SCM ENGINEERING SERVICES

Technical Report on

CONJUGATE HEAT TRANSFER ANALYSIS OF A 3 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. Fluid 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 Star-CD. 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 Proam module of Star-CD software.

Boundary layers were made near the fluid wall-surfaces.

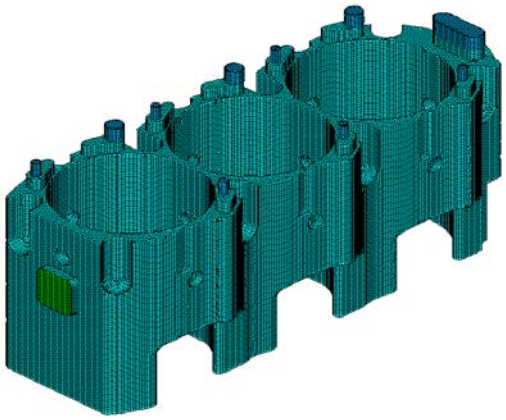

Fig 1. Mesh for fluid region

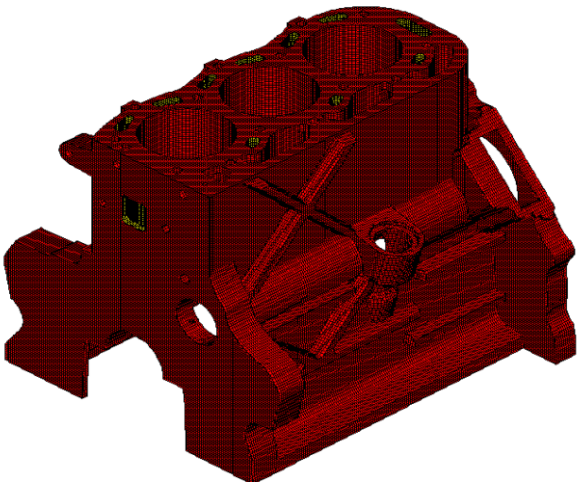

Fig 2. Finished mesh for solid region

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 4500 iterations.

5. RESULT OVERVIEW

The flow field at the inlet of the cooling circuit is **not symmetric**, due to the structural asymmetry and downstream flow, which is dominated by several small cross-sections and the several complicated stretches flow path.

The evident of **flow field re-circulation** was noted at the following locations:

1. Near to the inlet and through the sizing of the holes.

2. Between the cylinders, especially between 1st and 2nd cylinder a big re-circulation was absorbed.

3. Just behind the 3rd cylinder, a **choked region** was observed and this resulted in an increase of the adjoining solid temperature.

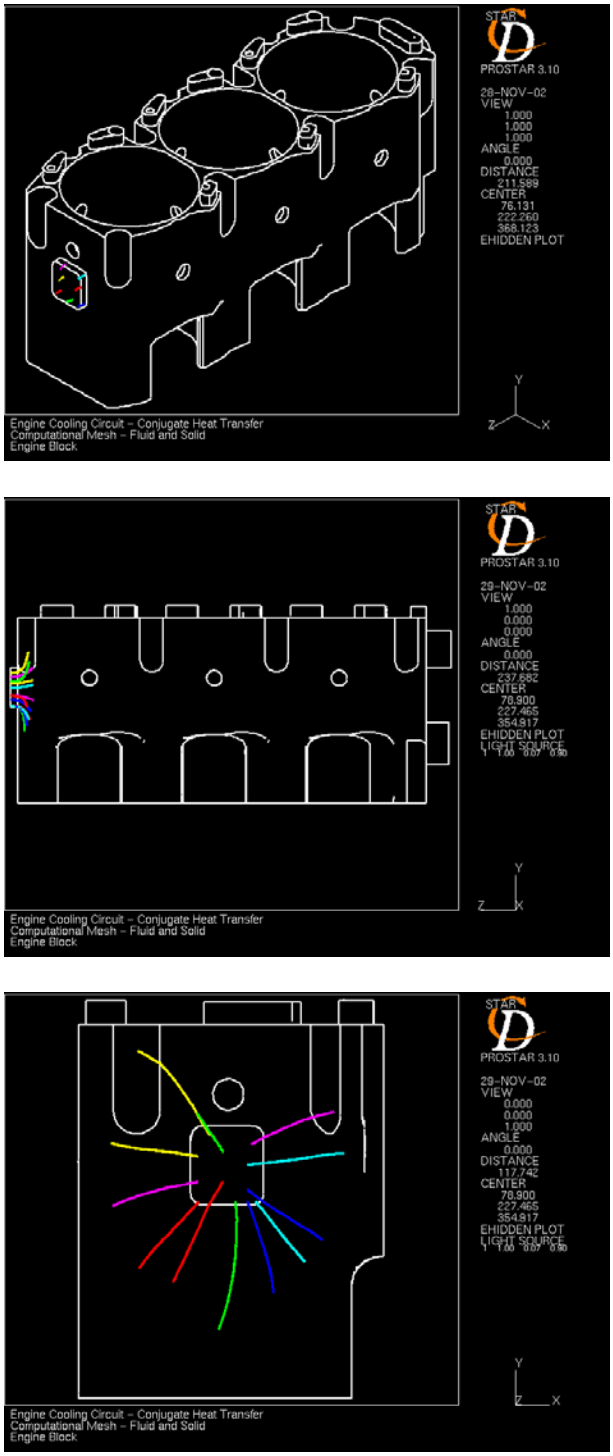

Fig. Particle tracks showing cooling water distribution near inlet.

6. CONCLUSION

- Low velocity re-circulation zone detected at both sides and all four corners of each cylinders horizontal cross-section, which is the sign of flow losses. It also affects the heat transfer rate.

- The heat transfer is directly proportion to the percentage of coolant flow in all those sections.