CFD ANALYSIS FOR OPTIMIZATION OF VELOCITY AND TEMPERATURE DISTRIBUTION WITH MODIFIED RADIATOR DESIGN AT DIFFERENT FAN RPM

CHAVAN D. K¹ & TASGAONKAR G. S²

¹Scholar, JJTU Rajasthan, Professor, Mechanical Engineering MMCOE, Pune, Maharashtra, India
²Professor, Navsahyadri Group of Institutes, Pune, Maharashtra, India

ABSTRACT

Computational Fluid Dynamics (CFD) analysis is made for the radiator & the goal is to predict a flow behavior of radiator with supporting velocity profile. CFD analysis is carried out in 2 phases. 1) Geometry and meshing 2) Steady state simulation. CFD models are prepared for radiator having fan, shroud & radiator tubes. Also velocity plot is done at different locations and at different rpm of fan.

Also radiator as a porous domain is studied and velocity plot is studied. It is observed that from CFD analysis, velocity of air increases from centre to extreme & further it remains constant. And it can be concluded that new proposed radiator design possesses almost constant velocity at the exit.

KEYWORDS: CFD, Porous Domain, Radiator, Velocity Plot, Optimization

INTRODUCTION

Computational fluid dynamics modeling was developed to predict the characteristics and performance of flow systems. Overall performance is predicted by breaking the flow system down into an appropriate number of finite volumes or areas, referred to as cells, and solving expressions representing the continuity, momentum, and energy equations for each cell. The process of breaking down the system domain into finite volumes or areas is known as mesh generation. The number of cells in a mesh varies depending on the level of accuracy required, the complexity of the system, and the models used. Equations solve for flow (x, y, and z velocities), energy (heat fluxes and temperatures), chemical reactions (reaction kinetics and species concentrations), and pressure based on various simplifications and/or assumptions (Anderson J. D, 1995). Some simplifications and assumptions are discussed below. If performed correctly, CFD modeling can accurately predict the performance of an entire system.

MODELING OBJECTIVE

The primary goal of this dissertation is to predict a flow behavior of radiator with supporting velocity profile. CFD simulation will be carried in 2 phase

Phase 1: Geometry and Meshing

- Geometry modeling will be done in Ansys Workbench
- Meshing will be done in Ansys Mesher.
  - Unstructured mesh is used for the analysis
  - Fine mesh at specific location to capture the flow behavior.
**Phase 2:** Steady State Simulation

- Steady state incompressible simulation is performed for different flow and atmospheric conditions.
- Flow and heat transfer will be solved considering different working scenarios.
- Appropriate turbulence models will be used depending on the Reynolds number.
- Flow rates and pressure drops will be observed and verified.

The validity of the models to various flow configurations for the radiator is assessed with the aid of comparisons with the corresponding experiments shown in the block diagram below.

**Figure 1: Flow Diagram of CFD Process**

**NUMERICAL MODEL DETAILS**

The commercial CFD Code Ansys Fluent was used as preprocessor for solving the flow physics. The computational Domain had to be represented as a finite volume model. Volume mesh is generated in Ansys Mesher and refinement is done at specific locations to capture the flow behavior. Unstructured volume grid with 1375000 tet cells is used for the simulation. In Fluent, appropriate scaling factors for scaling the mesh/geometry, and setting up the appropriate turbulence model, fluid properties, solver controls, convergence monitors.

All simulations were performed using the Standard k-ε turbulence model. For stability of the solution the linear upwind scheme was used in the beginning, once stabilized a second order upwind differencing scheme was used.

**Boundary Condition**

**Inlet and Outlets**

For the inlet and outlet the static pressure is specified, which is fixed to 0.
Side and Top Walls

Pressure outlet is considered.

Porous Medium

The radiator is modeled as porous medium to account for the pressure drop in the radiator.

MRF Zone

In a steady-state approximation in which individual cell zones can be assigned different rotational speeds. The flow in fan zone is solved using the moving reference frame equations.

Extended fluid domain is considered for the CFD analysis to capture the fluid behavior. The domain away from the radiator is open to atmosphere. The length of the domain is decided so that the flow near the fan and inside the radiator is not getting affected.

