SCM ENGINEERING SERVICES

Technical Report on

CFD ANALYSIS FOR COOLING FAN OF A HEAT EXCHANGER

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, the multi rotating frames for an airduct consisting a fan, Computational Fluid Dynamics (CFD) Analysis. The parts considered for the analysis are simplified heat exchanger as velocity inlet, a fan mounted in a separate box and an exhaust duct. Fluid channel is a rectangular box (inlet) followed by the rotating fan and the exhaust duct (outflow).

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 Pro-AM (Star-CD). The boundary conditions definition and post processing - for setting problem and analyzing results, Analysis was done 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
- Splitting the geometry into four parts for logical meshing
- Leaving a small gap at the interface of blade edge surface
- Surface generation by closing edges

3.1.2 Mesh generation

The geometry was split in four separate parts in order to have a logical mesh and limit the number of cells to the minimum. These four parts were meshed and imported to the same file.

Initially a surface mesh was generated in Pro-surf and imported to Pro-AM for further meshing.

To capture the boundary layer wake, a boundary layer extrusion was considered near wall boundaries.

3.1.3 Meshing Considerations

Considering the various topology and grid quality requirements a trimmed cell hexahedral mesh option was beard in mind. The geometry was shrunk to the boundary layer thickness (1 mm) and then meshed with the hexahedral type cells. This followed by the boundary layer extrusion at the end.

3.1.4 Meshing criterion

Overall geometry was divided in four parts as indicated below,

1. Fan enclosure
2. Simplified uniform inlet
3. Outlet duct
4. Fan

Fan enclosure

Meshed with cylindrical coordinate system with hexahedral cells at center core containing (510104 cells)

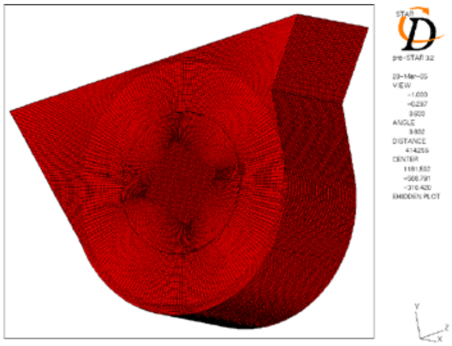

Fig 1. Fan enclosure separated and meshed

Simplified uniform inlet

The heat exchanger was removed and simplified rectangular box type geometry was considered as inlet.

Meshed, in Cartesian coordinate system, using pro-star. Fill ratio was considered to have finer mesh near the fan axis (300000 cells)

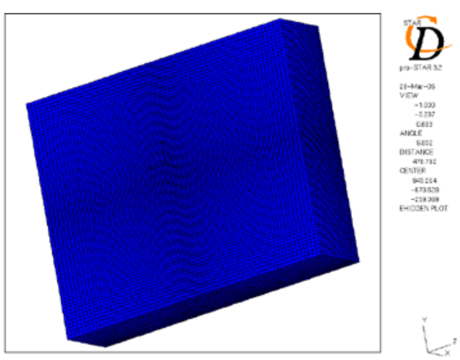

Fig 2. Simplified heat exchanger as uniform inlet, separated and meshed

Outlet duct

Outlet duct was meshed in Cartesian coordinate system. The finished mesh contains (182598 Cells)

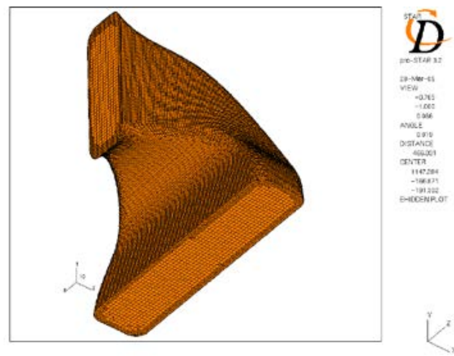

Fig 3. Exhaust duct separated and meshed

Fan

Fan geometry was meshed in cylindrical coordinate system. It was found feasible to mesh the interface area with pro-surf in the same cylindrical coordinates. Finished mesh contains 162768 cells including interface.

Fig 4. Fan separated and meshed

All the above-mentioned parts were then imported to a single file and coupled. As all uncoupled cells treat as wall (default), the parts were properly coupled.

