

CFD Analysis Study on Suction-Passage Flow of Hydrogen Reciprocating Compressor

**Gyeonghwan Lee¹, M. Sq. Rahman¹, Kyujin Shim², Chungseub Yi³,
Hanshik Chung⁴ and Hyomin Jeong⁴**

¹Graduate School, Department of Mechanical and Precision Engineering,
Gyeongsang National University, Tongyeong, KOREA

²SINTEF Fisheries and Aquaculture, 7465 Trondheim, NORWAY

³Samwon Tech co.Ltd., Gimhae, KOREA

⁴Department of Mechanical and Precision Engineering, Gyeongsang National
University, Institute of Marine Science, Tongyeong, KOREA

ABSTRACT: Numerical analysis information is very useful to improve fluid system. General information about an internal gas flow is presented by numerical analysis approach. In hydrogen compressing system, which has an important role in hydrogen energy utilization, CFD can be a powerful tool to observe the flow quickly and clearly. Flow characteristic analysis, including pressure and turbulence kinetic energy distribution of hydrogen gas coming to the cylinder of a reciprocating compressor are presented in this paper. Suction-passage model is designed based on real model of hydrogen compressor. Pressure boundary conditions are applied considering the real condition of operating system. The result shows pressure and turbulence kinetic energy are not distributed uniformly along the passage of the hydrogen system. Path line or particles tracks help to demonstrate flow characteristics inside the passage. The existence of vortices and flow direction can be clearly found. Based on this result, the design improvement, such as reducing the varying flow parameters and flow reorientation could be done. Consequently, development of the better hydrogen compressing system will be achieved.

Keywords: CFD, Hydrogen Reciprocating Compressing System, Suction-Passage Flow, Pressure Distribution, Turbulence Kinetic Energy

NOMENCLATURE

g_m gravitational acceleration component
[m/s²]
 p piezometric pressure = $p_s - p_0$ $g_m x_m$ [Pa]
 p_s static pressure [Pa]
 s_i momentum source component [kg/m²s³]
 s_m mass source component [kg/m³s]
 t time [s]
 u_i absolute fluid velocity component in
direction x_i [m/s]

x_i cartesian coordinate ($i = 1, 2, 3$)
 x_m coordinate relative to datum where p_0 is
defined
 ρ density [kg/m³]
 ρ_0 reference density [kg/m³]
 δ_{ij} kronecker delta, is unity when $i = j$ and
zero otherwise.
 τ_{ij} stress tensor component [Pa]

1. INTRODUCTION

Hydrogen can be produced from a variety of sources, including fossil fuels; renewable sources such as wind, solar, or biomass; nuclear or solar heat-powered thermo chemical reactions; and solar photolysis or biological method [1].

It is considered to be a prime fuel in supply and security, transition to hydrogen economy, environmental betterment, and social, societal, technological, industrial, economical and governmental sustain abilities in a country. Thus, green energy based hydrogen system can be one of the best solutions for accelerating and ensuring global stability and sustainability. Therefore[,] the production of hydrogen from non-fossil fuel sources and the development and application of green energy technologies become crucial in this century for better transition to hydrogen economy [2].

The assertion of “hydrogen is considered a promising future fuel for vehicles” is based on three main arguments: the potential reducing greenhouse gases from the transport sector, greater energy supply security, i.e. hydrogen can be produced from many energy sources and hence the risk of shortage of supply may be reduced; the potential of zero local emissions with the use of fuel cells.

The absence of hydrogen infrastructure is seen as major obstacle to the introduction of hydrogen FCVs. A full scale hydrogen infrastructure with production facilities, a distribution network and refuelling stations is costly to build. The venture of constructing a hydrogen refuelling infrastructure constitutes a long-term, capital-intensive investment with great market uncertainties for FCVs. Therefore, reducing the financial risk is major objective of any long-term goal to build a hydrogen infrastructure [3].

The relative simplicity of vehicle design for the hydrogen fuel cell vehicle must be weighed against the added complexity and cost of developing a hydrogen refuelling and infrastructure. Unlike gasoline and natural gas, hydrogen is not widely distributed to

consumers today, and refuelling a large number of hydrogen vehicles poses significant challenges. The question is often asked, ‘Where is the hydrogen going to come from?’ Indeed, hydrogen infrastructure is sometimes perceived as a ‘show-stopper’ for hydrogen vehicles [4].

