Computational fluid dynamics analysis of nozzle used in diesel engine

Bhanu Pratap Pulla

Lecture in Mechanical Engineering Department, Addis Ababa Science and Technology University, Ethiopia.

ABSTRACT

The main aim of the work is to design a hole type nozzle used in a diesel engine. Model of the nozzle is done in Pro/Engineer. Design of the nozzle is changed by changing the diameter of the nozzle and modelled. The flow through diesel fuel injector nozzles is important because of the effects on the spray and the atomization process. Modelling this nozzle flow is complicated by the presence of cavitation inside the nozzles. There are two main types of injector nozzle, hole and pintle. Hole-type nozzles are commonly used in direct injection engines. They can be single-hole, or multi-hole, and they operate at very high pressures, up to 200 atmospheres. They give a hard spray, which is necessary to penetrate the highly compressed air. The fuel has a high velocity and good atomization which is desirable in open combustion chamber engines. CFD analysis is done on both the design to evaluate the heat transfer rate for different velocities. Material used for nozzles is heat treated alloy steel. Modelling is done in Pro/Engineer and CFD analysis is done in ANSYS.

Keywords: Dynamics; Diesel Engine.

INTRODUCTION

The internal flow through diesel fuel injector nozzles is very important for reducing emissions, although the nature of the connection between the injector and emissions appears to be quite complicated (Montgomery [1]). The typical nozzle is roughly 250 microns in diameter and the fluid is moving at several hundred meters per second making experimental measurements of the flow difficult. Further more cavitation within the nozzle complicates both experiments and computations. Studying real fuel injectors is also difficult because the nozzle geometry is often some variant of the plain orifice. We use a numerical model for predicting cavitation nozzle flow and apply it to a variety of nozzles under realistic injection conditions. By using a variety of geometries we hope to make some practical observations which will further our understanding of fuel injector flow.

Despite the importance of injector nozzle flow, the difficulty of directly observing such a small, fast, two-phase flow has limited the number and quality of available experimental data. A few, very valuable, experiments have directly observed the internal nozzle flow. Bergwerk [2] performed some of the early work in cavitating flow through small nozzles. Nurick [3] provided a model of cavitating nozzle mass flow and validated it with measurements in relatively large, low speed flows. Hiroyasu [4] took photographs of low speed cavitating nozzle flow and observed a very significant connection between the nozzle flow and the downstream spray. Chaves [5] obtained photographs and made some measurements in small, cavitating nozzles and also looked at the subsequent sprays. The cavitating nozzles could be scaled up for mass flow rate measurements so long as a cavitation parameter was controlled. These works provide some qualitative understanding of the cavitating nozzle, measurements of some gross flow quantities, and a modicum of quantitative information about the effect of varying geometry. A few attempts have also been made to predict cavitating nozzle flow numerically. Delannoy and Kueny [6] used a TVD scheme and a barotropic
equation of state to predict inviscid cavitating flow through a venture. They assumed density was a continuous function of pressure where both pure phases were incompressible and that the phase change could be fitted by a sine curve. They had to use a very low ratio of vapor to liquid density for stability reasons, which may have affected their results. They had some success in matching the qualitative behaviour of the venturi but did not correctly predict the shedding frequency of bubbles or their length. Later, Chen[9] developed a few different models of cavitating flow.

In one case he closed the hydrodynamic equations by assuming that the growth phase of the bubbles occurred at the vapor pressure and that the substantial derivative of density could be given by an empirical relation during the collapse phase. The pure liquid phase was treated as incompressible. This model was applied to a sharp nozzle and appeared to give reasonable results but was not fully explored. Chen[9] also constructed a model where the density was given by an analytical relation based on Rayleigh bubble collapse in a bounded domain. Chen assumed that the fluid contained small spherical bubbles with an assumed number density. It is doubtful, however, that the fine distributions of spherical bubbles which are assumed by the latter model exist in injector nozzles, due to the small nozzle diameter. Chaves[5] instead observed long bubble films along the nozzle walls. The photographs of larger nozzles taken by Sorteriu[6] show clouds of small bubbles and are more consistent with the model proposed by Chen. Avva[10] presented a model of cavitating flow which was based on an energy equation for a homogenous mixture of phases, assuming no interphase slip. The fluid properties were evaluated using standard thermodynamic tables, thereby assuming thermodynamic equilibrium on the sub-scale level. By starting with an energy equation, the momentum and density equations could be closed without further modelling assumptions.

The authors treated the liquid phase as incompressible. They did not give much detail about the numerical differencing scheme they used and what steps were taken to handle the rapid density changes of the fluid. They did report some convergence difficulties and were not able to apply their model to the high speed flow which typifies diesel injector nozzles. Schmidt[11] has recently developed a two-dimensional, transient model which is specifically intended to predict small, very fast, nozzle flows as in diesel fuel injectors. The model uses a barotropic equation of state, but includes the compressibility of both the liquid and the vapor phase. A third-order shock capturing scheme is applied to the density equation to avoid density oscillations at the liquid-vapor interface. The numerical model has been described in detail and was validated with experimental data in Ref. [11]. A description of this model will be given below.

