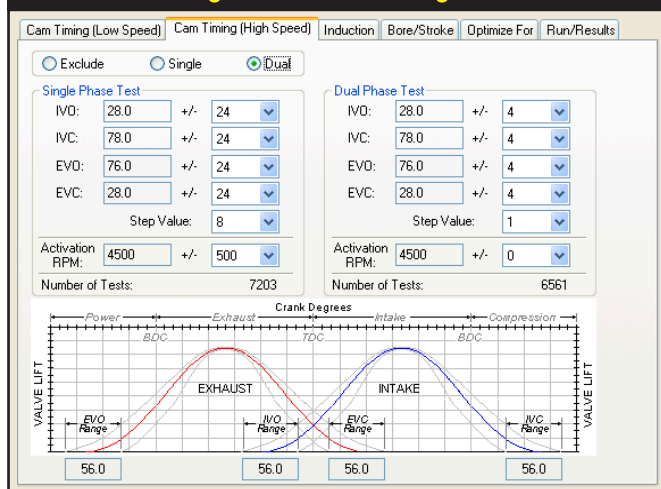
The *Cam Timing* tabbed pages establish a *Single-* or *Dual-Phase*™ test of cam-timing changes on power or torque output. Select either the *Single-* or *Dual-Phase* radio button and enter the testing criterion in the *Single-* and/or *Dual-Phase Test* boxes. The ranges of individual cam-timing values evaluated during *ProIterator*™ testing are displayed just below the "twin-hump" cam-timing diagram.

of the *ProIterator*™.

Cam-Timing Tabbed Pages (Standard and Variable-Valve Timing)

The first two tabbed pages establish a *Single-* or *Dual-Phase*™ test of cam-timing changes on power or torque output. The *Low Speed* lobe is also used for standard, non-VVT (Variable Valve Timing) cams, while the *High-Speed* tab is only used for the high-speed lobe on VVT applications. You can quickly setup tests by clicking on the **Cam Timing** button in the *QuickStart Preset* group (described previously). This will perform a *Dual-Phase* test of 8,962 simulations that will take about 5 minutes for the FE model on 4Ghz and faster computers.

If you did not press one of the **Preset Buttons**, the *Cam-Timing Tabbed Pages* default to *Exclude* (perform no cam-timing tests) as shown by radio-button controls at the top-left of the pages. To setup cam-timing Iteration, select either the **Single-** or **Dual-Phase** radio button and enter the testing criterion in the **Single-** and/or **Dual-Phase Test** boxes, located below the radio buttons. The number of degrees through which valve-timing events will be evaluated are shown just below the twin-hump cam-timing diagram.

> **Note:** When performing Iterative testing on **Variable Valve Timing** cams of the dual-lobe type), it is more efficient to test each lobe (low and high-speed) separately and select the best of each test for the final camshaft specifications. While the *ProIterator* can run "combined" tests of both lobe profiles at once, this often results in a test series requiring a long time to complete.

Cam Centerline Testing (compatible with Lobe-Profile Files)

The CamTiming tabbed page can be instantly "converted" into a Cam-Centerline testing tool by just clickiing the *Iterate Centerline Testing Only*

Using The *ProIterator*™

Cam Centerline Analysis
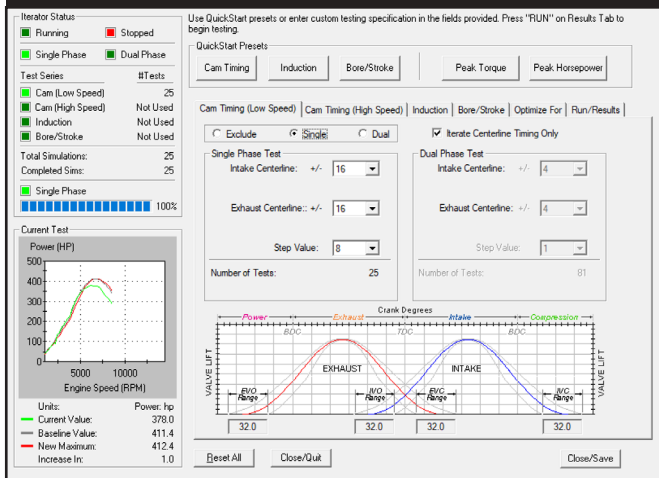
The *CamTiming* page can be "converted" into a Cam-Centerline testing tool by just clicking the *Iterate Centerline Testing Only* check box. This redefines the data entry fields into Centerline timing-range fields. Run the Iterator, and in about a minute (depending on the testing ranges) a full cam centerline analysis will be performed. This testing is particularly helpful for dual-overhead cam engines, and it's the only cam Iteration possible with Lobe-Profile files.

check box. This redefines the previous valve-event fields into Intake and Exhaust Centerline range modification fields. Run the Iterator, and in about a minute (depending on the ranges you've selected) a full cam centerline analysis will be performed and the Iterator will select the top 10 best performing lobe-centerline values based on your criterion. This testing is particularly helpful for dual-overhead cam engines where cam centerlines can be easily modified with adjustable belt/chain drive sprockets.

Induction Tabbed Page

The third tabbed page sets up a standard, *Single-or Dual-Phase* test of the

Induction "Tabbed" Data Page

The *Induction* tabbed page sets up a *Single- or Dual-Phase* test of the effects of various induction system components and dimensions. To perform an *Iterative* test of Induction components, select the *Single or Dual* radio button and select the manifolds/induction-systems that you would like to add to the test series. Note: Runner dimensions only apply to WA simulations.

effects of various Induction Manifolds and (if the WA Simulation is selected) Runner Lengths and Dimensions on power or torque output. By default, the *Induction Tabbed Page* is set to *Exclude* (perform no induction tests) as shown in the radio buttons at the top-left of the page. To perform an Iterative test of induction systems, select *Include* and check the manifolds/ induction-systems that you would like to include in the test series.

You can also quickly setup an Induction system test of Intake Runner Lengths and Areas by clicking on the **Induction** button in the *QuickStart Preset* group (top of dialog box, described earlier).

> **Note:** When selecting any of the *Individual Runner* induction systems, take note of the two additional radio-buttons: **Use Existing Runner Temp** and **Use Temp Compensation During IR Testing** that compensates for the lower temperatures of typical IR systems.
>
> In most cases, you should enable *Use Temp Comp..* since this allow the simulation to alter runner temperatures, providing a more realistic environment when modeling a variety of manifolds that include both plenum and IR systems.

Bore/Stroke Tabbed Page

This tabbed page establishes a *Single-* or *Dual-Phase*™ test of bore-and-stroke dimensional changes on power or torque output. By default, the *Bore/Stroke Tabbed Page* is set to *Exclude* (perform no bore-and-stroke modifications) as shown in radio buttons at the top-left of the page. To include a Bore/Stroke Iteration, select either the **Single-** or **Dual-Phase** radio button and enter the testing criterion in the **Single- and/or Dual-Phase Test** boxes, located just below the radio buttons. You can quickly setup an comprehensive test by clicking the **Bore/Stroke** button in the *QuickStart*

Bore/Stroke "Tabbed" Data Page
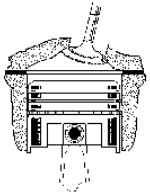
The *Bore/Stroke* tabbed page establishes an *Iterative* test of bore-and-stroke dimensional changes. Perform a Bore/Stroke *Iteration* by selecting either the *Single-* or *Dual-Phase* radio button and entering the testing criterion. You can chose to *Maintain Current Displacement* or let engine displacement vary throughout Bore/Stroke testing (see text for details).

Preset group (described earlier).

Note About Bore/Stroke Iteration: You can chose to *Maintain Current Displacement* or let engine displacement vary throughout Bore/Stroke Iterative testing. By checking the **Maintain Current Displacement** box, the **Stroke** within both **Phase-Test** boxes will switch to **(Auto)**, indicating that *Stroke* will be allowed to vary as much as required to keep displacement constant while the **Bore** varies from its current value throughout the indicated **(±)** *Range*. Alternately, you can choose to allow *Bore* (<u>rather than Stroke</u>) to vary as much as required to keep displacement constant while *Stroke* changes from its current value throughout its **(±)** *Range*.

Follow these steps to change the **(Auto)** variable: 1) With *Maintain Current Displacement* checked, 2) set the *Bore (±) Range* value to zero, 3) then set the *Stroke (±) Range* to any desired value (except zero). The **(Auto)** function will switch to *Bore.*

Optimize For Tabbed Page

This tabbed page establishes the <u>desired results</u> from Iterative testing. By default, the **ProIterator™** will search for the combination of components that produces *Peak Horsepower*. Alternately, you can select *Optimize For Peak Torque*. In addition to these two options, two additional **Optimize** choices are available: *Maximum Area Under The Horsepower Curve* or *Torque Curve.* While the peak torque and horsepower choices focus on absolute maximum values, the areas-under-the-curves selections will find parts combinations that produce the greatest "volume" of horsepower or torque within the selected rpm range.

In general, peak horsepower searches may find optimum components for narrow-rpm-band racing (like drag-racing), and maximum area under the curve

The *Optimize For* tabbed page establishes the goals for the Iterator. By default, the *ProIterator™* will search for *Peak Horsepower*. While *Peak Torque* and *Peak Horsepower* choices focus on absolute maximum values, the *Areas Under The Curve* selections locate engine setups that produce the greatest "volume" of horsepower or torque. Think of this as the maximum horsepower or torque throughout the selected rpm range (over time).

Optimize For "Tabbed" Data Page
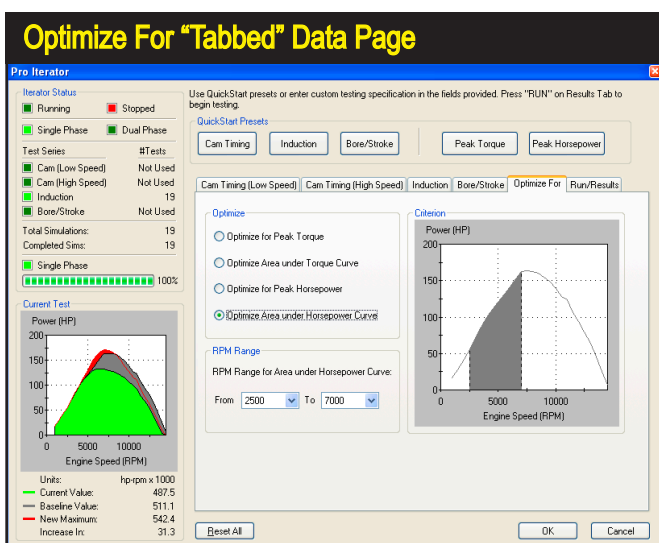
Using The *ProIterator™*

may find the best components for wide-rpm band racing (like road racing). Below the *Optimize* settings box, the **RPM Range** choices let you set the lower and upper limits through which the *Iterator* will search for optimum power or torque combinations. When the *Iterator* is searching for peak values, the *RPM Range* will be illustrated as dotted vertical lines on the *Criterion* graph. When either *Area Under The Curve* choice is selected, the *RPM Range* values will be displayed as a "bounded area" under the horsepower or torque curves.

> **Note:** Optimizing engine components for maximum area under the curve is a unique way to look at engine power output. There is little published data on this method of evaluating engine power or torque, nor is there much research available on which racing applications may benefit from this analysis. Rumor has it, though, that racing teams have used this method to find a winning edge. Now you can use this powerful analysis method in the *ProIterator™* to your advantage!

Run/Results Tabbed Page

Use this final tabbed page to begin an *Iterative* test series, to view testing progress, and to display the top ten results. Once you have selected testing parameters (on the *Cam Timing*, *Induction*, and *Bore/Stroke* tabbed pages) and set the optimization criterion, click the **Run** button to begin the *Iterative* test series. As the *ProIterator™* locates promising results, they are displayed in the **Best Results** graph as vertical bars. A horizontal "baseline" within the graph indicates the power level of the current engine (established from components selected in the Component Categories on the Main Program Screen). As the *Iterator* finds better and better component combinations, the bars continue to increase in height (the graph axis will rescale as needed).

Run/Results "Tabbed" Data Page
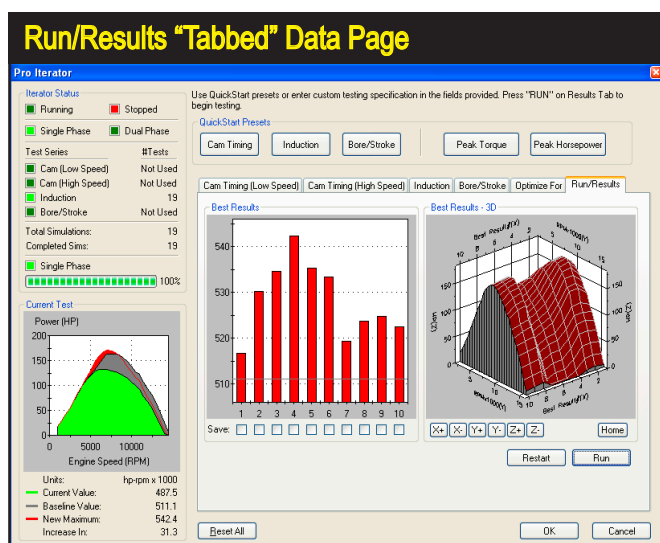
The *Run/Results* tabbed page will begin *Iterative* testing, display testing progress, and let you save any of the top ten results. When testing is complete, save any (or all) results by clicking specific *Save* boxes located below the vertical bars in the *Best Results* graph, then click *OK* to spawn (create) these engines within the simulation.

Using The *ProIterator™*

If the *Iterator* finds combinations that produce more power or torque than the baseline engine, the vertical results bars will exceed the baseline and continue to grow taller. The top ten horsepower or torque curves that match the bar-chart results are displayed on the **Best Results—3D graph**. You can view these curves from any prospective using the **X+**, **X-**, **Y+**, **Y-**, **Z+**, and **Z-** buttons (**Home** returns the 3D graph to its original position).

During the calculation of an *Iterative* test series, you can view the number of completed and remaining simulations in the **Iterator Status** box (you can stop testing at any time by pressing the **Stop** button; press **Run** to continue testing or **Restart** to clear current results). Save any (or all) of the top ten results by clicking the individual **Save** boxes located below the vertical bars in the *Best Results* graph. After selecting which results to save, click the **OK** button at the bottom of the **ProIterator™** dialog box. The *Iterator* will close and "spawn" (create) new engines based on the components that were used in the selected tests. You can switch to any of these new engines by clicking on the **Engine Selection Tabs** at the bottom of the *Main Component Screen* (continue modifying and testing as you would with any simulated engine).

The **ProIterator™** will almost always find more power or torque. To pinpoint these improvements, you can quickly setup back-to-back comparisons with the original, baseline engine: Right-click on the power/torque graph of any Iterator-generated engine, select **Properties**, then include the baseline engine in one of the four **Data Sets** shown on the *Graph Data* page. The baseline engine curves will be included on the current graph, and the key-legend at the bottom of the graph will be updated.

Spawned Engines Shown In Engine Selection Tabs
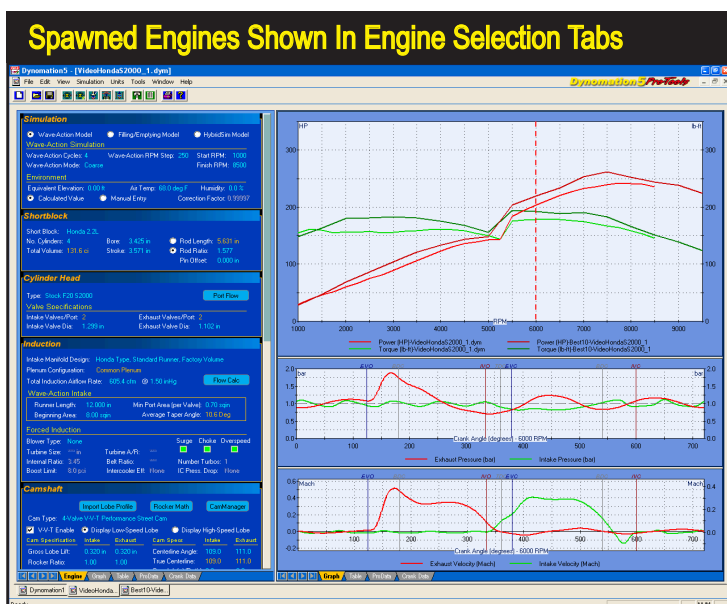
When you close the *Iterator* screen, new "spawned" engines will be created and displayed in the *Engine Selection Tabs* at the bottom of the Main Program Screen. Each new engine can be brought into the foreground by clicking on its *Selection Tab*. Iterator-spawned engines can be analyzed, tested, and modified in any way, just like any other engine in the simulation.

Using The *ProIterator*™

Close/Save State (bottom)

Closes the *Iterator* and saves the current testing setup (changing engine com-
ponents of this saved engine may force the *Iterator* to reset to its default). Use
the ***Close/Save State*** button when you wish to continue Iterative testing on
the same engine using similar components and Iterator settings.

Close/Quit (bottom)

Closes the Iterator without saving the current state.

Reset All **Button** (bottom)

Resets the current setup to default within the Iterator, but leaves it open so
you can begin to enter a new Iteration setup.

ProIterator™ Testing—A Quick Walkthrough

The first step in performing an *Iterative* test is to design the baseline engine.
When the engine has been simulated, you may begin an *Iterative* test series. Open
the *ProIterator*™ by choosing **ProIterator™ Testing...** from the **Tools** menu or by
clicking the **ProIterator™ Icon** in the **Toolbar**. Use the tabbed data-entry-and-display
pages to establish components and/or engine-systems for testing. *Alternately, to
quickly setup a test series* you can use the *QuickStart* buttons at the top of the
dialog box.

For this walkthrough, click on the **CamTiming** *QuickStart Preset* button. This
establishes a *Dual-Phase* test based around current camshaft timing specifications.

Setting Up A Camshaft Iteration
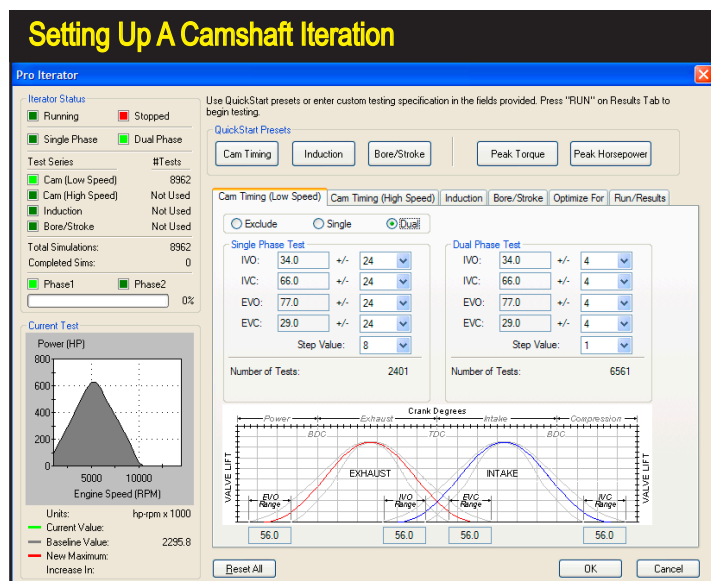
For this walkthrough, click on the *CamTiming* QuickStart Preset button. This sets up a Dual-Phase test around current camshaft timing specifications. Notice that 2,401 tests will be performed on the first phase, and 6,561 tests will be run in *Phase-2* using the best result from the *Phase-1* as the starting point. See text for more information on how this Iterative test evaluates cam timing.

Using The *ProIterator*™

Notice that 2,401 tests will be performed during the first phase, and 6,561 tests will be run in *Phase-2* using the best result from the *Phase-1* as the starting point. During Phase-1 simulations, individual cam timing points (IVO, IVC, EVO, EVC) will be varied through a range of +/- 24 degrees in 8-degree steps. That's seven tests for each timing point, making the total number of tests performed during Phase-1 equal to 2,401 (7 x 7 x 7 x 7). When this first series is complete, the simulation will select the best result and begin *Phase-2* testing. This time, each valve event will be varied through a range of +/- 4 degrees in 1-degree steps. This means that nine tests will be needed for each timing point, resulting in 6,561 tests in this series (9 x 9 x 9 x 9). The total number of tests required to complete the entire Iterative test is the sum of each series, or 8,962. Any of the timing point values, their +/- ranges, or the step sizes can be altered to suit custom requirements.

> **Note:** As you select *QuickStart Preset* buttons or enter data on individual tabbed pages, the *Iterator Status* box (upper-left of dialog) shows the component groups that have been included and the number of tests that must be performed to complete the current series. Since the program will typically perform 1 to 10 simulation tests per second (depending on the speed of your computer, the rpm range of the tests, etc.), keep in mind that it will take about an hour to execute 18,000 tests.
>
> Keep the testing criterion limited and the range and step values large to minimize calculation times. Also, set the simulation rpm range as narrow as possible (set this in the *Simulation Category*).

Before Iterative test can begin, the results criterion you would like to optimize must be specified. For this walkthrough, press the **Peak Horsepower** button at the top of the Iterator dialog. When you click the button, the *Optimize For* tabbed page is selected and is setup with *Optimize For Peak Horsepower* within the RPM

Selecting Results Criterion
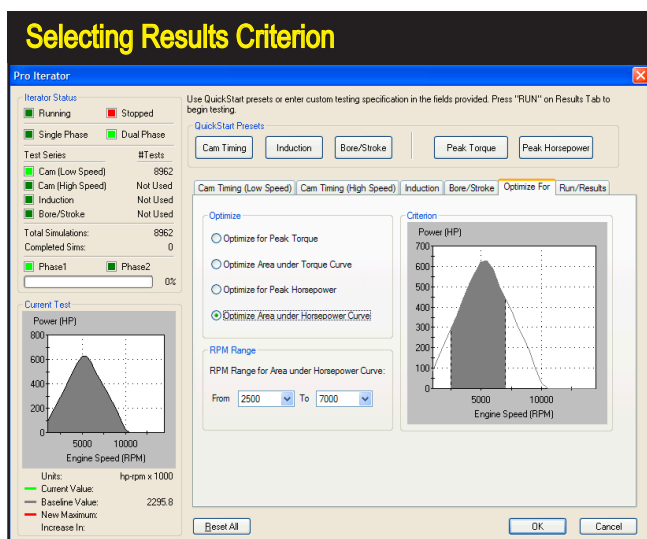
Before the Iterative test can begin, the <u>desired results</u> must be specified in the *Optimize For* tabbed page. To speed the process, you can click on either the *Peak Torque* or *Peak Horsepower* QuickStart Preset buttons. For this walkthrough, click on the *Peak Horsepower* button. Then <u>change</u> the Optimize selection to *Optimize Area Under Horsepower Curve*. That will display a shaded area under the power curve that is now the target for Iterative testing.

Using The *ProIterator*™
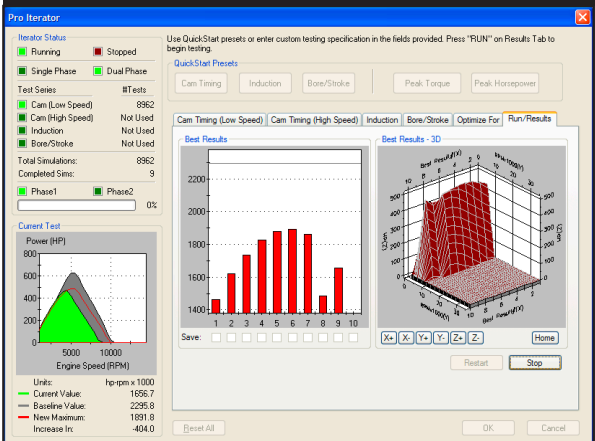
When all parameters have been entered, switch to the *Run/Results* tabbed page. Click the *Run* button to begin the test series. As the *ProIterator™* finds promising results, they are displayed in the *Best Results* graph as vertical bars. A horizontal "baseline" on the graph (shown here near the top of the bar graph) indicates the power level of the baseline engine. As the Iterator finds better and better component combinations, the bars continue to increase in height.

range of 2500- to 7500-rpm. Change the **Optimize** selection to *Optimize Area Under Horsepower Curve*. Notice that the graph display switches to show an area envelope under the curve that will be the focus of this Iterator test series.

> **Note:** If you make any mistakes during setup selections on any of the tabbed pages, you can correct any error individually or click the *Reset All* button, located at the bottom of the screen, to clear all Presets and return the tabbed-pages to their default settings.

Now that all parameters have been entered, switch to the *Run/Results* tabbed page. Click the **Run** button to begin the test series. As the *ProIterator™* finds promising results, they are displayed in the *Best Results* graph as vertical bars. A horizontal "baseline" on the graph indicates the power level of the initial engine

Iterator Test Series Complete
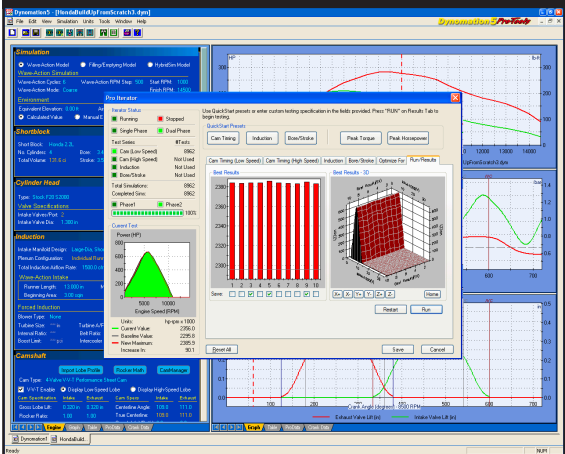
When testing is complete, pick your choices among the top ten results by clicking the Save boxes located below the vertical bars in the Best Results graph. Then click the OK button at the bottom of the ProIterator™ dialog box. When the Iterator closes, the simulation will spawn (create) all the engines you have selected and display them in the *Engine Selection Tabs* at the bottom of the Main Program Screen

Using The *ProIterator*™
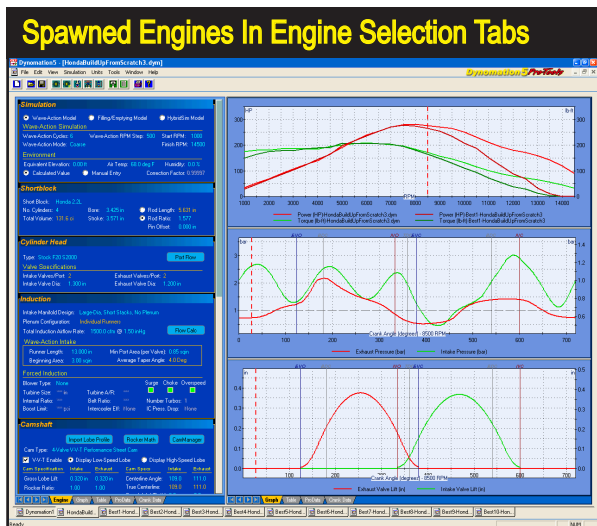
When the Iterator closes (after you have selected one or more engines from the Best Results list) the program will create the selected engines and display them in the *Engine Selection Tabs* at the bottom of the Main Program Screen. Each test engine can be brought into the foreground by clicking its Tab. Iterator-generated engines can be analyzed, tested, and modified in any way, just like any other engine in the simulation.

(built from components on the Main Component Screen). As the *Iterator* finds better and better component combinations, the bars continue to increase in height (and the graph axis will rescale as needed). When the *Iterator* finds combinations that produce more power under the curve than the baseline engine, the results bars will cross over the baseline indicator. The top ten results are accumulated on the **Best Results—3D graph**. You can view these curves from any prospective using the **X+**, **X-**, **Y+**, **Y-**, **Z+**, and **Z-** buttons (**Home** returns the 3D graph to its original orientation).

> **Note:** You can stop the testing process at any time by pressing the **Stop** button; press **Run** to continue testing or **Restart** to clear current test results.

When testing is complete, select the choices from the top ten results by clicking the individual **Save** boxes located below the vertical bars. Then click the **OK** button at the bottom of the **ProIterator™** dialog box. When the *Iterator* closes, the simulation will add all the engines you've selected to the current display and place them in the **Engine Selection Tabs** at the bottom of the **Main Program Screen** (see *Engine Selection Tabs* described earlier in this manual). Each test engine can be brought into the foreground by clicking on its Tab. Iterator "found" engines can be further analyzed, tested, and modified in any way, just like any other engine in loaded in the simulation. In fact, you can begin a *new Iterator* test using any of the spawned engines as a new baseline to further "home in" on desired results.