One of the most important processes in the hydrogen gas handling is the compressing system. This process is essential in all step of hydrogen gas energy utilization: production, storage, distribution until using. In addition since hydrogen will replace the role of fossil fuel, so all approach done in this field should be rewarded. This paper presents the study about gas flow characteristics in suction passage of hydrogen reciprocating compressor.

2. METHODOLOGY

2.1 Reciprocating Compressor

Reciprocating compressor is one of the most popular machines which are used in industry. The effective and accurate diagnosis of possible faults which degrade compressor performances are required to help in both reducing maintenance costs and increasing the plant efficiency [5].

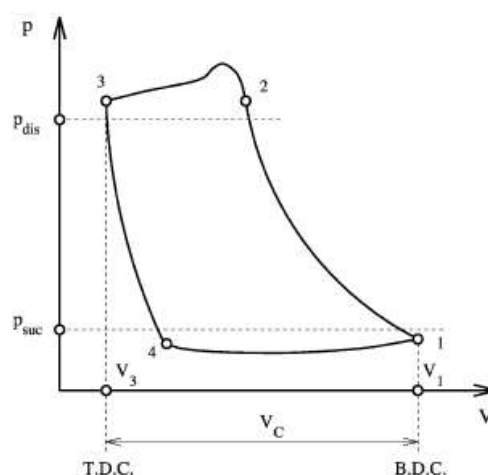


Figure 1: Indicator diagram of a reciprocating compressor.

Figure 1 presents a schematic view of a reciprocating compressor and the indicator diagram for a typical cycle. When the piston moves downwards, it reaches a position

where low pressured vapor is drawn in through the suction valve, which is opened automatically by the pressure difference between the cylinder and the suction chamber. The vapor keeps flowing during the suction stroke as the piston moves towards the bottom dead center (BDC), filling the cylinder volume with vapor at suction pressure, p_{suc} . The suction process is represented by curve 3-1 in the indicator diagram. After reaching the BDC, the piston starts to move in the opposite direction, the suction valve is closed, the vapor is trapped, and its pressure rises as the cylinder volume decreases. Eventually, the pressure reaches the pressure in the discharge chamber, p_{dis} , and the discharge valve is forced to open. After the opening of the discharge valve, the piston keeps moving towards the top dead center (TDC), represented by point A. It should be noted that suction and discharge processes do not take place at constant pressure. This phenomenon is associated with the dynamics of the valves and the restriction imposed by the valve passage areas. This appears on the indicator diagram, with compression continuing after pressure p_{dis} is reached and [again] the same [is] happening for the expansion stroke after pressure p_{suc} [6].

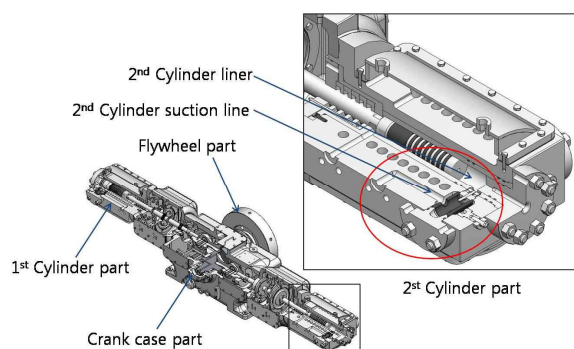


Figure 2: 2nd suction passage in the reciprocating compressor.

In industrial application, hydrogen gas compressing system is frequently used as the transferring or storing force device. It needs to be in high pressure condition; therefore high working compressors are required. For

this purpose, usually reciprocating type is used. Some aspects affecting performance of the compressor occur in the suction passage. Since this is the gateway of the gas before entering the cylinder to be compressed. Here the gas is firstly conditioned by means of its structure.

2.2 Mathematical Model

The necessary reduction in the computational cost is sought here through two approaches. The simulation methodology combines differential and integral formulations for the governing equations. Differential equations are solved to capture flow details during the opening of the suction valve, which are crucial to correctly predict its dynamics. The flow through the suction valve could be solved in the same way but, following Matos *et al* [7-10].

Eddy viscosity turbulence models are based on the analogy between the molecular gradient-diffusion process and turbulent motion (Boussinesq model). The Reynolds stresses and turbulent scalar fluxes in these models are directly linked to the local gradients of the mean flow field through a turbulent viscosity determined by a characteristic turbulence velocity scale and length scale are very numerous, ranging from prescribed profiles to the profiles to the popular two-equation models. Special models are also employed to characterize the flow near walls.