Results of Finite Element Analysis

FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested. In practice, a finite element analysis usually consists of three principal steps:

Pre-processing: The user constructs a model of the part to be analysed in which the geometry is divided into a number of discrete sub regions, or elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical "pre-processor" to assist in this rather tedious chore. Some of these pre-processors can overlay a mesh on a pre-existing CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.

Analysis: The dataset prepared by the pre-processor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations

$$K_i x_j x_{uj} = f_i,$$

where $u$ and $f$ are the displacements and externally applied forces at the nodal points. The formation of the $K$ matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types. One of FEA's principal advantages is that many problem types can be addressed with the same code, merely by specifying the appropriate element types from the library.

Post processing: In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results.

Modal Analysis: A modal analysis is typically used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a harmonic response or full transient dynamic analysis.
Stainless Steel

Figure 1.1: Imported Model-1

Ansys Model

Figure 1.2: Fluid Pressure Condition-2

stainless steel

Figure 1.3: Pressure analysis-1

Figure 1.4: Pressure analysis-2

Figure 1.5: Velocity curve-1

Figure 1.6: Velocity curve-2

Figure 1.7: Reference conditions

Figure 1.8: Rate of change of curve
Modal analyses, while being one of the most basic dynamic analysis types available in ANSYS, can also be more computationally time consuming than a typical static analysis. A reduced solver, utilizing automatically or manually selected master degrees of freedom is used to drastically reduce the problem size and solution time.

Harmonic Analysis

Used extensively by companies who produce rotating machinery, ANSYS Harmonic analysis is used to predict the sustained dynamic behaviour of structures to consistent cyclic loading. Examples of rotating machines which produced or are subjected to harmonic loading are: Turbines, Gas Turbines for Aircraft and Power Generation, Steam Turbines, Wind Turbine, Water Turbines, Turbo pumps, Internal Combustion engines, Electric motors and generators, Gas and fluid pumps, Disc drives A harmonic analysis can be used to verify whether or not a machine design will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations.

Computational Fluid Dynamics Analysis of By Reducing Diameter of Nozzle

The ANSYS/FLOTRAN CFD (Computational Fluid Dynamics) offers comprehensive tools for analysing two-dimensional and three-dimensional fluid flow fields. ANSYS is capable of modelling a vast range of analysis types such as: air foils for pressure analysis of airplane wings (lift and drag), flow in supersonic nozzles, and complex, three-dimensional flow patterns in a pipe bend. In addition, ANSYS/FLOTRAN could be used to perform tasks including:

- Calculating the gas pressure and temperature distributions in an engine exhaust manifold
- Studying the thermal stratification and breakup in piping systems
- Using flow mixing studies to evaluate potential for thermal shock
- Doing natural convection analyses to evaluate the thermal performance of chips in electronic enclosures

Conducting heat exchanger studies involving different fluids separated by solid regions FLOTTRAN analysis provides an accurate way to calculate the effects of fluid flows in complex solids without having to use the typical heat transfer analogy of heat flux as fluid flow. Types of FLOTTRAN analysis that ANSYS is able to perform include:

Laminar or Turbulent Flows, Thermal Fluid Analysis, Adiabatic Conditions, Free surface Flow, Compressible or incompressible Flows, Newtonian or Non-Newtonian Fluids, Multiple species transport

**Structural Analysis and Results**

In our project, we have designed and modelled a nozzle used in diesel engines. We have performed different heat treatment processes on the nozzle (i.e.) Carburizing at 900°C, hardening at 850°C, sub-zero treatment at -60°C and tempering at 180°C.

Heat treatment processes are performed to increase the performance of the nozzles. We have checked the fluid flow in the nozzle by CFD analysis by applying fluid flow at different inlet velocities of 15.5m/sec and 50m/sec. We have done CFD analysis to check the fluid flow in the nozzle. The maximum velocity developed in the nozzle is 97.027 m/sec and stagnation pressure is 35155 N/m² when the inlet velocity is 15.5m/sec. The maximum velocity developed in the nozzle is 52.007 m/sec and stagnation pressure is 0.165e17 N/m² when the inlet velocity is 50m/sec. In this project we are studied about the Design evaluation and optimization of a nozzle used in diesel engine. In every step we are studied about the different types of nozzles and performances of the nozzle.

**CONCLUSION**

We have designed and modelled a nozzle used in diesel engines. We have performed different heat treatment processes on the nozzle (i.e.) Carburizing at 900°C, hardening at 850°C, sub-zero treatment at -60°C and tempering at 180°C. Heat treatment processes are performed to increase the performance of the nozzles. We have checked the fluid flow in the nozzle by CFD analysis by applying fluid flow at different inlet velocities of 15.5m/sec and 50m/sec. We have done CFD analysis to check the fluid flow in the nozzle. The maximum velocity developed in the nozzle is 97.027 m/sec and stagnation pressure is 35155 N/m² when the inlet velocity is 15.5m/sec. The maximum velocity developed in the nozzle is 52.007 m/sec and stagnation pressure is 0.165e17 N/m² when the inlet velocity is 50m/sec.

**REFERENCES**