Tips For Running Efficient Iterative Tests

Setting up an *Iterative* series only takes a few seconds, however, if you include too many test parameters, ranges that are too wide, or step values that are too small, you will create an *Iterator* series that contains too many tests. If you create

Using The *ProIterator*™

An Extremely Long Iterator Test Series
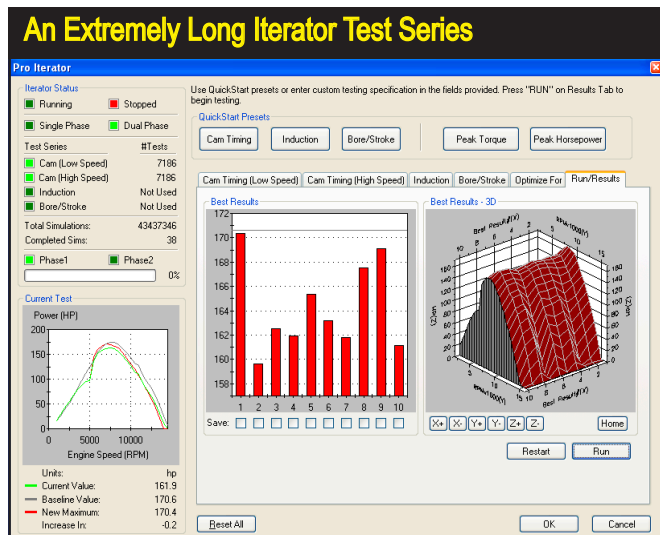
Narrowly-focused or multiple-component tests may require several thousand, or even millions of test cycles to complete. A test series as large as the one shown here, can require several days of calculation time depending on the speed of your computer. The same results can be obtained by a more carefully designed test that takes a fraction of the time to complete. Use wide first-phase ranges and steps to keep the number of Iteration cycles to a minimum.

an excessively long test series even fast computer systems may require days or months to complete. In these cases, the simulation may request that you increase step values for selected parameters.

The best way to find optimum components, especially cam timing, is to setup a Dual-Phase™ test that uses large step values (20 degrees or more) to "get in the ballpark" on the first phase, then focus the second *Iteration* phase with a narrower range of values (perhaps just a 2 to 4 degrees) and a smaller step value (perhaps 1 degree) to precisely locate the best timing.

Narrowly-focused tests may still require many thousands of test cycles to complete. In these cases, it is possible to continue to use your computer while the Iterator is still running. Simply use the **Start** menu to begin other applications or use **Alt-Tab** to switch between applications (see your Windows documentation for more information on program switching).

Note: You may also "minimize" the program and regain your desktop during an *Iterator* test.

Wave-Action Filling & Emptying Simulations

Version 6

BUILT-IN CALCULATORS

The simulation incorporates several built-in calculators that will assist your data entry and analysis. Some are simple, one-step calculators, others are powerful, multi-step "math engines" that make short work of otherwise error-prone calculations. This chapter presents an overview of these calculators.

THE INDUCTION-FLOW CALCULATOR

The **Flow Rate** field in the **INDUCTION** category establishes the overall flow for the simulated induction system, typically limited by the throttle size. Determining the flow rate for injectors or carburetors, especially if the flow rating is unknown, can be problematic. In these cases, the built-in *Flow Calculator* can help determine the appropriate total induction flow for the simulation. Open the *Flow Calculator* by clicking on the **Flow Calc** button in the **INDUCTION** category. The calculator presents two distinct calculation models: 1) the *Known Airflow Per Carb/Throttle,* and 2) *Known Throttle-Bore Diameter*.

For example, to calculate the flow for an induction system when the throttle-bore

Induction-Flow Calculator
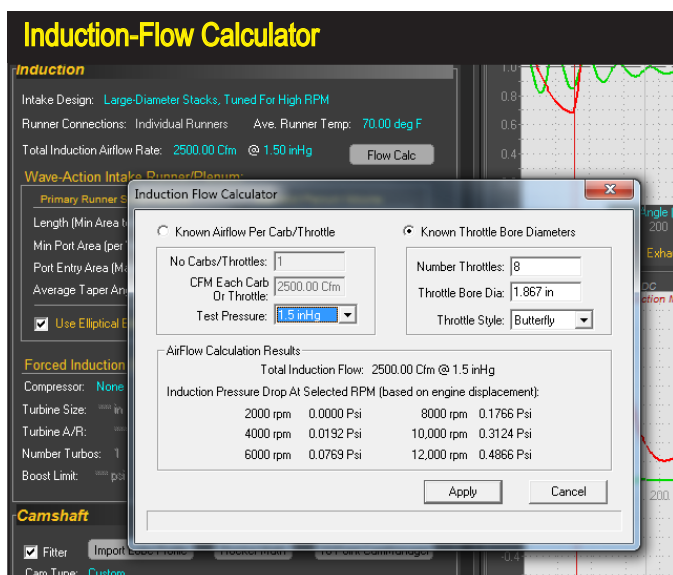
If you are modeling an induction system that uses multiple carburetors or injector "stacks," you can use the built-in *Flow Calculator* to help determine total induction flow. Enter the number of throttles, the throttle bore diameter or the flow per throttle. The calculator will estimate the airflow for the entire system (flow for each throttle times the number of throttles).

Built-In Calculators & Tools

Carburetors and injector stacks use two common throttle mechanisms: 1) The "standard" throttle plate that pivots on a shaft through the middle of the throttle bore, and 2) A slide-valve that when in the wide-open throttle position, is retracted entirely from the throttle bore. These carburetors are commonly called "FlatSide" since the chamber to accommodate the slide valve gives the carburetors a "flat," appearance on the sides.

diameter is known, select the right-hand radio button. First, make sure the pressure-drop value matches your application (typically, 1.5-inHg is used for four-barrel carburetors, injectors, etc., while 3.0-inHg is primarily used for 2-bbl carburetors. Enter the number of "stacks" or "barrels," the diameter of the throttle-bores, and select between *Butterfly* or *Flat-Side* throttle configurations *(a Flat-Side is a slide-valve throttle that, when opened fully, provides no throttle restriction to flow)*. The calculator will estimate the appropriate airflow for the overall induction system (flow for each throttle times the number of throttles). To apply the calculated airflow to the simulated engine, press the **Apply** button.

THE AIRFLOW PRESSURE-DROP CONVERTER

The simulation will accept induction airflow in CFM measured at a pressure drop of either 1.5- or 3.0-In/Hg (or in metric units). For those instances where an induction system, injector, or carburetor was flow tested at a different pressure drop, or whenever you would like to convert flow values from one pressure-drop rating to

The Airflow Pressure-Drop Converter is a general-purpose tool that will convert airflow to/from any pressure-drop standard. Activate the Airflow Converter by either selecting *Airflow Pressure-Drop Converter...* from the Tools menu or by clicking on the *Airflow Converter* Icon in the Toolbar.

Airflow Pressure-Drop Converter

Opens Airflow Pressure-Drop Converter

Built-In Calculators & Tools

MODE 1—Convert To 4-bbl Standard

MODE 1: When the calculator is first activated, the *Airflow Ratings Standard* is set to *4-Barrel Rating at 1.5-in/Hg Pressure Drop*. To convert from any known airflow to this flow standard, enter the known airflow and pressure drop in the *Known Airflow* category. The airflow at the 4-barrel rating standard will be displayed in the *Calculated Airflow* category.

another, the **Airflow Pressure-Drop Converter** easily performs these conversion functions. This tool can also convert flow ratings measured in inches-of-mercury (in/Hg) of depression to and from airflow values rated in inches-of-water (in/H$_2$O).

Note: A pressure drop of 1.5-in/Hg is equivalent to 20.3-in/H$_2$O.

The **Airflow Pressure-Drop Converter** has three basic modes of operation: 1) Convert flow to the 4-Barrel Standard pressure-drop rating, 2) convert to the 2-Barrel Standard pressure-drop rating, and 3) calculate airflow between any two pressure-drop rating levels. Each of these methods are described below. Activate the **Airflow Pressure-Drop Converter** by either selecting **Airflow Pressure-Drop Converter...** from the **Tools** drop-down menu or click on the **Airflow Icon** located in the **Toolbar**.

Using The Airflow Pressure-Drop Converter

Mode 1:
Convert Any Flow To 1.5-in/Hg—The 4-Barrel Standard.

When the calculator is first activated, the *1.5-in/Hg Airflow Ratings Standard* "radio button" is selected. The *RESULTS* category also defaults to a fixed pressure drop of *1.5-in/Hg* or *20.3-in/H$_2$O* (these pressure drops are identical). To convert any known airflow measured at any pressure drop to the *1.5-in/Hg*, 4-barrel standard, enter the measured airflow and pressure drop in the *INPUT* category (if needed, you can switch between Inches-of-Mercury[Hg] and Inches-of-Water[H$_2$O] by clicking on the appropriate radio buttons in the *INPUT* and *RESULTS* categories). The converted airflow will be displayed in the *Calculated Airflow* field. You can change to any of the previous fields (by clicking on them or using the Tab or SHIFT-Tab keys) to explore their effects on calculated airflow. At any time, you can click *Ap-*

Built-In Calculators & Tools

MODE 2: Switch the *Airflow Ratings Standard* to *2-Barrel Rating at 3.0-in/Hg Pressure Drop*. Enter the known airflow and pressure drop in the *Known Airflow* category. The airflow at the 2-barrel rating standard will be displayed in the *Calculated Airflow* category.

MODE 2—Convert To 2-Barrel Standard

Pressure Drop and Airflow Converter

Convert To Standard 2-Barrel Flow Rating

RESULTS Airflow Ratings Standard
- ○ Any Pressure Drop (cannot be directly applied to simulation)
- ○ 4-Barrel Carb. Rating of 1.5 inHg Pressure Drop (Default)
- ● 2-Barrel Carb. Rating of 3.0 inHg Pressure Drop

Apply To Sim

Cancel

INPUT Original Airflow/Pressure Drop
Pressure Drop Units
- ● Inches Mercury (inHg)
- ○ Inches Water (inH2O)

Airflow
Pressure Drop: 1.50 inHg
Airflow Rate: 600.00 Cfm

RESULTS Airflow/Pressure Drop
Pressure Drop Units
- ● Inches Mercury (inHg)
- ○ Inches Water (inH2O)

Calculated Airflow At 3.0 Inches/Hg (2-Barrel Std.)

Airflow
Pressure Drop: 3.00 inHg
Calculated Airflow: 848.50 Cfm

Any Known Airflow

---

ply To Sim to load the new calculated airflow into the ***Total Induction Airflow Rate*** field in the *Induction* category. Alternately, you can press **Cancel** to discard all entries and close the *Airflow Pressure-Drop Converter.*

Mode 2:
Convert Any Flow To 3.0-in/Hg, The 2-Barrel Standard.

Switch the ***RESULTS—Airflow Ratings Standard*** category selection to the radio button marked ***2-Barrel Carb of 3.0-in/Hg Pressure Drop***. This changes the ***RESULTS Airflow/Pressure Drop*** category to a fixed *3.0-in/Hg* (40.7-in/H_2O) pressure drop. To convert any airflow measured at any pressure drop to the *3.0-in/Hg*, 2-barrel standard, enter the known airflow and pressure drop in the **INPUT** category (you can switch between *Inches-of-Hg* and *Inches-of-H_2O* by clicking on the appropriate radio buttons in the **INPUT** and **RESULTS** categories). The converted airflow will be displayed in the ***Calculated Airflow*** field. You can change any of the previous fields (by clicking on them or using the Tab or SHIFT-Tab keys) to explore their effects on calculated airflow. At any time, you can click ***Apply To Sim*** to load the new, converted airflow into the ***Total Induction Airflow Rate*** field in the *Induction* category. Alternately, you can press **Cancel** to discard all entries and close the *Airflow Pressure-Drop Converter*.

Mode 3:
Convert Any Airflow To Equivalent Flow At Any Pressure-Drop.

Switch the ***RESULTS—Airflow Ratings Standard*** category to the radio button marked ***Any Pressure Drop***. This allows the **INPUT** and **RESULTS** to be set to any pressure drop measured in *Inches of Hg* or *Inches of H_2O*. Enter the known

Built-In Calculators & Tools

MODE 3: Switch the *Airflow Ratings Standard* to _No Rat-ings Standard_. Enter the *Known Airflow* at any pressure drop measured in Inches of Hg or H$_2$O. Then enter the desired pressure drop in the *Calculated Airflow* category. The equiva-lent airflow will be displayed in the *Airflow Rate* field.

Pressure Drop and Airflow Converter

Convert Between Any Two Flow Rating Systems

RESULTS Airflow Ratings Standard

Close

- ⦿ Any Pressure Drop (cannot be directly applied to simulation)
- ○ 4-Barrel Carb. Rating of 1.5 inHg Pressure Drop (Default)
- ○ 2-Barrel Carb. Rating of 3.0 inHg Pressure Drop

INPUT Original Airflow/Pressure Drop

Pressure Drop Units
- ○ Inches Mercury (inHg)
- ⦿ Inches Water (inH2O)

Airflow

Pressure Drop: 30.0 inH2O

Airflow Rate: 1022.00 Cfm

RESULTS Airflow/Pressure Drop

Pressure Drop Units
- ○ Inches Mercury (inHg)
- ⦿ Inches Water (inH2O)

Convert Airflow To Any Pressure Drop

Airflow

Pressure Drop: 60.0 inH2O

Calculated Airflow: 1445.30 Cfm

Any Airflow @ Any Pressure Drop

airflow and pressure drop in the **INPUT** category. Then enter the desired pressure drop in the **RESULTS** category. The calculated equivalent airflow will be displayed in the **Calculated Airflow** field. You can modify any of the fields (by clicking on them or using the Tab or SHIFT-Tab keys) to explore their effects on calculated airflow.

> **Note:** Since the **Total Induction Airflow Rate** field in the *Induction* category accepts only airflow values rated at either 1.5- or 3.0-in/Hg (20.3- or 40.7-in/H$_2$O), the **Apply To Sim** button is not available when **No Ratings Standard** is selected. If you wish to use the new calculated values in a dyno test, select either the **1.5-in/Hg** or **3.0-in/Hg Pressure Drop** in the **RESULTS—Airflow Ratings Standard** group at the top of the calculator dialog.

Rocker-Math Calculator

The Camshaft Category includes an in-depth **Rocker-Math Calculator** tool that will help you determine how and why changes in *Rocker Ratio* can have subtle affects on valve timing.

> **IMPORTANT:** *Follow this simple rule when using Rocker Math:* Enter the baseline *Rocker-Ratio* specifications (used by the cam manufacturer to determine their published valve-event timing, duration, etc.) in the Camshaft Category **FIRST!** Then use the *Rocker Math Calculator* to determine how changes to the initial values affect cam timing, valve lift, and engine power.

The *Rocker-Math Calculator* analyzes the subtle changes to valve-event timing that can occur when rocker-arm ratios are modified. When the intake or exhaust rocker-arm ratio is changed, overall valve lift is altered directly by the increase or decrease in ratio. It is clear that an increase in rocker ratio will open the valve further and increase valve acceleration. But what is less obvious is that rocker-ratio

Built-In Calculators & Tools

Rocker Math Calculator Overview

To properly use the Rocker Math Calculator: *Enter the original manufacturer's cam specification for Rocker Ratio in the Camshaft Category FIRST!* Then use the *Rocker Math Calculator* to determine how CHANGES to the baseline ratios will affect valve timing, valve lift, and engine power. In this photo, the cam manufacturer's specified rocker ratio of 1.5 was increased to 1.75 for the intake and 1.6 for the exhaust. The intake timing changed more than one degree and the exhaust duration has increased about 0.6 degrees.

Enter new rocker ratios in the boxes (1). Changes in valve lift are displayed in (4). Changes in seat-to-seat valve timing and duration are shown in the *Effects On Valve Timing* area. If you click *Apply,* the new rocker ratios, and changes in timing and valve lift will be transferred to the *Camshaft Component* category and used in the simulation.

Rocker-Math Calculator Data Entry

Built-In Calculators & Tools

Rocker Math Timing Changes

When you click the *Apply* in the Rocker-Math Calculator, valve event timing changes are transferred to the *Camshaft Component* category and placed in specific *Cam Modeling Input* fields dedicated to rocker-math changes. To Reset or Undo these changes, re-enter the original rocker ratios in the Rocker-Math Calculator. You can also click *Reset* in the Camshaft Component category, however, this only resets the timing changes to zero; the rocker ratios must be manually returned to their original values.
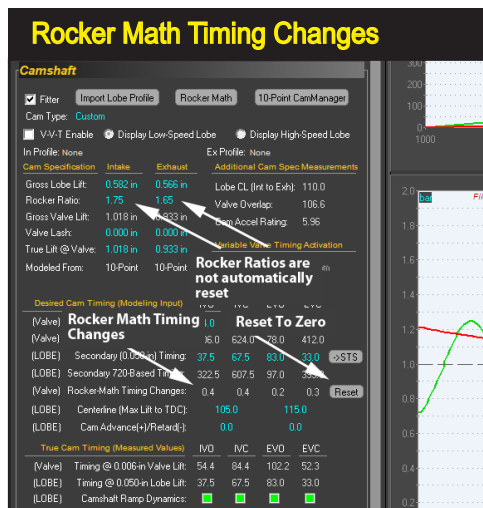
changes can also affect valve-open duration. When you click the **Apply** button after entering rocker-ratio changes in the Calculator, valve event timing changes are transferred to the *Camshaft Component* category and placed in specific fields only used for rocker-math changes.

Note-1: If you wish to "undo" Rocker-Math Calculator changes (after they have been applied to the simulation), either reopen the calculator and reenter the original rocker ratios, or use the *Reset* button next to the *Rocker Math Timing Changes* fields the Camshaft Category, then manually reset the rocker ratios to their original values.

Note-2: Changes in valve lash (independent of rocker-ratio) can have a significant affect on valve timing, duration, and engine output. You can model these changes directly in the *Camshaft Category* by simply changing the intake and/or exhaust lash values and reviewing the timing and lift changes in the *True Cam Timing* fields. In some cases, reducing lash by 0.005-inch to 0.010-inch can (depending on cam design) add 10 degrees or more to seat-to-seat valve duration!

THE *CAM-MATH QuickCalculator™*

The basic eight valve events (IVO, IVC, EVO, & EVC in both Seat-to-Seat and 0.050-inch timing) are required for the simulation to pinpoint opening and closing points of the intake and exhaust valves. IVO and EVO signal the beginning of mass flow in the intake and exhaust ports. The closing points, IVC and EVC, mark the end of mass flow. Unfortunately, some cam catalogs and other information sources only offer the lobe-center angles and duration values, leaving the conversion to IVO, IVC, EVO, and EVC up to you.

While these conversions can be accomplished in the *CamManager*, we also

Built-In Calculators & Tools

Opening The Cam-Math QuickCalculator

The *Cam-Math QuickCalculator* is available from the Tools menu. If a V-V-T cam is currently simulated, you can select between two versions of the QuickCalculator: low-speed and high-speed versions. Use *Cam-Math QuickCalculator* to easily modify Lobe Center Angles or Duration values.

Opens Low-Speed or High-Speed CamMath QuickCalculator
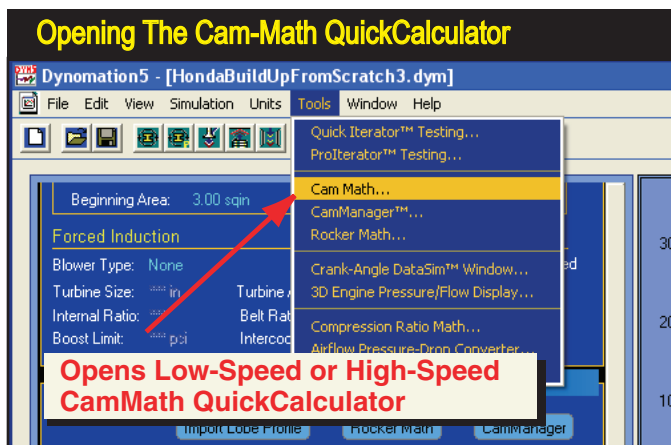
include a simple *Cam-Math QuickCalculator™* that easily converts the lobe-center angle, intake centerline, and the duration values into IVO, IVC, EVO, and EVC valve events ready for use in the simulation. One click of the **Apply** button transfers event-timing values into the **CAMSHAFT** category for use as *Primary* (Seat-To-Seat) or *Secondary* (0.050-inch) timing events. Furthermore, if you are using a VVT cam, you can specify if you would like to transfer the event timing to the low- or high-speed cam.

> **Note:** In order for the *Cam-Math QuickCalculator* to determine valve opening and closing events, <u>BOTH</u> the *Lobe-Center Angle* <u>AND</u> the *Intake Centerline*

Cam-Math QuickCalculator™

The *Cam-Math QuickCalculator* allows direct entry and conversion of cam-manufacturer published data. Since this Cam-Math Calculator opens with the timing events currently used in the simulation, it also simplifies changing the lobe centerline angle and duration values.

must be entered. Without the *Intake Centerline*, there is no way to determine how the cam is "timed" or "indexed" to the crankshaft. Many (unfortunately not all) cam manufacturer catalogs provide sufficient information to determine valve-event timing. If you have a catalog or have visited a website that does not provide this information, try another cam manufacturer.

If IVO, IVC, EVO and EVC cam-timing values were already entered in the **CAMSHAFT** Category, the *Cam-Math QuickCalculator* will display the lobe-center angle, intake centerline, and duration values for the current cam and accept any changes you would like to make. As you modify the data fields, the corresponding IVO, IVC, EVO and EVC points will be calculated and displayed. You may then either accept the calculated values and transfer them to the **CAMSHAFT** category by clicking one of the *Apply* buttons or discard the new values and close the *Cam-Math QuickCalculator* by clicking *Cancel*.

<div align="center">

Using Non-SAE Standard
Lift Heights

</div>

Dynomation6 and DynoSim6 also include a built-in Cam-Timing alternate-lift calculator that converts valve-event timing taken at any lift height (e.g., 1.0mm, etc.) to SAE standards of 0.006-inch/0.050-inch lift values. Refer to *Using Non-SAE Standard Lift Heights* in the *CamShaft Chapter* for more information.

<div align="center">

THE COMPRESSION-RATIO CALCULATOR
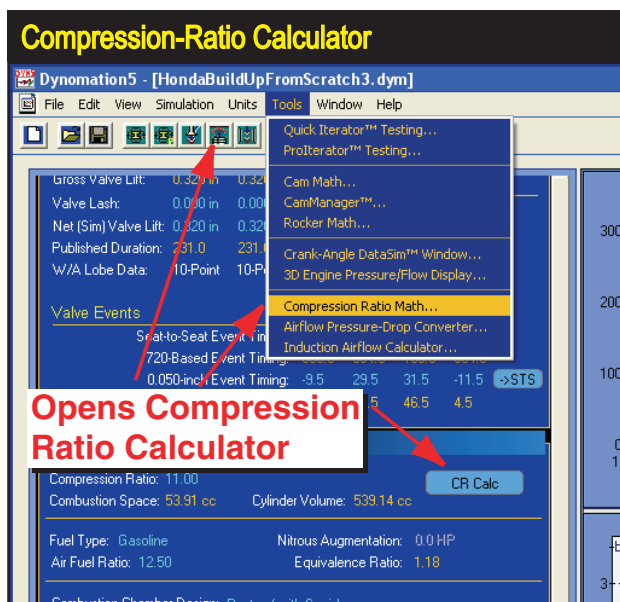
</div>

Compression-Ratio Calculator

Opens Compression Ratio Calculator

Activate the *Compression-Ratio Calculator* by clicking on the *CR Calc* button in the *Combustion Component* category, by selecting *Compression Ratio Math* from the Tools menu, or by clicking on the *Compression Ratio Icon* in the Toolbar.

Built-In Calculators & Tools

You can select and test a wide range of compression directly in the **Combustion Component** category. However, many engine builders need to directly enter combustion-chamber volumes, head-gasket thickness, etc., to calculate or modify the compression ratio. The **Compression-Ratio Calculator** quickly performs these functions. In addition, this tool is more than a simple calculator, it adjusts itself to the needs of the engine builder; changing the way it functions depending on whether combustion volumes are known values or need to be measured directly from engine components.

After you have specified the bore, stroke, and number of cylinders for the engine under test, activate the **Compression-Ratio Calculator** by clicking the **CR Calc** button in the **COMPRESSION** component category (see photo, below). When the calculator is first activated, it defaults to the *Known Volumes* mode. This is the most straightforward model for calculating compression ratio. Simply enter the required values in the **Compression-Ratio Calculator** and the compression-ratio will calculated and displayed.

Using Known
Dome/Dish/Deck/Chamber Volumes

If an engine builder is provided with the exact volumes displaced in the dome and valve pockets by the piston manufacturer, and the volumes of the combustion chamber, the deck height, and the specifications for the head gaskets are also known, a simple, numeric-only method can be used to calculate compression ratio.

CR Math Calculator—Known Volumes Mode

When the *Compression-Ratio Calculator* is first activated, it defaults to the *Known Volumes* mode. Simply enter the needed values in fields 1 through 6, and the *Compression-Ratio* will displayed.

Built-In Calculators & Tools

To measure how far down the bore the piston is positioned at TDC, use a dial indicator and stand. Enter a <u>positive number</u> for "<u>down-the-bore</u>" distances and a negative number if the piston protrudes above the deck surface. A typical value might be +0.040, indicating that the piston comes to a rest at TDC 0.040-inch <u>below the deck surface</u>.

This procedure is explained in this section. However, in those cases where piston specifications are unknown (not provided by the manufacturer or machine work has been performed on the dome/pockets), the engine builder must directly measure dome/pocket volumes. In these situations, refer to the next section *Burette-Measured Volumes*.

Known-Volumes Mode

Here are the steps for using the compression-ratio calculator in the *Known-Volumes Mode*. Start off by verifying that the upper radio button **Piston Dome/Deck/Relief Specs Known** has been selected. Next, enter the combustion-chamber volume (in cubic centimeters—cc's) in the *(1)* **Head Chamber Volume** data box. Next, enter the volume displaced by the piston dome in *(2)* **Dome Volume** and the volume of the valve reliefs in one piston into *(3)* **Valve Reliefs Volume**. If your piston manufacturer provided one value for both of these volumes, enter this combined volume in the *(2)* **Dome Volume** field and enter zero in field *(3)*.

> **Note:** If any of these values are unknown, they must be manually measured (with a burette; see the next section for *Burette-Measured Volumes*).

The next data entry field is *(4)* **Deck Clearance @ TDC**. This dimension indicates how far down the bore the piston is located when positioned at TDC. Enter a <u>positive number</u> for "down-the-bore" distances and a <u>negative number</u> if the piston protrudes above the deck surface. A typical value might be +0.040-inch, indicating that the piston comes to a rest at TDC positioned 0.040-inch <u>below the deck surface</u>.

> **Important Note:** A *positive* **Deck Clearance @ TDC** indicates the piston is positioned below the deck surface and this volume <u>adds</u> to the combustion space at TDC; a negative number indicates the piston protrudes above the

deck surface at TDC and <u>reduces</u> the combustion space.

The next two data-entry boxes are used to calculate the volume added to the combustion space by the head gasket that is compressed between the cylinder head and the block-deck surface. The data box marked *(5)* accepts the **Head Gasket Bore** diameter (in the appropriate Metric or U.S. units). Most head gaskets have a "bore-circle" or "bore diameter" larger than the cylinder-bore diameter. For gaskets with bore-circles of odd shapes, simply estimate the bore circle by averaging the larger and smaller dimensions. Finally, enter the compressed *(6)* **Head Gasket Thickness**. This dimension is often available from the head-gasket manufacturer. When the compressed thickness is entered, the **Head Gasket Volume** and the **Compression Ratio** are calculated.

At this point, you can change any data-input values and determine their effects on compression ratio. At any time, you can click **Apply** to load the calculated compression ratio into the **COMPRESSION** category and save all entered values for the simulated engine. Alternately, you can press the **Cancel** button to discard all entries and leave any previously entered compression ratio-value unchanged.

