

Steady and Transient CFD Approach for Port Optimization

Surendra Gaikwad and Kunal Arora
Mindware Engineering

Vamshi Korivi and Su K. Cho
Chrysler LLC

Copyright © 2008 SAE International

ABSTRACT

The intake and exhaust port design plays a substantial role in performance of combustion systems. The port design determines the volumetric efficiency and in-cylinder charge motion of the spark-ignited engine which influences the thermodynamic properties directly related to the power output, emissions, fuel consumption and NVH properties. Thus intake port has to be appropriately designed to fulfill the required charge motion and high flow performance.

While turbulence intensity and air-mixture quality affect dilution tolerance and fuel economy as a result, breathing ability affects wide open throttle performance. Traditional approaches require experimental techniques to reach a target balance between the charge motion and breathing capacity. Such techniques do not necessarily result in an optimized solution. Progress in development of Computational Fluid Dynamics (CFD) tools, Design of Experiment (DOE) and optimization techniques combined with increased computational power led to the development of new methodologies over the past decade. Such advancements have the potential to deliver optimized solutions.

Recent releases of engineering CAD packages, CATIA V5 and Pro-Engineer, enable both parametric modeling and associative design update. This paper demonstrates a coupling procedure of CFD with engineering CAD software using process integration and design optimization software (PIDO). CATIA V5, ICEM-CFD meshing tool and FLUENT-UNS CFD code were integrated to run through many port designs using ISIGHT. The automatic coupling was aimed at optimizing the port layout for a certain cost function such as flow restriction or charge motion, subject to manufacturing and packaging constraints. Accomplishing this task necessitates running the executables of various software using macros and scripts. This integrating methodology utilized best design practices for an intake port, and numerous numerical experiments were attempted.

Based on the above mentioned DOE approach, a few designs were selected along with designs optimized based on charge

flow and tumble (turning ability of flow). Further, a steady state analysis at full lift was performed with finer mesh to confirm the coarse mesh results. A sensitivity chart of flow and tumble with respect to each design parameter was derived so as to better understand the design parameter response on the combustion system.

But the steady state approach has its own limitations. It has been shown in the past, high tumble during intake stroke necessarily guarantee high tumble/turbulence at ignition. After the intake valve closing, the chamber and piston shape play bigger role in preserving the tumble. To determine the effectiveness of optimized shape, it is essential to do transient analysis. The transient analysis allows the detailed analysis of the in-cylinder system with more realistic assumptions and boundary conditions. This leads to better understanding of the flow characteristic at various valve lifts as well as evaluation of significant quantities like turbulent kinetic energy (TKE) and tumble values at the ignition.

This paper is a continuation of the effort initiated by Korivi et. al. [1,5] in terms of extending the optimization approach to transient analysis. In the present study, all the selected designs were further meshed using ES-ICE for the transient simulations. ES-ICE coupled with STAR-CD provides very prominent tool for moving mesh transient in-cylinder simulation. A novel approach was employed for faster turn around time in which chamber along with exhaust port was meshed only once and intake ports were meshed separately. Various parameters like turbulent kinetic energy, tumble ratio, valve curtain utilization area along with flow field inside the chamber were visualized.

INTRODUCTION

While structural analysis computations have been used in the automotive industry for a long time, the role of CFD in decision making is growing rapidly in the recent years. In the past, most of the development in CFD is contributed by the aerospace industry. The automotive industry is predominantly applications oriented and hence relies heavily on the use of commercial CFD codes to address their specific problems.



Documents

Document Sharing Hub

Currently CFD application in automobile sector includes flow around body, cooling circuit, climate control, exhaust system, combustion, under hood etc. But with the advancements and low cost of high performance computational hardware, effective and faster mesh generation tools, advanced commercial solvers and post-processing tools, CFD is finding ever-increasing role in more complex regime like multi-physics, fluid-solid interactions, and aero-acoustics.

Design optimization using CFD tools have been widely used in the powertrain area of the automobile. Intake and exhaust ports, intake and exhaust manifolds, induction systems, exhaust systems, water cooling jackets, in-cylinder combustion system, and catalytic converters are amongst the components optimized.

Design of a cylinder head for an automotive engine is important since it impacts performance, fuel economy, and emissions. Conventional process (Fig. 1) for cylinder head port development highly depends on a flow room where a preliminary flow box is conceptualized based on past experiences and is usually made of rigid and stable material to preserve dimension integrity over the development period. The flow box is then evaluated on a flow bench for airflow potential at various valve lifts under a specified pressure differential. Successive modifications/iterations are made to the port area schedule, valve seat details and chamber using clay or epoxy to arrive at a desirable solution.