$k - \epsilon$ turbulence model comprising transport equations for the turbulence kinetic energy k and its dissipation rate ϵ . In these models, k and ϵ are chosen as typical turbulence velocity scale and length scale, respectively. The options differ from each other in one of the following respects:

- The form of the equations
- The treatment of the near-wall region
- The relation between Reynolds stresses and the rates of strain.

Those models that use a linear relationship between Reynolds stresses and strains are classified as non linear.

Conservation of mass

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = s_m \quad (1)$$

Conservation of momentum (Navier-Stokes equation)

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j u_i - \tau_{ij}) = -\frac{\partial p}{\partial x_i} + s_i \quad (2)$$

$$\tau_{ij} = 2\mu s_{ij} - \frac{2}{3}\mu \frac{\partial u_k}{\partial x_k} \delta_{ij} - \overline{\rho u_i' u_j'} \quad (3)$$

$$s_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (4)$$

Turbulence kinetic energy

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} \left[\rho u_j k - \left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] = \quad (5)$$

$$\mu_t (P + P_B) - \rho \varepsilon - \frac{2}{3} \left(\mu_t \frac{\partial u_i}{\partial x_i} + \rho k \right) \frac{\partial u_i}{\partial x_i} + \mu_t P_{NL}$$

where

$$P \equiv S_{ij} \frac{\partial u_i}{\partial x_j} \quad (6)$$

$$P_B \equiv -\frac{g_i}{\sigma_{h,t}} \frac{1}{\rho} \frac{\partial \rho}{\partial x_i} \quad (7)$$

$$P_{NL} = -\frac{\rho}{\mu_t} \overline{u_i' u_j'} \frac{\partial u_i}{\partial x_j} - \left[P - \frac{2}{3} \left(\frac{\partial u_i}{\partial x_i} + \frac{\rho k}{\mu_t} \right) \frac{\partial u_i}{\partial x_i} \right] \quad (8)$$

$P_{NL} = 0$ for linear models and σ_k is the turbulent Prandtl number. The first term on the right-hand side of equation (5) represents turbulent generation by shear and normal stresses and buoyancy forces, the second viscous dissipation, and the third amplification or attenuation due to compressibility effects. The last term accounts for the non-linear contributions.

Turbulence dissipation rate

$$\begin{aligned} & \frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_j} \left[\rho u_j \varepsilon - \left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] = \\ & C_{\varepsilon 1} \frac{\varepsilon}{k} \left[\mu_t P - \frac{2}{3} \left(\mu_t \frac{\partial u_i}{\partial x_i} + \rho k \right) \frac{\partial u_i}{\partial x_i} \right] + \\ & C_{\varepsilon 3} \frac{\varepsilon}{k} \mu_t P_B - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} + C_{\varepsilon 4} \rho \varepsilon \frac{\partial u_i}{\partial x_i} + \\ & C_{\varepsilon 1} \frac{\varepsilon}{k} \mu_t P_{NL} \end{aligned} \quad (9)$$

Where σ_ε is the turbulent Prandtl number and $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, $C_{\varepsilon 3}$ and $C_{\varepsilon 4}$ are coefficients whose values are given below. The turbulent viscosity μ_t is obtained via equation (10), with f_μ set equal to unity.

$$\mu_t = f_\mu \frac{C_\mu \rho k^2}{\varepsilon} \quad (10)$$

$$S_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \quad (11)$$

$$C_{\varepsilon 1} = 1.44 \quad C_{\varepsilon 2} = 1.92 \quad C_{\varepsilon 3} = 0 < \text{or} > 1.44 \\ C_{\varepsilon 4} = -0.33 \quad \sigma_k = 1.0 \quad \sigma_\varepsilon = 1.22$$

SIMPLE (Semi Implicit for Pressure-Linked Equations) algorithm can be explained as follow. If a steady-state problem is being solved iteratively, it is not necessary to fully resolve the linear pressure-velocity coupling, as the changes between consecutive solutions are no longer small. The main procedures for doing the SIMPLE algorithm are:

- An approximation of the velocity field is obtained by solving the momentum equation. The pressure gradient term is calculated using the pressure distribution from the previous iteration or an initial guess.
- The pressure equation is formulated and solved in order to obtain the new pressure distribution.
- Velocities are corrected and a new set of conservative fluxes is calculated

2.3 Model Development

Model for simulation is made regarding the fluid passage in the suction line. The passage is shown in Figure 2. Then the model was built considering the fluid area only, all dimensions are exactly same with the real compressor used by hydrogen plant.

