SCM ENGINEERING SERVICES

Technical Report on

COOLANT FLOW ANALYSIS IN A 6 CYLINDER ENGINE

Executed By:

S Sunil Krishna
sunilkrishna.s@scmengg.com

S Sampath Kumar
sampath.s@scmengg.com

Technical Director

Dr. Subhash Deo
subhash@scmengg.com

SCM Engineering Services
Plot #156, First Floor,
Motilal Nagar,
Begumpet,
Hyderabad 500019,
India.
Ph: 040 – 65449094
Email: projects@scmengg.com
URL: http://www.scmengg.com

CONTENTS

1. INTRODUCTION

The present study involves, CFD simulation of three-cylinder internal combustion engine to predict the heat transfer rate through the circulating cooling water by using conjugate heat transfer method. The study was divided into two phases:

1. Coolant flow analysis within the cylinder block considering only fluid flow in the water jacket for the given inlet velocity to detect dead zones and pressure drop.

2. Conjugate heat transfer analysis considering both fluid flow in the water jacket and metallic structure of the cylinder block.

Heat transfer through water jacket and metallic structure of the cylinder block was considered for the analysis.

The study was carried out in the procedure given below,

1. CAD cleaning
2. Meshing
3. Flow Analysis

2. SOFTWARE AND HARDWARE USED

2.1 Software

The analysis consists of Pre–processing, analysis and post processing followed by the result synthesis. Mesh generation being critical to the analysis was done using ICEM-TETRA. Boundary conditions definition and post processing results analysis were carried out using Star-CD.

2.2 Hardware

The hardware used for mesh generation, pre processing, post processing and the Star-CD solver was Intel based windows platform. The work was carried out on P4 processor, with 2 GB RAM.

3. GEOMETRY AND MESH GENERATION

3.1 CAD clean up and mesh generation

3.1.1 CAD Clean Up

In order to simplify and making the geometry ready for meshing, a CAD clean up procedure was done using CATIA and steps indicated below:

- Removing the unwanted surfaces and holes from the original igs file supplied.
- Surface trimming and using other techniques to get a closed surface
- Surface generation by closing edges

3.1.2 Mesh generation

Mesh was generated using TETRA module of ICEM - CFD software.

Boundary layers were made near the fluid wall-surfaces.

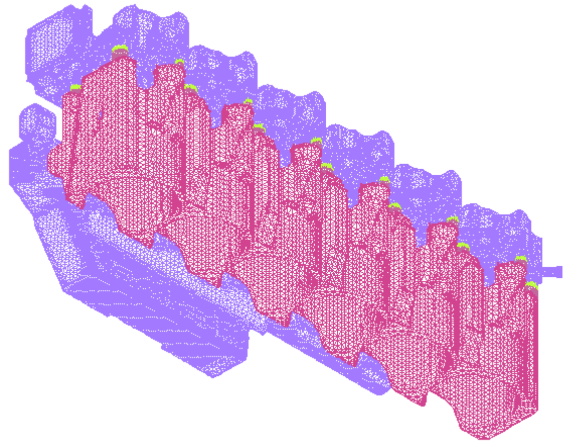

Fig 1. Mesh for solid region

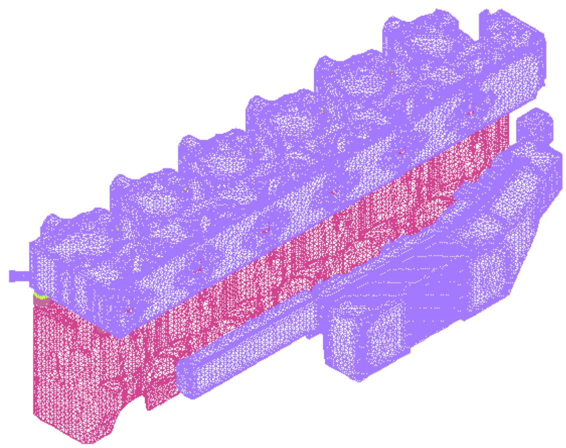

Fig 2. Different view of the mesh

4. PROBLEM SET UP AND SOLVING

Spatial Discretisation Schemes used:

Upwind and MARS Differential Schemes.

Convergence:

Mass, Momentum and Enthalpy convergence became to the order of 1.0e-5 after 3800 iterations.

5. RESULT OVERVIEW

- A strong re-circulation was found near the inlet region and few weak re-circulation zones in the sharp corners.
- The maximum velocity magnitude was found in sized hole zones, which is joining the cylinder block to the cylinder head.
- On examining the cells correspond to maximum residuals for velocity, the cells were found near the inlet and the sized hole portions.
- Pressure drop across the cooling circuit ~ 0.3 E+5 Pa

6. CONCLUSION

- The asymmetric flow pattern in the cooling circuit of the asymmetric geometry.
- Low velocity re-circulation zone detected at the both sides and all four topological corners of each cylinders horizontal cross-section.
- The growth of low velocity cells from 0.05 m/s starts from sixth cylinder block and moves towards the first cylinder block but is vice-e-versa in case of cylinder head.
- Most of cells of cooling circuit has got coolant flow velocity less than 0.5m/s, which is alarming to the cooling problem.
- The mass flow rate from oil cooler to cylinder block shows following the stack rule in flow process.