

DESIGN EVALUATION AND OPTIMIZATION OF A NOZZLE USED IN DIESEL ENGINE

V. Mallikarjuna^{*}

G. Nagaraju^{**}

B. Rama Bhupal Reddy^{***}

ABSTRACT:

A nozzle is a mechanical device designed to control the direction or characteristics of a fluid flow as it exits (or enters) an enclosed chamber or pipe via an orifice. A nozzle is often a pipe or tube of varying cross sectional area and it can be used to direct or modify the flow of a fluid (liquid or gas). Nozzles are frequently used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that emerges from them. The flow through diesel fuel injector nozzles is important because of the effects on the spray and the atomization process. Modeling this nozzle flow is complicated by the presence of cavitations 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. The main aim of the paper is to design 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 modeled. CFD analysis is done on both the design to

^{*} Assistant Professor, Department of Mechanical Engineering, Global College of Engineering, Science & Technology, Kadapa, A.P., India.

^{**} Assistant Professor, Department of Mechanical Engineering, Sreenivasa College of Engineering & Technology, Laxmipuram, Kurnool, A.P., India.

^{***} Associate Professor, Department of Mathematics, K.S.R.M. College of Engineering, Kadapa, A.P., India.

evaluate the heat transfer rate for different velocities. Material used for nozzles is heat treated alloy steel. Modeling is done in Pro/Engineer and CFD analysis is done in ANSYS.

Key words: CAD/CAM/CAE, ANSYS, CFD, Nozzle

1. 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. 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 cavitations within the nozzle complicate 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 cavitations 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. A model of cavitations nozzle mass flow and validated it with measurements in relatively large, low speed flows. The photographs of low speed cavitations nozzle flow and observed a very significant connection between the nozzle flow and the downstream spray. Obtained photographs and made some measurements in small, cavitations nozzles and also looked at the subsequent sprays. The cavitations nozzles could be scaled up for mass flow rate measurements so long as a cavitations parameter was controlled.

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 modeling 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. The model uses a bar tropic 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. A description of this model will be given below.

2. NUMERICAL MODEL

The first key assumption of this model is that fuel is the only fluid present. There is no differentiation between phases, so only one density variable is required. Thus, the basic equations of continuity and momentum are the same as in single-phase flow.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \bar{V} + \omega H_{\rho} = 0 \quad (1)$$

$$\frac{\partial \rho u}{\partial t} + \nabla \cdot \rho u \bar{V} + \omega H_x = -\frac{\partial p}{\partial x} + \nabla \mu \nabla u + \omega H_{xv} \quad (2)$$

$$\frac{\partial \rho v}{\partial t} + \nabla \cdot \rho v \bar{V} + \omega H_y = -\frac{\partial p}{\partial y} + \nabla \mu \nabla v + \omega H_{yv} \quad (3)$$

ρ is the fluid density and (u,v) are the components of the velocity vector. The terms which are denoted by H are the axis Symmetric acceleration and viscous terms. The variable w which appears in front of the axis symmetric terms is either zero or unity and is used to handle calculations in Cartesian or polar coordinates. The viscosity used in the viscous terms represents is estimated to be linearly proportional to density, as in Kubota [12]. The subscript l denotes the liquid phase.

$$\mu = \mu_l \left(\frac{\rho}{\rho_l} \right) \quad (4)$$

These equations require an additional expression for pressure. A simplified, barotropic equation is used for closure:

$$\alpha^2 \frac{D\rho}{Dt} = \frac{DP}{Dt} \quad (5)$$

α is the speed of sound and can be written as a function of density by assuming the two phases are homogeneously mixed on the sub-cell level. For pure vapor, when the density is less than that of a saturated vapor, the sound speed is treated as a constant. For the pure liquid phase, the

variation in sound speed is also neglected. We have analytically integrated this result to give pressure as a function of density. The final result is stated below:

$$p = p_l^{sat} + p_{gl} \cdot \log \left[\frac{\rho_g \cdot \alpha_g^2 \cdot \rho_l + \alpha \cdot \rho_l}{\rho_l \cdot \alpha_g^2 - \alpha \cdot \rho_l} \right] \quad (6)$$

Where the subscript g refers to the vapor phase and l refers to the liquid phase and where p_{gl} is a constant given by

$$p_{gl} = \frac{\rho_g \cdot \alpha_g^2 \cdot \rho_l \cdot \alpha_l^2 - \rho_l}{\rho_g^2 \cdot \alpha_g^2 - \rho_l^2 \cdot \alpha_l^2} \quad (7)$$

α is the void fraction, which is also a function of density:

$$\alpha = \frac{\rho - \rho_l}{\rho_g - \rho_l} \quad (8)$$

This analytic result means that only three partial differential equations must be solved: Equations (1), (2), and (3). Pressure is evaluated analytically from density by equation (6). Aside from the numerical advantages of this closure, this is a more physically reasonable model than assuming incompressibility of the pure phases. Compressibility naturally limits acceleration and pressure during bubble collapse.

Manufacturing Process

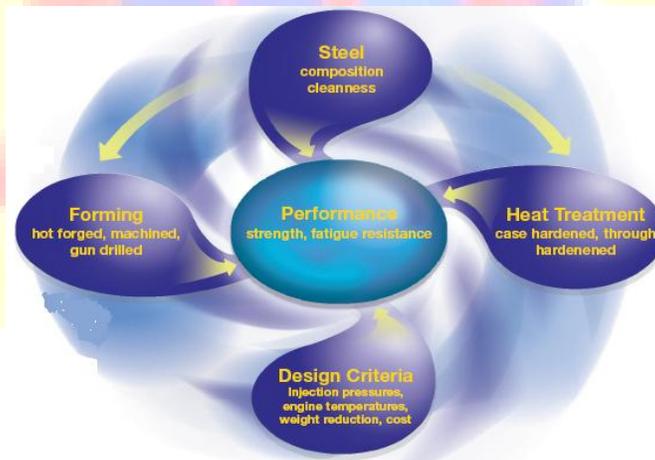


Figure 1 : Manufacturing Process

Performance Requirements

1. High mechanical strength to tolerate the high system pressures

2. Resistance to the fatigue stresses imposed by the fuel delivery cycle
3. Durability at high combustion temperatures
4. Nozzle body holders are also subjected to high fatigue stresses from pulsating fuel pressures. Additionally, weight and packaging constraints are demanding smaller components.

Nozzle Body Holders and Unit Injector Bodies Characteristics

1. High mechanical strength to resist the ever increasing fuel pressures.
2. Fatigue resistance, especially in regions of high stress.

Manufacturing Requirements

Nozzle production is a high volume operation which demands:

1. Good machinability
2. Appropriate heat treatment response to give the final properties.
3. Nozzle body holders and unit injector bodies can be either from bright bar or hot forged then machined.

A heat treatment process follows machining. Optimal processing of these components requires:

1. Good machinability with special reference to gun drilling of small holes
2. Minimum distortion on heat treatment

Material Requirements

For nozzles, Corus has developed a low carbon case Harden able steel fulfilling the essential requirements for:

1. Excellent machinability in the pre-heat treated, annealed condition.
2. High strength and fatigue resistance after heat treatment
3. Strength at elevated temperatures

Nozzle body holder and unit injector bodies have more demanding machinability requirements.

For These Applications Corus Provides:

1. Bars with exceptional straightness which are stress and defect free to avoid hole drift during gun drilling
2. Controlled resulphurised or leaded steels to further improve machinability.

In Addition, All Components Demand:

1. Clean steels for good fatigue resistance from a low. Overall inclusion content

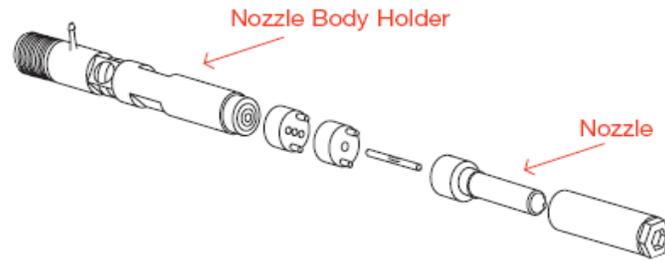


Figure 2 :Nozzle Body

Material Selection and Development

Diesel injection engineering is developing rapidly in response to the increasing service demands. Corus works closely with Manufacturers to develop steels that meet the needs for higher operating pressures, temperatures and reliability.