Using Burette-Measured Volumes

If you are using pistons with domes, dishes, or valve-pockets/reliefs of unknown volumes, determining the compression ratio requires a bit more work. Each of these volumes must be accurately measured so that the net effect of all "positive" (domes) and "negative" (pockets, reliefs) can be included in the calculations.

Burette-Measured Mode (see photo next page)
Start off by verifying that the lower radio button **Measured Piston Dome/Reliefs With Burette** is selected. Enter the combustion chamber volume (in cubic centi-meters—cc's) in the first *(1)* **Head Chamber Volume** data field. The next entry, *(2)* **Piston Down From TDC For Burette Measurement**, is a more-or-less arbitrary distance down the bore (measured from the deck surface) of sufficient depth so that the <u>entire piston dome is located below the deck surface</u>. Typical values may be 0.100-inches or 0.250-inches depending on the height of the piston dome (any distance is acceptable as long as the entire dome is positioned below the deck surface). At this depth, a <u>direct measurement</u> is made of the **Volume Above The Piston** in the cylinder (sometimes called *Dome/Deck Volume*). This measurement is taken by the engine builder using a liquid-filled burette to fill the space above the piston (a flat Plexiglas plate is often used to seal the top of the bore; grease is typically used to seal the piston to the bore). For pistons with positive dome volumes, the volume of liquid dispensed will be less than the volume for a simple cylinder of the same height. The liquid volume dispensed from the burette is entered in the **Measured Liquid Volume Above Piston** field. The difference between this volume and the volume of a simple cylinder (of a height equal to the value entered

Built-In Calculators & Tools

CR Math Calculator—Burette Measured Mode

in field **(2)**) is the **Calculated Deck Volume At TDC**, a volume equivalent to the sum of the dome, dish, relief, and deck volumes of the piston.

> **Important Note:** A <u>negative</u> **Calculated Deck Volume At TDC** indicates that the total dome/deck/relief volumes <u>reduce</u> the combustion space and will, therefore, <u>increase</u> the compression ratio over a flattop piston. A <u>positive</u> value indicates that the sum of all dome/dish/relief volumes will <u>increase</u> the combustion space volume and <u>decrease</u> the compression ratio over a similar flattop piston (with the same deck height at TDC).

The next two data-entry fields are used to calculate the volume added to the combustion space by the head gasket compressed between the cylinder head and the block deck surface. The data box marked *(3)* accepts the **Head Gasket Bore** diameter (in the appropriate Metric or US units). Most head gaskets have a "bore-circle" or "bore diameter" larger than the cylinder-bore diameter. For gaskets with bore-circles of odd shapes, simply estimate the bore circle by averaging the larger and smaller dimensions. Next, enter the compressed *(4)* **Head Gasket Thickness**. This dimension is often available from the head gasket manufacturer. When the compressed thickness is entered, the **Head Gasket Volume** and **Compression Ratio** are calculated and displayed.

At this point, you can change any data-input values and determine their effects on compression ratio. At any time, you can click **Apply** to load the new calculated compression ratio into the **COMPRESSION** component category and save all entered values with the simulated engine. Alternately, you can press the **Cancel** button to discard all entries and leave any previously entered compression ratio value intact.

Built-In Calculators & Tools

Measuring Dome/Deck Volume

Measure the volume above the piston while the highest portion of the piston dome is positioned below the deck surface. Enter this value in field **(5)** *Volume Above Piston*. The difference between this volume and the volume of a simple cylinder [of a height equal to the value entered in field **(4)**] is the *Deck Volume At TDC*. This volume is equivalent to the combined volumes in the dome, dish, and valve reliefs in the piston. A negative *Deck Volume At TDC* indicates that the dome reduces the combustion space and will increase the compression ratio over a flattop piston. A positive value indicates that the sum of all dome/deck/dish/relief volumes will increase the combustion volume and decrease the compression ratio over a flattop piston.

Wave-Action Filling & Emptying Simulations

Version 6

PRINTING

PRO-PRINTING™ ENGINE DATA AND RESULTS CURVES

The simulation can print a complete report of dyno-test results, including engine components, cylinder head airflow data, exact engine-test result values, 2D graphic displays of several engine-test variables and much more. Called *ProPrinting™*, this professional printout comprises a multi-page, dyno-test document of the currently-selected engine

ProPrinting™ features include custom page graphics, a cover page with the name and address of your business (or your personal name and address) and logo, a table of contents, optional text printed at the bottom of each page (can be a disclaimer, copyright notice or any other text you wish), even an optional glossary of common terms can be printed with your engine simulation report.

The full-color *ProPrinting™* report is "assembled" within the simulation and de-livered to your default Internet browser (e.g., *Edge, Chrome, FireFox, Opera, etc.*). Printing can be performed from within your Browser.

ProPrinting™
Functions And Setup

The *ProPrinting™* capabilities are another exclusive feature of this simulation. The ProPrint™ dyno-test report consists of fully-formatted data pages built from an HTML template. This template is "filled" with data from your current simulation which is sent to your default Internet browser for viewing and subsequent printing. Your Internet Browser is used for print-preview and to perform the complex process of converting the HTML into specific information that your printer can understand. By using the graphic capabilities of HTML, the simulation is able to produce a fully-formatted, professional test report.

ProPrinting™ setup and function are controlled by two choices from the program **File** menu:

1) ProPrint Display/Print In Internet Browser
Selecting ProPrint from the FILE menu starts the process of building the HTML document and, when completed (takes a few seconds), opens your default

ProPrinting™ **Features**

ProPrinting™ Menu Selections

There are two *ProPrint™* choices in the File menu that will help you setup report formats and initiate report printing. *ProPrint*—Opens your Internet browser so you can review each page. *ProPrint Setup*—Opens a dialog box that allows you to select the information that is included on the pages of the ProPrint report and do other customizations (see photo, below).

browser with the Dyno Report on screen. Use your printer's *Properties* to set the print *Range* (by entering **X-Y** in the *Print Range* field, with **Y** the starting page and **X** the ending page). Printing can be started (or canceled) from this dialog box.

2) ProPrint Setup...

Opens a dialog that allows you to customize **Pro-Printing™** features. You can

Pro-Printing™ Setup Options
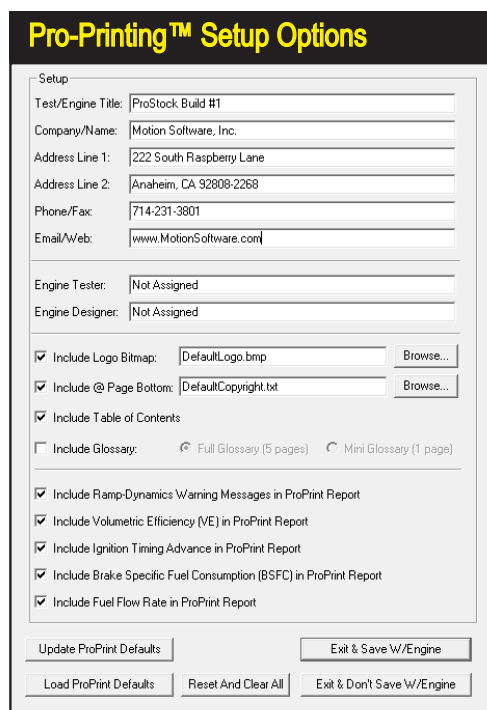
The *ProPrinting™ Setup Options, available* in simulation program FILE menu, allow you to include your *Logo*, Company Name and many other personal preferences on ProPrint report pages. Use the *Save* button to save your preferences, which will be applied to all subsequent engine simulations. The files for the *Default Logo.bmp* and the *DefaultCopyright.txt* are located in *Dynomation6/Manuals & Videos/ProPrint* or the *DynoSim6/Manuals & Videos/ProPrint* subdirectory. You can modify these files to suit your requirements.

ProPrinting™ Features
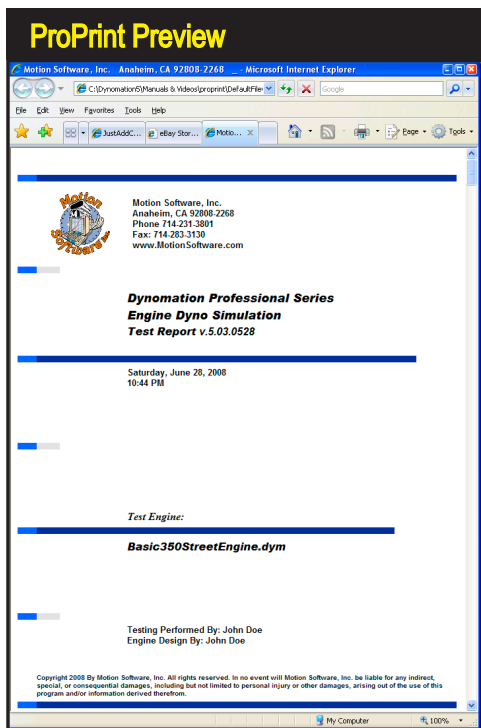
ProPrint Preview

Print Preview **displays your ProPrint report in your default Internet browser for you to review (works with most browsers). The first page is shown, and you can scroll through the rest of the document with the scroll bar. Select** *Print* **from the** *File* **or** *Settings* **menu to send the report to your printer.**

add your name, address, your company logo, specialized (copyright) text, a table of contents, and even add a short or long glossary to your *ProPrint* report (for your customers that may need help understanding technical terms). Use the **Save** button to save your preferences, which will be applied to all subsequent engine simulation reports. The files for the *Default Logo.bmp* (should be square with a size near 100 by 100 pixels) and the *DefaultCopyright.txt* are located in

ProPrint Preview—Multiple Pages
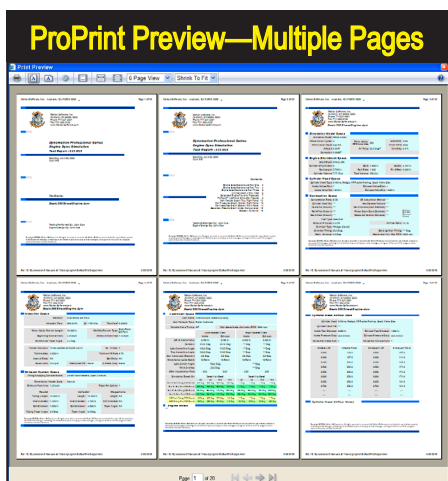
Some browsers support a multiple page *Print Preview.* **Here is a six-up display; in some cases you can select any number of pages you wish.**

ProPrinting™ Features

Dynomation6/Manuals & Videos/ProPrint or *DynoSim6/Manuals & Videos/ ProPrint* subdirectory (within your main program directory: *C:/Dynomation6* or *C:/DynoSim6*). You can modify these files to suit your requirements.

Custom ProPrinting™ Options

ProPrinting™ Setup will let you configure **ProPrinting™** features. If you activate **Include Logo**, the logo file *DefaultLogo.BMP* will be displayed and printed at the top of ProPrint pages. If you activate **Include @ Bottom Of Page**, the *Include* file (default is *DefaultCopyright.txt*) will be printed at the bottom of most pages.

The location of the *Default Logo.bmp* and *DefaultCopyright.txt* is the **Dynomation6/ Manuals & Videos/ProPrint** or **DynoSim6/Manuals & Videos/ProPrint** subdirectory. You can modify the **Include @ Bottom Of Page** text to display any non-formatted text. Use a non-formatting word processor (we recommend *Windows Notepad*) to modify the text to fit your needs. Keep the length under about 50 words. The *DefaultLogo.BMP* file can be replaced with your own logo. Keep the file in a square, BMP graphic format, and it should be no larger than approximately 100 by 100 pixels.

ProPrinting™
Page Descriptions

The complete **ProPrinting™** report generated by the simulation consists of 22+ pages (you can decide what pages to print using the print dialog and the ProPrint™ Setup dialog). The "core data" pages consist of four pages numbered 3 through 6, page 7 displays the simulation log, results tables are printed on pages 8 through 13, pages 14 though 17 are results graphs, finally, pages 18 through 22 provide a Glossary listing. Here is brief description of these pages:

Page 1 & 2—These pages are **Title** and **Table-of-Contents** pages. The *Table of Contents* page can be eliminated from the report by un-checking **Include Table Of Contents** in the *ProPrint™ Setup* dialog box (opened from the **File** menu within the simulation).

Page 3—Displays the first group of *Component Category* selections for the simulated engine (because of space considerations, they are not in the same order as shown on-screen).

Page 4—Displays the *Cylinder Head AirFlow* specifications for the simulated engine.

Page 5—Displays the *CamShaft Component Category* for the simulated engine.

Page 6—Displays the *Combustion* and *Exhaust Component Categories* for the simulated engine.

Page 7—This page or pages displays the Simulation Log (showing simulation issues and/or other generated data during the last engine simulation).

Page 8—Contains the first table of rpm-based simulation results, including Engine Speed, Flywheel Power and Torque, Indicated Power, Frictional Power,

ProPrinting™ Features

and Pumping Power.

Page 9—Contains the second table of rpm-based simulation results, including BASE Force On Piston, Piston Speed, Volumetric Efficiency (VE), Mechanical Efficiency, and Fuel Conversion Efficiency.

Page 10—Contains the third table of rpm-based simulation results, including indicated mean-effective pressures IMEP, BMEP, FMEP, and PMEP, plus Ignition Timing.

Page 11—Contains the fourth table of rpm-based simulation results, including Induction Airflow, Intake Manifold Pressure, Intake Port Pressure, Brake Specific Fuel Consumption (BSFC), and the Fuel Usage Rate.

Page 12—Contains the fifth table of rpm-based simulation results, including Total Retained Mass Flow, Lost Mass From Each Cylinder, Total Charge Loss, Charge Loss To Intake Reversion, Charge Lost To Exhaust Flow, Exhaust Spoilage.

Page 13—Contains the sixth table of rpm-based simulation results, displaying Nitrous Augmented Power and Torque.

Page 14—Contains a duplicate of the top-right graph of rpm-based simulation results as shown on the main program screen.

Page 15—Contains a duplicate of the center-right graph of crank-angle-based simulation results as shown on the main program screen.

Page 16—Contains a duplicate of the bottom-right graph of crank-angle-based simulation results as shown on the main program screen.

Page 17—Contains a duplicate of the left graph of crank-angle-based simulation results located under the Component Categories on the main program screen.

Page 18 to 22—These pages contain a *Glossary Of Terms* that may help explain the "language" of engine simulations to your customers or others you may provide *ProPrint* reports.

You can completely customize ProPrinting™ pages by editing the HTML *DefaultFileInput.html*. This file is the template that the simulation uses as the basis for ProPrint reports. The simulation key fields, such as _DATE, _TIME, etc., are replaced with data from the current engine test. Do not modify these keys or your reports may not include all simulation data!

ProPrinting™ **Features**

ProPrinting™ Report Modifications (Optional, Advanced)

You can customize *ProPrinting™* pages by editing the HTML **DefaultFileInput.html** (see *Important Note*, below). This *Input* file is the HTML <u>template</u> that the simulation uses as the basis for ProPrint reports. The simulation adds current engine data and sends the result as *DefaultFileOutput.html* to your default Browser. To make permanent changes to *ProPrinting™* report pages, edit the *Input* file (make sure you make a backup copy of the original file first). You will notice key fields that begin with an underscore, such as _DATE, _TIME, etc. During processing, these data keys are located by the simulation and replaced with data from the current engine test. Do not modify these keys or your reports may not include all simulation data!

> **Important Note:** Many HTML editors have their own methods of generating HTML code and may include characters and formatting that affect how reports are printed. If you are not familiar with HTML, don't attempt to modify the report Input File. Because of the unique characteristics of HTML, modifying ProPrinting™ HTML files is NOT supported by Motion Software, Inc.
>
> Also, your modified *DefaultFileInput.html* **may be overwritten** when the simulation is updated using the *Motion Updater*, so keep a backup copy of your modified file to prevent loosing your work.

SIMULATION ASSUMPTIONS

Applies To FE and WA Simulations, Except as Indicated

Dynomation6 and DynoSim6 closely simulate the conditions that exist during an actual engine dyno test. The software will predict the torque and horsepower that a dynamometer would measure while the engine and dyno are run through a programmed step test.

It is well known that engine power can vary considerably from one dyno test to another if environmental and other critical conditions are not carefully controlled. In fact, many of the discrepancies between real-world dyno tests are due to vari-abilities in what many assume are "fixed" conditions.

Among the many interviews conducted during the research and development of this simulation, dyno operators and engine owners readily acknowledged the pos-sibilities of errors in horsepower measurements. Unless the dyno operator and test personnel are extremely careful to monitor and control the surrounding conditions, including calibration of the instrumentation, comparing results from one dyno cell to another (or even one test run to another) is error prone.

Controlling these same variables in an engine simulation program is infinitely easier but, nevertheless, just as essential. Initial conditions of temperature, pressure, energy, and methodology must be established and carefully maintained throughout the simulation process. Here are some of the assumptions within this simulation that establish a modeling baseline:

Fuel:
1) The fuel is assumed to have sufficient octane to prevent detonation. Detona-tion is not modelled.
2) The air/fuel ratio is established by the user and will not vary unless the user has entered A/F data in a user-defined "Fuel-Ratio Table." For more on Air/Fuel ratio modeling, see *Air/Fuel Ratio Modeling* later in this chapter.
3) Ignition timing can be set at MBT (Minimum advance for Best Torque) by clicking the *Estimate Ignition Timing* checkbox in the *Combustion Category*, however, if MBT not set, ignition timing (and power) may not be optimum in

the simulation.

Environment:

1) Depending on the Atmospheric Standard in use (set in the Environment Group in the *Simulation Category*, inducted air is 77-degrees (F), dry (0% humidity), and of 29.234-in/Hg atmospheric pressure unless changed by the user.

2) The simulation assumes that engine, oil, and coolant have been warmed to operating temperature.

Methodology:

1) The simulated engine is put through a series of "step" tests, during which the load is adjusted to "hold back" engine torque and maintain a steady-state speed as the throttle is opened wide. The load is adjusted to allow the engine speed to rise to the first test point. The engine is held at this speed at wide-open-throttle (WOT) and a power reading is taken. Then engine speed is allowed to increase to the next step and a second power reading is taken. This process continues until the maximum testing speed is reached.

   Note: Since some engines, especially those with cam timing designed for all-out drag racing, are not able to run at full throttle under load at very low engine speeds, the power generated at some of the lower rpm points may register as zero in these circumstances.

2) Since the testing procedure increases engine speed in discrete steps, and engine speed is held steady during the power measurement, measured engine output does not reflect losses from accelerating the rotating assembly (the effects of rotational inertia in the crank, rods, etc.). These processes affect power in most "real-world" applications, such as road racing and drag racing, where engine speed is rapidly changing throughout the race.

Using Imported Cam Profile Data

"Profile" Indicates Lobe Profile Data Is Being Used.

"->10-pnt" Buttons will convert Profile data to 10-Point modeling

If cam timing for any of the lobes used in the current simulation are based on an imported *Profile* data, the W/A Lobe Data field display will switch from *10-Point* to *Profile* in the CAMSHAFT category. Profile data is the closest match to "real-world" camshaft profiles and offers the highest modeling accuracy. 10-Point data doesn't follow "true" cam profiles as well as Profile data, but has the advantage of being extremely flexible by allowing modifications to any cam-timing events and lift data.

Simulation Assumptions & Considerations

Camshaft Modeling:

As detailed in the Component Category chapter of this User Guide, the simulation uses two distinct types of cam data for simulation modeling: 1) *10-Point Timing* and 2) *Lobe-Profile* lift data. Profile data is the closest match to a specific "real-world" lobe profile and offers the highest modeling accuracy. 10-Point data, while not as precise in modeling a specific lobe, has the advantages of being extremely flexible by allowing modifications to cam-timing events and lift data; very helpful during the process of finding optimum cam timing for specific applications.

Air/Fuel Ratio Modeling:

Air/Fuel ratio has a strong influence on engine output. Fuel and oxygen molecules combine during combustion to produce heat, pressure, and residual gases. If all the carbon and oxygen atoms combine within the cylinder during combustion, no carbon monoxide or unburned hydrocarbons will remain. This "complete burning" is called stoichiometric combustion (and is a theoretical optimum condition that is not achieved in the real world). The air/fuel ratio that allows complete combustion is called the *stoichiometric air/fuel ratio*, and it is approximately 14.6:1 for gasoline. However, for maximum power a richer mixture is required, often around 12.5:1.

> **Note:** The air/fuel ratios for maximum power for all fuels modeled by the simulation are listed in the **Air/Fuel Ratio** drop-down menu within the **Combustion** category.
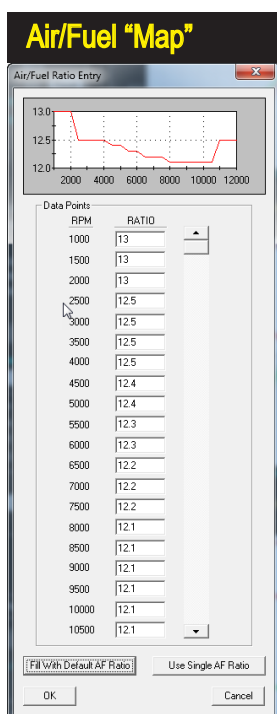
The simulation can model a varying air/fuel ratio throughout the simulation rpm test range. The *Air/Fuel Ratio Map* is available by selecting *Table* within the *Combustion* category.

Simulation Assumptions & Considerations

Forced-Induction Modeling Techniques

The simulation uses the greatest care in modeling the actual processes involved in forced-induction systems, however, the level of complexity is quite high. The bottom line: Trying strange values or combinations will probably generate less accurate results. Also, don't forget the adage "Garbage in; garbage out," as the more accurate the simulation inputs reflect the actual engine buildup, the better the results.

Finally, when *Nitrous-Oxide* augmentation is used with either Gasoline or Methanol, additional fuel flow is required to maintain the correct air(oxygen)/fuel ratio. The simulation will <u>automatically</u> increase the fuel volume to maintain the air/fuel ratio displayed in the **Combustion** category.

> **Note:** The fuel flow rate is shown in the **ProTools Table** display, along with Brake Specific Fuel Consumption (BSFC).

Forced-Induction Modeling:

There are three basic types of supercharger models available: the *Roots/Screw Modelling*, *Centrifugal Modelling*, and the *Turbocharger Modelling*. In addition, there is also an intercooler model that can be coupled with any of the supercharging systems. Each of these forced-induction models function in unique ways.

Three basic steps are performed within all of these forced-induction models: 1) Calculation of the rotational speed of the compressor, 2) Determination of the degree of air pressurization, and 3) Calculation of the effects of pressurized air on engine output. This process is very dynamic and complex, particularly in turbocharging, as it requires constant feedback between the supercharger model and the simulated engine results (such as intake and exhaust airflow rates and temperatures, power consumed by the supercharger, heat generated during charge compression, etc.) to obtain accurate results.

Roots And Screw Superchargers:

Air compression within Roots and screw superchargers is calculated from the rated volume-per-rotation of the compressor impeller, the speed of the compressor, and the efficiency of the compressor (that constantly varies as a function of pres-

Simulation Assumptions & Considerations

Roots Supercharger Simulation
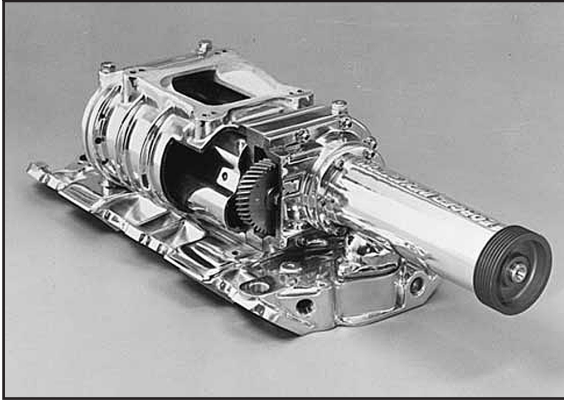
Roots superchargers do not directly compress the air but rather pump a fixed volume of air into the induction system for each revolution of the impeller. Induction pressure is calculated as the difference in the amount of air being delivered to the intake manifold relative to the amount of air being "swallowed" by the engine.

sure, temperature, the design of mechanical components, and rotational speed). Roots and screw compressors are often driven by the engine crankshaft through a belt-and-pulley or gear-train system. So for this type of supercharger, the speed of the compressor impeller can be calculated directly from the rotational speed of the engine.

A Roots compressor does not strictly compress the air, but rather "moves" a fixed volume of air mass into the induction system for each revolution of the impeller. It is here the air is compressed: pressure is a result of the difference in the amount of air being pumped into the manifold relative to the amount of air being "swallowed" by the engine. Because this pressure is dependant on engine speed, it is possible for a Roots blower to function under vacuum conditions, or lower than ambient pressure, when the engine is drawing in more air than the compressor can supply. This typically occurs at low engine speeds.

Air compression in a screw-compressor system is very similar to that of a Roots, except that a screw compressor produces a small amount of internal compression. Peak pressure regulation in the induction system of Roots or screw superchargers is controlled with a pop-off valve that releases overpressure to the atmosphere. However, the most efficient operation is obtained by carefully selecting the correct blower displacement and belt ratios so that pop-off valve operation is not required.

The work required to pump additional air into the induction system comes with a price: The energy required to operate a Roots or screw compressor is substantial. The total power consumed depends on many factors, including ambient pressure, output pressure, temperature, compressor size, compressor friction, engine displacement, engine and supercharger rpm and operational efficiency.

Centrifugal Superchargers:

Like Roots/screw superchargers, a centrifugal supercharger is driven directly from the engine, usually by the crankshaft through a series of gears or pulleys and belts. The centrifugal supercharger consists of impeller that spins at high speed.

Simulation Assumptions & Considerations

Screw Supercharger Simulation

Air pumping in a screw-compressor system is very similar to that to a Roots, except that a screw compressor produces a small amount of internal charge compression and is typically more efficient.
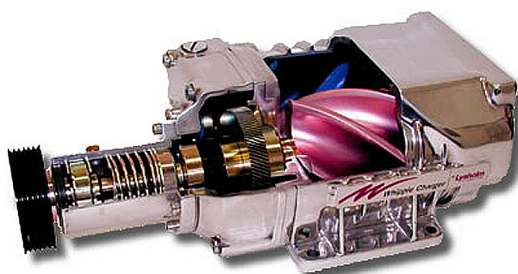
The blades on the impeller draw-in and accelerate air into the supercharger housing. The degree of compression is dependant on the speed, the size of the impeller wheel, the wheel trim (the shape of the blades), and housing (often called the "scroll") size. The pressure delivered to the engine is calculated from a "compressor map" (consisting of measured performance data for that specific supercharger supplied by the manufacturer). The map contains information about the efficiency of the compressor at various pressures and speeds. Numerical data from the map is dynamically extracted during the simulation, and data points are interpolated to ensure the highest accuracy in predictions of induction temperature and pressure.

To limit maximum output pressure, typical centrifugal systems will use a pop-off valve that opens to release the additional pressure to the inlet side of the supercharger. The power consumed by a centrifugal compressor is not as great as that for a roots or screw supercharger, but it is, nevertheless, a significant source of "lost" power.

Centrifugal Simulation
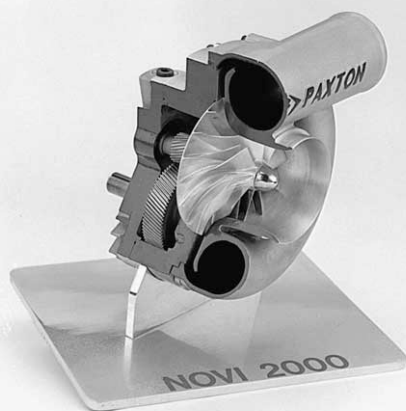
Centrifugal superchargers are driven directly from the engine, usually by the crankshaft through a series of gears and/or pulleys and belts. The centrifugal supercharger consists of impeller that spins at high speed. The blades on the impeller draw-in and accelerate air into the surrounding housing. In addition to shaft speed, the pressure delivered to the engine is also determined from data obtained from a "compressor map" (consisting of measured performance data supplied by the manufacturer for that specific compressor).