The flow box is then mounted on a tumble meter where charge motion is measured. This tumble meter has honey comb at the $2/3^{\text{rd}}$ bore plane and torque is measured on the honeycomb, which is representative of the turning ability of the fluid. The charge motion inside the cylinder can be divided into three components namely – tumble, swirl and cross-tumble. Tumble is the predominant motion inside a four-valve chamber, while swirl is typical of a two-valve engine.

Once an acceptable tradeoff between charge motion and flow is achieved by flow box iteration, the internal surfaces are to be reverse-engineered. Experienced flow room technicians and designers are essential to develop port designs that result in anticipated improvements.

Numerical approach using steady state CFD was implemented for a port with few design variables in [2] and a need is state for transient analysis. A parametric approach with six design variables for an intake port was described in reference [3]. Morphing of an intake port approach without manufacturing constraints was evaluated in [4].

Engine designers have been interested in quantifying various design parameters that affect volumetric efficiency and charge motion. While experimental techniques can yield such parameter sensitivity [7], such approach is exhaustive, expensive and time consuming.

Here, an effort has been initiated to create a flow box inside the CAD system with CFD analysis capability in an iterative process accounting for manufacturing and packaging

constraints for a real engine in a production environment. This was executed in an automated fashion including the best practices built into the CAD template. This procedure will reduce the uncertainty of the reverse engineering process in which the iterative loop is carried out automatically and CAD design is optimized for performance that can be prototyped and validated.

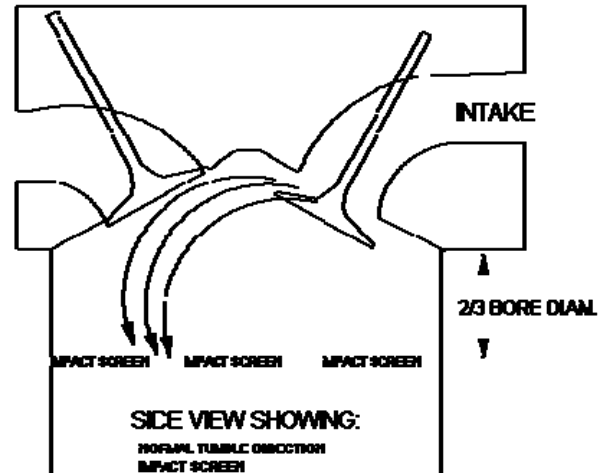


Fig 1. Flow room test procedure

In this paper, a novel CFD approach is used in which steady state analysis was done, at peak and half valve lifts, to determine the candidate designs for detailed transient simulation.

The ideal approach would be to perform automated DOE using full transient approach. But if transient analysis was performed using all the given design parameters, the simulation time would have been increased by a factor of 20. Steady state approach gives a faster way with much reduced computational time to determine scarce but important flow characteristics at particular instant. So based on DOE using steady state analysis at full and half valve lifts, optimized design based on cfm and tumble value was selected. But the role of port is limited only when valve is open. Once the valve is closed, the chamber and piston shape play highly crucial role in preserving the flow characteristics during the compression cycle. This makes the transient analysis mandatory to determine the validity of optimized design. So, few candidate designs based on experience together with optimized design were selected and transient analysis was performed. This approach provides faster and effective way for port optimization.

IMPLEMENTATION DETAILS

Process Integration Details: Automatic analysis without much human interaction using ISIGHT [8] resulted in process automation. All of the software that were being used for CAD, meshing, and CFD solver were run through a UNIX script without GUI as outlined in Fig. 2. Initial design variables passed onto the CATIA script that launches the CATIA session to update the model and this model is fed into the

meshing software. The resultant mesh from the software is exported to the analysis software where the problem is set-up to run and submitted for analysis through a load share software on a High Performance Computer. The load share software monitors job submittal and status. Once the job is done, the analysis software will process the results and feeds both the cost function and the extracted information to the ISIGHT software. This process is continued until the DOE is completed.

Once the optimized design was obtained using various optimizing tools available in ISIGHT, few more designs, e.g. design with maximum volumetric efficiency, were selected for further evaluation. A detailed steady state analysis with finer mesh was done using FLUENT to predict the CFM and tumble value more accurately. More details regarding the steady state implementation are given in [5].

Using ES-ICE, the whole cylinder except ports was meshed for transient analysis. The ports for all the selected designs were meshed separately using ICEM and connected to the cylinder mesh in ES-ICE. A transient simulation was performed in STAR-CD yielding flow field information along with temporal variation of TKE, tumble etc.