Nozzles: The 2Ni2Cr steel developed by Corus for injector nozzles has been tailored to meet the combined requirements of mach inability, strength, fatigue and high temperature durability. This steel is in use in the latest common rail systems, operating at pressures of up to 1800 bar and temperatures in excess of 200°C.

Nozzle Body Holders and Unit Injector Bodies

Medium carbon steels, sometimes resulphurised and / or leaded (e.g. C45Pb) are used for the less demanding applications where mach inability overrides fatigue considerations. Through hardening alloy steels (e.g. 42CrMo4) are needed for higher fuel Pressure systems where fatigue strength is more critical. On heat treatment these steels develop the required mechanical Strength and retain adequate mach inability. Special clean steel practices are necessary to minimize inclusion content and further Improve fatigue properties. The table 1 shows typical chemical compositions (wt %) for diesel injector steels.

Table 1: Chemical Compositions

Designation	C	Mn	S	Cr	Mo	Ni	Pb	Component
C45Pb	0.45	0.65	0.045	-	-	-	0.20	Nozzle body
42CrMo4	0.42	0.90	0.040	1.10	0.30	-	-	holder
2Ni2Cr	0.175	0.50	0.035	2.00	-	2.00	-	Nozzle

Material Properties

Consistency of Harden ability (Nozzle Bodies): Calculated from the chemical composition, the ideal Diameter (DI) is a recognized measure of harden ability. Using fully computerized control of steel making and models of Harden ability response, Corus' enhanced 42CrMo4 steel for Nozzle bodies achieves a 10mm range of DI.

Very clean steels have well dispersed and relatively few Inclusions. Oxygen content is an appropriate measure of Cleanness. Low oxygen levels are more difficult to achieve in Low carbon steels. Through close control of steel making Practice Corus' enhanced 2Ni2Cr nozzle steel consistently achieves oxygen levels below 10ppm. These are well below Industry standards of 12-15ppm and more comparable with the much higher carbon 1%CCr bearing steels for which fatigue resistance, and therefore cleanness, is paramount.

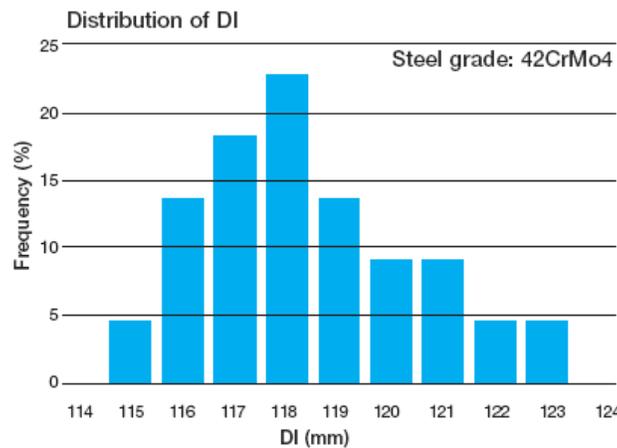


Figure 3: Consistency of Steel Cleanness (Nozzles)

Nozzle Calculations

Pressure Variation and Baseline Case: The typical geometry for representing a fuel injector nozzle has an L/D of about 4.0 and a sharp inlet corner. We have chosen this geometry as a starting point to allow for comparison to available experimental data. The inlet corner was not perfectly sharp, but rounded to one-fortieth of the Nozzle diameter to avoid a singularity in the transformation from physical to computational coordinates. Usually nozzle experiments are run with an atmospheric usually nozzle experiments are run with an atmospheric downstream pressure and a variable upstream pressure. We have performed calculations at upstream

pressures from 7.5 MPa up to 100 MPa. We have fixed the downstream pressure at 5 MPa in order to cover a wide range of the cavitations parameter K .

$$K = \frac{P_0 - P_v}{P_o - P_2} \tag{9}$$

The numerical results agree closely with correlation. The fact that the code accurately predicts the bulk mean velocity as measured by coefficient of discharge and the velocity at the centerline suggests that the exit velocity profile is essentially correct. An important feature of these two quantities is that the centerline velocity is substantially higher than the bulk mean velocity. This means that the exit velocity profile is not flat, as one would expect for fully developed turbulent pipe flow. The common uniform velocity assumption of nozzle exit flow appears to be incorrect for short cavitating nozzles.

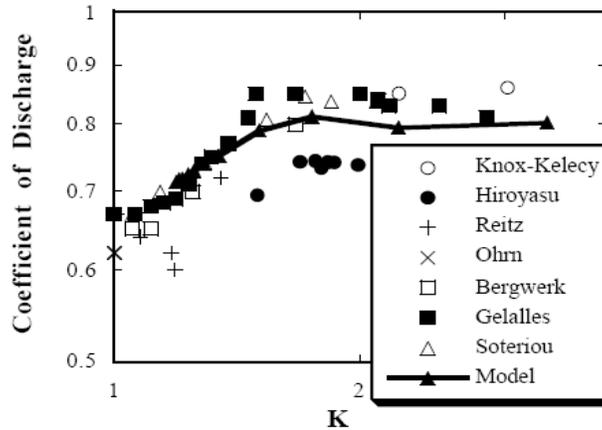


Figure 4: Predicted coefficient of discharge compared to experimental data. K is a cavitation parameter, defined in equation 9.

CORNER RADIUS: The baseline computation was performed with a very sharp inlet corner, typical of most

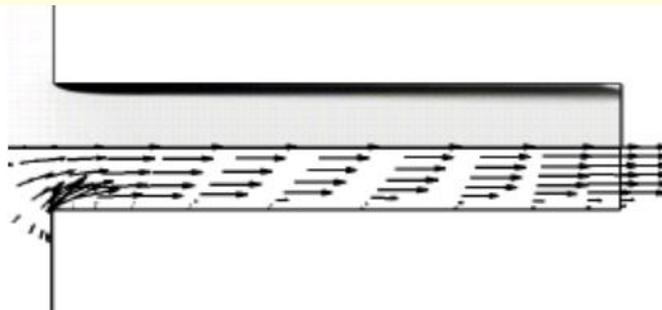


Figure 5: Baseline Case: Calculated density and momentum at 100 MPa upstream pressure, $r/D=1/40$, $L/D=4.0$.

Longer vectors denote higher momentum. Black shading represents vapor density and white is liquid density case. This computation is used as a baseline for comparing to other geometries. The upstream boundary is two diameters upstream of the corner and is not visible in this or the other flow field images. The calculation was performed on a 140 by 40 point grid, assuming axis-symmetry. The “mirror image” shown in these figures was created during post-processing. In the flow field images, only every 10th vector in the stream wise direction and every 5th vector in the transverse direction is shown. Instantaneous images were used to represent the results, since the flow eventually stabilized in all of the calculations. The calculations were run long enough for a sound wave to move from the inlet to the exit boundary and back thirty times.

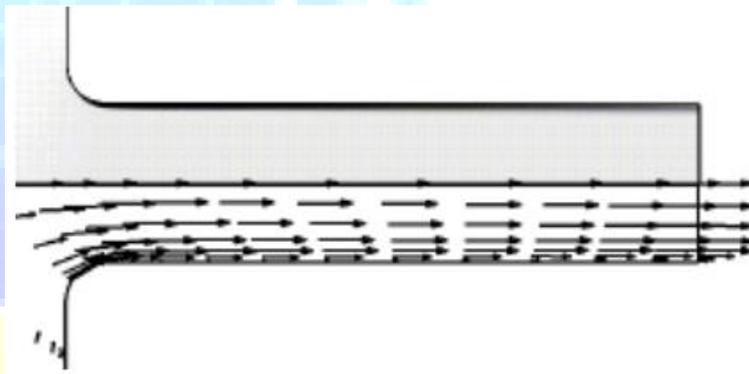


Figure 6: Flow through a nozzle with $r/D = 1/4$. Longer vectors denote higher momentum. Black shading represents vapor density and white is liquid density.