Simulation Assumptions & Considerations

Centrifugal/Turbocharger Sims

$p_0 = 961$ mbar
$T_0 = 293$ K

Maximum permissible speed

Choke line

Surge line

Pressure ratio

$\eta_{isv} = 0.75$
0.70
0.68
0.65
0.60

$200 \cdot 10^3$
$180 \cdot 10^3$
$160 \cdot 10^3$
$140 \cdot 10^3$
$120 \cdot 10^3$
$90 \cdot 10^3$
$60 \cdot 10^3$

$\dfrac{n \cdot \sqrt{T_0/T_1}}{\text{min}^{-1}}$

Volume flow rate $\dot{V}_1 \cdot \sqrt{T_0/T_1}$ [m^3/s]
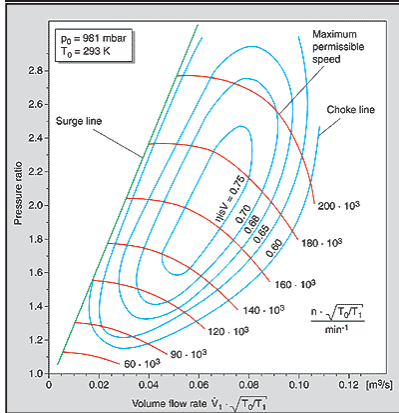
Both centrifugals and turbochargers use an efficiency "map" of the compressor, but turbochargers also use a second map for the exhaust turbine. Internal impeller speeds are based on a intricate balance between the energy used to spin the turbine and the energy needed to compress incoming air. The simulation uses an "iterative" process to balance energy generated with energy consumed to determine a stable internal shaft speed.

Turbochargers:

Turbochargers are, basically, centrifugal superchargers with the compressor element connected to a turbine wheel driven by exhaust pressure. And while calculation of compressor speed is straightforward on a centrifugal supercharger, the same cannot be said for a turbocharger. Internal shaft speed (connecting the turbine and compressor wheels) is determined by (among other things) an analysis of exhaust flow, turbine wheel size, turbine housing shape, and the efficiency read from the turbine "map" (in addition to the compressor map!). Internal wheel speeds are based on a balance between the energy used to spin the turbine and the energy needed to compress incoming air. The simulation uses an "iterative" technique to find a shaft speed that balances energy generated with energy consumed.

Turbocharger Simulation

Turbochargers are, basically, centrifugal superchargers with the compressor element connected to a turbine wheel driven by exhaust pressure. But that said, they are anything but simple in operation or simulation. The exhaust pressure drives the turbine and turbine speed is balanced against the power required by the compressor wheel. The turbocharger routines within the simulation find the optimum balance at each rpm point.

Simulation Assumptions & Considerations

Turbocharger Exhaust Wastegates
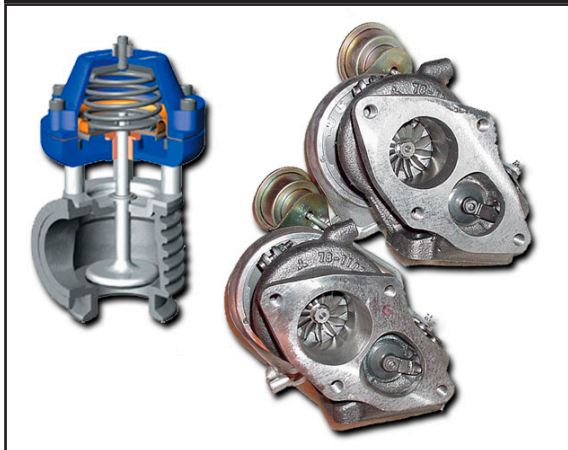
To limit maximum intake manifold pressure and to minimize back-pressure in the exhaust system, turbochargers use a pressure-limiting technique different from roots, screw, or centrifugal superchargers. When maximum boost pressure has been reached, excess exhaust-gas flow is diverting around the turbine by a "wastegate" valve. Since diverted exhaust gasses no longer impinge on the turbine, the speed of the turbine can be effectively controlled, and bypassed exhaust gas flow reduces backpressure and improves pumping efficiency.

The differences between the energy required to compress the incoming air and the energy available in the exhaust system is also the basis from which "backpressure" in the exhaust system is calculated. Backpressure robs the engine of its ability to purge itself of exhaust gasses and prevents efficient cylinder filling with fresh charge. To limit maximum intake manifold pressure and to minimize backpressure in the exhaust system, the turbocharger uses a pressure-limiting technique different from roots, screw, or centrifugal superchargers. When maximum boost pressure has been reached, excess exhaust-gas flow is diverting around the turbine through a "wastegate" valve. Since diverted exhaust gasses no longer impinge on the turbine, the speed of the turbine can be effectively reduced and controlled, and the bypassed exhaust gas flow reduces backpressure and improves engine efficiency.

> **Note:** The turbocharging algorithms used in the simulation were designed to model the properties of a "stable-state" turbo system. In other words, transient factors, like turbo "lag" that can occur when the engine is rapidly increasing in rpm, are not modeled. The simulation will provide a "steady-state" analysis of a turbocharged engine, that is, a power prediction when the engine reaches a steady-state condition on the dynamometer at wide open throttle at each test rpm.

Intercoolers:

The temperature of the air entering the engine from a supercharging system is determined by several factors, including: 1) The *Gas Law* description of how air will always increase in temperature as it is compressed, and 2) The efficiency of the compressing device; the lower the efficiency, the greater the temperature rise.

Intercoolers, sometimes referred to as "aftercoolers," are used to reduce the temperature of the compressed air exiting the supercharger before it enters the engine. Intercoolers are rated by their ability to absorb heat, and the ability to move air through the maze of passages within the intercooler (determines the amount of

Simulation Assumptions & Considerations

Intercooler Modeling

Higher induction temperatures, common on blowers with pressure ratios of 2.0 or higher, can cost horsepower _and_ reliability. An Intercooler cools the inducted air the same way that heat is removed from the engine itself: with a radiator. Intercooling is the key to power and reliability in high-boost systems.

pressure reduction across the intercooler—some efficient intercoolers have pressure drop of less than 10%). The higher the intercooler efficiency, the more heat the intercooler removes from the compressed charge and the lower the pressure drop loss applied to the induction system.

Simulation File Compatibility

Motion Software users have created countless numbers of engine simulation files in several different file formats that have "evolved" as the simulation has improved over time. Many of these engine files are available on the Internet on enthusiast sites (with extensions of .DYN, .DYM, and .SDY). The simulation can now can open engine files created in most previous Windows-based versions of Dynomation, DynoSim and the DeskTop Dyno. If you are a user of the DOS-based Dynomation sold in the mid 1990's, you can import these files, too.

Importing Engine-Simulation Files
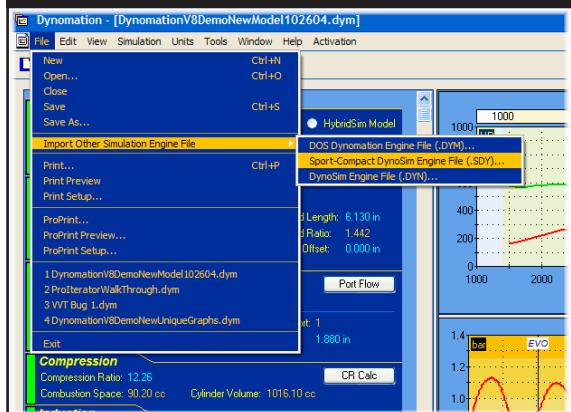
Motion Software simulations have been used successfully for many years. Many of our engine simulation files are available on the Internet on enthusiast sites (search for _.DYN_, _.DYM_, _.SDY_ and _.DXML_ files). Version-6 simulations can import engine files created in many earlier program versions, including older DOS-based Motion simulation files.

Simulation Assumptions & Considerations

Importing older engine simulation is easy, just use the *Import Other Simulation Engine Files* selection from the **File** menu (don't try to *"Open"* these files directly; they must be **Imported**).

> **Note:** You can install engines simulated in this program into vehicles using DragSim and test 1/8- or 1/4-mile performance. And using FastLapSim you can test any simulated engine on virtually any closed-course track. Find out more about the other Motion Software simulations on www.MotionSoftware. com and on www.ProRacingSim.com.

Dynomation6 Wave-Action And DynoSim6 Filling-And-Emptying Functional Differences		
Motion Engine Simulation Capabilities	**Filling-And-Emptying Simulation** DynoSim6 and Dynomation6	**Wave-Action Simulation** Dynomation6 Only
Dyno-Testing RPM Range	1000 to 14500 rpm	1000 to 14500 rpm
Bore Range Limits	2.00 to 7.00-in	2.00 to 7.00-in
Stroke Range Limits	1.50 to 7.00-in	1.50 to 7.00-in
Forced-Induction Modeling (1)	Yes	Yes
Extended-Data Displays (2)	Yes	Yes
ProPrinting™ Extended Dyno Reports (2)	Yes	Yes
ProData™ Tables (4)	Yes	Yes
Graph DataZones™ (5)	Yes	Yes
Prolterator™ Testing (6)	Yes	Yes
Analyze Area Under Power And Torque Curves (7)	Yes	Yes
Full Wave-Action Simulation (8)	No	Yes
Analyze Induction & Exhaust Pressure Waves (9)	Simplified Modelling	Full Analysis
Model Intake & Exhaust Runner Lengths & Taper Angles (10)	No	Yes
Megaphone/Normal Headers (11)	No	Yes
Transparent 3D-Engine Pressure-Analysis Display (12)	Yes	Yes
Graph Reticle Synchronized With 3D Engine Analysis (13)	Yes	Yes
Import Earlier Engines, Port Flow And CamFiles (14)	Yes	Yes
Import FlowPro Flow-Bench Files (15)	Yes	Yes
Import CamPro+, S96, CamDoctor Lobe-Profile Files (16)	Yes	Yes
Supplied With 7000+ Lobe-Profile Library (17)	Optional	Included
View 3D-Engine Mass-Flow Analysis (18)	Yes	Yes

Dynomation6 and *DynoSim6* use two distinct simulation models: The *Filling-And-Emptying* (used in DynoSim6) and the *Wave-Action* (used in Dynomation6). Each of these simulation methods have their strong points, and like using the right tool for the job, selecting the best simulation model will help you find optimum component combinations in the shortest time and at the lowest cost. Refer to this chart for a feature comparison of our engine simulations.

Simulation Assumptions & Considerations

Dynomation6 and DynoSim6
Simulation Features

Our Version6 engine simulations utilize two distinct simulation models: The *Filling-And-Emptying* (in DynoSim6) and the *Wave-Action Model* in Dynomation6 (Dynomation6 can use either model depending on user selection). Each of these simulation methods have their strong points, and like using the right tool for a specific job, selecting the best simulation model will help you find optimum component combinations in the shortest time. Referring to the chart on the previous page, here is a description of each feature and capability included in the software:

(1) Forced-Induction Modeling: Adds sophisticated turbocharger, roots and centrifugal supercharger modeling. Display manifold pressures, modify belt ratios, pressure ratios, A/R ratios, intercooler efficiency, and more! Also includes wastegate modeling.

(2) Extended-Data Displays: Professionals often need additional data beyond power, torque, elapsed time, speeds, and other "basic" measurements to optimize engine and vehicle performance. To provide this data, internal physics models are extended, "in-depth" data is calculated and made available for graphic and table-based displays. Extended data types depend on the simulation, but examples include various engine pressures, frictional power consumption, trapped cylinder mass, etc.

(3) ProPrinting™: ProPrinting produces custom, full-color, multi-page test reports, including cover pages, tables of contents, your name and/or company name and logo, and even an optional glossary. ProPrinting produces an easy-to-read, comprehensive, professional printed test reports that you will be proud to display, present to customers, or just to keep a testing journal.

(4) ProData™ Graphic Displays And Tables: Includes the calculation and display of even more simulation data often needed by professional racers and/or engine developers.

(5) Graph DataZones™: DataZones allow you to display additional graph data and/or show ranges for target data values. DataZones can clearly illustrate dangerously high pressures, engine speeds, or almost anything you wish to highlight. DataZones can be displayed in any color(s), and you can automatically select a gradient range between beginning and ending colors.

(6) ProIterator™: *Iterative™* testing, an exclusive feature of Motion Software products, is an automated method of performing multiple tests and locating optimum combinations. The *ProIterator™* provides maximum flexibility in selecting components and setting up testing criterion. Search for optimum power/torque/acceleration, or locate the component combination that produces maximum "area-under-the-curve" for horsepower, torque, acceleration, or other variables.

Simulation Assumptions & Considerations

Once you have established the testing criterion, the *ProIterator* will begin a series of tests and display the best 10 results, all without user intervention.

(7) Analyze Area Under Data Curves: Part of the *ProIterator™*. While peak values of torque, horsepower, acceleration, and other simulation data focus on the "absolute maximum," the areas under these curves represent the greatest "volume" of power or acceleration. Think of this area as the maximum horsepower throughout the rpm range. The *ProIterator* can search for the 10 best combinations that produce the greatest area under any portion of a selected data curve.

(8) Full Wave-Action Simulation: Finite-amplitude, wave-dynamics analysis using the Wave-Action, Method-Of-Characteristics mathematical model, Dynomation6 simulates the complex interactions of pressure waves in the intake and exhaust passages. Since these waves significantly affect the flow characteristics within engine ducting, a comprehensive analysis of pressure pulses and how they interact with runner length and taper angles is an essential element in accurately determining mass flow within the IC engine.

(9) Analyze Induction And Exhaust Pressure Waves: Comprehensive graphs and data displayed in tables, synchronized to the graph-reticle position (indicating the instantaneous crank-angle), are provided in the Dynomation6 engine simulation. DynoSim6 also offers these same features, however, pressure waves are pre-calculated in DynoSim and so offer lower resolution than the wave presentation in Dynomation6. These pressure-wave data-sets are clearly displayed and can be compared with pressure pulses at different engine speeds or even with different engine designs.

(10) Model Intake/Exhaust Runner Lengths And Taper Angles: The "secrets" to engine performance often lie in the subtle elements of engine design, like the lengths and shapes of induction and exhaust passages. Dynomation6 allows anyone to test a wide variety of runner lengths and taper angles to determine, not only how these critical elements affect engine power throughout the powerband, but also to select optimum shapes for virtually any application.

(11) Megaphone And Normal Header Design Modeling: The Dynomation6 engine simulation models single exhaust pipes per cylinder, "standard" headers with collectors, and megaphone exhaust systems. All tubing lengths, diameters, and taper angles can be tuned for a variety of applications. Megaphone exhaust systems are often used in racing motorcycle applications, but are sometimes applied to performance automotive applications.

(12) Transparent 3D-Engine Pressure-Analysis Display: Provides a unique and powerful way to visualize the motion of pressure waves within a running engine. A cutaway view of a engine, rendered in real time using sophisticated 3D graphics, synchronizes valve and piston motions to the pressure-crank angle diagram. As you move the reticule across the diagram—to any crank angle within

the 720 degrees of the 4-cycle process—the motion of the pistons, valves, and pressure waves are displayed. This unprecedented technology allows you to "see inside" the engine, providing the ultimate tool for pressure-wave understanding and analysis.

(13) Graph Reticule Synchronized With Engine Data: Moveable reticule lines on graphs that are synchronized with the underlying data. By moving the reticule over the graph, the exact data can be displayed.

(14) Import DynoSim & SC-DynoSim Engine, Cam, & Flow Files: Import and test engine, cam, and flow files from virtually all other Motion Software engine simulations. This makes testing and analysis much easier for those who have developed a library of data files from other Motion simulations.

(15) Imports FlowPro Flow-Bench Test Files: This simulation models intake and exhaust port flow from measurements taken at a wide variety of pressure drops and valve-lift points. To accommodate the professionals that uses Audie Technology software on their flow bench, it also directly reads FlowPro files.

(16) Import CamPro, CamProPlus, S96, and CamDoctor Cam-Profile Files: Supports the direct import of a number of commonly used cam-profile formats, including CamPro, CamPro+, S96, and Cam-Doctor files. A comprehensive Import dialog helps you assign centerline values and determine valve-open duration from lifter-rise measurements.
Motion Software offers a ProFile Library that contain 3500+ modern lobe profile files specifically designed for these engine simulations.

(17) Supplied With 5000+ Lobe-Profile Library: The *Dynomation6* software package includes a separate CD with an extensive *Lobe-Profile Library* of more than 5000 modern lobe profile files from stock to ProStock applications! The Lobe-Profile Library is also compatible with DynoSim6, but is an optional item (can be purchased at any time).

(18) View 3D Engine Mass-Flow Analysis: Includes a cutaway view of a engine allowing you to view the motion of the pistons, valves, and pressure waves within the engine. In addition, a 3D visualization of mass flow into and out of the cylinders is also provided.

Also: Automatic Software Updates—Includes Motion's Automatic Software Update tool that automatically checks for updates (you can initiate a update check at any time by selecting *Check For New Version* from the **HELP** menu in the program). Users not only receive the typical bug fixes and minor enhancements, but often Motion Software, Inc., provides new features and modeling capabilities. More than 100 software enhancements have been made available FREE OF CHARGE for both Dynomation and DynoSim within the past few of years!

Wave-Action Filling & Emptying Simulations 6

WAVE-DYNAMICS ANALYSIS

If you read the previous chapters in this Users Manual, you've discovered some of capabilities of this simulation and its ability to use gas dynamics analysis in IC engines. This chapter takes you considerably deeper into the theory of finite-amplitude waves and offers help in interpreting wave-dynamic results. We will examine how exhaust and intake runner lengths "tune," you'll see the effects of blowdown, you'll discover pressure waves as they "bounce" off of the open and closed ends of passages, you'll uncover how stepped headers affect cylinder pressures, and much more.

DynoSim Note: *DynoSim6* analyzes pressure waves, similar to the *Wave-Action* model provided in *Dynomation6*, however, it does not resolve runner lengths, passage dimensions, minimum and maximum areas, etc., as is possible with Dynomation6. Pressure waves and runner dimensions in DynoSim6 are pre-calculated, reducing data input, but because passage dimensions and the instantaneous states of pressure waves are not a part of the analysis, DynoSim6 is not able to determine all of the complexities that can occur in unique and racing engines. Despite this, the advanced pressure-wave algorithms used in DynoSim6 are remarkably accurate in modeling stock and high-performance engines of all types, even Diesel engines!

A review of this chapter will broaden your understanding of the displayed pressure waves and particle flows illustrated in the graphs in DynoSim6 and Dynomation5. Using this knowledge will allow you to take advantage of many of the insights possible from a look "inside" a running engine.

THE IC ENGINE: AN UNSTEADY FLOW MACHINE

The air/fuel mixtures and exhaust gasses that move within the passages of the internal-combustion (IC) engine behave in an *unsteady* manner. In other words, the gases are constantly changing pressure, temperature, and velocity throughout the four-cycle process of induction, compression, expansion, and exhaust. For example, when the intake valve is closed, the gas velocity at the valve is zero. When the valve

Wave-Dynamics Analysis

begins to open, a difference in pressure between the cylinder and the port begins to accelerate gas particles into (or out of) the cylinder. This gas motion—and all other gas particle motion within the engine—starts and stops, squeezes and decompresses, heats up and cools down. To analyze and simulate these actions, designers and programmers must rely on the discipline of *Unsteady Gas Dynamics*. A basic knowledge of this subject and the ability to visualize (and calculate using simulations) wave interactions inside the IC engine is probably the single most important "tool" available to the modern engine builder.

The application of Unsteady Gas Dynamics does not require that we cover the development of gas-flow equations (thank goodness!!), but a general description of the mechanisms that apply to the IC engine are essential.

Acoustic Waves Vs. Finite-Amplitude Waves

The sounds we hear around us are actually small pressure disturbances in the air. We call these pressure "pulses" *acoustic waves*. The pressure amplitude (volume) of these waves is very small. As an example, the volume at which you will begin to experience pain from sound occurs around 120 decibels and creates a peak pressure of only 0.00435psi above the ambient, undisturbed air. Since sea-level air pressure (barometric pressure) is about 14.7psi, then the *pressure ratio* at which sound become painful is:

Pr = P / Pa
  where
P = 14.7 + 0.00435 Psi
Pa = 14.7 Psi

Finite Amplitude Wave Strengths

Finite-Amplitude Waves   **Sound Waves**

Finite Amplitude Waves In IC Engine Are 10,000 Times Stronger Than Sound Waves
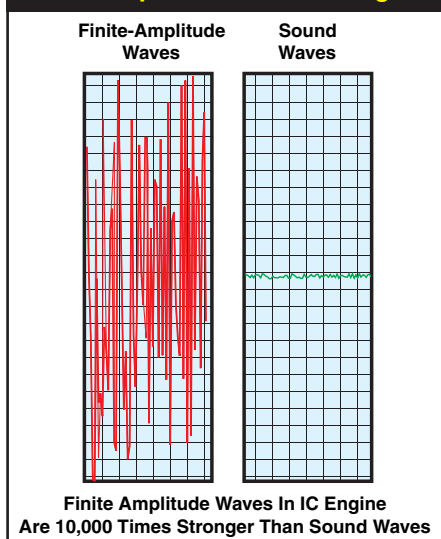
Very loud acoustic waves create small pressure disturbances. *Finite-amplitude waves* found in the IC engine, however, create pressure disturbances almost 10,000 times greater than painfully-loud sound waves. Because of this huge difference in amplitude, finite-amplitude waves exhibit their own, unique characteristics, requiring a radically different mathematical analysis than sound waves.

Wave-Dynamics Analysis

so:

Pr = 14.70435 / 14.7

Pr = 1.0003 or a 0.03% Increase In Pressure

that's three one-hundredths of one percent increase over atmospheric pressure!

Obviously, very "loud" acoustic waves create very small pressure disturbances. There are waves that produce substantially higher pressure ratios than even loud sound waves. These powerful energy-charged waves are called *finite-amplitude waves*. Pressure disturbances at these higher intensities can be found in the induction and exhaust passages of the IC engine. Remarkably, pressure ratios of 2.5 can be readily measured (that's a pressure ratio almost 10,000 times greater than painfully loud sound waves—certainly something you would never want to "hear"). This enormous difference in intensity between acoustic and finite-amplitude waves gives some insight into why the misnomer "acoustic theory" (still commonly used) for calculating optimum "tuned lengths" of intake and exhaust passages is misapplied (more on this later).

Compression And Expansion Waves

Finite-amplitude waves take two forms within the IC engine: *Compression* and *Expansion*. The *Compression* wave is a positive pressure disturbance that will always have a pressure ratio greater than one. The *Expansion* wave is a strong drop below ambient pressure, and therefore will always have a negative pressure ratio (less than one). Expansion waves are known by other names, such as "rarefaction waves" or "suction waves," however, they all refer to the identical phenomenon.

Compression and expansion waves act in similar—but uniquely different—ways as they move through IC engine passages. Understanding how these waves move and how they interact with their surroundings is an important part of understanding how gasses move inside the IC engine. The first piece of the puzzle is illustrated in

Compression And Expansion Waves

Figure 1

Compression Wave Motion

+ Pa -

Particle flow direction
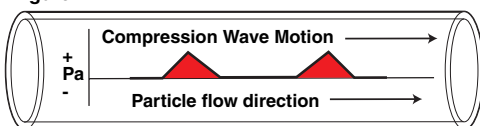
Compression waves drive particles in same direction as wave

Figure 2

Expansion Wave Motion

+ Pa -

Particle flow direction
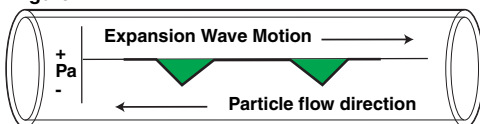
Expansion waves drive particles in opposite direction of wave

As positive compression waves travel from left to right through a pipe (top), they drive gas particles in the same rightward direction. When expansion waves (lower) travel from left to right, they pass through gas particles in the pipe and propel them in the *opposite* direction, in this case, leftward.

Wave-Dynamics Analysis

Figure-1. This drawing depicts a positive compression wave traveling from left to right through a pipe. As the pressure waves travel rightward, they drive gas particles in the same rightward direction. However, the velocity of the gas is considerably slower than the speed of the pressure waves. There are many analogs to this in everyday life. For example, consider logs that are pushed ashore by waves on the surface of a lake. The waves wash through floating logs driving them forwards but at a much slower rate than the speed of the waves.

An expansion wave is shown in **Figure-2**. This low-pressure wave is also traveling from left to right. However, as it passes through gas particles, its lower pressure gradient draws particles toward the wave, in effect, driving particle flow in the *opposite* direction, in this case, leftward.

In addition, both compression and expansion waves change their character when they encounter sudden transitions in area. Area changes within engine ducting occur at an open end, a closed end, or at transition to smaller or larger diameter passages. Perhaps the most familiar area change occurs at the end of an exhaust header pipe. The primary tube either abruptly ends at the opening to the atmosphere, or it empties into a larger-diameter collector that opens to the atmosphere several inches downstream. Beyond this very visible (and easily heard) pipe transition, there are many other changes in area within engine ducting. For example, where the intake runner transitions into the manifold plenum or directly into the atmosphere (in the case of injector stacks), a substantial area change is produced. Furthermore, the intake and exhaust valve sits at the end of a pipe that is either closed or partially open, again creating significant changes in areas. What happens when a finite-amplitude wave reaches one of these area transitions?

First, let's examine a case where pressure waves reach the closed end of a pipe. **Figure-3** illustrates a positive-pressure compression wave striking the closed end of a passage. It returns with its original profile intact. The only change is that the pressure wave is now moving in the opposite direction. Consider how this affects gas particle motion. When the compression wave travels rightward, it nudges the gas

Wave Interaction With Closed Pipe End

Figure 3

Compression waves before reaching closed end of pipe.

Compression waves after reflecting from closed end. Note that the reflected waves are still positive.

Figure 4

Expansion waves before reaching closed end of pipe.

Expansion waves after reflection from closed end. Note that the reflected waves return as expansion waves.
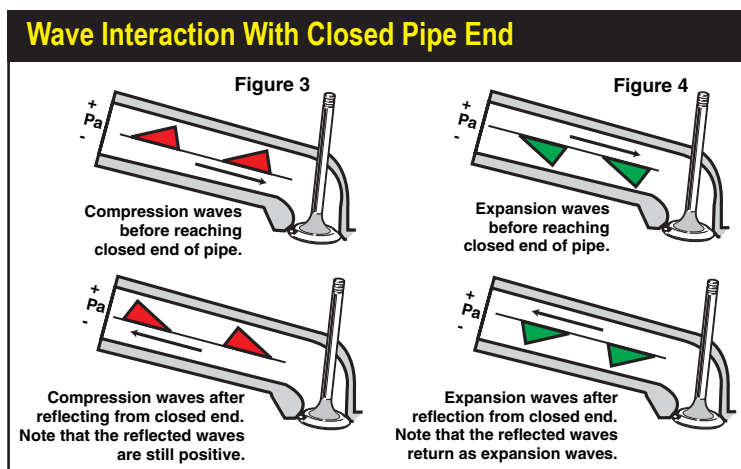
Positive pressure compression waves striking the closed end of a passage return with their original profile intact. Expansion waves behave similar to compression waves. Net particle motion is zero.

Wave-Dynamics Analysis

A leftward-moving compression wave is reflected as a _rightward-moving expansion wave_ when it reaches the open end of a pipe (figure 5). Particle movement is leftward in both cases. A leftward moving expansion wave is reflected as a _rightward-moving compression wave_ (figure 6). Particle movement is rightward in both cases.

Wave Interaction With Open Pipe End

Figure 5

+ Pa -

Compression waves before reaching open end of pipe.

+ Pa -

Compression waves after reflecting from open end. Note that the reflected waves return as expansion waves.

Figure 6

+ Pa -

Expansion waves before reaching open end of pipe.

+ Pa -

Expansion waves after reflection from open end. Note that the reflected waves return as compression waves.
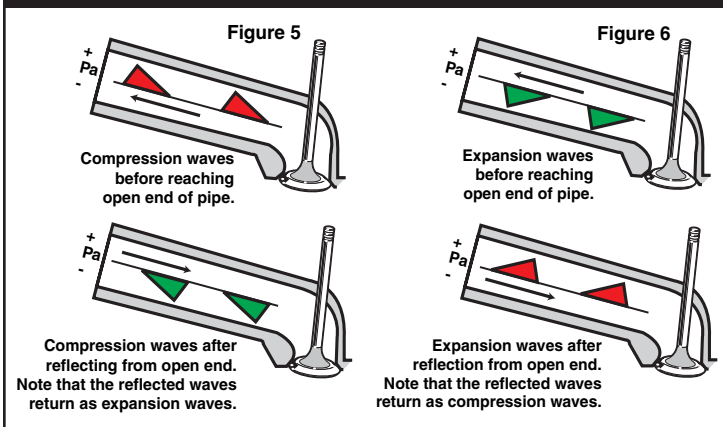
particles toward the right. After reflection, the compression wave nudges the particles leftward. The gas particles are returned to their original position; there is no net flow in the pipe. This is exactly what you would expect to find in a pipe that is closed at one end! **Figure-4** depicts an expansion wave as it travels rightward within the same closed-end pipe. Behaving similar to a compression wave, the reflected leftward moving wave remains an expansion wave with the same profile. Net particle motion is, again, zero.