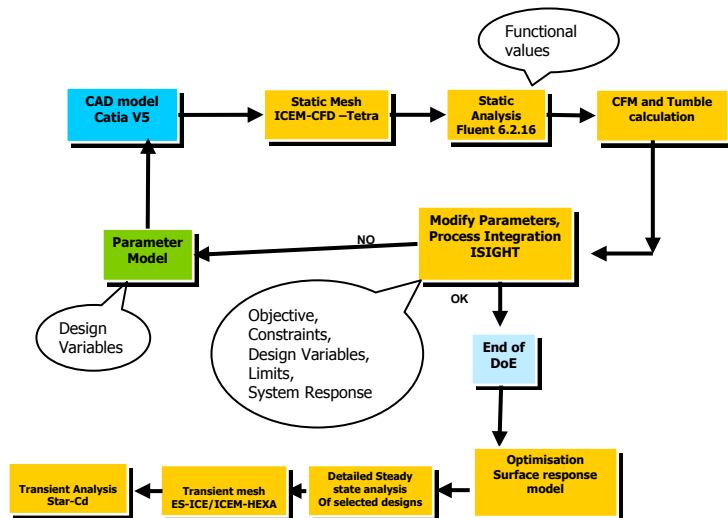


Fig 2. Process Integration Details.

Parameterization of the model: Design variables were selected as a result of session between design engineers and flow room technicians. Due to packaging constraints, several port design specification including valve angle, inner seat diameter and head/manifold interface cross sectional area, were not changed in this study. Seven independent design variables were selected as shown in Fig 3, including:

1. port to valve angle
2. short turn radius
3. chimney height
4. throat size
5. septum location
6. window height
7. floor to roof ratio

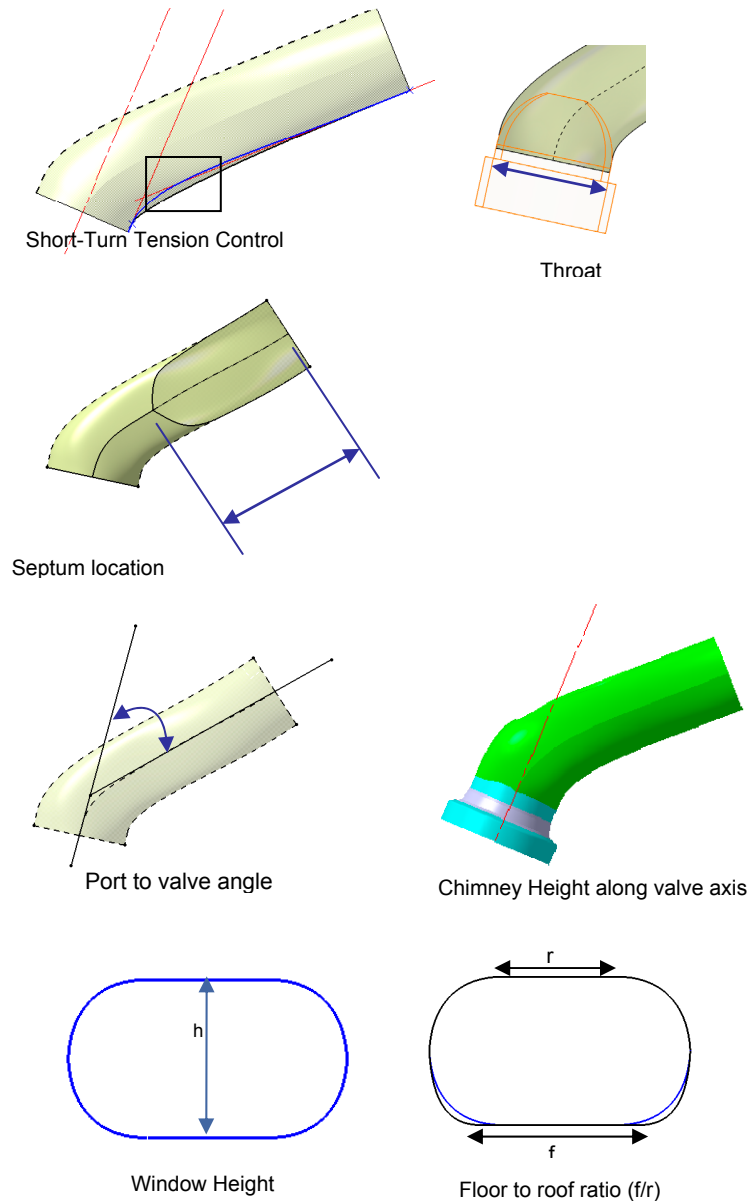


Fig 3. Design Parameters

While the seven design control variables define the overall shape of the port, detailed local shapes are controlled by parameters defined in each section. The port interaction with the surrounding architecture such as spring seats and push rods were also accounted for.

CAD models were generated at a specific valve lift with an ideal entrance and bore extension based on the ASME standard to alleviate flow entrance conditions impact on the flow development. The area scheduling was maintained by monitoring various cross-sectional areas at different locations while generating different CAD models. Also, port window area was kept constant while changing the window height and floor to roof ratio.

DOE: The CAD geometry from CATIA V5 was imported automatically into ICEM-CFD software. The geometry was