An instantaneous calculation of the flow through a rounded inlet is shown below in Figure 6. The computed cavitations region tended to become increasingly thinner as the inlet became more rounded. However the predicted region remained very long, continuing to reach to the outlet. The thinness of the bubble indicates a small degree of contraction. This reduced contraction is reflected in the coefficient of discharge.

Figure 7 clearly shows the high sensitivity of discharge to corner radius. Even the sharpest corner used for this study, $r/D=1/40$ still shows the effect of rounding. As can be seen in Figure 5, the corner appears quite sharp, but the coefficient of discharge is noticeably higher than the theoretical limit of 0.611. Additional rounding continues to increase the coefficient of discharge

for all the cases shown in Figure 6. Another important effect of the rounding can be seen by plotting the exit momentum profile. The plot in Figure 7 of ru for the baseline and $r/D=1/4$ case shows the effect of rounding on the exiting flow. As a reference velocity, we use the Bernoulli velocity. The exit flow is more uniform than in the baseline case and is issuing at a higher coefficient of discharge.

$$u_b = \sqrt{\frac{2\Delta p}{\rho}} \tag{10}$$

The fluid separates from the inlet corners and forms a vena contracta, which concentrates the momentum into the center of the nozzle. The fluid momentum profile at the exit still maintains this concentration, which appears as the plateau in Figure 7. Throughout this plateau, or “core”, the velocity is roughly equal to the theoretical maximum, the Bernoulli velocity.

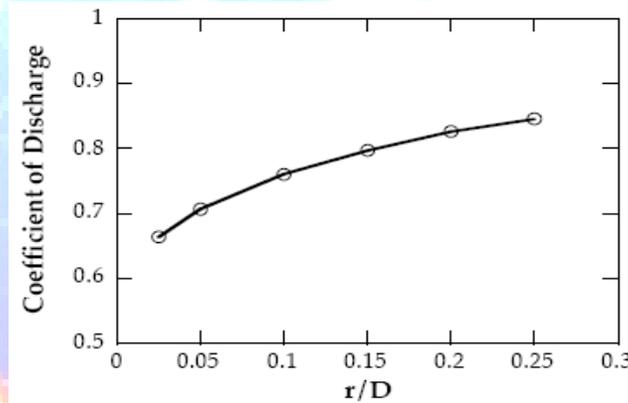


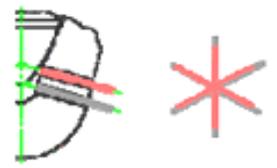
Figure 7: Coefficient of discharge versus the radius of the inlet corner

Investigated Nozzle Layout

Within the overall research project numerous multi layer nozzles with grouped nozzle holes were investigated. The chosen approach for layer assigned injection rate measurement and accompanying in nozzle flow simulation and optical validation shall be presented at the example of a multi layer nozzle with parallel nozzle holes. The main parameters of the investigated nozzle are shown in Table 2.

Table 2: Geometric data and layout of the investigated nozzle ADD NOZZLE INLET

Hole Number “Upper Row”	Cone Angle “Upper Row”	HFR “Upper Row” [cm 730s	Twist Angle	k-factor [*]	Overall HFR “Upper Row” [cm

[*]	[*]	@100 bar]	[*]		730s @100 bar]
3	153	105	0	1.5	370
Hole Number "lower Row" [*]	Cone Angle "lower Row" [*]	HFR "lower Row" [cm 730s @100 bar]			
3	153	185			

Injection Rate Measurement

The realized injection rate is one of the key input parameters for spray and overall combustion simulation. For the investigated multi layer nozzles, where the nozzle holes are arranged on an upper and a lower row, the overall injection rate is not sufficient to describe the nozzle behavior as it cannot be assumed that the injection rate is comparable for holes of the upper and lower row. Especially for nozzle concepts with varying hole-diameters for upper and lower row and during the needle opening phase a significant difference in injection rate can be expected. In order to analyze the impact of multi layer nozzle concepts on spray propagation, the lower chamber of the injection rate measurement tool is connected to an injection rate analyzer with high resolution data logging capabilities and a precise fuel mass measurement. A comparison of injected fuel mass via upper and lower nozzle layer is shown in Figure9.

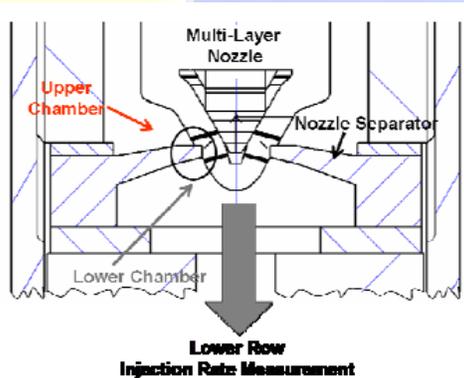


Figure 8: Tool for separate layer injection rate measurement

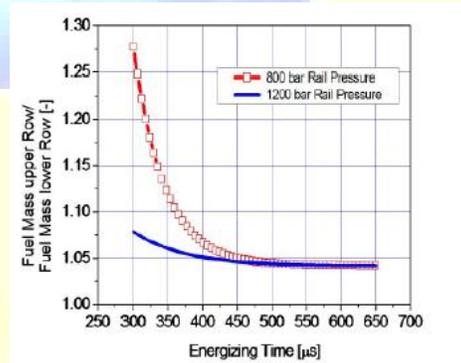


Figure 9: Ratio of Fuel Mass injected via upper and lower nozzle row

The separate injection quantity measurements for the upper and lower layer show that especially for low rail pressure levels, e.g. 800 bar and injection durations with energizing times less than 400 micro seconds the fuel quantity that is injected via the upper row is up to 25% higher in

comparison to the fuel quantity that leaves the lower layer. A measured injection rate profile for a typical pilot injection as shown in Figure 8 with an energizing time of 300 micro seconds and a rail pressure of 800 bar confirms the results of the fuel mass measurements

3D – CFD In Nozzle Flow Simulation

In order to better understand the experimental results a 3D CFD simulation model was created with Star CD. To reduce calculation time and meshing effort a sector mesh for one pair of nozzle holes was set up. Figure 11 shows the realized mesh.

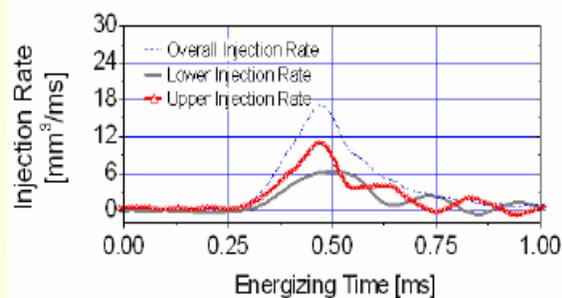


Figure 10: Injection Rate for upper and lower layer (800 bar, 300 micro seconds energizing time)

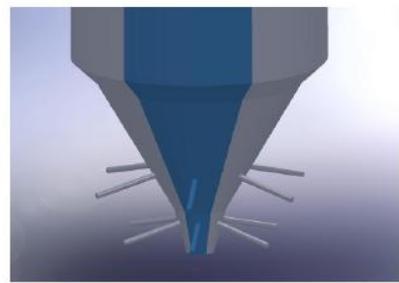


Figure 11: Sector mesh for 3D CFD Simulation Model

Steady state flow calculations were carried out for three different needle lift positions:

- 1) almost closed / slightly open
- 2) mid-open position
- 3) completely open

As especially for low rail pressures and short energizing times, differences in injection rate and subsequent injected fuel mass could be identified for upper and lower row, high emphasis was put on simulations with a mesh corresponding to needle lift position 1.