Now let's explore the interesting actions of finite-amplitude waves as they move within an open-end pipe. **Figure-5** shows the arrival of a compression wave at a transition to a larger area. Notice that leftward-moving compression wave is reflected as a _rightward-moving <u>expansion</u> wave_. This has fascinating implications for particle movement. Initially, while the compression wave traveled leftward, it helped propel gas particles in the same leftward direction, toward the pipe end. When the expansion wave is created and it begins rightward movement, it continues to drive particles in the leftward direction (because expansion waves move particles in the opposite direction of wave travel). _Finite-amplitude compression waves moving <u>toward</u> the open_

Simple Kadenacy Theory

Incorrect "Kadenacy" Theory For Exhaust Gas Dynamics

High Pressure "Slug"
(similar to compressed spring)

Low Pressure Follows
(similar to expanded spring)
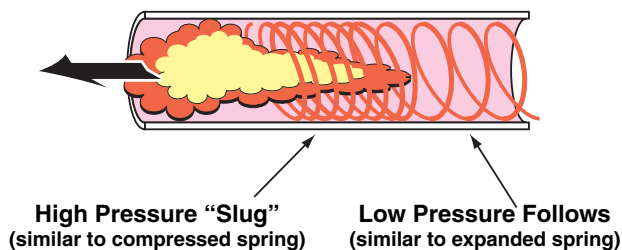
In an effort to explain gas flow in the exhaust system, it was believed that when the exhaust valve opened, a high pressure "slug" of gas blasted out of the port and down the header pipe. As this slug moved, it created a low-pressure "wave" behind it. This Kadenacy theory was conclusively disproved nearly 60 years ago. Despite this, it is still widely believed by engine "experts" to this day!

end of a pipe provide a "double assist" to particle movement in the same direction. Now consider the same open-end pipe, but this time a leftward moving expansion wave is illustrated in **Figure-6**. As the expansion wave approaches the open end, it moves particles in the opposite direction, away from the end of the pipe. When the expansion wave reaches the pipe end, it is reflected as a compression wave and moves rightward, driving particles in the same rightward direction. S*o expansion waves "double-assist" particle movement <u>away</u> from the open end of the pipe.*

These pressure wave phenomena, particularly as they apply to the exhaust system, were not understood until the 1940s. Until that time, it was assumed that a high-pressure gas particle "slug" moved through the header pipe and created a vacuum behind it that helped to draw out residual gases. This "Kadenacy" theory— named after its inventor—is analogous to the compression waves traveling through a "Slinky™" coil-spring toy; a tight group of coils (representing high pressure waves) moves along the spring followed by a more open group of coils (representing low-pressure waves). Despite the fact that this theory was conclusively proven to be incorrect over 60 years ago, it is still believed by some engine "experts" to this day!

Pressure Waves And Engine Tuning

The goal of the high performance engine builder is to tune the lengths of IC engine passages so that the reflected waves reach the cylinder at the most effective times, either assisting exhaust-gas outflow or induction-charge inflow. If there ever was a statement that qualified for the adage "easier said than done," then this is it! There are many factors that affect the arrival of these waves. A short list would include valve timing, cam profiles, piston speed, pipe lengths, valve discharge coefficients, and cylinder blowdown pressures. To make matters worse, the peaks of finite amplitude waves travel faster than the base of the waves, causing wave profiles to distort as they travel through engine passages (see **Figure-7**). This can ultimately cause the waves to "tumble" over themselves, forming shock waves and converting their energy into heat. All of these complex interactions occur simultaneously and make it easy to see why the simple *acoustic* formulas that engine builders have traditionally used to determine "tuned lengths and pipe diameters" are not applicable.

Finite-Amplitude Wave Distortion

Figure 7

Velocity (Aa + Ap)

Note Shock Wave Forming

V(Aa+Ap) > V(Aa)

+

Pa

−
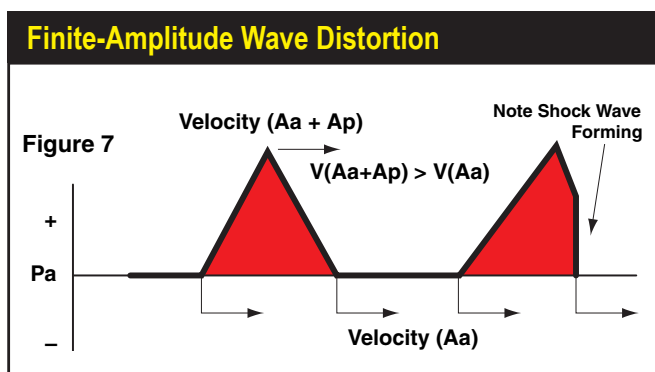
Velocity (Aa)

The peaks of finite amplitude waves travel faster V(Aa + Ap) than the base of the waves (Aa) acoustic velocity, causing wave profiles to distort as they travel through engine passages. This can ultimately cause the waves to "tumble" over themselves forming shock waves and converting their energy into heat.

Wave-Dynamics Analysis

There are only two practical methods to determine effective pipe lengths and diameters. One is to build the engine, install it on a dyno, connect pressure-reading transducers to the intake and exhaust passages and cylinders and record pressure data. By analyzing these pressure signatures and running a series of tests with various component combinations, the engine builder can develop an effective engine for the desired purpose. The problem with this method is the associated high costs in time and money. Another method of finding effective pipe lengths and "zeroing in" on optimum component combinations is to *simulate* the pressure waves and particle flow that occurs within the IC engine. The problem with this method was that, until recently, an affordable four-cycle, pressure-wave simulation program for personal computers simply did not exist.

In 1993, V.P. Engineering introduced Dynomation DOS, a PC computer program that applied the *Method Of Characteristics* mathematical solutions to the analysis of finite-amplitude waves within the 4-stroke racing engine. This early DOS version of Dynomation allowed engine builders to analyze pressure-waves and particle-flow motions (although at a snail's pace). It displayed intake, exhaust, and cylinder-pressure curves along with intake and exhaust flow velocities and directions. The experienced engine builder used this information to develop better component combinations.

Now with introduction of *Dynomation6* from Motion Software, this powerful simulation method runs under Windows about 10 to 100 times faster, displays simulation results in custom graphics, includes 3D engine renderings, calculates mass flow in ports and passages, includes more sophisticated modeling, forced induction, multiple fuel modeling and much more.

PRESSURE-TIME HISTORIES

Pressure-Wave Measurement

Figure 8

Pressure Transducers
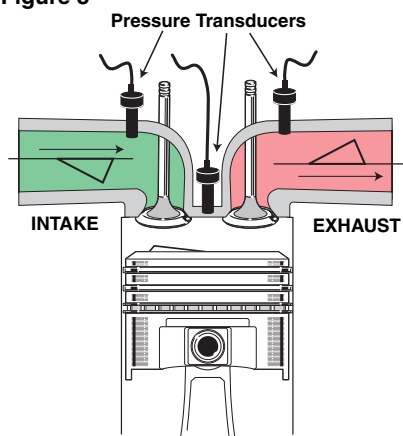
INTAKE   EXHAUST

Engine pressures can be directly measured with special transducers precisely positioned in the intake and exhaust passages and directly in the cylinders. The intake and the exhaust transducers are often located near the valves. The cylinder pressure transducer is located in the cylinderhead, protruding into the combustion chamber.

Wave-Dynamics Analysis

The direct measurement of engine pressures can reveal a great deal about engine function. An analysis of engine pressures—throughout the complete four-cycle process—reveals the combined effects of all mechanical components, plus the thermodynamic effects of heat transfer, the results of all finite-wave interactions, and mass flow of induction and exhaust gasses within the engine. The measurement of these pressures requires the precise placement of pressure transducers in the intake and

Pressure Crank-Angle Diagram

Figure 9 — Pressure Crank-Angle Graphs
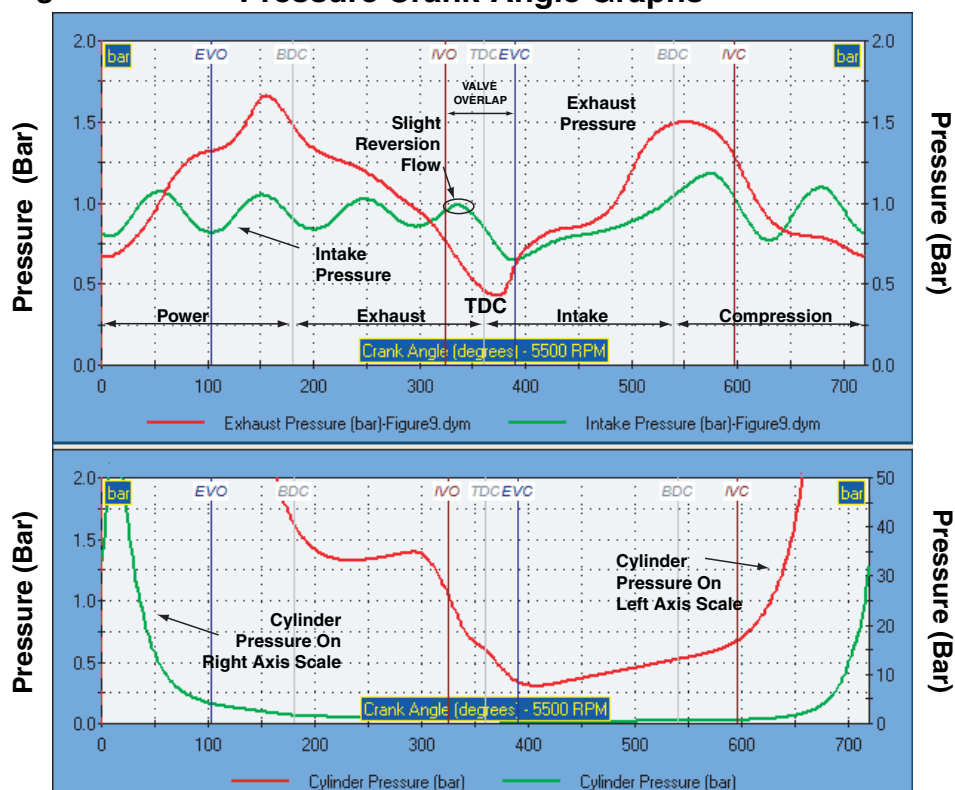
These graphs are called pressure crank-angle diagrams. They trace out the pressures measured throughout all four engine cycles, as if recorded by (simulated) transducers located in the ports and cylinders (see photo, previous page). In the upper graph, notice when the exhaust valve opens (EVO). The formation of the primary exhaust pulse occurs. Cylinder pressure (shown in the lower graph) quickly falls and begins to approach exhaust port pressure. The pressure wave created at EVO returns to the cylinder during overlap. It creates subatmospheric pressures that stops reversion and helps induct fresh charge. If the exhaust system was designed correctly, the expansion wave will continue to assist induction flow throughout overlap until the exhaust valve closes (EVC).

Wave-Dynamics Analysis

exhaust passages, and directly in the cylinders of the engine. **Figure-8** shows typical locations. The intake and the exhaust transducers are often positioned close to the valves. The cylinder pressure transducer is located in the cylinderhead, protruding into the combustion chamber. The typical pressures recorded by these transducers is illustrated by the graphs shown in **Figure-9**.

This plot is referred to as a pressure crank-angle graph or pressure time-history diagram. The horizontal axis displays the crank position throughout the 4-cycles from 0-720 degrees. The zero point on the left of the graph starts at TDC just before the power stroke. Moving rightward, the first vertical line indicates the exhaust valve opening point (EVO), typically occurring slightly after 100 degrees. Continuing rightward, the next vertical line pinpoints intake valve opening (IVO) around 330 degrees. The exhaust valve closing point (EVC) occurs just under 400 degrees. The distance between the IVO and EVC graphically indicates the valve-overlap period. Around 600 degrees the intake valve closes (IVC). Finally, the diagram ends at 720 degrees with the piston back at TDC, marking the end of the compression stroke. The vertical axis of the diagram indicates pressure in terms of the pressure ratio (or *Bar*). A value of 1.0 represents standard atmospheric pressure (Pa). A value below this indicates subatmospheric pressure or *expansion* waves. Values above 1.0 are positive pressures above atmospheric and represent *compression* waves.

Note: The axis for the left and right vertical axis can be independently set. Notice that the left and right vertical axis on the upper graph both display up to 2.0 Bar, while the lower graph has independently scaled vertical axis set at 2.0 and 50.0 Bar, even though both axis display *Cylinder Pressure*. This allows a broader view of this widely changing pressure.

Now let's trace out the pressures measured throughout all four engine cycles (on **Figure-9**) as recorded by the three simulated transducers. Keep in mind that the pressures illustrated in this diagram reflect an engine that is "in-tune." (We'll explore out-of-tune pressures later.) When the exhaust valve opens (EVO), the formation of the primary exhaust pulse occurs. This high-pressure compression wave is created when the exhaust valve releases cylinder pressures (produced by the combustion of fuel and air) into the exhaust port and header pipe system. Cylinder pressure quickly falls and begins to approach exhaust port pressure. Before IVO, an intake pressure wave—created during the previous intake cycle—bounces between the closed inlet valve and the open end of the runner. This is shown by the oscillating wave (green) on the diagram alternately becoming an expansion and compression wave, with pressures centering around 1-Bar, or atmospheric. When the intake valve opens, cylinder pressure is almost always higher than the pressure in the induction system. For the short period of time that this pressure differential exists (intake pressure pulse indicated as *Reversion Flow* on graph), a reverse flow of exhaust gasses begins to move into the induction tract. Very little reverse flow generally occurs since piston speed is quite slow near TDC. During this time, the pressure wave that was generated when the exhaust valve opened reached the end of the header pipe and has now returned to the cylinder during the valve overlap period (between IVO and EVC) as

an expansion wave. This wave creates subatmospheric pressures that stops reversion and helps draw in fresh charge from the induction system <u>before</u> the piston begins to move down from TDC on the intake stroke. If the exhaust system (and several other engine parameters) are designed correctly, the expansion wave will continue to assist induction flow until the exhaust valve closes (EVC), as shown in **Figure-9**.

With the ending of the overlap period, the intake stroke begins in earnest. Throughout at least the first half of this period, inlet pressure closely follows cylinder pressure. Just before IVC a pressure surge or "hump" in the induction pressure begins to increase cylinder pressure. This surge is caused by a strong compression wave arriving from the induction system (created during the previous intake stroke), helping to "ram" fresh charge into the cylinder and minimize back flow. The arrival of this critically important compression wave is determined by runner length and shape, engine speed, and other factors. After IVC, the cylinder pressure continues to rise as the piston heads for TDC on the compression stroke.

Gas Flow Vs. Engine Pressures

Up to this point, you may have assumed that there exists a direct relationship between the pressures measured in the ports and the flow of gasses. Furthermore, you may have assumed that when a pressure transducer registers a compression wave within a passage that opens on the right, gas flow would move in a rightward direction. In some instances gas flow does not coincide with pressures. **Figure 10** will help shed some light on this confusing issue. As compression **Wave-1** moves rightward it propels gas particles toward the right. A second compression wave, **Wave-2,** moves leftward in the same passage and propels gas particle toward the left. When **Waves 1** and **2** pass through each other they form the superposition **Wave-3 (1 + 2)**.

Pressure-Wave Vs. Gas-Particle Flow

Figure 10

Wave-1   Wave-2

+

Pa

1   1 + 2   2

Superposition
Wave-3

-
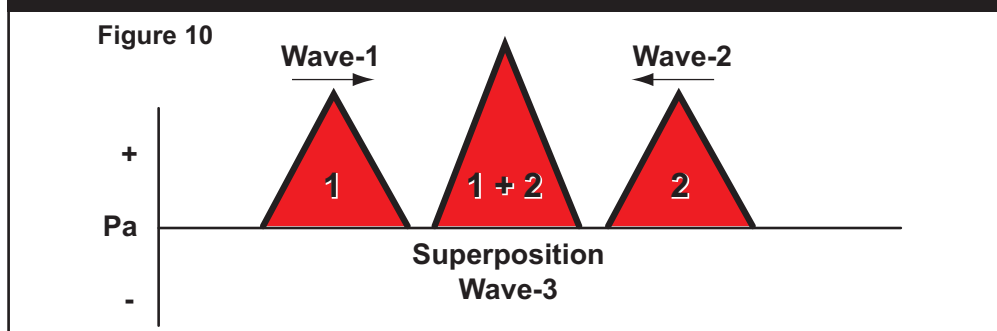
A direct relationship between measured pressures and the flow of gasses within the ports does not always exist. Consider compression Wave-1 moving rightward and propelling gas particles toward the right. Wave-2 moves leftward in the same passage and propels gas particles toward the left. When Wave-1 and Wave-2 pass through each other they form the superposition Wave-3 (1 + 2). Gas particles come to rest, but a pressure transducer measuring this "new" wave would register a significant pressure increase.

Wave-Dynamics Analysis

Pressure-Wave Measurement
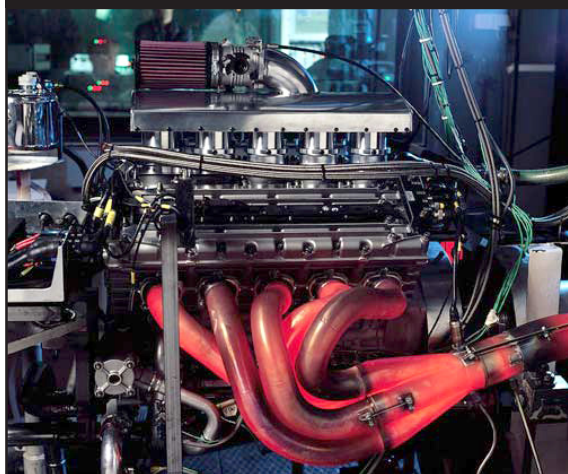
The first part of effective intake tuning begins with good exhaust system tuning. When the exhaust valve opens it generates a powerful positive pressure wave that travels to the end of the header pipe and returns to the cylinder as a strong expansion wave. When the timing is right, this suction wave will arrive during valve overlap and draw out burnt gases while starting the inflow of fresh charge, optimizing horsepower.

Gas particles that were being propelled towards each other by the separate waves will collide and come to rest within **Wave-3**. A pressure transducer measuring this "new" wave will register a pressure increase. It would be easy to assume that the higher pressure represented an increase in particle velocity.

This phenomenon makes the interpretation of pressure data difficult and easily misleading. Fortunately, Dynomation6 and DynoSim6 also calculate and display *particle-flow velocity and direction* in addition to pressures for each degree of crank rotation. With the display of port flow velocities, the simulation provides the engine builder a comprehensive picture of port and cylinder pressures, intake and exhaust particle flow velocities, and the direction of that flow.

INTAKE TUNING

The first part of effective intake tuning begins with good exhaust system tuning. As mentioned previously, when the exhaust valve opens it generates a powerful positive pressure wave that travels to the end of the header pipe and returns to the cylinder as a strong expansion wave. When the timing is right, this suction wave will arrive during valve overlap and draw out burnt gases while starting the inflow of fresh charge. This scavenging effect lowers cylinder pressures just before the exhaust valve closes, early in the intake stroke. After EVC, the piston begins to move rapidly down the bore and when it reaches maximum speed, between 73- and 82-degrees after TDC, the rapid increase in cylinder volume will "yank" down the already low pressure in the cylinder. On the pressure-crank angle diagram this will occur between 400- and 440-degrees and is usually the lowest point on the cylinder pressure trace. This sudden drop in pressure generates a strong suction wave that travels toward the open end of the intake runner where it is reflected as a positive compression wave. Depending on the length of the runner and the speed of the engine, this compres-

Wave-Dynamics Analysis

Reflected Waves In Intake Port

Figure 11

Intake Pressure Crank-Angle Graph
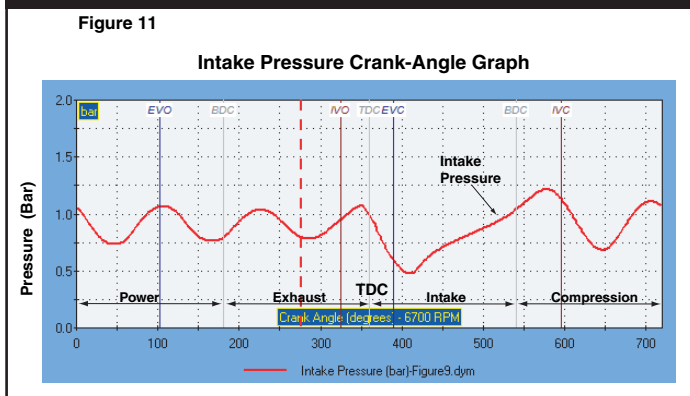
Typical intake pressures throughout the full 720 degrees of crankshaft rotation show how each successive wave reflection returns with slightly lower peak pressure. To gain the greatest benefit from the reflected wave at peak engine speed, it is often best to tune the induction system to utilize the *second or third* returned pulse.

sion wave will return to the cylinder just before the intake valve closes. The arriving intake compression wave provides two benefits: 1) As the wave moves through the port it pushes particles in the same direction, helping to fill the cylinder. When the compression wave reaches the intake valve just before IVC, 2) the positive pressure overcomes the buildup of pressure in the cylinder created by the piston moving up the bore on the early part of the compression stroke. This forces additional charge into the cylinder while delaying charge reversion until just before intake valve closes (an effect sometimes described as ram tuning). This complex series of tuning events adds the winning edge to many racing engines, but the exhaust system, induction system design, and all cam timing events must be synchronized to produce these effects.

When this critical synchronization is lost, intake charge will be driven back into the induction system before IVC. And reversion is a power killer. The simulation can accurately predict when this will occur in the port-flow velocity diagram (we'll provide specific examples of this later in the chapter).

After IVC the strong expansion wave created by the now completed intake stroke oscillates between the pipe end and the closed intake valve. It changes its sign from expansion to compression each time it encounters the open end of the runner. **Figure-11** shows the typical intake pressure trace throughout the 720 degrees of crankshaft rotation. Each successive reflection returns with slightly lower peak pressure (displayed on the pressure graph from zero to IVO). To gain the greatest benefit from the reflected wave at peak engine speed, it is often best to tune the induction system to utilize the *second or third* returned pulse, since these retain the highest energy levels (engine timing and the physical length of the passages make it difficult to utilize the first pulse).

At lower engine speeds there will be more time for pressure reflections to return from the induction system, so additional waves will appear in the pressure crank-angle diagram. It is also possible to increase the number of pulsations at any given rpm by decreasing the length of the inlet runner. Similarly, a longer runner will decrease the

number of pulsations since it takes more time for each pulse to return to the intake valve. Optimum runner length to return a second pulse at IVC can be "ballparked" with the following equation:

Second Pulse Length (inches) = 108,000 / RPM

Note: This is an approximation of pulse length, as it varies with charge density, pressure, cam timing, runner temperature, and other factors.

This formula works best for engines that operate at relatively high rpm. Unfortunately, at lower engine speeds this equation predicts a tuned length that is too long to be practical. Under these conditions, tuning to the third, fourth, or possibly even the fifth returned pulse requires a much shorter and more-practical length runner. The famous "Long Ram" of the early 1960s was designed by Chrysler to boost lower-speed torque on some of its big-block engines (used in vehicles like the 300G). It was based on second and third pulse tuning, requiring runners so long the carburetors wound up positioned over the exhaust manifolds of the adjacent bank of cylinders. The runner lengths could have been much shorter if the manifold was tuned to the third or fourth induction pulses. Here are good approximations for runner lengths tuned for the third, fourth, and fifth returned induction pulse:

Third Pulse Length (inches) = 97,000 / RPM
Fourth Pulse Length (inches) = 74,000 / RPM
Fifth Pulse Length (inches) = 54,000 / RPM

Engine designers and builders can use these equations to obtain good initial estimates for further "fine tuning" in the simulation.

Because of the nature of "pulse-tuning," an intake manifold goes in and out of

Manifold Designed Using Dynomation
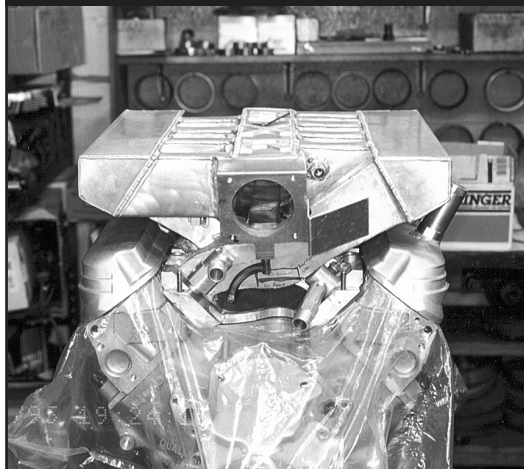
This custom intake manifold was designed with the help of Dynomation software. The manifold boosts low- and mid-range power on smallblock Ford engines. Dyno tests confirmed expected performance. Without simulation software, intake manifold design is strictly a trial-and-error process involving considerable time and money without any guarantee of success.

tune as the engine accelerates through the rpm range. The manifold transitions from fifth, to fourth, to third, and then, possibly, to second pulse tuning at maximum engine speed. As the engine passes through each of these tuning points, the runner phases in and out of sync with the expansion wave created when the piston reaches maximum speed on the intake stroke. At lower speeds, when the engine tunes on the fifth or higher pulse, little benefit is derived from the weak returning pulse. However, by the time the engine reaches sufficient speed so that the third or second pulse arrives just before IVC, a substantial pressure differential strongly assists cylinder filling and prevents charge reversion.

> **Note:** Since the second pulse delivers the strongest returning wave, the question arises: Why not use second-pulse tuning for all manifold designs? The answer is that the un-tuned gap between the second and third pulse is much greater than the gap between the third and fourth pulse. This means that engines using manifolds based on second-pulse tuning will have narrow, peaky, power bands, while third-pulse manifolds will tend to produce wider, although potentially lower, power curves.

Intake Runner Taper Angles

The taper angle of the inlet runner has a significant effect on the optimum tuned length. As runner taper angle increases, the returning pressure waves speed up, requiring a higher engine speed to regain the same level of "tune." For example, a 13-inch runner with a 4-degree taper angle will tune at approximately the same engine speed as a 10-inch straight runner.

Measuring Port/Runner Taper Angle

Port Entry Area

Port Taper Angle

Minimum Area
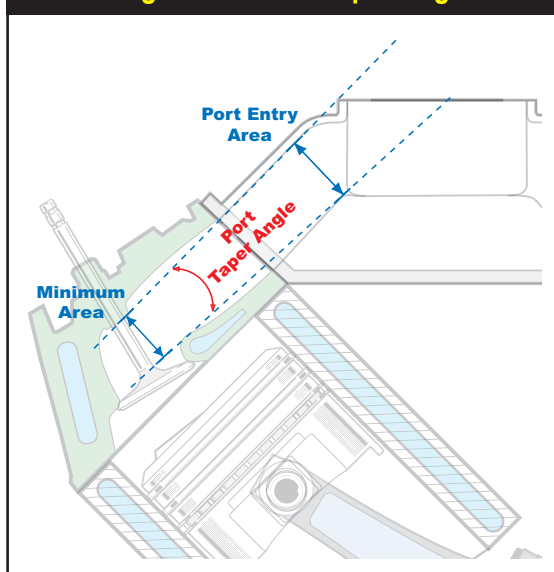
The taper angle of the inlet runner has a significant effect on the optimum tuned length and peak power. As runner taper angle increases, the returning pressure waves speeds up, so a higher engine speed is needed to regain the same level of "tune." There are several complex reasons why taper angles optimize power, but the most straightforward are: 1) A tapered port has a greater volume to "store" air/fuel charge and feed the cylinder during the main portion of intake stroke, and 2) A taper angle acts somewhat like a stepped-header for the induction system, generating a series of positive pressure waves that return to the valve and assist cylinder filling.

