Using CFD to Simulate a Cylinder Head Flow Test

Cody McKinley

### Undergraduate Research Poster Symposium: April 11, 2006

Introduction:

- A cylinder head provides passageways for air to enter and exit the engine
- Computational Fluid Dynamics uses computer simulation to analyze fluid dynamic problems
- A CFD model has been created and is capable of duplicating actual flow numbers from a test bench
- Analysis of "Problem Areas" in ports by viewing contours and pathlines of pressure and velocity
- Alteration of the port shapes to optimize air flow
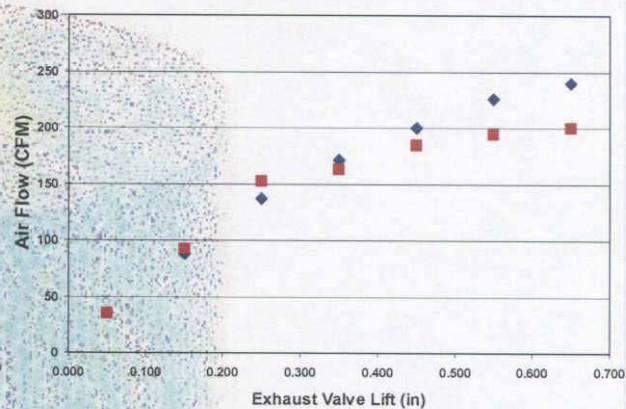
Methods:

- Obtain actual flow bench test data for cylinder head at standard 28" H_2O pressure differential
- Obtain computerized models of cylinder head and port shapes using laser scanner
- Create valves, valve seats, and cylinder bore with Pro/Engineer
- Mesh assembly in Gambit (preprocessing)
- Import mesh into Fluent to run CFD analysis
- Comparative Analysis

Intake Flow Model Correlation
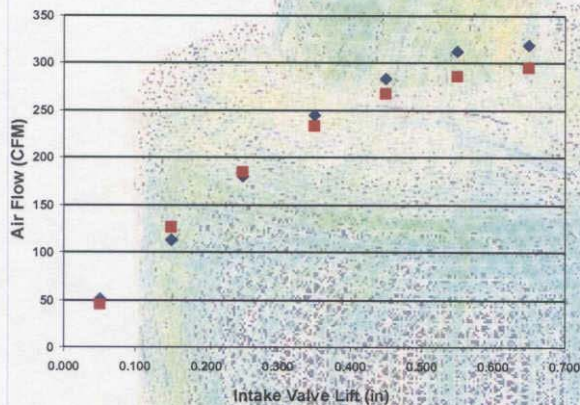
Exhaust Flow Model Correlation

Lift (in)	Actual Intake (CFM)	Fluent Intake (CFM)	% Error	Difference (CFM)
0.050	51	46	10.5	5.4
0.150	113	126	11.7	13.3
0.250	180	185	2.5	4.6
0.350	245	234	4.7	11.5
0.450	284	268	5.6	15.9
0.550	313	286	8.5	26.7
0.650	319	296	7.3	23.2

Lift (in)	Actual Exhaust (CFM)	Fluent Exhaust (CFM)	% Error	Difference (CFM)
0.050	36	35	2.9	1.1
0.150	88	92	4.8	4.2
0.250	137	152	11.1	15.2
0.350	171	163	4.9	8.4
0.450	200	184	7.8	15.6
0.550	226	195	13.9	31.4
0.650	240	200	16.5	39.7

Conclusion:

- Model predicts air flow at low valve lifts accurately
- Larger lift flows are not quite as accurate
- Laminar flow model predicts cylinder head flow patterns despite the turbulent Reynolds number
- Intake port is designed properly and flows well
- Problem areas spotted in exhaust port

Benefits:

- Savings in time and money during the engineering design process of the cylinder head
- Keeps companies up to date with technology
- Controlled engine air flow management
- Boosted power and superior engine performance
- Can lead to reductions in exhaust pipe emissions