Pro/Engineer Wildfire Benefits:

- Unsurpassed geometry creation capabilities allow superior product differentiation and manufacturability

- Fully integrated applications allow you to develop everything from concept to manufacturing within one application
- Automatic propagation of design changes to all downstream deliverables allows you design with confidence
- Complete virtual simulation capabilities enable you to improve product performance and exceed product quality goals
- Automated generation of associative tooling design, assembly instructions, and machine code allow for maximum production efficiency

Pro/Engineer can be packaged in different versions to suit your needs, from Pro/Engineer Foundation XE, to Advanced XE Package and Enterprise XE Package, Pro/Engineer Foundation XE Package brings together a broad base of functionality. From robust part modeling to advanced surfacing, powerful assembly modeling and simulation, your needs will be met with this scalable solution. Flex3C and Flex Advantage Build on this base offering extended functionality of your choosing.

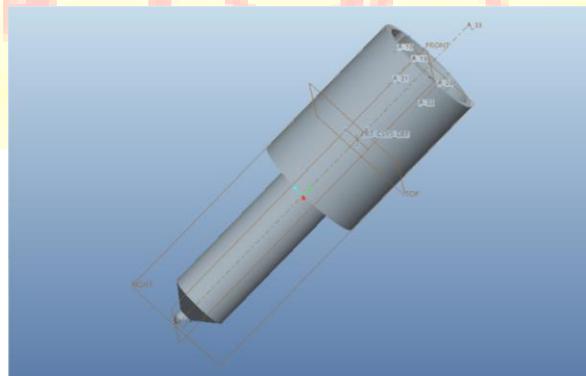
4. DIFFERENT MODULES IN PRO/ENGINEER

- Part Design
- Assembly
- Drawing
- Sheetmetal

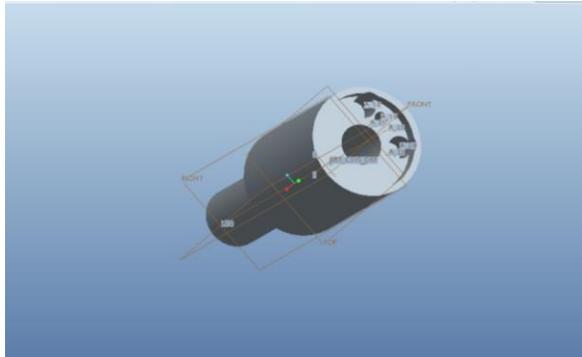
Nozzle Body



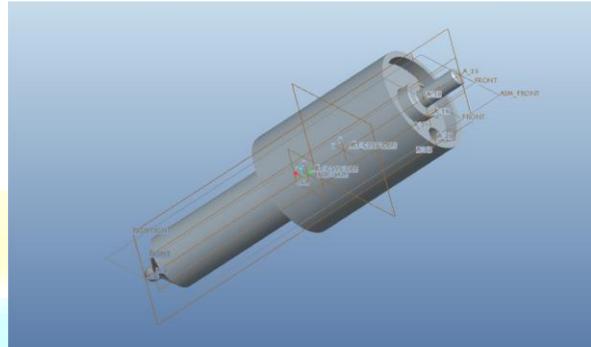
Nozzle Pin



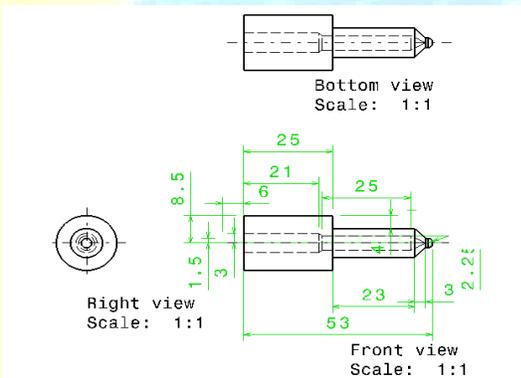
Assembly



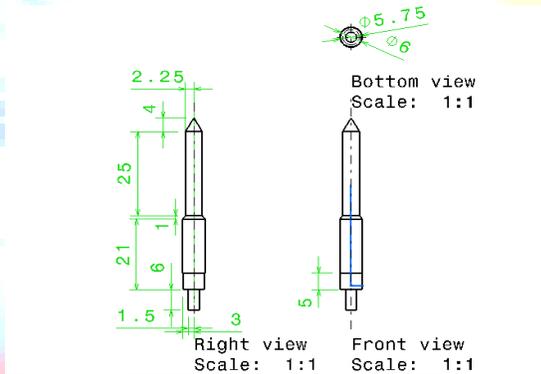
2D Drawings



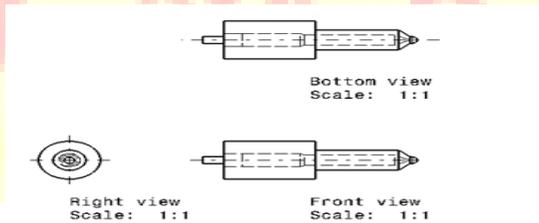
Nozzle Body



Nozzle Pin



Assembly



Number	Part Number	Quantity
1	Body	1
2	pin	1

5. GENERIC STEPS TO SOLVING ANY PROBLEM IN ANSYS

Like solving any problem analytically, you need to define (1) your solution domain, (2) the physical model, (3) boundary conditions and (4) the physical properties. You then solve the

problem and present the results. In numerical methods, the main difference is an extra step called mesh generation. This is the step that divides the complex model into small elements that become solvable in an otherwise too complex situation. Below describes the processes in terminology slightly more attune to the software.

Build Geometry

Construct a two or three-dimensional representation of the object to be modeled and tested using the work plane coordinate system within ANSYS.

Define Material Properties

Now that the part exists, define a library of the necessary materials that compose the object (or project) being modeled. This includes thermal and mechanical properties.

Generate Mesh

At this point ANSYS understands the makeup of the part. Now define how the modeled system should be broken down into finite pieces.

Apply Loads

Once the system is fully designed, the last task is to burden the system with constraints, such as physical loadings or boundary conditions.

Obtain Solution

This is actually a step, because ANSYS needs to understand within what state (steady state, transient... etc.) the problem must be solved.

Present the Results

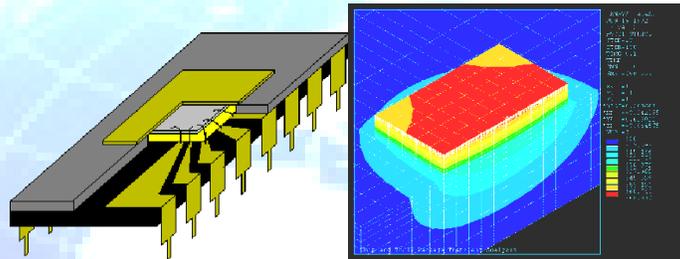
After the solution has been obtained, there are many ways to present ANSYS' results, choose from many options such as tables, graphs, and contour plots. Structural analysis is probably the most common application of the finite element method as it implies bridges and buildings, naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools.

- **Static Analysis** - Used to determine displacements, stresses, etc. under static loading conditions. ANSYS can compute both linear and nonlinear static analyses. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

- **Transient Dynamic Analysis** - Used to determine the response of a structure to arbitrarily time-varying loads. All nonlinearities mentioned under Static Analysis above are allowed.
- **Buckling Analysis** - Used to calculate the buckling loads and determine the buckling mode shape. Both linear (eigenvalue) buckling and nonlinear buckling analyses are possible.

In addition to the above analysis types, several special-purpose features are available such as Fracture mechanics, Composite material analysis, Fatigue, and both p-Method and Beam analyses.

6. THERMAL



ANSYS is capable of both steady state and transient analysis of any solid with thermal boundary conditions.