Wave-Dynamics Analysis

Effect Of Taper Angles

The taper angle of the inlet runner has a significant effect on the optimum tuned length. Inlet-runner taper angles can be varied in Dynomation6 not only to develop better-functioning induction systems, but also to gain a better understanding of how this important tuning element affects engine power. Here is a comparison between straight runners (darker curves) and 3-degree taper runners. Notice that exhaust velocity is higher while intake velocity is lower. With tapered runners the engine produces more horsepower at peak engine speed.
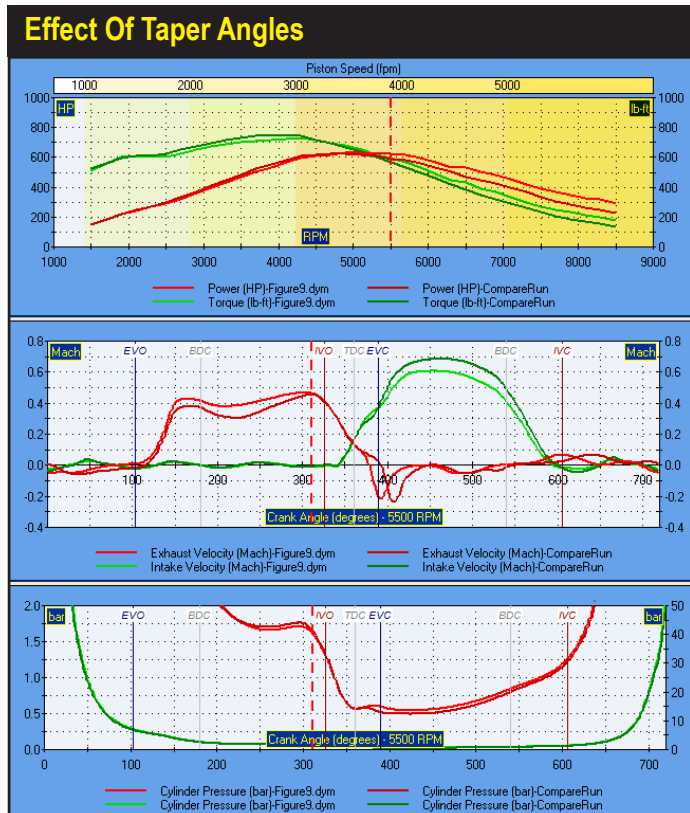
Straight runners often provide more power at lower engine speeds and work quite well for stock engines. However, to obtain peak performance from a high-speed racing engine, intake runners must incorporate a taper of 2 to 4 degrees (included). There are several complex reasons why taper angles optimize power, but the most straightforward are: 1) A tapered port has a greater volume to "store" air/fuel charge and feed the cylinder during the main portion of intake stroke, and 2) a taper angle acts somewhat like a stepped-header for the induction system, generating a series of positive pressure waves that return to the valve and assist cylinder filling.

Inlet-runner taper angles can be varied in Dynomation6 not only to develop better-functioning induction systems, but also to gain a better understanding of how this important tuning element affects engine power. A particularly revealing test can be done by overlaying port velocities and cylinder pressures for various inlet taper angles.

Port Flow Velocities

Another very important element of intake and exhaust tuning is port flow velocity. The speed at which gasses move through the ports essentially is controlled by port area and piston speed. For example, if intake port velocity is too low, gas particles flowing into the cylinder near IVC will not have sufficient momentum to overcome the

rise in cylinder pressure as the piston moves up the bore. Fresh charge will flow back out of the cylinder into the inlet tract. Furthermore, when the ports are too large, all pressure waves generated by piston motion and exhaust gas outflow are diminished. This will decrease the effectiveness of returning pressure waves during scavenging and cylinder filling. In fact, port size is so important that we can make this blanket statement: Developing an engine with optimum port sizes will potentially produce more power than the same effort put forth in determining optimum tuned runner lengths!

Research has shown that maximum port velocity through the minimum cross-sectional area of the runner should fall between 0.5 to 0.6 Mach, or about 600- to 720-feet-per-second. Port velocities higher than 0.6 Mach produce no further increases in volumetric efficiency. Since it is impossible for any port to generate optimum flow velocities at all engine speeds, it is essential that the engine builder accurately determine the required operating range for the engine and develop ports that flow appropriately at these speed ranges. If a broad power range is required, port areas should be sized so that maximum engine speeds produce port velocities of about 0.75 Mach. This will place optimum velocities (between 0.5 and 0.6 Mach) at lower engine speeds, optimizing volumetric efficiency vs rpm. If high-speed, maximum power is the goal, then port velocity at peak engine speed should be no higher than 0.6 Mach.

"In-Tune" Induction System

Figure 12

Intake Pressure Crank-Angle Graph
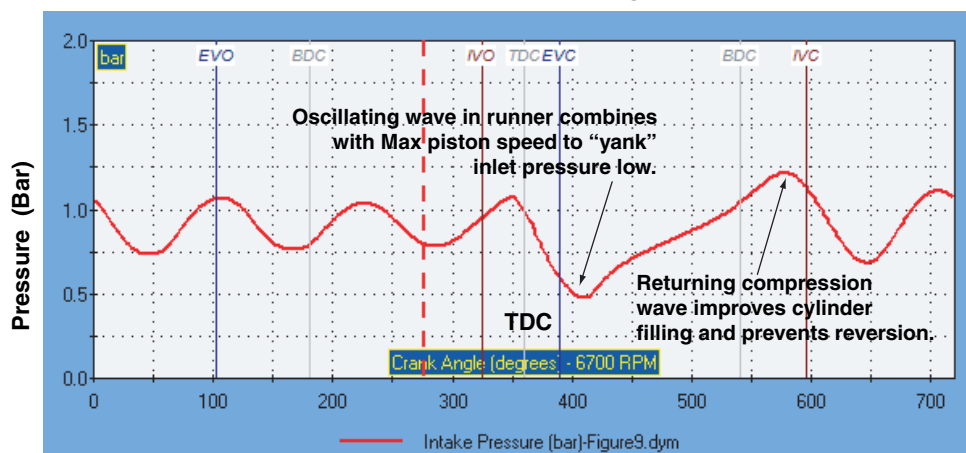
This is a good example of "in tune" induction pressures. The intake pressure wave is oscillating inside the runner and arrives at the intake valve during maximum piston speed (highest volume change) approximately 60- to 80-degrees after TDC. The "trough" of this wave combines with the suction created by the piston moving down the bore to "yank" the pressure wave to a very low amplitude. This low pressure generates a strong expansion wave that reflects back from the open-end runner as an equally strong compression wave that arrives at IVC. This not only drives additional charge into the cylinder, but reduces or eliminates reversion.

Wave-Dynamics Analysis

During your design sessions, here's some important concepts to keep in mind. It is often possible to make port areas small enough—and velocities high enough—to completely eliminate intake reversion at IVC. When this is accomplished, it often produces insufficient port volume and constricted flow. A better approach, producing higher peak power, is to allow some reversion to occur. It is also possible to design ports that flow between 0.5 or 0.6 Mach and do not show an increase in power. This situation can occur when intake valve or port restriction becomes too great. In these cases, the engine designer will simply have to experiment with several different port sizes, valve diameters, and valve lifts to find the optimum combination. The following formula can help you ballpark the required minimum port cross-sectional areas:

Minimum Area (Square Inches) = RPM x STROKE x BORE^2) / 190,000
This formula is an approximation that tends to provide conservative values. **(RPM**, **STROKE**, and **BORE** are in inches)

One reason this formula can only be an approximation of port area is that port velocity is also dependent on the valve-discharge coefficient. This value is the measured volume of flow across the valve divided by the perfect, theoretical (isentropic) flow. The discharge coefficient will always be less than one and is essentially a rating

Exhaust And Cam Timing In Sync

Figure 13
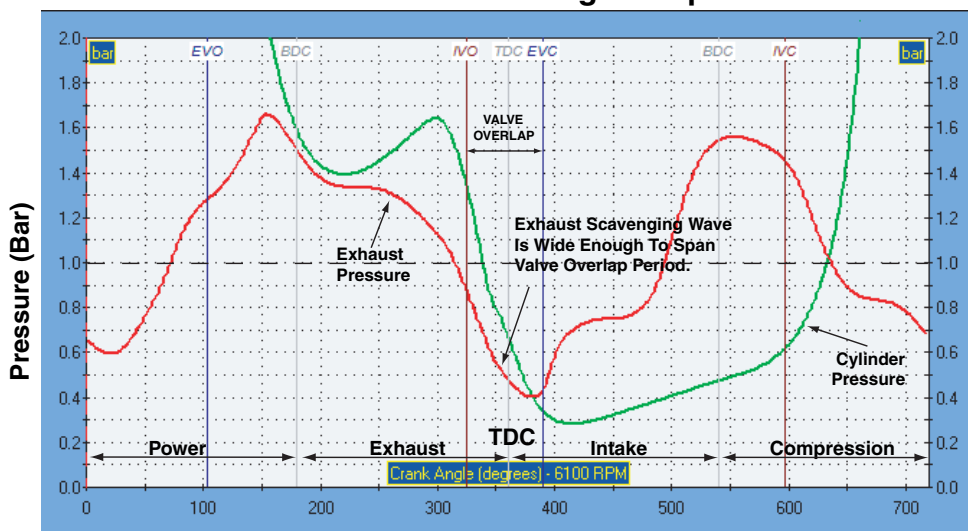
These engine pressures depict an engine with an exhaust system and cam timing that are "in-tune." The scavenging wave is wide enough to cover the overlap period, assisting exhaust gas outflow and preventing charge reversion.

Wave-Dynamics Analysis

for the efficiency of the valve. Specific discharge coefficient values are unique for each valve and cylinderhead design. The simulation allows the direct input of flow-bench data and will calculate the discharge coefficients for specific cylinder head combinations. Since about 30% of flow loss through the port occurs at the valve, an improved discharge coefficient can result in substantial performance gains.

EXHAUST TUNING

Incomplete removal of exhaust gases from the cylinder reduces engine performance in two ways. First, volumetric efficiency is decreased when residual gases occupy space within the cylinder that could have been filled with fresh charge. Second, higher cylinder pressures, resulting from incomplete exhaust blowdown, force residual burnt gasses into the inlet tract upon IVO. This "spoils" the fresh charge pulled back into the cylinder during the intake stroke.

As rpm increases, the pressure-crank angle diagram curves "stretch" out to the right, as shown by these exhaust traces. If the scavenging wave is wide enough and the overlap is short enough, the exhaust system will remain in tune throughout a wide rpm range. Good scavenging is still taking place at higher rpm in Figure-14a, but relatively little scavenging remains at even higher rpm in Figure-14b.

Exhaust System Scavenging Vs. Increasing RPM

Figure 14a — Pressure Crank-Angle Graph

Figure 14b — Pressure Crank-Angle Graph
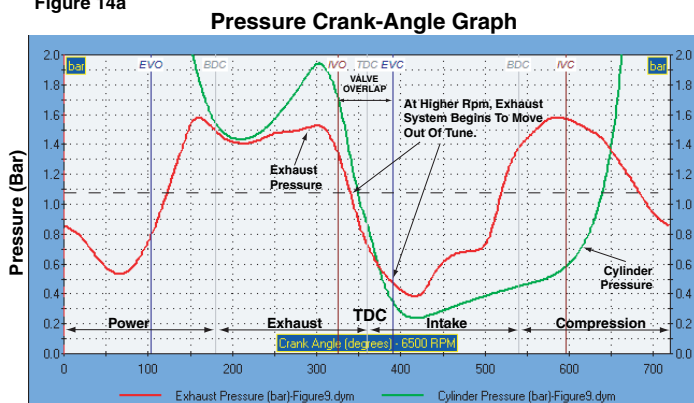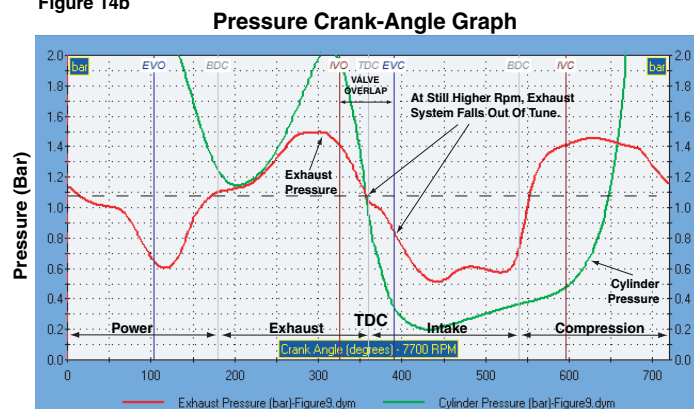
Wave-Dynamics Analysis

When the compression wave produced at exhaust valve opening (EVO) reaches the end of the header pipe (at the transition to the collector), a lower-strength expansion wave is reflected back to the cylinder. The positive compression wave, now of lower intensity, continues to move through the collector. When it reaches the atmosphere boundary, a second expansion wave is generated that moves back through the collector and into the primary header pipe. If the header system and cam timing are properly coordinated, the first expansion wave will return to the cylinder at, or slightly before, IVO. This will assist exhaust-gas scavenging and improve the flow of fresh charge from the induction system. Then, slightly later, the second expansion wave will arrive and continue the scavenging process until EVC. A well-designed collector can effectively <u>broaden</u> the scavenging wave to "cover" the entire overlap period. When this occurs, the engine and exhaust system "in tune." If a significant portion of the overlap period is not accompanied by a scavenging wave, the engine and exhaust system fall "out of tune."

Engine pressures illustrating an in-tune condition are pictured in **Figure-13**. As rpm increases, the pressure crank-angle diagram traces "stretch" to the right. If the scavenging wave is "wide enough" (i.e., of sufficient duration) and the overlap is short enough, the exhaust system will remain in tune even though engine rpm has increased. Notice that good scavenging is still taking place at higher rpm in **Figure-14a,** however, at an even higher rpm in **Figure-14b** the engine falls out of tune. A wide scavenging wave is particularly desirable in engines with long overlap periods. A scavenging wave that maintains a low pressure differential at the exhaust valve

Exhaust Header Lengths Vs. Induction Flow

Figure 15

Pressure Crank-Angle Graphs
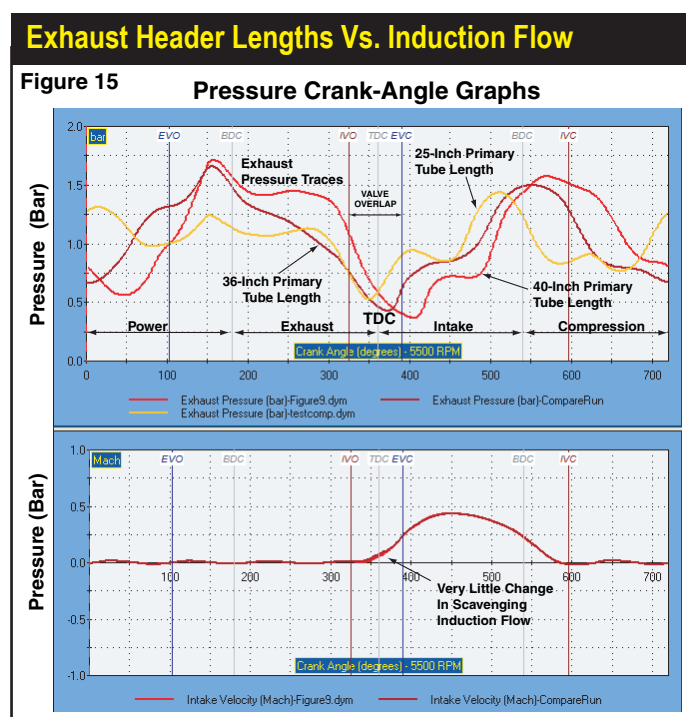
Exhaust tuning works by using the proper header pipe length to "time" the arrival of the suction wave to occur during the overlap. However, optimum primary pipe and collector lengths are not very critical for low- to medium-performance engines. This graph shows how induction flow changed as header length was varied from 25 inches to 40 inches (engine power changed by only about 2%). However, high-output, high-speed engines with long overlap periods require much more precise tubing lengths to optimize power. As power levels increase, the margin for error decreases.

throughout the entire overlap period will assist exhaust scavenging, prevent reversion, and ensure high power levels over a wider range of engine speeds.

This fact brings to light an important point about exhaust-header tuning. *Optimum primary pipe and collector lengths are not very critical for low- to medium-performance engines* (see **Figure-15**). In these applications, overlap periods are short and cylinder pressures are often not very high at the end of the exhaust stroke (lower performance engines produce a smaller volume of exhaust gasses and blowdown more effectively in shorter time periods). However, high-output, high-speed engines with long overlap periods require much more precise tubing lengths to optimize power. As power levels increase, the margin for error decreases.

An Interesting Note About Collectors: When exhaust pulses from multiple cylinders enter a collector, primary and reflected waves interfere and interact with one another. The influence of this interaction is sometimes overstated.

Peak exhaust velocity typically occurs <u>twice</u> during the exhaust cycle. The first peak-flow event, at approximately 60 degrees after EVO, is produced by maximum flow during <u>blow-down</u> (Figure-16a). The second event occurs at maximum piston speed on the exhaust stroke, about 73- to 82-degrees before TDC, and is the maximum flow during exhaust <u>pumping</u> (Figure-16b). If the flow at the second peak is the greater of the two, it usually indicates excessive restriction in the exhaust system; in this case, caused by engine overspeed (reducing the time for exhaust-gas blowdown) in a very-large displacement V8.

Peak Exhaust Flow Velocities Vs. Engine RPM
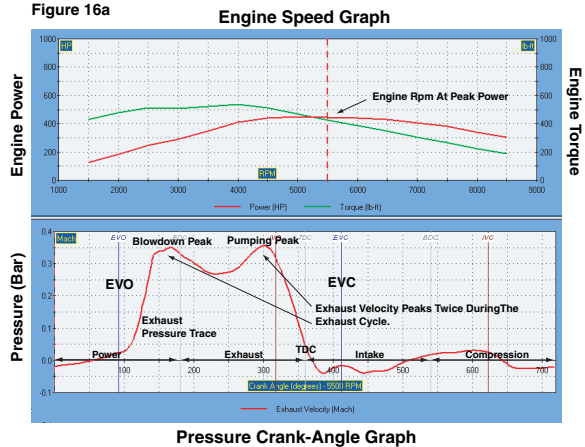
Figure 16a — Engine Speed Graph — Pressure Crank-Angle Graph

Figure 16b — Engine Speed Graph — Pressure Crank-Angle Graph
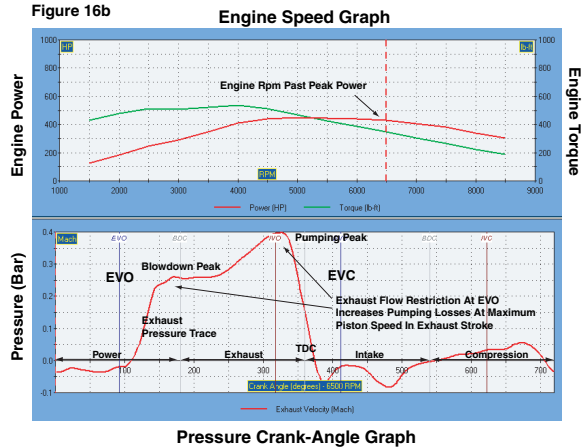
Wave-Dynamics Analysis

These "interference" waves are considerably lower in magnitude and have much less effect than the primary and first reflection waves from cylinder blowdown. Dr. Jon C. Morrison of the *Institute of Mechanical Engineers* conducted experiments on interference effects in collectors and concluded that multicylinder wave interference had an essentially insignificant affect on performance.

Exhaust Flow Velocities

Another important aspect of exhaust tuning is optimizing gas particle velocity throughout the exhaust system. When the exhaust valve opens (EVO), the piston is still moving downward on the power stroke. However, a large percentage of combusted gasses "blowdown" during the period between EVO and BDC. This occurs despite the fact that exhaust valve lift is relatively low. This phenomenon highlights the importance of low-lift flow characteristics of the exhaust valve and port for optimum performance. If the header system is too restrictive—possibly from insufficient pipe diameter—or if low-lift valve flow provides a restriction—often from improper port and valve seat/face contours—to allow sufficient blowdown, the piston will have to physically "pump" out the remaining gasses during the exhaust stroke. This substantially increases pumping work and cylinder pressures at IVO (which, in turn, reduce effective scavenging and create reversion into the induction system). On the other hand, when primary pipe diameters are too large, the returning scavenging wave is weakened and is too defuse to effectively assist exhaust gas outflow and initiate induction inflow during overlap. Optimum exhaust pipe diameters, like intake runner areas, are essential in producing flow velocities that achieve a balance between minimizing restriction and optimizing finite-amplitude-wave actions.

Peak exhaust velocity typically occurs _twice_ during the exhaust cycle. The first peak-flow event, at approximately 60 degrees after EVO, is produced by maximum flow during blowdown. The second event occurs at maximum piston speed on the exhaust stroke, about 73- to 82-degrees before TDC, and is the maximum flow during exhaust pumping. In most well-designed engines, the first peak will be the greater of the two (see **Figure-16a**); if the flow at the second peak is greater, it can indicate excessive restriction in the exhaust system (see **Figure-16b**). When the exhaust system is designed properly, the level of the first peak should fall between 0.45 to 0.5 Mach. If flow rates over 0.5 Mach are recorded, the exhaust system is too restrictive. If flow values fall below 0.45 Mach (as is the case in the two examples shown in Figures 16a & 16b), blowdown will not generate expansion waves strong enough to effectively scavenge the cylinder during overlap. Similar to intake tuning, it is possible to establish optimum flow velocities yet not measure a performance gain. When this occurs, the problem may be excessive exhaust-valve restriction. Fortunately, this can be easily detected by reviewing the pressure-crank angle diagram or the pressure-volume diagram (discussed later in this chapter).

Exhaust Tubing Lengths

Wave-Dynamics Analysis

While it is impossible to exactly predict primary lengths and diameters without using extremely complex formulas, it is helpful to the engine designer to have approximate starting points. The book *Performance Tuning In Theory and Practice* by A. Graham Bell, has excellent empirical formulas for header pipe length and diameters that provide reasonably accurate estimates for most applications. These formulas work well because they incorporate EVO timing within the calculation:

HPL (Header Length Inches) = ((850 x (360 - EVO)) / RPM) - 3
HPD (Header Diameter Inches) = ((SCID x 16.38) / ((HPL + 3) x 25)) x 2.1

Where **EVO** is *Exhaust Valve Opening Timing* measured in the 720-degree system. **SCID** is the displacement of a single cylinder in cubic inches. **HPL** and **HPD** are lengths and diameters in inches. Solve math within nested parentheses first.

These equations only give good length and diameter estimations when the ap-

Exhaust Collector Length Vs. Exhaust Scavenging Wave Shape

Figure 17

Pressure Crank-Angle Graph
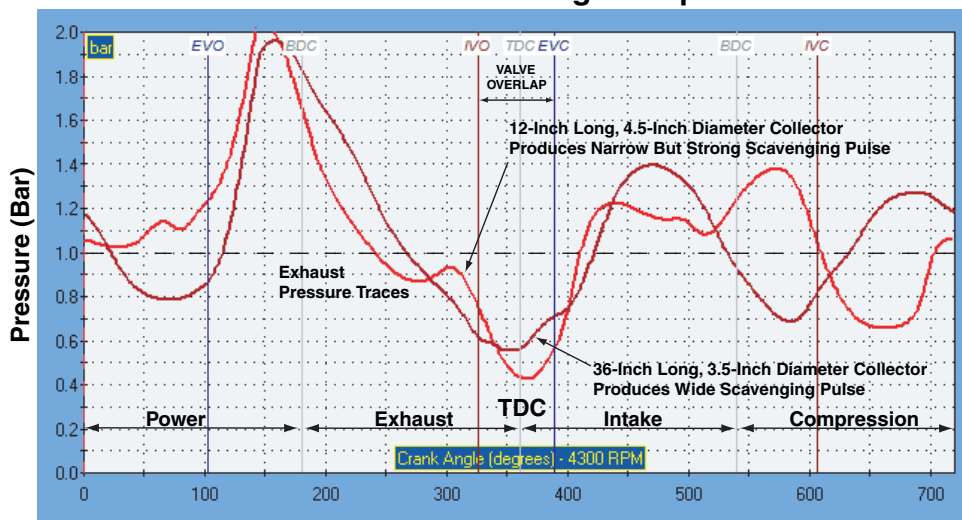
In this test, we changed the collector from 12-inch long, 4.5-inch diameter to 36-inch long, 3.75-inch diameter. Generally, increasing the collector length will broaden the scavenging wave. Smaller collector diameters place more tuning emphasis on the expansion wave generated at the open end as opposed to the header/collector boundary. The short-length, large-diameter collector (light-red line) produced the lowest and narrowest scavenging pulse. This design tends to develop more low- to mid-range power. The longer collector produces a wider scavenging wave of somewhat lower amplitude, a better overall choice for high engine speeds.

propriate value for rpm is used. If the engine has been designed for typical street-driving, the selected rpm should correspond with peak-torque speed. Racing engines should use either the engine speed at peak horsepower or some value in between.

While Bell's formulas derive reasonably accurate estimations of primary tubing lengths and diameters, unfortunately, there are no reliable empirical formulas for collector length and diameter. However, there are rules-of-thumb to keep in mind when choosing starting values for these dimensions. Longer collectors drive peak torque lower in the engine rpm range. This happens because, at higher engine speeds, expansion waves generated at the end of a long collector take too much time to travel back to the cylinder, failing to assist exhaust scavenging. Longer collectors should be used to widen powerbands at lower engine speeds, whereas, shorter collectors optimize peak power at higher engine speeds.

Choosing collector *diameters* requires further development of a previous theory. Recall that when a compression or expansion wave reached the open end of a pipe, a reflected wave was generated by this sudden change in area. What was not discussed, however, was that the amplitude of the reflected wave is dependent on the extent of area change. If a wave reaches an open-end pipe that discharges into the atmosphere (effectively, has an infinite change in area), then the reflected wave will be of maximum possible amplitude; nearly equal to the originating wave but of the opposite sign. If the area change is small, for instance, a 1.00-inch pipe joining a

Stepped Headers Can Lower Cylinder Pressures

Figure 18

Pressure Crank-Angle Graph
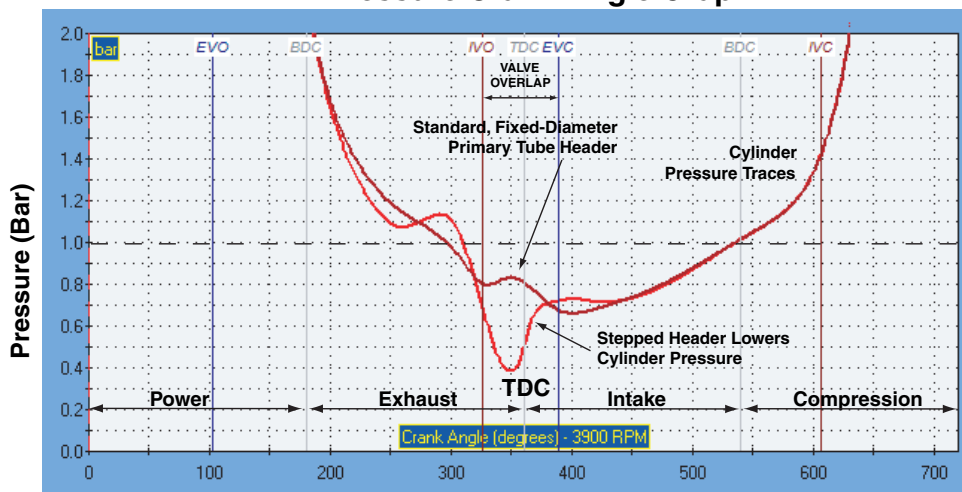
These cylinder pressure curves illustrate how stepped headers work. Stepped headers return small expansion waves at each step. These scavenging waves help exhaust gases flow out of the cylinder, particularly helpful on large-displacement, high-speed engines. Notice the lower cylinder pressures produced with stepped headers (lighter-red line). The lower pressures reduce pumping work.

