CFD Analysis of Variable Helical Inlet Port for a Six Cylinder Engine

Mahalakshmaiah Vaddani[1] Dr.V.Krishnareddy[2]
[1] M.tech student, [2]Professor, dept of mechanical engineering, kits, markapur, A.P, INDIA

ABSTRACT

The helix angle at inlet port decides the spread. For improving the spread we are going to perform CFD analysis on ports having different helix angle. This is the Port simulation project. All the works are carried out through simulation software such as NX-CAD & ANSYS. Improving the efficiency of the Six Cylinder Engine by Redesign of the Port called Helical Port. Due to this swirl motion of air inside the cylinder takes place. So that relative velocity between air and fuel is increased and get a rich mixture for combustion to increase efficiency. The Port will be modeled in NX-CAD Software, and the flow characteristics can be determined by CFD Analysis using Flotran 142 element in a module in ANSYS Software.

Key words: six cylinder engine, C.F.D.Analasys,NX-CAD,flotran142in Ansys

INTRODUCTION

The combustion where take place at the inside of the cylinder. Water is used as a coolant. This Six Cylinder Hino engine is mainly used in Trucks / Trailers for moving Cargo containers from shipyard to the customer place. This Six Cylinder engine is used in CHEETAH-231 a truck manufactured by M/s. ASHOK LEYLAND, Ennore.

The efficiency of the engine is dependent upon the complete mixing of air and fuel inside the cylinder. This complete mixing can be achieved by creating relative velocity between the fuel and air inside the cylinder. For that we have to give turbulence motion to the air inside the cylinder.

METHODOLOGY

Project can be distributed in to two PHASES

PHASE-ONE

- The existing straight inlet port will be modeled in NX-CAD.
- CFD Analysis will be performed on the existing port to simulate the flow in to the block.

PHASE-TWO

- Redesign of existing straight port to helical port.
- CFD Analysis will be performed on helical port to find spread into the Engine Block.
- Port wall will be modeling in NX-CAD

Modeling Part:

The Path will be taken as a Trajectory. It is a 3D curve drawn by two projection method using curve option. In the two projection method the front view of the 3D curve can be drawn on the front plane and top view of the 3D curve can be drawn on the Top plane finally getting the 3D curve.

Along the 3D curve the points will be created wherever we need the different cross section using offset coordinate system in datum points option from the datum features.

SWEEP:

A sweep is created by two sections. The first session is the trajectory and the second is the cross section. Trajectory is the path along which the cross section is swept but the cross section should be uniform.

BLENDS:

A blended feature consists of a series of at least two planar sections which joints together at the edges with transitional surface to form a continuous feature.

Swept Blend:

Trajectory is a path along which the different or non uniform cross section can be swept among above three since using swept blend option in this project for creating the solid model the 3D curve taken as a trajectory.

The different cross section drawn along the trajectory where to corresponding points can be created. Finally getting the solid model of straight inlet port.

FLOTRAN CFD ANALYSIS

The ANSYS FLOTRAN derived product and the FLOTRAN CFD (Computational Fluid Dynamics) option to the other ANSYS products offer you comprehensive tools for analyzing two-dimensional and three-dimensional fluid flow fields. Using either product and the FLOTRAN CFD elements FLUID141 and FLUID142, you can achieve solutions for the following:

a) Lift and drag on an airfoil
b) The flow in supersonic nozzles Complex,
c) three-dimensional flow patterns in a pipe bend
d) In addition, you can use the features of ANSYS and ANSYS FLOTRAN to perform tasks including:
e) Calculating the gas pressure and temperature distributions in an engine exhaust manifold
f) Studying the thermal stratification and breakup in piping systems
g) Using flow mixing studies to evaluate potential for thermal shock
h) Doing natural convection analyses to evaluate the thermal performance of chips in electronic enclosures
i) Conducting heat exchanger studies involving different fluids separated by solid regions