The model is divided to three areas. First is the Inlet Channel Area; where become the first gate for gas entering the compressor. This has circular profile. The second area is support valve. The gas from previous area is distributed to three channels here during pass from the outer to the center area. Then the gas is flow to the valve area. This is a narrow space which connects the support valve and the cylinder area. In the real device, there is a moving part named valve plate which has the function to stop or let the gas flowing. But in this study, we take its maximum open condition considering the suction maximum condition was taken into account. The last area is cylinder area. Actually, this is also varying due to the movement of cylinder. But, as written before, the maximum suction is assumed. Hence the maximum cylinder volume or the Bottom Death Center (BDC) of the piston was modelled.

One of commercial CFD programs which is 'STAR-CCM+' was used for grid generation. The result is the model which number of cell 101,665 polyhedral mesh cells (Figure 3).

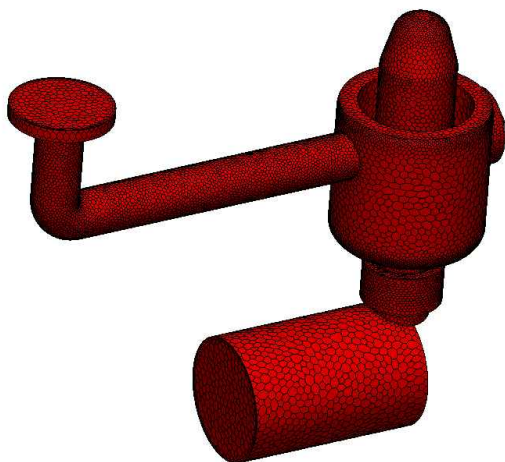


Figure 3: Grid of the 3D CAD model.

The CFD solver used in this study is STAR-CCM+. In the present study, the 3D, steady state, finite volume method was used. Steady state condition was chosen at Bottom Death Center (BDC) of piston, this happened at the same time with the valve are fully open. Then the gas flow through the line can be assumed, at a very short time, as a steady state with both inlet and outlet boundary condition.

The flow also was assumed as an incompressible fluid regarding it is an expansion phase. Material applied is H₂ gas according to the real application. For turbulence modelling, k- ϵ turbulence model, explained in mathematical model part, was used. Since the study is limited on pressure and flow analysis only, therefore considering thermal model was ignored.

3. RESULTS AND DISCUSSION

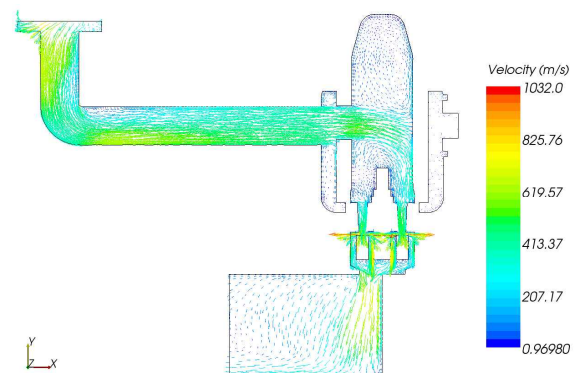


Figure 4: Velocity magnitude at middle plane section.

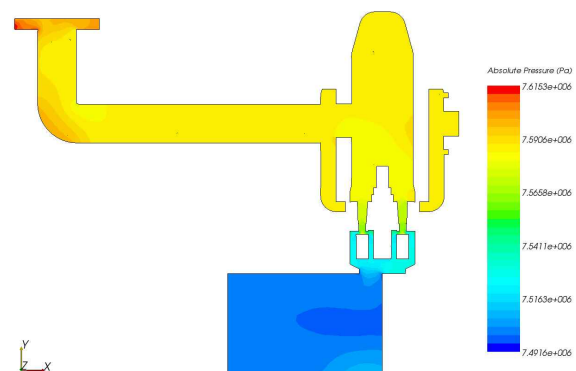


Figure 5: Absolute pressure at middle plane section.