1.05-inch pipe, a reflection wave will still be generated, but it will have a very small amplitude. The lesson to be learned from this for collector design is that *smaller collector diameters decrease the area change and reduce the amplitude of the first reflected wave*. As collector diameter is decreased, <u>higher amplitude</u> waves are generated at the <u>open-end</u> of the collector and <u>smaller reflected waves</u> are generated at the <u>primary/collector pipe boundary</u>. Since the open end is farther away from the cylinder, smaller collectors often boost low- to mid-rpm power. Larger diameter collectors generate stronger waves at the primary/collector boundary, and since this is closer to the cylinder, larger collectors generally boost high-rpm power (see **Figures 17** and **18** that illustrate some of these principals).

While there are no simple formulas to accurately calculate these variables, the following formula can provide "broad ballpark" figures for high-speed, racing engines:

Collector Diameter = 1.9 x HPD   **Collector Length = .5 x HPL**
   (**HPL** and **HPD** are in inches)

VALVE EVENTS AND TUNING STRATEGIES

The four basic valve-event timing points (IVO, IVC, EVO, EVC) can be grouped into three categories based on their influence on engine performance:

1) **EVC** and **IVO** are the least important individually, but the "angular" distance between these two events comprise the overlap period that has a significant effect on power.

2) **EVO** is the next most important timing point since it determines the beginning of the exhaust cycle which initiates cylinder blowdown.

3) **IVC** is the most critical event since it fixes the balance between cylinder filling and intake reversion, each having a potent effect on engine output.

Combining your knowledge of finite-amplitude waves with this information on valve timing will allow you to use the simulation to design or select camshafts (and choose related intake and exhaust components) for virtually any application.

Each valve-timing point is described below, including tips about event timing and how incorrect timing can affect wave-dynamic tuning. The reader is encouraged to use the simulation to recreate these phenomena and examine the pressure and velocity data to better understand how valve-event timing affects engine performance.

(IVO) Intake Valve Opening—This valve event marks the beginning of the intake process and valve overlap. IVO is probably the least sensitive timing point of all four valve events. However, when the exhaust system returns a broad, strong expansion wave, earlier IVO timing can boost performance. If a strong scavenging wave is not present, the same early IVO timing will cause exhaust gasses to flow into the induction system (reversion). This will substantially reduce power from not only fresh-charge "spoiling," but also charge heating. There are several additional conditions, along with IVO, that influence intake reversion. Low engine speeds usually produce low port velocities. Since induc-

Wave-Dynamics Analysis

Valve-Timing Events
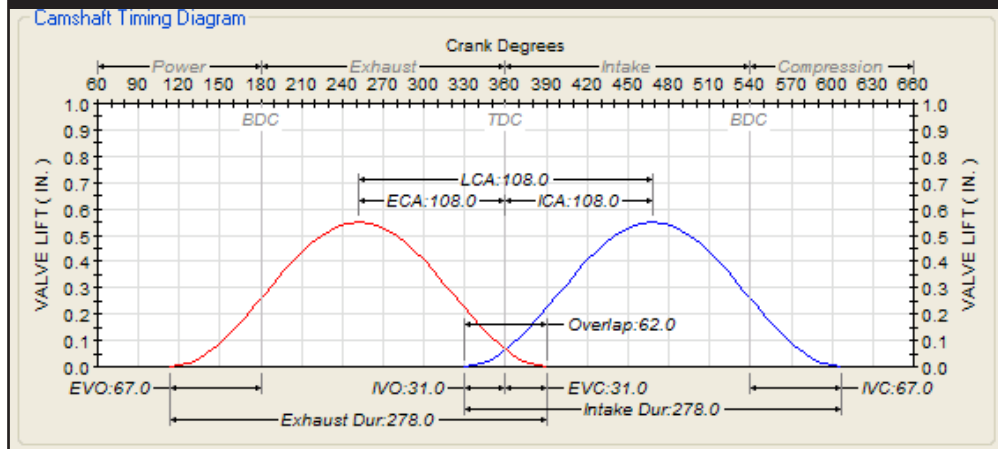
The four basic valve-event timing points (IVO, IVC, EVO, EVC) can be grouped into three categories based on their influence on engine performance: 1) EVC and IVO are the least important individually, but together comprise the overlap period that has a significant effect on power, 2) EVO is the next most important timing point since is determines the beginning of the exhaust cycle which initiates cylinder blowdown, and 3) IVC is the most critical event since it fixes the balance between cylinder filling and intake reversion, each having a potent effect on engine output.

tion gasses have less momentum when they are moving at slower speeds, it is easier for the piston—moving upward at IVO—to force residual exhaust gasses into the inlet track. Furthermore, low intake manifold pressures that occur during part-throttle operation or with restricted carburetion create large pressure differentials between the intake port and the cylinder, again contributing to reversion. A similar condition can occur at wide-open throttle when IVO is too early, as cylinder pressures are still quite high. Engines with these conditions might benefit from later IVO timing.

(EVC) Exhaust Valve Closing—This event marks not only the end of the exhaust process, but also the ending of the overlap period. At low rpm EVC strongly influences the degree to which exhaust gasses back-flow into the cylinder from the exhaust system. This low-speed back flow happens for two reasons. First, at EVC, the piston is beginning to move down the bore on the intake stroke, and since exhaust gas outflow velocity is low, the piston can begin to draw exhaust back into the cylinder. The second reason is more interesting. The negative scavenging wave created at the end of the header is always followed by another, <u>positive</u> compression wave. At low engine speeds both the scavenging and compression waves arrive prematurely. If the <u>compression</u> wave arrives during overlap, it forces exhaust gases *back into the cylinder*. One solution to this problem is the reverse-megaphone "collector"

Wave-Dynamics Analysis

often used on motorcycle exhaust systems. This device dampens the follow-up compression wave and reduces exhaust reversion at mid to low rpm. This discovery was made by accident (true for most discoveries!) when motorcycle racers tried using a reverse-megaphone in an effort to strengthen their flimsy exhaust systems.

At higher engine speeds, later EVC will shift the power curve towards higher rpm at the expense of low-speed torque. <u>At peak power the exhaust valve should close just slightly before exhaust back-flow occurs.</u> Don't forget that exhaust system tuning also has a significant influence on reversion. So, final EVC timing should only be determined after the exhaust system has been optimized.

(EVO) Exhaust Valve Opening—This timing point always occurs during the latter part of the power stroke and signals the start of the blowdown process.

Pressure-Crank-Angle Diagram

Figure 20
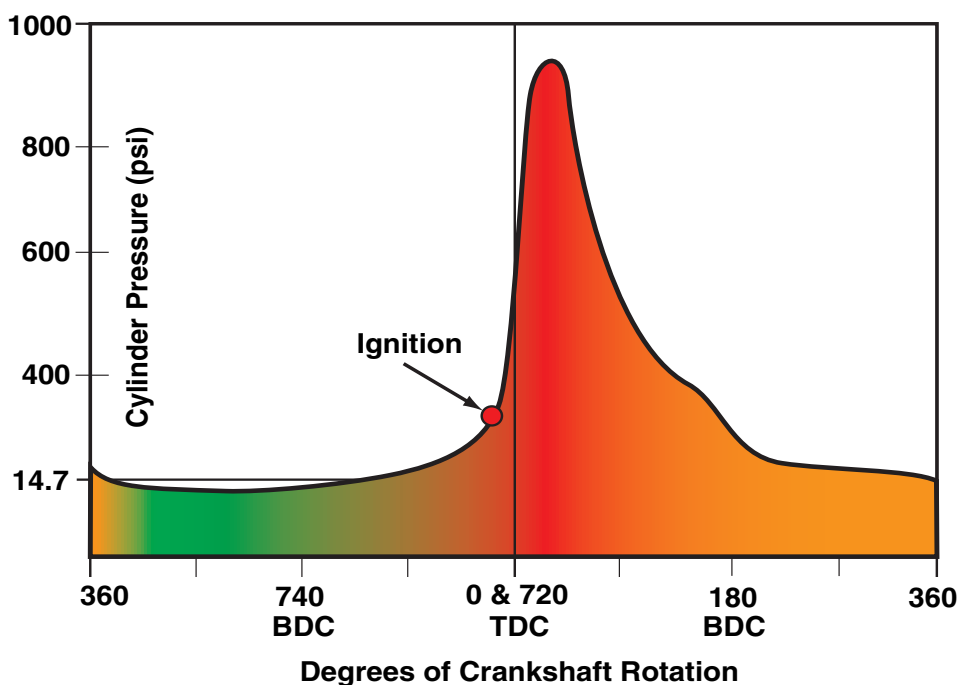
The *Pressure Crank-Angle* diagram is one of two common ways that cylinder pressures are illustrated. This method plots cylinder pressures against the crank position during the 720-degrees of the four engine cycles. On this graph (adjusted 360-degrees "forward" from the graph standard used in the simulation) it is easy to see widely varying cylinder pressures from induction through peak-ignition pressure.

Wave-Dynamics Analysis

Optimum EVO timing permits a reduction in cylinder pressure that would otherwise result in exhaust gas reversion into the induction system at IVO and during overlap. However, at higher engine speeds it is almost impossible for some reversion not to take place. <u>If there is no reversion flow during overlap at high speed, it is likely (but not certain) that EVO is too early</u>, and cylinder pressure that could have been used to push the piston during the power stroke is being prematurely expelled from the engine, reducing thermal efficiency and horsepower. <u>If EVO occurs too late, insufficient blowdown will increase pumping work during the exhaust stroke</u> and, in turn, reduce horsepower. Late EVO will also increase cylinder pressures at IVO contributing to reversion and additional losses in power.

> **Note:** Thermal efficiency research has shown that within the normal range of EVO timing, this event has less affect on low-speed power than either IVC or EVC. *Therefore, EVO timing should be early enough to satisfy high-speed performance requirements, but no earlier, or thermal efficiency will suffer.*

(IVC) Intake Valve Closing—This is the most important valve event in the 4-cycle, IC engine. IVC occurs well into the compression stroke and intake flow is easily susceptible to reversion if induction velocity and/or pressure-wave tuning do not provide sufficient energy to hold back increasing cylinder pressures. This is especially true at low to mid-range engine speeds where port velocity is still sluggish. Late IVC reduces volumetric efficiency at lower rpm and potentially increases volumetric efficiency at higher rpm. Optimum IVC at maximum engine speeds depends on port velocity, pressure wave tuning, and piston speed.

Using Pressure Diagrams

The simulation can display additional graphs that illustrate otherwise "hidden" engine operating conditions. One of these important graphs illustrate cylinder pressure as a function of crank angle throughout the 4-cycle process—called a *Pressure-Crank-Angle Diagram* shown in **Figure-20** (to recreate this graph, use either the center or bottom graph on the right program pane, redefine the **X** axis to *Crank Angle*, **Y1** to *Cylinder Pressure*, and **Y2** axis to *None*). Cylinder pressure can also be graphed as a function of cylinder volume—called a *Pressure-Volume*, or *PV Diagram* as shown in **Figure-21** (using either the center or bottom graph, redefine the **X** axis to *Cylinder Volume*, **Y1** to *Cylinder Pressure*, and **Y2** axis to *None*).

The Pressure-Crank-Angle Diagram

The *pressure crank-angle* diagram plots cylinder pressure against the position of the crankshaft during all four cycles (when this graph is reproduced, the zero point is fixed at TDC just after ignition on the power stroke). Ambient or atmospheric pressure forms the baseline for the vertical "Cylinder Pressure" axis, below which vacuum

Wave-Dynamics Analysis

Figure 21

Pressure-Volume Diagram

1 - 2 Intake Stroke
2 - 3 Compression Stroke
3 - 4 Ignition
4 - 6 Power Stroke
6 - 1 Exhaust Stroke

Pressure In Cylinder

TDC

BDC

V_2

V_1

Piston Displacement

$(V_1 - V_2)$

Clearance Volume

Piston

Cylinder

TDC

BDC
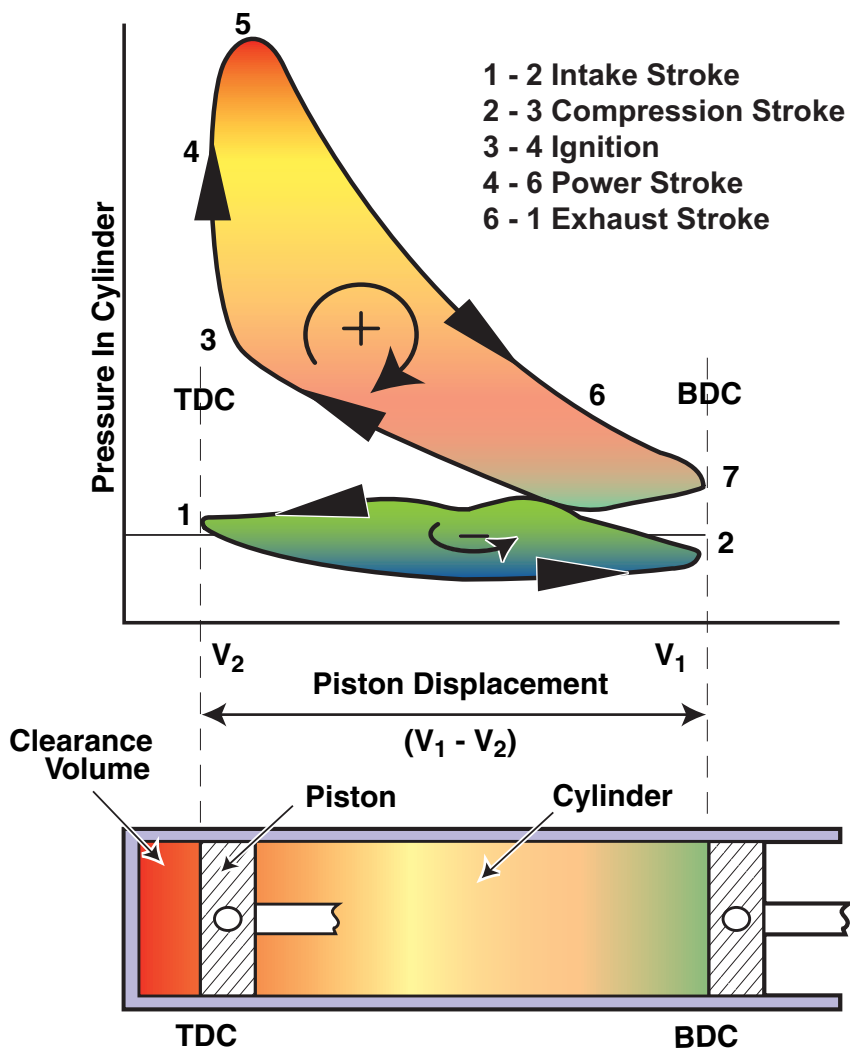
The PV diagram compares cylinder pressure to displaced volume and illustrates the same cylinder pressures displayed in the pressure crank-angle diagram. The PV diagram has the remarkable feature of isolating the *work consumed* from the *work developed* by the engine. The area within the lower loop, drawn in a counterclockwise direction, represents the work consumed by pumping losses (exhaust and intake stroke). The upper loop area, drawn in a clockwise direction, indicates the work produced by expanding gasses after combustion (compression and power stroke). The PV diagram can be a useful tool for optimizing valve-event timing, with the pumping loop providing the most helpful information.

levels are plotted. As the intake stroke begins, cylinder pressure decreases to near ambient as exhaust venting concludes. Pressures then change to a slight vacuum during the intake stroke (or a much stronger vacuum during part-throttle operation) until the cycle ends with the closing of the intake valve. Now the trapped charge is compressed and cylinder pressures rise to a few hundred pounds per square inch (psi). Typically, a few degrees before the piston once again reaches TDC, the spark plug fires, igniting the mixture and sending cylinder pressures up to several hundred or even thousands of psi within about 40 degrees of crank rotation. If the ignition point was timed properly, peak cylinder pressure will occur about 12 to 14 degrees after TDC. This timing allows cylinder pressures to be most efficiently converted to rotational force or torque by the rod/crank assembly. As the piston continues to move down the cylinder and pressures decrease, the exhaust valve opens, beginning a dramatic drop in pressure. Finally, cylinder pressures stabilize at a few psi during the exhaust stroke when remaining burnt gasses are forced out of the cylinder.

The Pressure-Volume Diagram

The pressure crank-angle diagram provides a straightforward view of the widely varying pressures in the cylinder. While it's easy to see "what's happening and where it's happening" with this diagram, another method of plotting the identical cylinder pressures offers a very different "look" and unique insight into the same data. This plot, known as a *pressure-volume* (*PV*, called a *Pee-Vee* or *indicator*) *diagram*, is very helpful in understanding the work produced-from and consumed-by the engine.

The PV diagram plots pressure against the displaced volume in the cylinder. Notice in the illustration (**Figure-21**) that the curve tends to wrap around and recon- nect with itself, forming a complete path. This curve represents cylinder pressures throughout the entire Otto-cycle process. It begins at the same point as the previ- ous crank-angle diagram—just after the intake valve opens—marked as a **1** on the graph. The cylinder and piston below the graph illustrate the position of the piston and the volume in the cylinder at each point on the PV. As the intake stroke continues, the piston moves down the bore and the plot indicates pressure below atmospheric continuing to point **2**. Here the piston begins to move back up the bore and pressure rises as the compression cycle begins. At point **3**, the fuel is ignited, rapidly increasing cylinder pressure. The piston reaches TDC at point **4**, and the power stroke begins. Cylinder pressure continues to rise to point **5**, ideally about 15 degrees after TDC. The exhaust valve opens at point **6**, just before BDC, assisting in cylinder depres- surization. The piston reaches BDC at point **7**. The exhaust cycle continues as the piston moves back up the bore maintaining some positive pressure in the cylinder until, at point **1,** the piston once again reaches TDC.

> **Note:** The lower "pumping" loops illustrated in the accompanying PV diagrams depict naturally-aspirated engines (it's called a "pumping" loop because the engine is either pumping in fresh charge or pumping out exhaust gasses; the average pressure of this loop is the pumping mean effective pressure—Pmep). *This pressure loop will look completely different for a supercharged engine*

Wave-Dynamics Analysis

PV Diagram Of Cylinder Misfire

Figure 21a   **Pressure-Volume Diagram Cylinder Misfire**

1 - 2 Intake Stroke
2 - 3 Compression Stroke
3 - 4 Misfire
4 - 6 "Power" Stroke
6 - 1 Exhaust Stroke
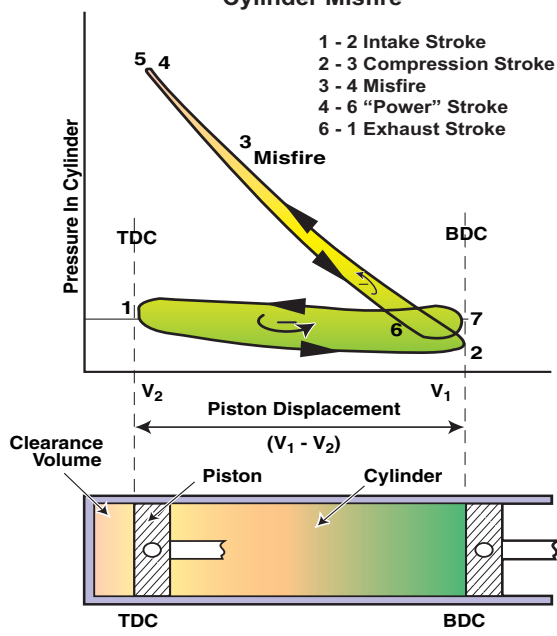
The work-isolating concepts of a PV diagram are evident with a cylinder misfire. In this case, the air/fuel mixture is drawn into the cylinder as usual, but it fails to ignite and boost cylinder pressure. The PV diagram of this event still consists of two loops; however, if we start at point *1* and trace through the sequence of pressures, we outline <u>two counterclockwise</u> loops. Both loops represent power consumed by the engine.

because positive (forced induction) pressures will act on the piston during the intake stroke.

Notice that the "pressure areas" have internal directional arrows that indicate the same sequence of events just described. Trace along the curve, again, from the starting point **1** around part of the "lower loop" marked with a minus sign. Notice that you are moving in a counterclockwise direction. As you move from point **2** through the junction between the upper and lower loops, continuing past points **3**, **4**, **5**, **6** and finally arrive at point **7**, this trip has taken you around the "upper loop," marked with a plus sign, in a clockwise direction. Finally, as you move past point **7** back to **1**, you trace out the upper half of the "lower loop," again in a counterclockwise direction. What does all the "clockwise stuff" have to do with anything? You may be amazed by the relationship. A PV diagram has the remarkable feature of isolating the *work consumed* from the *work developed* by the engine. The area within the lower loop, drawn in a *counterclockwise direction, represents the work consumed* by the engine "pumping" the charge into the cylinder and forcing the exhaust gasses from the cylinder. The upper loop area, drawn in a *clockwise direction, indicates the work produced* by the engine from pressures generated by expanding gasses after combustion.

Look again at the crank-angle diagram, **Figure-20**. Would you have thought the relatively "simple" curve in that illustration would contain so much more "hidden"

Wave-Dynamics Analysis

Pumping Loop Vs. RPM

Figure 22

Pressure-Volume Diagram
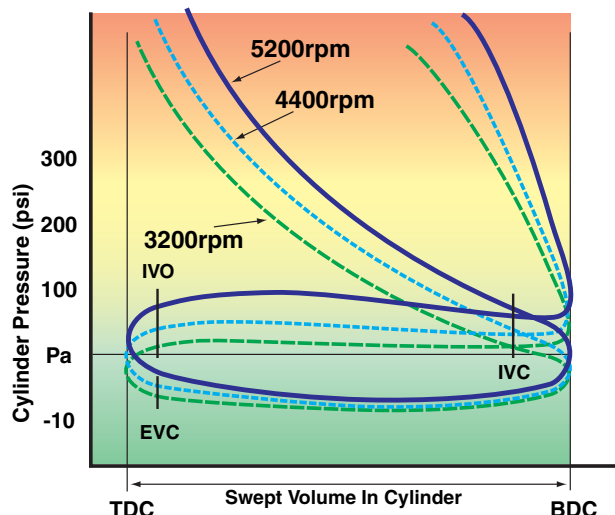
This PV diagram illustrates how the pumping loop changes as engine speed increases from 3200rpm to 5200rpm. Notice how the lower portion of the loop (the intake cycle) remains relatively constant, while the upper portion of the loop (the exhaust cycle) dramatically rises as rpm increases. The changes in exhaust pressure are caused by the decreasing _time_ available to blowdown the cylinder as engine speed increases. Higher pumping pressures increase pumping work (increasing Pmep) and that consumes horsepower.

information?

Let's test the work-isolating concepts of PV diagram with a cylinder misfire (see **Figure-21a**). In this case, the air/fuel mixture is drawn into the cylinder as usual, but it fails to ignite and boost cylinder pressure. The PV diagram of this event still consists of two loops (more or less). However, if we start at point **1** and trace through the sequence of pressures, we outline _two counterclockwise loops_. So both loops represent power _consumed_ by the engine. The upper loop encloses very little area since most of the power used to compress the charge is returned when the piston is forced back down the bore by the compression pressure on the "power" stroke.

> **Note:** The pressure in the cylinder as the piston moves down the bore on the "power" stroke of a misfire is slightly less than the measured pressure on the compression stroke because the charge has given up some heat, and therefore pressure, to the cylinderwall and other surfaces.

Optimizing Valve Events

The pressure-volume diagram can be a useful tool for locating optimum valve event timing, with the pumping loop providing the most insightful information. **Figure-22** shows a close-up view of a typical pumping loop. This period is also called the "valve-open cycle," since at least one of the valves is open throughout the entire loop. This illustration also shows how the pumping loop changes shape as engine speed increases from 3200rpm to 5200rpm. The lower, pumping loop obviously "balloons." However, notice that the lower portion of the lower loop (the intake cycle) remains

Wave-Dynamics Analysis

Early Exhaust And Intake Valve Closing

Figure 23

Pressure-Volume Diagram
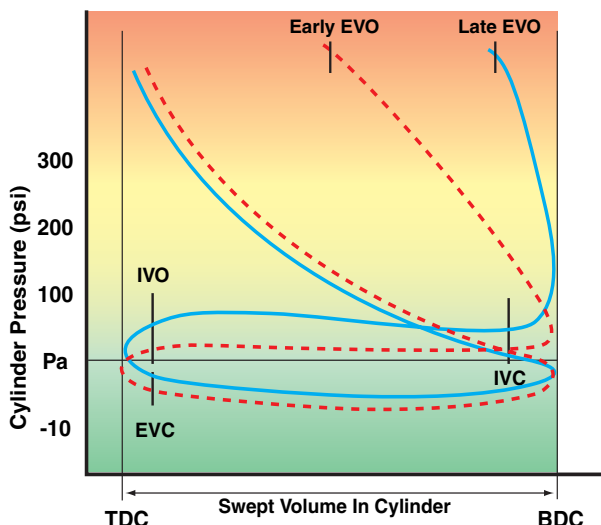
Here are the effects of early and late exhaust valve openings. Early EVO (dotted line) causes the pressure curve to slope strongly from left to right as cylinder pressure decreases during blowdown. If EVO is too late the pressure curve takes much more of a vertical "dive" and tends to level out considerably above Pa.

Effects Of Early And Late Valve Timing Events

Figure 24

Pressure-Volume Diagram
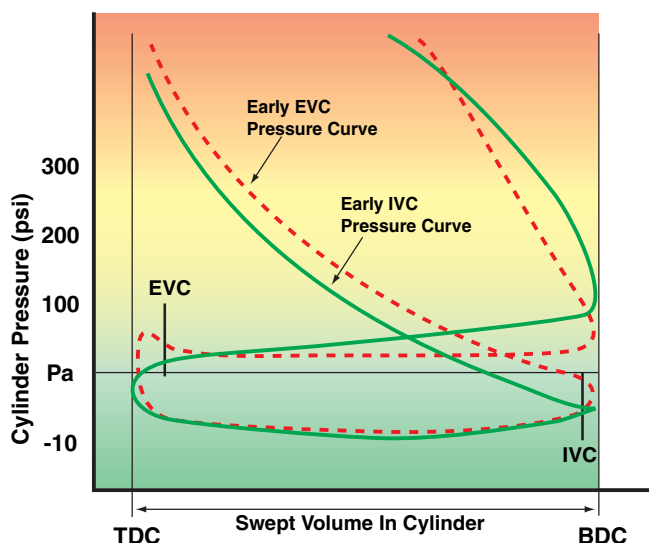
This PV plot shows the sharp rise in cylinder pressure from too early EVC timing (dotted line). The "spike" at the left end of the loop is caused by the upward moving piston beginning to re-compress exhaust gasses. Also illustrated is too early IVC timing (solid line). Notice the "pointed tail" at the lower right. This is produced when the intake valve closes near BDC and prevents the inflow of fresh charge normally caused by the ram tuning effects of the induction system.

relatively constant, while the upper portion of the loop (the exhaust cycle) dramatically rises as rpm increases. The increasing exhaust pressures are the result of the decrease in time available to blowdown the cylinder as engine speed increases. Higher pumping pressures increase pumping work (increasing Pmep) and that consumes horsepower.

The 5200rpm curve in **Figure-22** also shows the beginning of "bulge" forming around maximum piston speed on the exhaust stroke. This is a sign of exhaust flow restriction, sometimes caused by insufficient exhaust valve size and/or valve lift. This "restriction" bulge also would be visible on the pressure-crank angle diagram where cylinder pressure drops to parallel inlet pressure, at about 80 degrees before TDC.

Figure-23 show how early and late exhaust valve openings affect cylinder pressure. Notice the slope of the blowdown pressure at the upper right hand corner of the diagrams. If EVO is too early (dotted line), the pressure curve slopes strongly from left to right as cylinder pressure decreases during blowdown. If EVO is late, the pressure curve takes a much more vertical "dive" and tends to level out considerably higher above **Pa**. Ideal EVO timing will produce a pressure plot that will slope from left to right at a relatively steep angle (similar to the 4400rpm curve in **Figure 22**).

Figure-24 shows the sharp rise in cylinder pressure from very early EVC timing (dotted line). The "spike" at the left end of the loop is caused by the upward moving piston beginning to re-compress exhaust gasses trapped in the cylinder when the exhaust valve closed too early. This figure also illustrates how early IVC timing can change the lower part of the pumping loop (solid line). Notice the "pointed tail" at the lower right. This is produced when the intake valve closes near BDC and prevents the inflow of fresh charge normally developed by the ram tuning effects of the induction system.