3-D MODEL OF HELICAL PORT

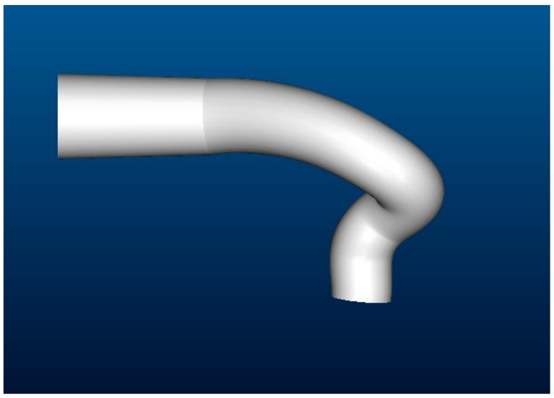

Fig 1 Solid model of the port of angle 160

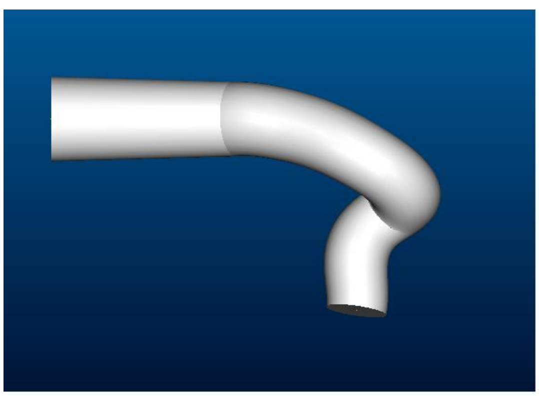

Fig 2 Solid model of the port of angle 220

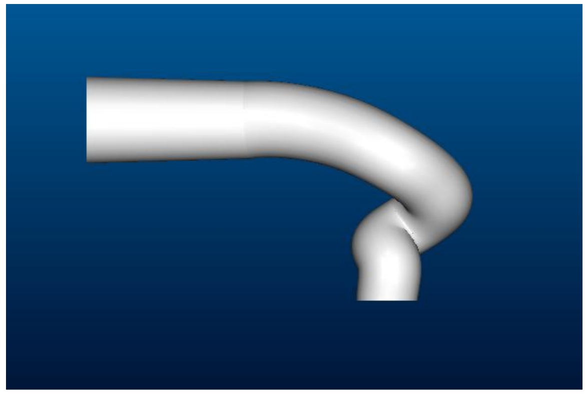

Fig 3 Solid model of the port of angle 300

MODEL CALCULATION
PRESSURE CALCULATION

Engine has a **20 hp** for a single cylinder.