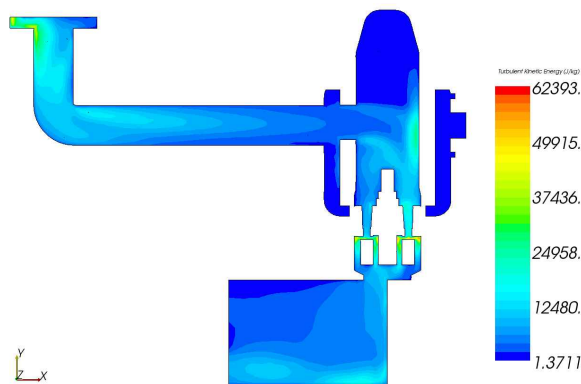


Figure 6: Turbulence kinetic energy at middle plane section.

CFD simulation gives numerical result that can be shown by vector and scalars. Parameters studied are velocity vector, absolute pressure, stream traces and turbulence kinetic energy. From the series of plotted picture, the general phenomena can be interpreted clearly and precisely.

Figure 4 shows distribution of velocity magnitude. The highest velocity is seen in the valve area. Because that valve line space is become narrow suddenly.

The pressure distribution can be seen in figure 5. The information about it is useful to analyze the passage structure stress load. The highest pressure is seen in the vertical inlet channel area and outside surface region of L-tube. Then the pressure is decreased at the horizontal inlet channel. The pressure is not changed entering the valve area. The pressure is decreased rapidly when it is passing through a valve. Finally, the minimum pressure is reached in the cylinder area. Leaving the valve, the pressure is concentrated at the bottom-end part of cylinder.

The turbulence kinetic energy is a dependent variable regarding the existence of turbulence in fluid flow. Consequently, the areas with high turbulence kinetic energy contain unstable flow. The distribution of it in the suction passage is presented in figure 6. The highest quantity of turbulence energy in the inlet channel is occurred at the center and then it reduces parabolically within radial direction. Flowing to the support valve area,

the turbulence energy fills the bottom part of it and lets the upper part as its minimum value area. Moving down through the valve part this energy is distributed uniformly until it reaches the cylinder part.

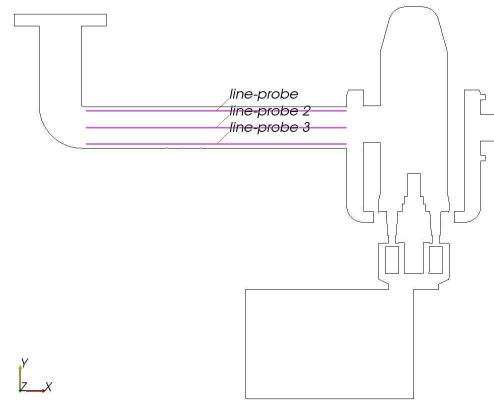


Figure 7: Line-probes of different Y axial values at the horizontal inlet channel.

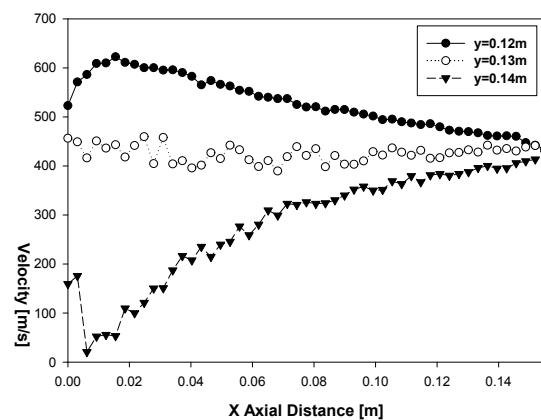


Figure 8: velocity of different Y axial values at the horizontal inlet channel.

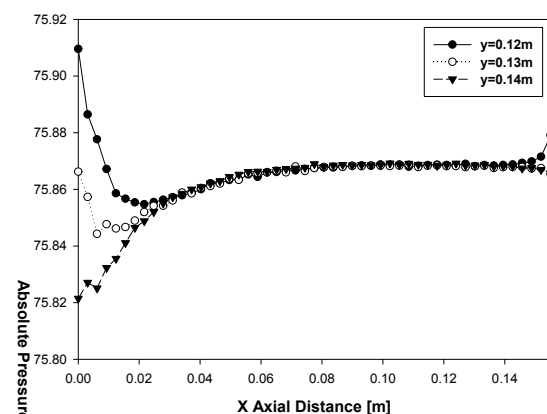


Figure 9: Absolute pressure of different Y axial values at the horizontal inlet channel.