4. PROBLEM SET UP

1. Fluid used was air and the properties were taken as default values in Star-CD.
2. Outlet boundary with no split.
3. Default wall boundary conditions with no slip.
4. MRF case with implicit scheme

4.1 Boundary Conditions

The following boundary conditions were supplied for Fluid analysis:

Mass flow rate = 3750m^3/hr

Hence, Velocity = 2.0842m/s.

Fan rotation speed = 1600 RPM

The inlet temperature was set to the default value i.e. 293K

Pressure at outlet was set to the default pressure i.e. 101.3 kPa.

4.2 Assumptions for the analysis

Fluid

a) The flow is treated as incompressible (low velocity) and hence the density is constant.

b) The problem is a steady state (MRF case).

c) The reference pressure and reference temperature was set to the atmospheric.

4.3 Processing

I. MRF case- Implicit scheme

II. Fan cells rotate at 1600 RPM in local cylindrical coordinate system

III. Rest of the cells are not rotating (at rest)

IV. Inlet boundary:

Velocity = 2.0842m/s

V. No flow split at outlet

4.4 Solver

i. Single precision

ii. Elapsed time = 30 hrs.

iii. No. of iteration = 1100

iv. Convergence e-04

v. K- epsilon turbulence model

5. RESULT SYNTHESIS & CONCLUSION

The analysis was carried out using Star-CD solver. Once the desired convergence was achieved the results were then written in a pst file format for further processing.

The flow pattern in the entire domain was expected to behave as follows,

a. Uniform flow at the inlet

b. Laminar flow in the convergent part of the nozzle.

c. High turbulent flow in the axial direction due to the fan rotation.

d. Few zones experiencing re-circulation in the fan enclosure, as there is only one-directional exit present.

6.1 Result synthesis

In comparison to the expectations the flow behavior obtained for the case discussed below:

The analysis shows re-circulation zones at various locations in the fluid domain. The main cause can be bounded fan region by fan enclosure.

Flow analysis

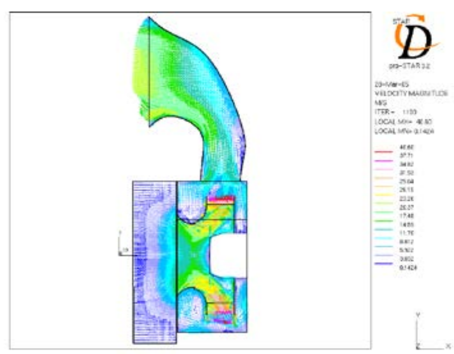

Fig 5. Velocity magnitude in the fluid domain

Above figure shows that the flow is uniform at the inlet and as it reaches the fan, velocities increase and maximum velocities are present at the blades. To see the clear picture of velocity distribution what is happening to the fan region, below is the plot from a different angle.

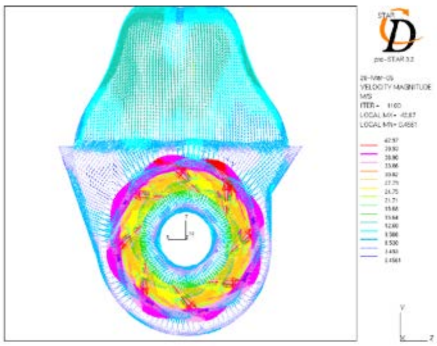

Fig 6. Velocity magnitude plot at the fan region

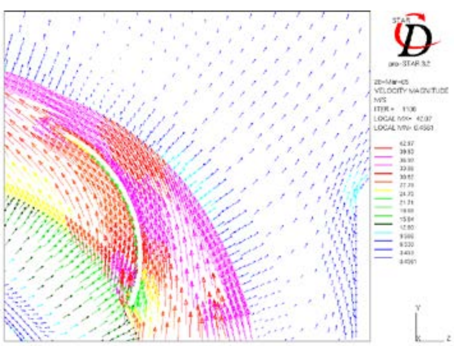

Fig 7. Velocity vectors near the fan blade

The above plot gives a clear picture and meets our expectations as mentioned above. The maximum velocity found to be 42.79 m/s. Also, we can see here the huge change in the magnitude after it passes the fan. Velocity vectors indicate that the flow is highly turbulent in this region.