Engine speed (N) = 2400 rpm

Engine power in watt = $\frac{(20 \text{ x } 2400)}{.75}$ = 64000 watt

$$B H P = \frac{2 \pi N T}{60}$$

$$T = \frac{B H P \text{ x } 60}{2 \pi N}$$

$$= \frac{64000 \text{ x } 60}{2 \pi \text{ x } 1000}$$

$$= 611.4 \text{ n-m}$$

Force = torque x distance

= 611.4 x .08

= 48.91

Pressure = force/ area

$= 48.1 / (\pi/4 \text{ x } (.105)^2)$

$= 5651 \text{ n/m}^2$

PORT INLET VELOCITY CALCULATION

Port inlet dia (d) = 40 mm = .04 m Crank radius (R) = 80 mm = .08 m

$\omega = 2 \pi N / 60$

$= (2 \text{ x } \pi \text{ x } 2400) / 60$

$= 251.32$

Crank velocity $(V) = R\omega$

$= .08 \times 251.32$

$= 20.1$ m /s

$AV = av$

$\pi \times (.16)^2 \times 20.1 = \pi \times (.04)^2 \times v$

Velocity at inlet of the port $(v) = 321.6$ m / s

CFD ANALYSIS OF THE STRAIGHT PORT

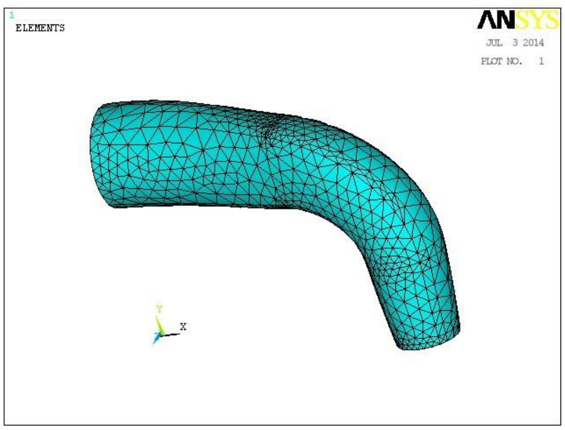

Fig.4 Meshed model of the straight port

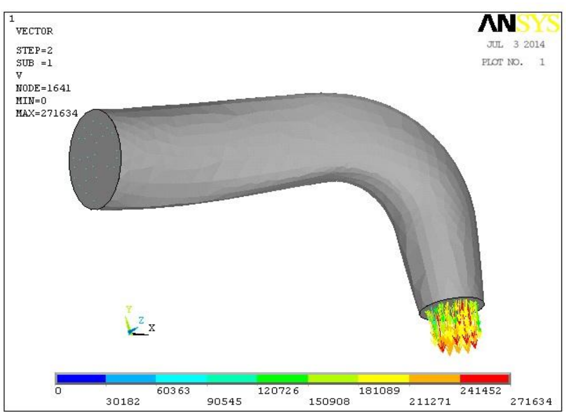

Fig.5 Vector distribution of the port

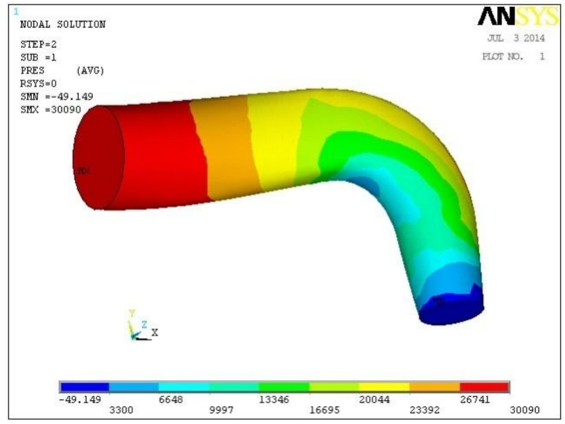

Fig.6 Pressure distribution 0f the port

SECOND PHASE OF THE PROJECT

Here we have to check which of the above mentioned Helix angles gives the better results. So here we have to find out the Flow Characteristics of each angle using CFD through ANSYS Software. Here am considering 3 angles for CFD Analysis. They are of angles 160,220,300.

During Second Phase, the port will be modified into helical port for getting swirl motion of air inside the cylinder. The modification can be based on the helix angles preferred for various configurations. A Parametric design will also be modeled where a variable will be governing the helix angle

In this we are considering the inlet port of angle 160 to find out the flow characteristic analysis using CFD. After the completion of the CFD Analysis of Straight Port we are going

Boundary Conditions are applied. After that the Velocity, Vector, Pressure Distributions are calculated.