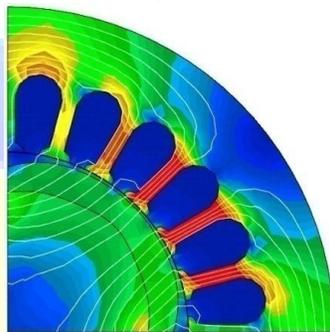
Steady-state thermal analyses calculate the effects of steady thermal loads on a system or component. Users often perform a steady-state analysis before doing a transient thermal analysis, to help establish initial conditions. A steady-state analysis also can be the last step of a transient thermal analysis; performed after all transient effects have diminished. ANSYS can be used to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. Such loads include the following:

- Convection
- Radiation
- Heat flow rates
- Heat fluxes (heat flow per unit area)
- Heat generation rates (heat flow per unit volume)
- Constant temperature boundaries

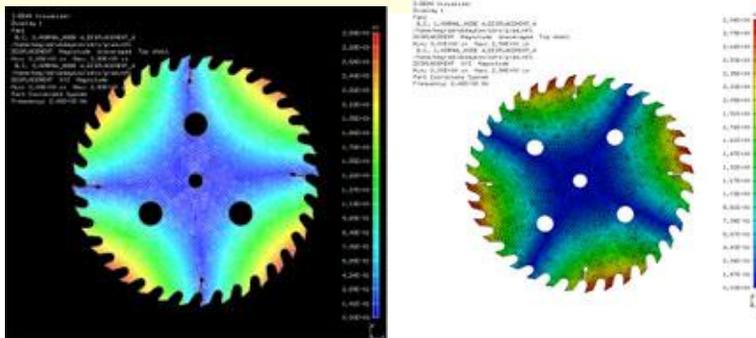
7. MAGNETIC

Magnetic analyses, available in the ANSYS/Multiphysics and ANSYS/Emag programs, calculate the magnetic field in devices such as:

- Power generators
- Magnetic tape/disk drives
- Transformers
- Waveguides
- Solenoid actuators
- Resonant cavities
- Electric motors
- Connectors
- Magnetic imaging systems
- Antenna radiation
- Video display device sensors
- Filters
- Cyclotrons



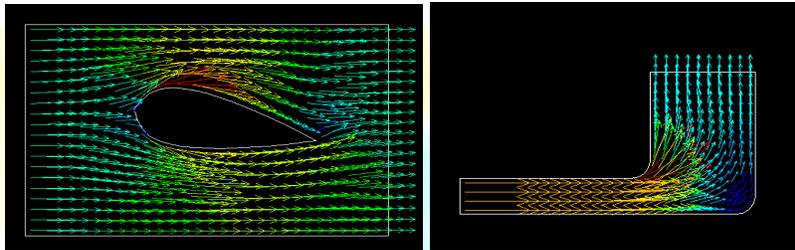
8. ACOUSTICS / VIBRATION



(Noise analysis and optimization of a Craftsman Table Saw Blade)

ANSYS is capable of modeling and analyzing vibrating systems in order to that vibrate in order to analyze

9. FLUID FLOW



The ANSYS/FLOTRAN CFD (Computational Fluid Dynamics) offers comprehensive tools for analyzing two-dimensional and three-dimensional fluid flow fields. ANSYS is capable of modeling a vast range of analysis types such as: airfoils 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.

FLOTRAN 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 FLOTRAN analysis that ANSYS is able to perform include:

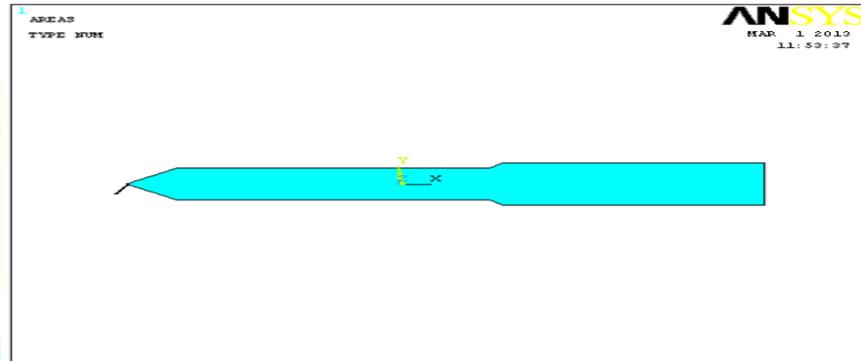
- Laminar or Turbulent Flows
- Thermal Fluid Analysis
- Adiabatic Conditions
- Free surface Flow
- Compressible or incompressible Flows
- Newtonian or Non-Newtonian Fluids

- Multiple species transport

10. CFD ANALYSIS OF NOZZLE

Main Menu > Preferences>FLOTTRAN CFD

Imported model



Element type

Main Menu > Preprocessor > Element Type > Add/Edit/Delete > ADD

select flotron cfd> 2d flotron 141

CLICK OK

Meshing

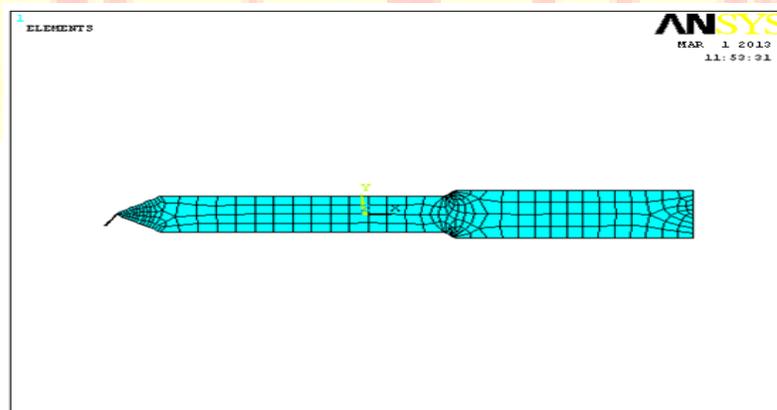
Main Menu> Preprocessor> Meshing> Mesh Tool

Select Smart size as a 6

Select mesh option from window

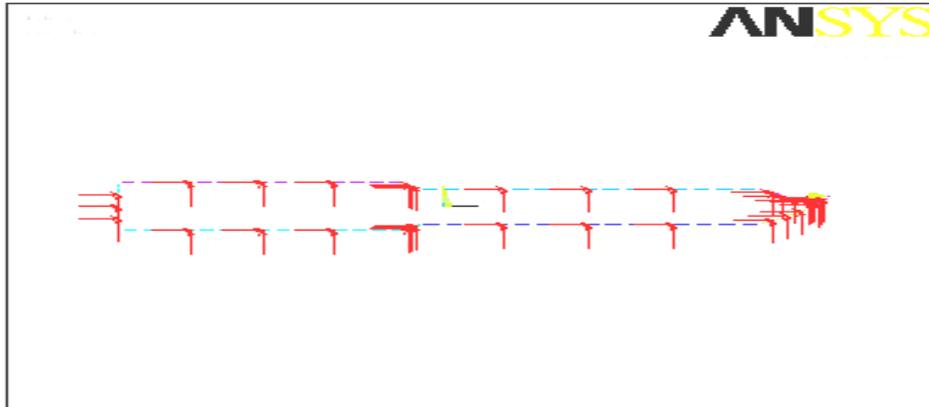
Select model

Meshed Model



Main Menu> Preprocessor> Loads > Define Loads> Apply> Fluid/CFD> Velocity> On
Lines

VELOCITY.....15.5 m/sec



Main Menu> Preprocessor> Loads> Define Loads> Apply> Fluid/CFD> Pressure DOF>
On Lines

Main Menu> Solution> FLOTRAN Set Up> Fluid Properties

NOMINAL VALUE_4

Main Menu> Solution> FLOTRAN Set Up> Execution Ctrl

Change reference conditions.

The reference pressure is changed from the default value of 101 KPa to maintain a consistent set of units. Likewise, the nominal stagnation and reference temperatures are changed from 293°K.