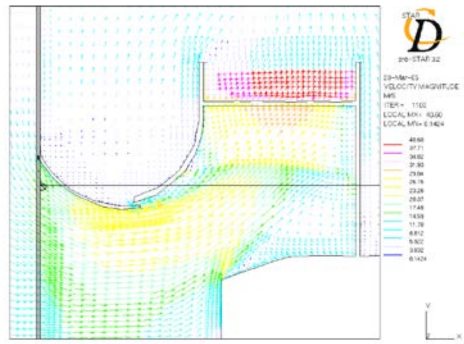

Fig 8. Re-circulation zone at the upper part of the fan

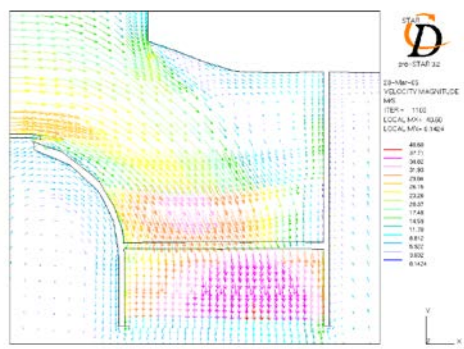

Fig 9. Re-circulation zone at the lower part of the fan

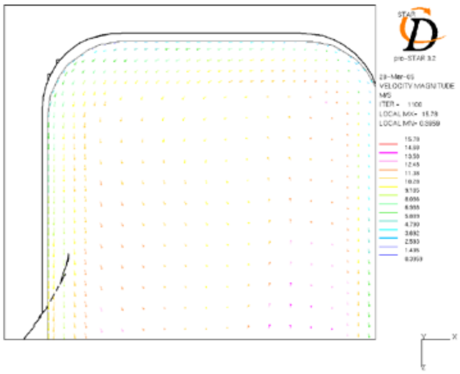

Figure 10. Re-circulation zone at the inlet to exhaust duct

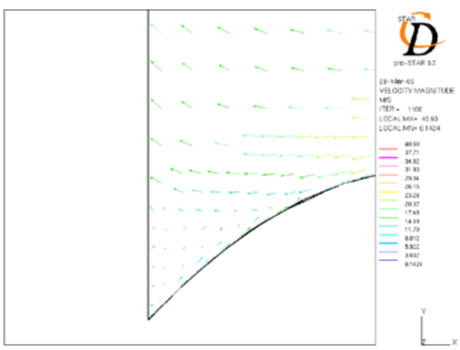

Fig 11. Re-circulation zone at the exit from exhaust duct

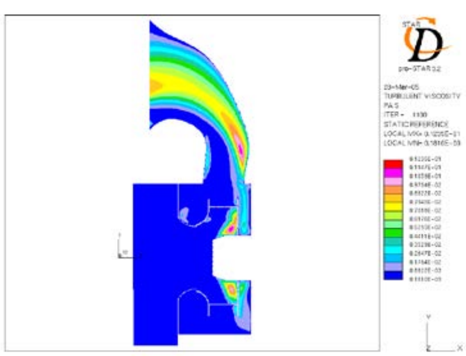

Fig 12. Turbulent viscosity across the fluid domain

Fig 13. Fluid particle track from the various locations

The above fluid particle track figure shows us the behavior of fluid motion that enters thru inlet and exits from the exhaust duct. The particle track the path followed by a single particle. To have a better idea about a particle behavior, single particle track was plotted and shown in Fig 14.

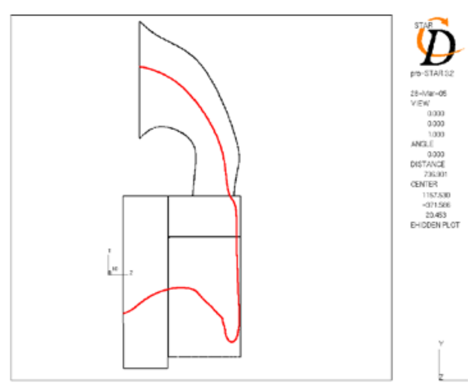

Fig 14. Single particle track

6.2 Conclusion

A conclusion was drawn on the basis of the heat exchanger fan CFD analysis, the geometry can be considered for optimization to avoid the above mentioned re-circulation zones. And followed by CFD analysis of the optimized geometry in order to have the best performance.

A serious optimization needed at the inlet of the exhaust duct and exit as well. It is recommended to reduce the exit area of the exhaust duct and avoid any divergent parts at the lower part.