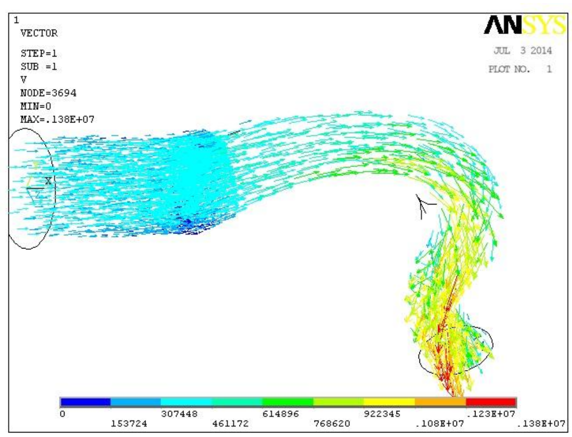

Fig7 Vector distribution of the inlet angle of 160

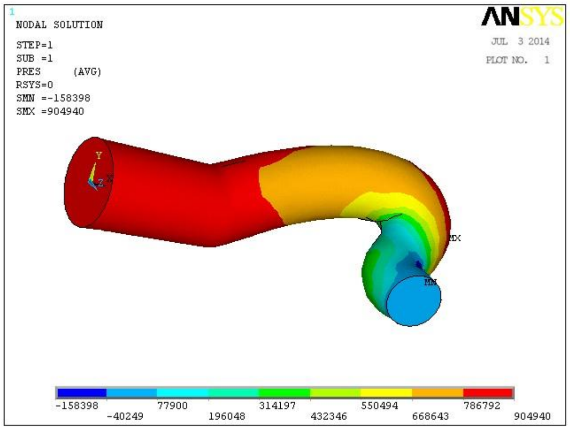

Fig8 Pressure distribution of the inlet port of angle 160

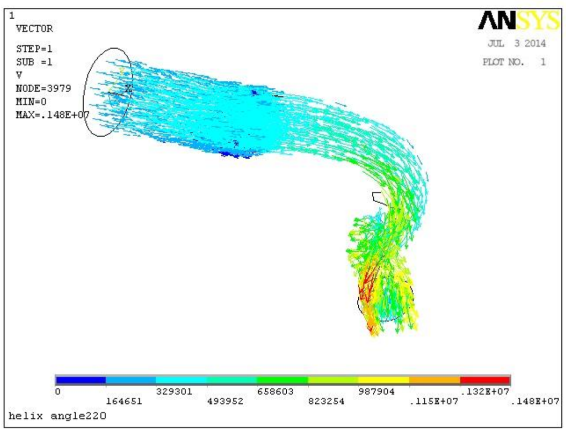

Fig9 Vector distribution of the inlet angle of 220

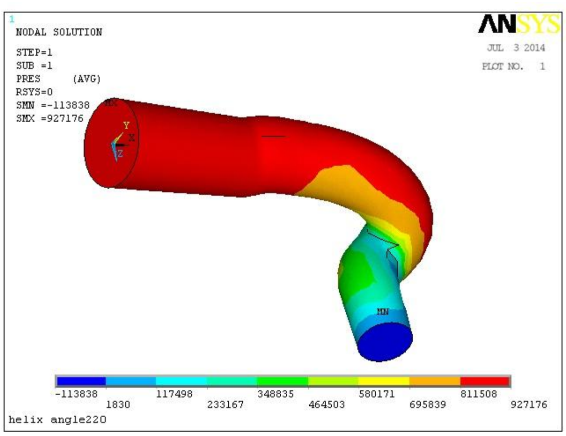

Fig10 Pressure distribution of the inlet port of angle 220

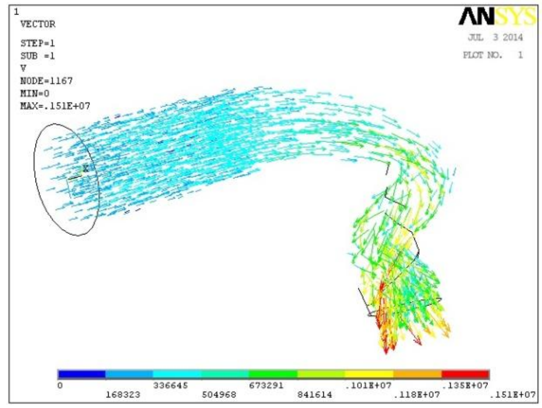

Fig11 Vector distribution of the inlet angle of 300

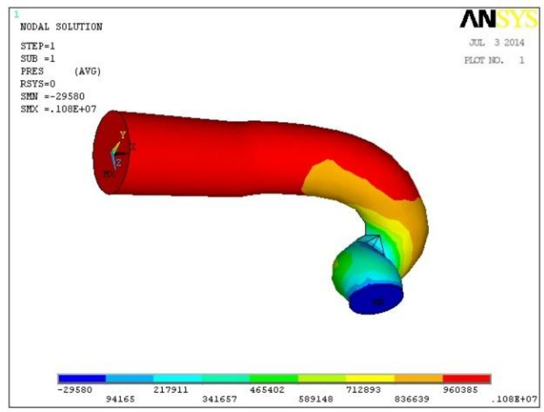

Fig12 Pressure distribution of the inlet port of angle 300

RESULTS AND DISCUSSIONS

SWIRL RATIO CALULATIONS:

For straight port,

$Vx = 5608.48$

$Vy = 1.941e5$

$V2 – 4.1217e4$

Swirl ratio, SR = Square root of

$$\left\{ \frac{Vx^2 + Vy^2}{Vz^2} \right\}$$

Square root of

$$\left\{ \frac{(5608.48)^2 + (1.941e5)^2}{(4.1217e4)^2} \right\}$$

SR = 0.047

For 160 Angle Helical Port,

Vx = 7.58 e4

Vy = 251220.1

Vz = 8.6e5

SR = 0.30512

For 220 Angle Helical Port,

Vx = 5.83e3

Vy = 2.98e4

Vz = 8.51e5

SR = 0.03568

For 300 Angle Helical Port,

Vx = 8.65e4

Vy = 2.39e5

Vz = 7.31e5

SR = 0.34770

PORTS	VELOCITY (OUTLET) MAX.	PRESSURE (INLET) MAX.	SWIRL RATIO SR
STRAIGHT PORT	245229	30090	0.047
PORT OF ANGLE 160	0.130E+07	904940	0.30512
PORT OF ANGLE 220	0.130E+07	927176	0.03568
PORT OF ANGLE 300	0.143E+07	0.108E+07	0.34770

Table 1 Table showing max. velocity (outlet), max. pressure (inlet) & swirl ratio.

CONLUSION