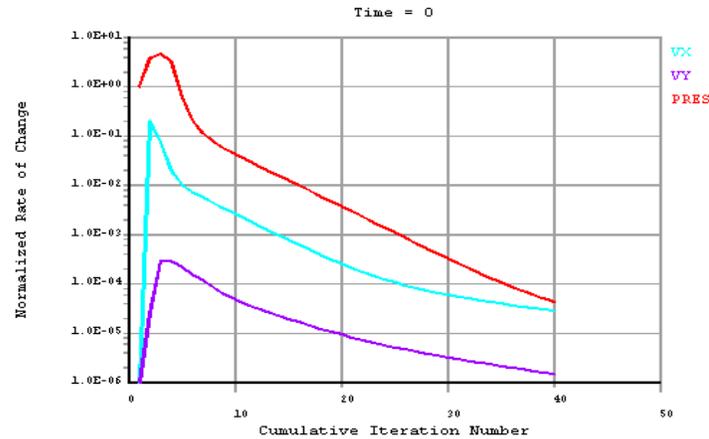
Main Menu> Solution> FLOTRAN Set Up> Flow Environment> Ref Conditions

Execute FLOTRAN solution.

Main Menu> Solution> Run

FLOTRAN

Close the information window when the solution is done.



Post processing

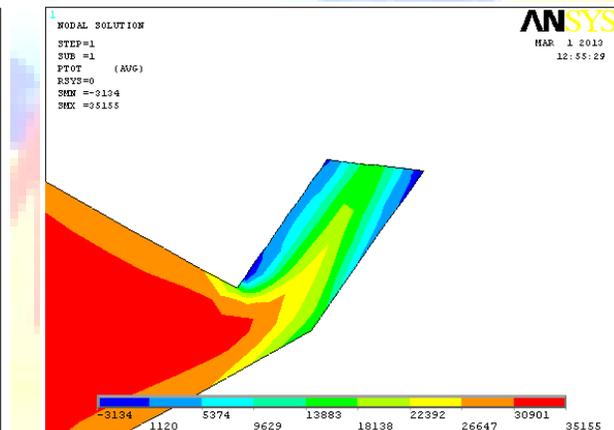
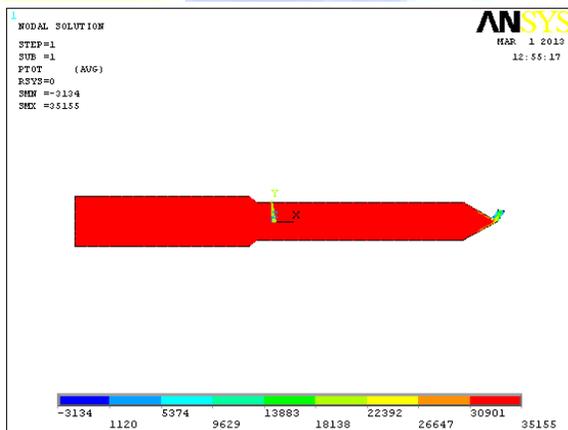
Read in the results for postprocessing.

Enter the general postprocessor and read in the latest set of solution results, and then create a vector plot.

Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu

Choose Other FLOTRAN Quantities

Choose Total Stagnation Pressure



Main Menu> General Postproc> Plot Results>Defi Trace Pt

OK

Main Menu> General Postproc> Path Operations> Define Path> By Nodes

Pick the lowest and then the highest point of the outlet

OK

Main Menu> General Postproc> Path Operations> Map onto Path

Enter VELOCITY as label.

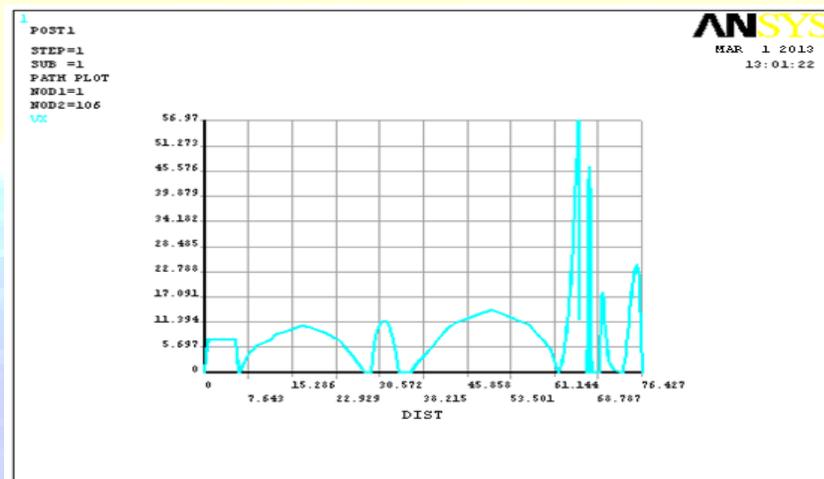
Choose DOF Solution.

1. Choose Velocity VX.
2. OK.

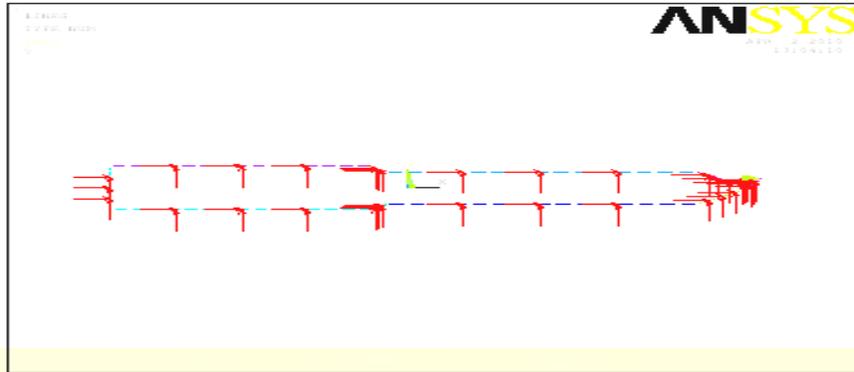
Main Menu> General Postproc> Path Operations> Plot Path Item> On Graph

Choose the label VELOCITY that you previously defined.

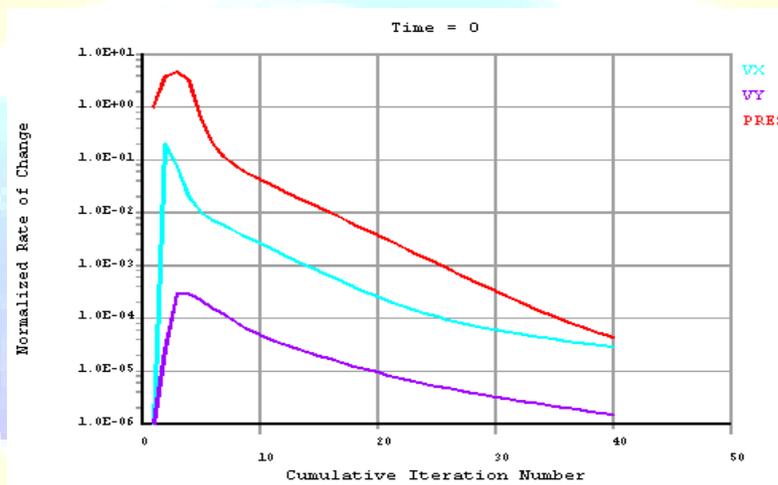
OK to create path plot.



1. Utility Menu> Plot> Lines
2. Main Menu> Solution> Define Loads> Apply> Fluid/CFD> Velocity> On Lines
3. Pick the inlet line (the vertical line at the far left).
4. OK (in picking menu)
5. Enter 50 for VX.
6. Enter 0 for VY.
7. OK.