> **One Final Note:** IVO is the least sensitive valve event, and its opening point (within the range of typical valve event timing) is often not distinguishable on the PV diagram.

Wave-Action Filling & Emptying Simulations 6

FAQ's

FREQUENTLY ASKED QUESTIONS

The following information may be helpful in answering questions and solving problems that you encounter when installing or using this simulation. If you don't find an answer to your problem here, send and email to: support@motionsoftware.com *(Motion Software provides Email technical service to registered users only).* We will review your problem and return an answer to you as soon as possible. *And make sure you have a look at our growing list of tech videos on YouTube: youtube.com/@motionsoftwaresim*

> **IMPORTANT:** Most Motion simulations include an Update tool that automatically checks for updates (plus, you can initiate a update check at any time by selecting *Check For New Version* from the **HELP** menu in the program). You'll not only receive the usual bug fixes and minor enhancements, but often updates include entirely new features and modeling capabilities.
>
> Many of these new features have been requested by our users (yes, we really do listen to our customers!). If you have ideas for enhancements or you have found issues with our software, please let us know. You'll probably see your requested feature "appear" in the next update!
>
> **Note:** Updating your Motion software will NOT erase or delete any of your engine files, cam files, or other user-created work!

GENERAL TROUBLESHOOTING

> Here are just a few of the things you might think about when trying to solve problems:

a) If the cam data that you are testing or reviewing in one of the graphs appears incorrect, make sure you are viewing the correct lobe (this is particularly important if you are using a *Variable-Valve-Timing* cam setup).

b) If a turbocharging system is not developing the boost or power you expect, remember that the *Boost Limit* is an <u>upper</u> limit. It does not force a too-large turbo to spin up more quickly and generate the desired boost. You need to select the correct size turbo, turbine, and A/R ratio to obtain the desired boost curve.

c) Take a few minutes to <u>*carefully*</u> review all your component selections. You may find that reviewing your engine component selections directs you to the source of

the problem. It's easy to look at a component specifications a hundred times and see the "right" number, only to look again and find that an error that was in "plain sight."

d) If you are certain that you have found an outright "bug," we are anxious to hear from you. It is our firm intention to make sure this engine simulation stays one of the most sophisticated, stable, and useful development tools available to the engine builder and enthusiast. Use the email address *support@motionsoftware.com* or *latherton@motionsoftware.com* to contact our development team. Always include your email address and attach any .dxml engine files (or any other related cam or flow files) that may us help diagnose the problem. Include a thorough explanation of the issues. We will make every attempt to review and reply to your problem as soon as possible.

INSTALLATION/BASIC-OPERATION QUESTIONS

Question: Received an "Error Reading Drive D" (or another drive) message when attempting to install the software from a CD. What does this mean?

Answer: This means your computer cannot read the disk in your CD-ROM drive. The disk may not be properly seated in your drive, the drive may be defective, or the disk may be damaged. If you can properly read other CDs in your CD-ROM drive, but our software distribution disk produces error messages, try requesting a directory of a known-good disk. If these operations produce an error message only when using the Motion CD, the disk is probably defective. Contact Motion Software, Inc., for a replacement (email address on the back cover of this manual). Replacement CDs or USB Security Keys are free for as long as you own our software (you must be the original purchaser). Just return the defective merchandise to Motion Software, Inc., and we will replace it at no charge. If you have lost the CD or the USB Key, please call for replacement pricing.

Question: "Could not locate HASP USB Security Key" error message was displayed when trying to run Dynomation6.

Answer: If Dynomation6 displays an error message indicating that the HASP Security Key is missing, here are some quick steps you can follow to isolate and correct common problems:

a) Make sure the Security Key is, in fact, properly connected to a functioning USB port on your computer or to a USB hub that is connected to your computer. If you connected the Key to a hub, try connecting it directly to a USB port on your computer system. The Security Key contains a small LED that lights up when it is properly connected and after it has located the correct software drivers on your system (these drivers are normally installed during Dynomation6 installation, see step "**c**" below).

b) Make sure your USB port is functioning correctly by disconnecting all other USB devices from your system. Then connect the Security Key to another USB port. If that works, try another USB device in the port that did not work to

determine if that port is defective.

c) Try re-installing or updating the Security Key drivers. Updates are posted on the Support page of the Motion Software, Inc., website (www.motionsoftware.com). Restart your computer after you install the new drivers.

And here's some additional tips for both **Dynomation6** and **DynoSim6** users:

d) Is your computer experiencing technical difficulties, such as non-functional devices, spontaneous rebooting, etc? If so, the device drivers for our Security Key may not properly install on your system. You should have a "stable" computer system and a "clean," virus-free Windows installation to properly use this software. You also should have the latest Windows updates installed on your system (available free from www.microsoft.com).

e) Try installing this software (and for Dynomation6, the Security Key) on another computer system to determine if your computer is at fault.

f) Make sure you are using the latest version of this simulation by opening the Windows **Start** menu, then choose *Motion-Dynomation6 Engine Sim (or Motion-DynoSim6 Sim)*, and finally click on the *MotionUpdater* icon in that folder. Your computer must be connected to the Internet to obtain the latest updates.

g) Refer to additional information on the Support page of the **www.MotionSoftware.com** website.

Question: The software won't start. It displays messages saying something like, "msvcrt.dll linked to missing export..."

***Answer*:** You need to update your version of Windows by using the Windows Update service built into Windows, or directly go to www.microsoft.com to obtain Windows updates. Consider updating from older Windows versions to Windows10 or Windows11.

Question: The simulation produced an *Assertion Failure* error. What should I do?

***Answer*:** Please note down all of the information presented in the error-message box, provide a quick synopsis of what lead up to the error, then send this information, along with the *Dynomation6-Run.log* or *DynoSim6_Run.log* data file (located in the C:/Dynomation6 or C:/DynoSim6 director) to Motion Software at support@motionsoftware.com. Thank you for your assistance in helping us improve this software. If you cannot send this email, contact support@motionsoftware.com for assistance.

SCREEN DISPLAY QUESTIONS

Question: Even though I have a 21-inch monitor, I can only see a portion of the simulation screen on my monitor. What can I do so that I don't have to scroll both horizontally and vertically?

Troubleshooting & Common Questions

Answer: The *screen resolution* of your monitor (not its size) determines how much of the simulation screen you can see without scrolling left and right. You can change *screen resolution* by **RIGHT CLICKING** on your desktop, then selecting **PROPERTIES** from the drop-down menu. Choose the **SETTINGS** tab and increase screen resolution by moving the **Screen Area** slider to the right. For more information about screen resolution, refer to the documentation that was supplied with your computer, your video graphics card, or with Windows.

BORE/STROKE/SHORTBLOCK QUESTIONS

Question: I cannot find the shortblock I want to model in the Shortblock menus?

Answer: The Shortblock menu is simply a list of most of the common engines. If you cannot find your shortblock in the menus, simply enter the bore, stroke and number of cylinders directly in the Shortblock Component Category. Send us an email and let us know what engines you would like to see in the menu (latherton@motionsoftware.com).

INDUCTION/MANIFOLD/FUELS QUESTIONS

Question: When I choose induction flow that is too large for an engine (for example 4000cfm on a Honda 4-cylinder), why does the power increase without showing "bog," or power drop at low speeds?

Answer: The simulation assumes an optimum air/fuel ratio regardless of the selected CFM rating. As a result, the program produces increasingly smaller positive results from larger-and-larger induction flows. However, you won't go wrong if you use common sense when selecting induction/carburetor flow capacities.

Question: The engine I am building uses two carburetors. How can I simulate the airflow?

Answer: You can simulate induction airflow from less than 100 to 7000cfm, rated at either the standard pressure drop of 1.5-inches or 3.0-inches of mercury (a pressure drop of 1 inch of mercury is equivalent to 13.55-inches of water). To simulate two carburetors, simply add the airflow and enter the total cfm value into the Induction category (or you can use the *Flow Calculator* if you are unsure about the rated flow of a venturi or throttle).

CAMSHAFT/VALVETRAIN QUESTIONS

Question: I built a relatively stock engine but installed a drag-race camshaft. The engine only produced 9hp @ 2000 rpm. Is this correct?

Answer: Yes. Very low power outputs at low engine speeds occur when racing camshafts are used without complementary components, such as high-flow cylinder heads, high compression ratios, and exhaust system components that match the performance potential of the cam.

Troubleshooting & Common Questions

Question: The simulation doesn't predict power unless I enter both seat-to-seat and 0.050-inch valve timing. Why is this necessary?

***Answer*:**   The simulation can use either Cam Profile data or 10-Point Timing (specs found on your cam card and in cam manufacturer's catalog/website). To optimize accuracy with 10-Point timing, it is essential to provide both seat-to-seat <u>and</u> 0.050-inch timing points. With both sets of timing points, the simulation can calculate lifter acceleration rates much more accurately.

Question: How does the simulation allow for the different acceleration rate cams used with hydraulic, solid, and roller lifters?

***Answer*:**   The simulation calculates a valve acceleration rate and a valve-motion curve using both the seat-to-seat and 0.050-inch cam timing specifications (see previous question). Note that the acceleration rate of cams is not directly linked to the type of lifters (e.g., mild street cams often used roller lifters), however, determining the acceleration rates mathematically from seat-to-seat and 0.050-inch timing points produces accurate valve-motion curves and acceleration rates. *Check out our cam tech videos on YouTube: youtube.com/@motionsoftwaresim*

Question: Can I change rockerarm ratios?

***Answer*:**   Yes. You can directly change the rocker ratios in the Camshaft Component category. You can also use the Rocker Math calculator to get insight into how rocker-ratio changes affect valve timing and lift.

Question: I found the published factory seat-to-seat valve timing for an engine that I am building. The IVC occurs at 112 degrees (ABDC). Something goes wrong when I enter the valve events into the simulation.

***Answer*:**   There are so many ways that cam specs can be described for cataloging purposes that it can be quite confusing (check out our helpful YouTube videos at motionsoftwaresim). The cam specs for your engine are a classic example of this lack of standards. The OEM cam listed in the factory manual is a hydraulic grind with seat-to-seat timing measured at *0.001-inch lifter rise*. Because the cam is designed for long life and quite operation, it has extended (quiet) opening ramps. This is the reason for the large number of crank degrees between the specified "opening" and "closing" points. In fact, during the first 35 degrees of crank rotation, the lifter rises less than 0.010-inch. If this wasn't the case, and the valve truly *opened* and *closed* at the factory specified timing points, the cam would have over 350-degrees duration, and it's unlikely the engine would even start! Simulation seat-to-seat timing should be based on the SAE standard of 0.006-inch <u>valve</u> rise. The 0.001-inch lifter-rise figures published in your factory manual do not indicate when mass flow begins in the ports and are useless for engine simulation purposes.

Troubleshooting & Common Questions

Question: My cam manufacturer's catalog does not list seat-to-seat, valve-event timing. But it does list seat-to-seat intake and exhaust duration, lobe-center angle, and intake centerline. Can I calculate the valve-event timing from these figures?

Answer: Yes. Use the *CamMath QuickCalculator™* built into the simulation to calculate the intake and exhaust opening and closing points. You'll need the following information:

1) **Intake Duration**
2) **Exhaust Duration**
3) **Lobe-Center Angle** (sometimes called lobe separation angle).
4) And the **Intake Centerline Angle**.

Question: I have been attempting to test camshafts from data obtained on the Internet. I can find the duration and lobe center angle. However, I can't find the seat-to-seat timing (they act like it's a trade secret!). Can I use the available data to run a simulation?

Answer: No. Our suggestion is to contact another cam manufacturer.

COMPRESSION-RATIO QUESTIONS

Question: The simulation calculated the total Combustion Volume at 92ccs. But I know my cylinder heads have only 75cc. What's wrong?

Answer: This confusion comes from assuming that the calculated **Total Combustion Volume** is the same as your measured combustion-chamber volume. The *Total Combustion Volume* is the entire volume that remains in the cylinder when the piston reaches top dead center.

Question: When using the Compression Ratio Calculator, the "Measured Volumes, Burette Method" mode Item-2 should, but does not, allow a zero entry. I have flattop pistons with zero deck clearance and 5cc valve pockets. Wouldn't this be the correct entry since I am running zero deck clearance?

Answer: The assumption in the "Measured Volumes, Burette Method" option is that there is a volume (the combination of the displacements in the domes/dishes/pockets) that cannot be easily calculated by the engine builder. The only practical way to measure this volume is to move the piston down the bore an arbitrary amount, say 0.250 (sufficient to ensure that the entire piston dome is below the deck surface), and measure the volume in the cylinder (with a liquid and a burette). This is then compared to the volume of a cylinder 0.250 inches long with the same bore diameter. The difference in volume is the volume taken up by the dome/dish/pockets.

However, on your engine, you know that the flattop pistons with valve pockets that you have will produce a zero deck height at TDC, and the displacement of the valve pockets is 5cc. Knowing this, you can select the "Known Volumes" model, set the deck clearance to zero, add 5cc for valve-pocket volume, enter

the combustion chamber volume, and the compression ratio will be calculated.

GENERAL QUESTIONS ABOUT RUNNING A SIMULATION

Question: The simulation displayed an error message "Program was unable to complete the simulation. A more balanced combination of components..." What went wrong?

***Answer*:** The combination of components you have selected produced a calculation error in the simulation process. This can be caused by using restrictive induction flow on large-displacement engines, using radical cam timing on otherwise mild engines, or large taper angles on intake and exhaust passages. Try increasing induction flow, selecting a cam with less duration, reducing the compression ratio, or try changing the *Wave-Action Simulation Mode* from **Normal** to **Fine** or **Optimized** (this decreases the Mesh size used to divide engine passages for analysis and provides a more effective damping of shock waves).

Question: When I run a simulation, part of the horsepower and torque graph goes off the end of the graph. What can I do to correct the display?

Answer*:** Open the **Graph Options** menu (right-click on the graph) and select ***Auto Range for the **Y1** or **Y2** variable.

Troubleshooting & Common Questions

MINI GLOSSARY

0.050-Inch Cam Timing Method—See **Cam Timing**, @ 0.050-inch.

ABDC or After Bottom Dead Center—Any position of the piston in the cylinder bore after its lowest point in the stroke (BDC). ABDC is measured in degrees of crankshaft rotation after BDC. For example, the point at which the intake valve closes (IVC) may be indicated as 60-degrees ABDC. In other words, the intake valve would close 60 degrees after the beginning of the compression stroke (the compression stroke begins at BDC).

Air-Fuel Ratio—The proportion of air to fuel: by weight: that is produced by the carburetor or injector.

ATDC or After Top Dead Center—Any position of the piston in the cylinder bore after its highest point in the stroke (TDC). ATDC is measured in degrees of crankshaft rotation after TDC. For example, the point at which the exhaust valve closes (EVC) may be indicated as 30-degrees ATDC. In other words, the exhaust valve would close 30 degrees after the beginning of the intake stroke (the intake stroke begins at TDC).

Atmospheric Pressure—The pressure created by the weight of the gases in the atmosphere. Measured at sea level this pressure is about 14.69psi.
Back Pressure: A pressure developed when a moving liquid or gaseous mass passes through a restriction. "Backpressure" often refers to the pressure generated within the exhaust system from internal restrictions from tubing and tubing bends, mufflers, catalytic converters, tailpipes, or even turbochargers.

BBDC or Before Bottom Dead Center—Any position of the piston in the cylinder bore before its lowest point in the stroke (BDC). BBDC is measured in degrees of crankshaft rotation before BDC. For example, the point at which the exhaust valve opens (EVO) may be indicated as 60-degrees BBDC. In other words, the exhaust valve would open 60 degrees before the exhaust stroke begins (the exhaust stroke begins at BDC).

Big-Block—A generic term that usually refers to a V8 engine with a displacement that is large enough to require a physically "bigger" engine block. Typical bigblock

engines displace over 400 cubic inches.

Blowdown or Cylinder Blowdown—Blowdown occurs during the period between exhaust valve opening and BDC. It is the period (measured in crank degrees) during which residual exhaust gases are expelled from the engine before the exhaust stroke begins. Residual gasses not discharged during blowdown must be physically "pumped" out of the cylinder during the exhaust stroke, lowering power output from consumed "pumping work."

Bore or Cylinder Bore—The internal surface of a cylindrical volume used to retain and seal a moving piston and ring assembly. "Bore" is commonly used to refer to the cylinder bore diameter, unusually measured in inches or millimeters. Bore surfaces are machined or ground precisely to afford an optimum ring seal and minimum friction with the moving piston and rings.

Brake Horsepower (bhp)—Brake horsepower (sometimes referred to as shaft horsepower) is always measured at the flywheel or crankshaft by a "brake" or absorbing unit. Gross brake horsepower describes the power output of an engine in stripped-down, "race-ready" trim. Net brake horsepower measures the power at the flywheel when the engine is tested with all standard accessories attached and functioning. Also see Horsepower, Indicated Horsepower, Friction Horsepower, and Torque.

Brake Mean Effective Pressure (bmep)—A theoretical average pressure that would have to be present in each cylinder during the power stroke to reproduce the force on the crankshaft measured by the absorber (brake) on a dynamometer. The bmep present during the power stroke would produce the same power generated by the varying pressures in the cylinder throughout the entire four-cycle process.

BTDC or Before Top Dead Center—Any position of the piston in the cylinder bore before its highest point in the stroke (TDC). BTDC is measured in degrees of crankshaft rotation before TDC. For example, the point at which the intake valve opens (IVO) may be indicated as 30-degrees BTDC. In other words, the intake valve would open 30 degrees before the intake stroke begins (the intake stroke begins at TDC).

Cam Timing @ 0.050-Lift—This method of determining camshaft valve timing is based on 0.050 inches of tappet rise to pinpoint timing events. The 0.050-inch method was developed to help engine builders accurately install camshafts. Lifter rise is quite rapid at 0.050-inch lift, allowing the cam to be precisely indexed to the crankshaft. Camshaft timing events are always measured in crankshaft degrees, relative to TDC or BDC.

Cam Timing @ Seat-To-Seat—This method of determining camshaft timing uses a specific valve lift (determined by the cam manufacturer) to define the beginning or ending of valve events. There is no universally accepted valve lift used to define seat-to-seat cam timing, however, the Society of Automotive Engineers (SAE) has

accepted 0.006-inch <u>valve</u> lift as its standard definition. Camshaft timing events are always measured in crankshaft degrees, relative to TDC or BDC.

Camshaft Advance/Retard—This refers to the amount of advance or retard from the manufacturers recommended timing that the cam is installed in the engine. Focusing on intake timing, advancing the cam closes the intake valve earlier. This setting typically increases low-end performance. Retarded cam timing closes the intake valve later which tends to help top end performance.

Camshaft Lift—The maximum height of the cam lobe above the base-circle diameter. A higher lobe opens the valves further, often improving engine performance. Lobe lift must be multiplied by the rocker ratio (for engines using rocker arms) to obtain total valve lift. Lifting the valve more than 1/3 the head diameter generally yields little additional performance. Faster valve opening rates add stress and increase valvetrain wear but can improve performance. High lift rates usually require specially designed, high-strength components.

Centerline—An imaginary line running through the center of a part along its axis, e.g., the centerline of a crankshaft running from front-to-back directly through the center of the main-bearing journals.

Duration or **Valve Duration**—The number of crankshaft degrees (or much more rarely, camshaft degrees) of rotation through which the valve lifter or cam follower is raised above a specified height; either <u>seat-to-seat valve duration</u> measured at 0.006-, 0.010-inch or other valve lifts (even 0.020-inch lifter rise), or duration measured at 0.050-inch lifter rise, called <u>0.050-inch duration</u>. Intake duration is a measure of all intake lobes, and exhaust duration indicates the exhaust timing for all exhaust lobes. Longer cam durations hold the valves open longer, often allowing increased cylinder filling or scavenging at higher engine speeds.

Exhaust Center-Angle/Centerline or **ECA—**The distance in crank degrees from the point of maximum exhaust valve lift (on symmetric cam profiles) to TDC during the valve overlap period.

Exhaust Valve Closing or **EVC**—The point at which the exhaust valve returns to its seat, or closes. This valve timing point usually occurs early in the intake stroke. Although EVC does not have substantial effects on engine performance, it contributes to valve overlap (the termination point of overlap) that can have a significant effect on engine output.

Exhaust Valve Opening or **EVO**—The point at which the exhaust valve lifts off of its seat, or opens. This valve timing point usually occurs late in the power stroke. EVO usually precedes BDC on the power stroke to assist exhaust-gas *blowdown*. The EVO timing point can be considered the second most important cam timing event

from a performance standpoint.

Filling & Emptying Simulation—This engine simulation technique includes multiple models (e.g., thermodynamic, kinetic, etc.), and by dividing the intake and exhaust passages into a finite series of sections it describes mass flow into and out of each section at each degree of crank rotation. The Filling And Emptying method can accurately predict average pressures within sections of the intake and exhaust system and dynamically determine VE and engine power. However, the basic Filling And Emptying model can not account for variations in pressure *within* individual sections due to gas dynamic effects.

Four-Cycle Engine—Originally devised by Nikolaus Otto in 1876, the four-cycle engine consists of a piston moving in a closed cylinder with two valves (one for inlet and one for outlet) timed to produce four separate strokes, or functional cycles: Intake, Compression, Power, and Exhaust. Sometimes called the "suck, squeeze, bang, and blow" process, this technique—combined with a properly atomized air/fuel mixture and a precisely timed spark ignition—produced an engine with high efficiency and power potential. Dynomation6 and DynoSim6 are designed to simulate the functional processes of a four-cycle engine.

Horsepower—Torque measures how much work (an engine) *can* do; and power is the rate-based measurement of *how fast* the work is being done. Starting with the static force applied at the end of a torque arm (torque), then multiplying this force by the swept distance through which the same force would rotate the torque arm one full revolution determines the power per revolution: Power Per Revolution = Force or Weight x Swept Distance. James Watt (1736-1819) established the current value for one horsepower: 33,000 pound-feet per minute or 550 pound-feet per second. So horsepower is currently calculated as: Horsepower = Power Per Revolution/33,000, which is the same as Horsepower = (Torque x 2 x Pi x RPM)/33,000, or simply: Horsepower = (Torque x RPM)/5,252. The horsepower being calculated by these equations is just one of several ways to rate engine power output. Various additional methods for calculating or measuring engine horsepower are commonly used (to derive friction horsepower, indicated horsepower, etc.), and each technique provides additional information about the engine under consideration.

Induction Airflow—The airflow rating (a measurement of restriction) of a carburetor or fuel injection system. Standard automotive four-barrel carburetors are rated by the measured airflow when the device is subjected to a pressure drop equal to 1.5-inches of mercury. Two-barrel carburetors are tested at 3.0-inches of mercury.

Intake Centerline Angle—The distance in crank degrees from the point of maximum intake valve lift (on symmetric cam profiles) to TDC during the valve overlap period.

Intake Stroke—One of the four 180-degree full "sweeps" of the piston moving in the

Mini Glossary

cylinder of a four-stroke, internal-combustion engine (originally devised by Nikolaus Otto in 1876). During the intake stroke, the piston moves from *TDC* to *BDC* and inducts (draws in by lowering the pressure in the cylinder) air/fuel mixture through the induction system. Note: The 180-degree duration of the intake stroke is commonly shorter than the period during which the intake valve is open, sometimes referred to as the true "Intake Cycle." The intake stroke is followed by the compression stroke.

Intake Valve Closing or **IVC**—Considered the most important cam timing event from a performance standpoint. The point at which the intake valve returns to its seat, or closes. This valve timing point usually occurs early in the compression stroke. Early IVC helps low-end power by retaining air/fuel mixture in the cylinder and reducing charge reversion at lower engine speeds. Late IVC increases high-speed performance (at the expense of low speed power) by allow additional charge to fill the cylinder from the ram-tuning effects of the induction system at higher engine speeds.

Intake Valve Opening or **IVO**—The point at which the intake valve lifts off of its seat, or opens. This valve timing point usually occurs late in the exhaust stroke. Although IVO does not have a substantial effect on engine performance, it contributes to valve overlap (the beginning point of overlap) that can have a significant effect on engine output.

Lobe-Center Angle or **LCA**—The angle in <u>cam degrees</u> from maximum intake lift to maximum exhaust lift. Typical LCAs range from 100 to 116 camshaft degrees (or 200 to 232 crank degrees).

Normally Aspirated—When the air-fuel mix is inducted into the engine solely by the lower pressure produced in the cylinder during the intake stroke; aspiration not aided by a supercharger.

Otto-Cycle Engine—See Four-Cycle Engine

Overlap or **Valve Overlap**—The period, measured in crank degrees, when both the exhaust valve and the intake valve are open. Valve overlap allows the negative pressure scavenge wave to return from the exhaust system and begin the inflow of air/fuel mixture into the cylinder even before the intake stroke begins. The effectiveness of the overlap period is dependent on engine speed and exhaust "tuning."

RPM—Revolutions Per Minute. A unit of measure for angular speed. As applied to the IC engine, rpm indicates the instantaneous rotational speed of the crankshaft described as the number of crank revolutions that would occur every minute if that instantaneous speed was held constant throughout the measurement period. Typical idle speeds are 300 to 800rpm, while peak engine speeds can reach as high as 10,000rpm or higher in some racing engines.

Mini Glossary

Simulation and **Engine Simulation**—A engine simulation process or program attempts to predict real-world responses from specific component assemblies by applying fundamental physical laws to "duplicate" or simulate the processes taking place within the components.

Smallblock—A generic term that usually refers to a V8 engine with a displacement small enough to be contained within a "small" size engine block. Typical smallblock engines displace under 400 cubic inches.

Stroke—The maximum distance the piston travels from the top of the cylinder (at TDC) to the bottom of the cylinder (at BDC), measured in inches or millimeters. The stroke is determined by the design of the crankshaft (the length of the stroke arm).

Top Dead Center or **TDC**—The position of the piston in the cylinder bore at its uppermost point in the stroke. Occurs twice within the full cycle of a four-stroke engine; at the start of the intake stroke and 360 degrees later at the end of the compression stroke.

Torque—The static twisting force produced by an engine. Torque varies with the length of the "arm" over which the twisting force is measured. Torque is a force _times_ the length of the measurement arm: _Torque = Force x Torque Arm_, where _Force_ is the applied or the generated force and _Torque Arm_ is the length through which that force is applied. Typical torque values are ounce-inches, pound-feet, etc.

Valve Head and **Valve Diameter**—The large end of an intake or exhaust valve that determines the working diameter. Valve head temperature can exceed 1200 degrees(F) during engine operation and a great deal of that heat is transferred to the cylinderhead through the contact surface between the valve face and valve seat.

Valve Lift—The distance the valve head raises off of the valve seat as it is actuated through the valvetrain by the camshaft. Maximum valve lift is the greatest height the valve head moves off of the valve seat; it is the lift of the cam (lobe height minus base-circle diameter) multiplied by the rockerarm ratio (in engines equipped with rockerarms).

Valve Motion Curve or **Valve Displacement Curve**—The movement (or lift) of the valve relative to the position of the crankshaft. Different cam styles (i.e., flat, mushroom, or roller) typically have different displacement curve acceleration rates. Engine simulation programs calculate a valve motion curve from valve event timing, maximum valve lift, and other cam timing specifications.

Volumetric Efficiency—An engine measurement calculated by dividing the mass of air inducted into the cylinder between IVO and IVC by the mass of air that would fill the cylinder at atmospheric pressure (with the piston at BDC). Typical values range

Mini Glossary

from 0.6 to 1.2, or 60% to 120%. Peak torque always occurs at the engine speed that produced the highest volumetric efficiency.
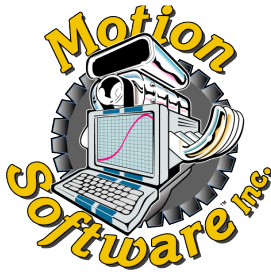
Dyno Testing Notes

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

Dyno Testing Notes

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

Dyno Testing Notes

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

_____

Motion Software, Inc.
222 South Raspberry Lane
Anaheim, CA 92808-2268

Support Phone: 714-231-3801
Web: www.MotionSoftware.com
Support Email: support@motionsoftware.com