This project titled "CFD ANALYSIS OF VARIABLE HELICAL INLET PORT FOR A SIX CYLINDER ENGINE" deals with the CFD Analysis in order to find out the flow characteristics for different Helical Angles and suggest the best one among them. This analysis is done in order to increase the Engine Efficiency & to reduce Fuel Consumption.

This project is mainly based on Analysis part of the inlet port. At first the existing straight port is modeled & analyzed. And then different Helical Angles have to be taken and the analysis part is carried out

After doing the Swirl Ratio calculations the best flow is the Helical Angle 300. Its Swirl Ratio is 0.34470. And the Swirl Ratio for Straight Port is 0.047. So compared to straight port, helical angle of 300 gives the good flow.

REFERENCES

1. Chen & J. C. Dent (1994) AN INVESTIGATION OF STEADY FLOW THROUGH A CURVED INLET PORT. SAE paper volume II 950818.
2. Dr.Stanley K. Widener (1995) PRAMETRIC DESIGN OF HELICAL INTAKE PORT. SAE paper 950818
3. H.Jasak, J.Y.Luo, B.Kaludercic & A.D.Gosman (1999) RAPID CFD SIMULATION OF INTERNAL COMBUSTION ENGINES. SAE paper 1999-01-1185.
4. Jun-ichi kawashima, Hiroshi Ogawa and yoshiyuki Tsuruhave (1998). RESEARCH ON A VARIABLE SWIRL INTAKE PORT FOR 4-VALVE I-SPEED D1 DIESEL ENGINES. SAE Paper 982680.

5. William Church and P.V.Farrell (1998) EFFECT OF INTAKE PORT GEOMETRY ON LARGE SCALE IN – CYLINDER FLOWS. SAE paper 980484.
6. www.autoindustry.com
7. www.cfdonline.com
8. www.eupen.com
9. www.ibh.co.uk/glossary.htm