Main Menu> Solution> Run FLOTRAN

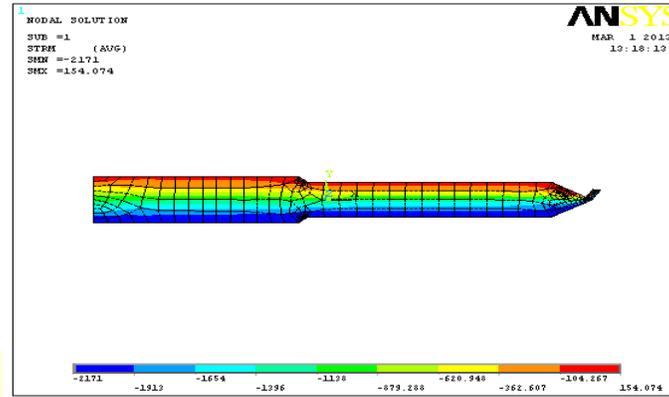


1. OK.
2. Main Menu> Solution> FLOTRAN Setup> Relax/Stab/Cap> MIR Stabilization
3. Enter 0.1 for the Momentum Equation.
4. OK.
5. Main Menu> Solution> Run FLOTRAN Once again, the Graphical Solution Tracker is displayed.
6. Close.

Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu

Choose Other FLOTRAN Quantities

Choose Total Stagnation Pressure



Main Menu> General Postproc> Plot Results>Defi Trace Pt

OK

Main Menu> General Postproc> Path Operations> Define Path> By Nodes

Pick the lowest and then the highest point of the outlet

OK

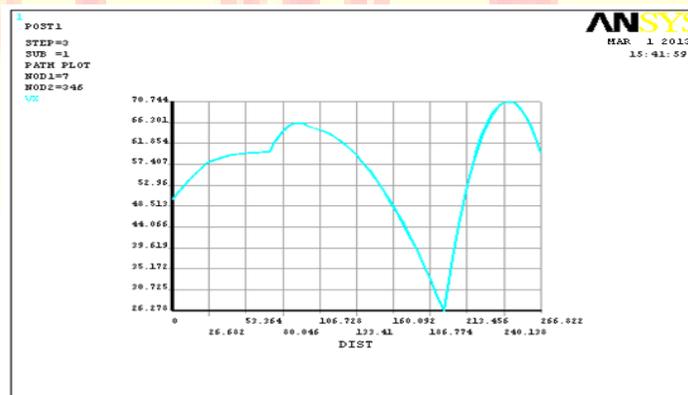
Main Menu> General Postproc> Path Operations> Map onto Path

1. Enter VELOCITY as label.
2. Choose DOF Solution.
3. Choose Velocity VX.
4. OK.

Main Menu> General Postproc> Path Operations> Plot Path Item> On Graph

Choose the label VELOCITY that you previously defined.

OK to create path plot.



CFD Analysis of by Reducing Diameter of Nozzle

Element type

Main Menu > Preprocessor > Element Type > Add/Edit/Delete > ADD

select flotron cfd> 2d flotron 141

CLICK OK

Meshing

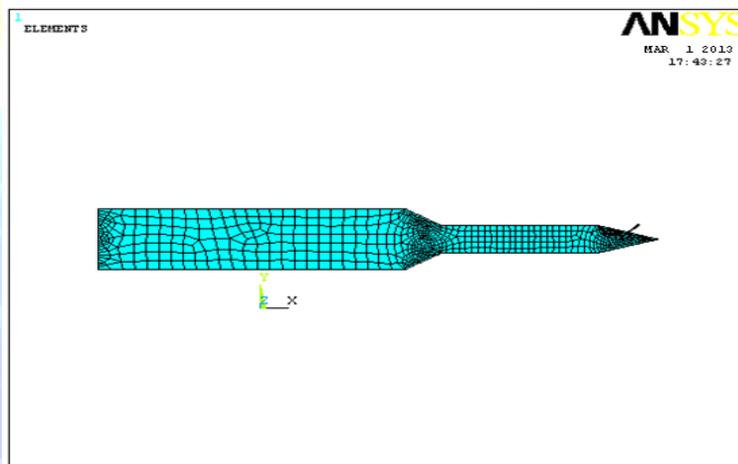
Main Menu> Preprocessor> Meshing> Mesh Tool

Select Smart size as a 6

Select mesh option from window

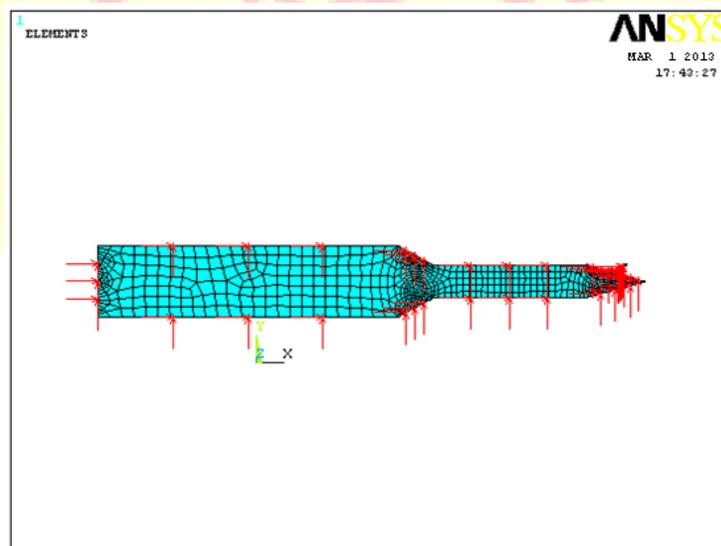
Select model

Meshed Model



Main Menu> Preprocessor> Loads > Define Loads> Apply> Fluid/CFD> Velocity> On Lines

PRESSURE.....50 m/sec



Main Menu> Preprocessor> Loads> Define Loads> Apply> Fluid/CFD> Pressure DOF> On Lines

Main Menu> Solution> FLOTRAN Set Up> Fluid Properties

NOMINAL VALUE_4

Main Menu> Solution> FLOTRAN Set Up> Execution Ctrl

Change reference conditions.

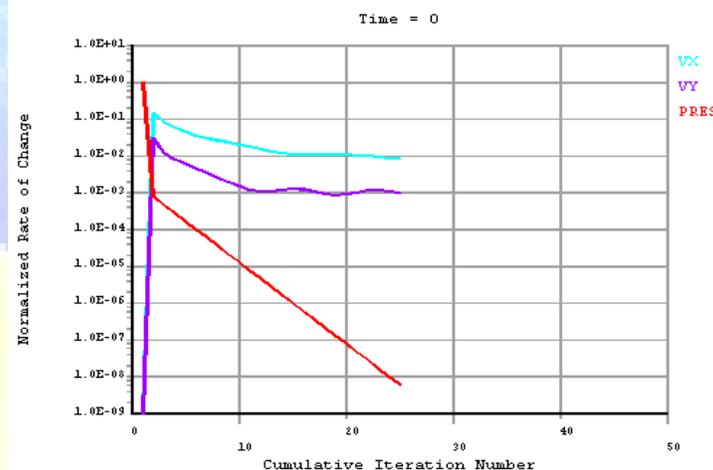
The reference pressure is changed from the default value of 101 KPa to maintain a consistent set of units. Likewise, the nominal stagnation and reference temperatures are changed from 293°K.

Main Menu> Solution> FLOTRAN Set Up> Flow Environment> Ref Conditions

Execute FLOTRAN solution.

Main Menu> Solution> Run FLOTRAN

Close the information window when the solution is done.



Post processing

Read in the results for post processing

Enter the general postprocessor and read in the latest set of solution results, and then create a vector plot.