Simplified radiator model shown above have fan, shroud and radiator with tubes. Radiator fan is rotating with different rpm is consider in the analysis.
RESULTS AT 1500 RPM

In figure 5 velocities before radiator and at the central plane of radiator is shown for 1500 rpm speed of fan before radiator.
Graph 1 shows velocity plot for horizontal and vertical line before radiator and in graph-2 plotting is done for diagonal lines.

![Graph 1: Velocity Plot Comparison with Experimental](image1)

1. Velocity Plot Comparison with Experimental
2. Velocity Plot Comparison with Experimental

Figure 7: Velocity Plot Comparison for CFD and Experimental at Different Location

CFD velocity result and experimental value are compared at different location in figure 7. In the graph L1 to L8 represent the line location as per the figure 4. Velocity trend is near about same for both CFD and experimental but the variation has been seen because of many assumption and measurement constraints. Radiator domain in CFD has tubes and remaining as a fluid domain, but in actual fins act as a resistance to the flow.

RESULTS AT 2000 RPM

In figure 9 velocities before radiator and at the central plane of radiator is shown for 2000 rpm speed of fan before radiator. High velocity region is visible compared to lower rpm.

![Graph 2: Velocity Plot along Horizontal and Vertical Line](image2)

1. Velocity Plot along Horizontal and Vertical Line
2. Velocity Plot along Diagonal Lines

Figure 8: Velocity Plot is Done at Different Location
1. Velocity Pattern before Radiator
2. Velocity Pattern at the Center of Radiator

Figure 9: Velocity Contour at Different Location

Graph 1 shows velocity plot for horizontal and vertical line before radiator and in graph-2 plotting is done for diagonal lines.

RESULTS AT 1500 RPM AND RADIATOR AS POROUS DOMAIN

1. Radiator Model without Tubes
2. Velocity Pattern before and after Radiator

Figure 10: Radiator Model and Velocity Contour at Different Location

In figure 10 radiator model with porous domain is consider and in the second plot velocities before and after radiator is shown for 1500 rpm speed.

1. Velocity Plot along Horizontal and Vertical Line
2. Velocity Plot along Diagonal Lines

Figure 11: Velocity Plot is Done at Different Location before Radiator
Graph 1 shows velocity plot for horizontal and vertical line before radiator and in graph-2 plotting is done for diagonal lines.

1. Velocity Plot along Horizontal and Vertical Line  
2. Velocity Plot along Diagonal Lines

Figure 12: Velocity Plot is Done at Different Location after Radiator

Graph 1 shows velocity plot for horizontal and vertical line after radiator and in graph-2 plotting is done for diagonal lines.

RESULTS AT 1500 RPM AND RADIATOR AS POROUS DOMAIN WITH MODIFICATION IN SHAPE

1. Radiator Model without Tubes  
2. Radiator Model Side View

Figure 13: Proposed Radiator Model

Proposed modified radiator model is shown in figure with porous domain. At the center radiator thickness is less compared to peripheral thickness.
In figure 15 radiator velocity before and after radiator is shown for 1500 rpm speed.

Graph 1 shows velocity plot for horizontal and vertical line before radiator and in graph-2 plotting is done for diagonal lines.

Figure 16: Velocity Plot is Done at Different Location after Radiator
Graph 1 shows velocity plot for horizontal and vertical line after radiator and in graph-2 plotting is done for diagonal lines.

CONCLUSIONS

- Velocity of air increases with the increase in rpm of radiator fan.
- CFD and experimental velocity values are compared at different locations in figure 4.6 for 1500 rpm.
- Fins are not considered in the simulation. It will also create some resistance to the flow. Radiator domain in CFD has tubes and remaining as a fluid domain.
- Variations in the results obtained by CFD and experimentation are observed near about 15 to 20% at different locations.
- It can be observed that average velocity of air entering before the radiator inlet at the center core is almost zero, and it increases towards the periphery, because of change in shape. (For proposed radiator with porous zone as shown in figure 4.11)
- Shape optimization can be done to improve the radiator efficiency i.e.,
  - Smaller fins are provided at the centre and length of the fins increases towards periphery.
  - Low velocity zones are created in the corners, hence eliminate corners and have circular radiator for optimum efficiency.
- The new proposed radiator shape with porous zone is shown figure 4.14.
- It is also observed from CFD analysis that, velocity of air increases from centre to extreme and further it remains constant.
- Further it can be concluded that the air entering in the radiator, with new proposed design possesses almost constant velocity at the exit.

REFERENCES