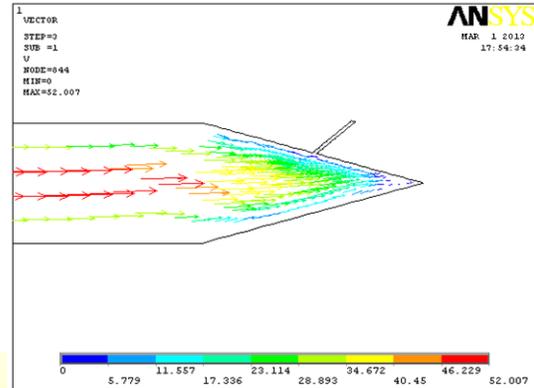
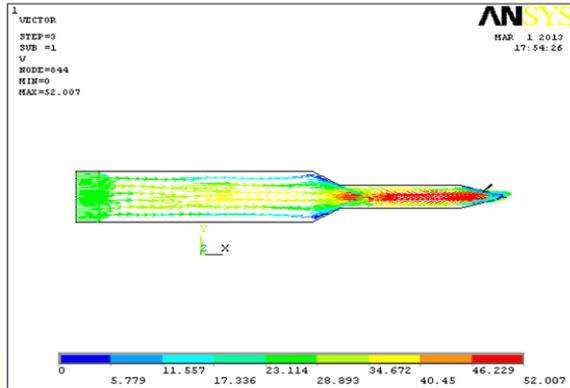
Main Menu> General Postproc> Read Results> Last set.

Main Menu> General Postproc> Plot Results> Vector Plot> Predefined

Choose DOF solution.

Choose Velocity V.

OK.

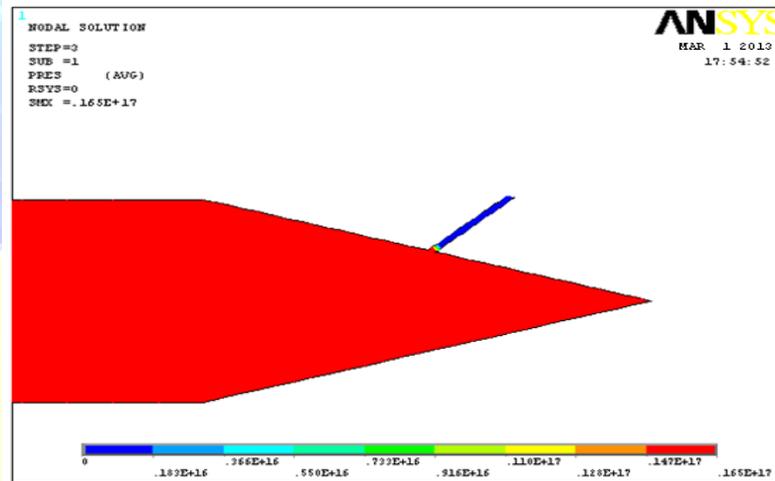


Plot total pressure contours.

Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solution

Choose Other FLOTRAN Quantities

Choose Total Stagnation Pressure



Main Menu> General Postproc> Plot Results>Defi Trace Pt

OK

Main Menu> General Postproc> Path Operations> Define Path> By Nodes

Pick the lowest and then the highest point of the outlet

OK

Main Menu> General Postproc> Path Operations> Map onto Path

1. Enter VELOCITY as label.
2. Choose DOF Solution.
3. Choose Velocity VX.

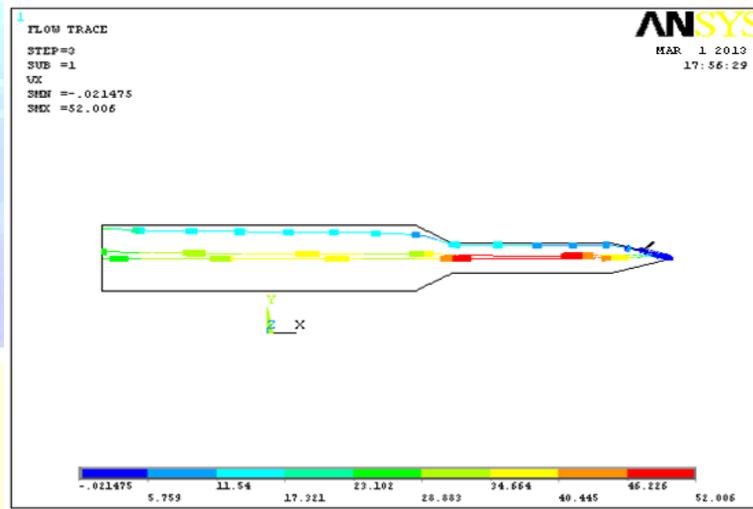
4. OK.

Main Menu> General Postproc> Path Operations> Plot Path Item> On Graph

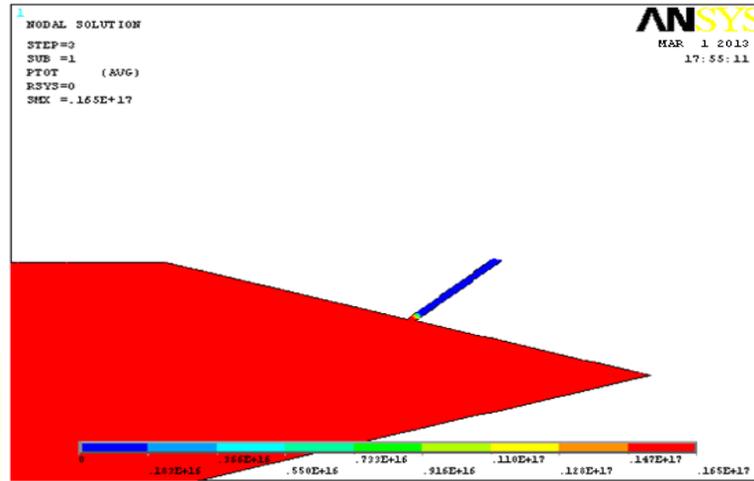
Choose the label VELOCITY that you previously defined.

OK to create path plot.

1. Utility Menu> Plot> Lines
2. Main Menu> Solution> Define Loads> Apply> Fluid/CFD> Velocity> On Lines
3. Pick the inlet line (the vertical line at the far left).
4. OK (in picking menu)
5. Enter 50 for VX.
6. Enter 0 for VY.
7. OK.



1. Main Menu> Solution> Run FLOTRAN
2. OK.
3. Main Menu> Solution> FLOTRAN Setup> Relax/Stab/Cap> MIR Stabilization
4. Enter 0.1 for the Momentum Equation.
5. OK.
6. Main Menu> Solution> Run FLOTRAN Once again, the Graphical Solution Tracker is displayed.
7. Close.



Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solution

Choose Other FLOTRAN Quantities

Choose Total Stagnation Pressure

Main Menu> General Postproc> Plot Results> Defi Trace Pt

OK

Main Menu> General Postproc> Path Operations> Define Path> By Nodes

Pick the lowest and then the highest point of the outlet

OK

Main Menu> General Postproc> Path Operations> Map onto Path

1. Enter VELOCITY as label
2. Choose DOF Solution.
3. Choose Velocity VX
4. OK.

Main Menu> General Postproc> Path Operations> Plot Path Item> On Graph

Choose label VELOCITY that you previously defined.

OK to create path plot.

11. CONCLUSIONS

In this paper, we have designed and modeled 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, subzero 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 97.027m/sec and stagnation pressure is 35155N/m² when the inlet velocity is 15.5m/sec. The maximum velocity developed in the nozzle is 52.007m/sec and stagnation pressure is 0.165e¹⁷ N/m² when the inlet velocity is 50m/sec.

REFERENCES

1. Handbook of Die Design by Ivana Suchy
2. Machine Design by R.S. Khurmi and J.K. Gupta, Eurasia Publishing House (P) Ltd., New Delhi
3. Pro/Engineer WildFire 2.0 by Steven G. Smith
4. Product Design and Manufacturing by R.C. Gupta and A.K. Chaitle
5. Theory of Machines by P.L. Ballney, Khanna Publishing, New Delhi.
6. Theory of Machines by R.S. Khurmi and J.K. Gupta, S.Chand & Co., New Delhi.
7. Theory of Machines by S.S. Rattan, Tata McGraw Hill, New Delhi.