Figure 8 shows velocity values at $y=0.12\text{m}$, 0.13 and 0.14m in the horizontal inlet channel. In case of $y=0.12\text{m}$, velocity is initially increased until $x=0.015\text{m}$ from 523m/s and is reached at 622m/s , then, it is decreased gradually from the highest velocity to 434m/s . There is a sudden fall of velocity from 159m/s to 21m/s within a very short horizontal distance of 0.006m . After this point the velocity is increased from this low to 425m/s velocity for $y=0.14\text{m}$. For $y=0.13\text{m}$, the initial velocity value is 456m/s and it is maintained the almost same value with little variation until $x=0.155\text{m}$ and reached the same velocity for $y=0.12\text{m}$ and $y=0.14\text{m}$. Finally, all y axial values are equalized at 0.155m . The velocity of this point is 430m/s .

Figure 9 shows pressure when y -axial values are 0.12m , 0.13 and 0.14m at the horizontal inlet channel. The initial pressure 75.91 bar is rapidly drop to 75.856 bar for $y=0.12\text{m}$ and it drops relatively less rapid for $y=0.13\text{m}$. But in case of $y=0.14\text{m}$, the initial pressure 75.82 is suddenly increase up to 75.856 bar at $x=0.03\text{m}$. In this figure, each of the values is presented similar pattern with gradual increasing trend from 0.03m to 0.14m x -axial distance with 75.856 bar and 75.868 bar.

4. CONCLUSION

It has seen clearly from above discussion that hydrogen gas flow in the suction passage has interesting special characteristics. The velocity, pressure and turbulence kinetic energy distribution are presented in this paper.

The CFD simulation results are as follows:

1. At $y=0.12\text{m}$ (Line-probe), velocity is decreased gradually from 622m/s to 434m/s . For $y=0.14\text{m}$ (Line-probe3), velocity is increased sharply from 21m/s to 425m/s . For $y=0.13\text{m}$ (Line-probe2), it is maintained the almost same value with little variation until $x=0.155\text{m}$ and reached the same velocity for $y=0.12$ and $y=0.14\text{m}$. Finally, all y axial

values are equalized at 0.155m . The velocity of this point is 430m/s .

2. Each of the values is presented similar pattern with gradually increasing trend from 0.03m to 0.14m x -axial distance with 75.856 bar and 75.868 bar.

This study can be use as a preliminary study for the design improvement.

5. ACKNOWLEDGEMENTS

This research was supported under the project (No. 10024337) and Program for the Training of Graduate Students in Regional Innovation System of the Ministry of Knowledge Economy and Second-Phase of BK 21 project, Korea.

REFERENCES

- [1]Technical report. United States Department of Energy, 2002. National hydrogen energy roadmap – production, delivery, storage, conversion, application, public education and outreach. United States of America.
- [2]Midili, A, Dincer, I. 2007. Key strategies of hydrogen energy systems for sustainability. International Journal of Hydrogen Energy 32. pp511-524. United States of America.
- [3]Forsberg, P, Karlstrom, M. 2007. On optimal investment strategies for a hydrogen refueling station. International Journal of Hydrogen Energy 32. pp647-660. United States of America.
- [4]Ogden, J, M. 1999. Developing an infrastructure for hydrogen vehicles: a Southern California case study. International Journal of Hydrogen Energy 24. pp709-730.
- [5]Elhaj, M, Gu, F, Ball, A, D, Albarbar, A, Al-Qattan, M, Naid, A. 2008. Numerical simulation and experimental study of a two-stage reciprocating compressor for condition monitoring. Mechanical System and Signal Processing 22. pp374-389.

- [6]Rovaris, J, B, Deschamps, C, J. Large Eddy Simulation Applied to Reciprocating Compressors. 2006. The Brazilian Society of Mechanical Sciences and Engineering Vol XVIII. No 2. April-June.
- [7]Matos, F, F, S, Prata, A, T, Deschamps, C, J. 1999. Numerical Analysis of the Dynamic Behaviour of Plate Valves in Reciprocating Compressors. Proc. International Conference on Compressor and Their Systems. London .pp453-462. UK
- [8]Matos, F, F, S, Deschamps, C, J. Prata, A, T. 2002. Numerical Simulation of Turbulent Flow in Reciprocating Compressors. Proc. of the 2002 Spring School on Transition and Turbulence. Florianopolis. pp10. Brazil.
- [9]Matos, F, F, S. 2002. Numerical Analysis of the Dynamic Behavior of Compressor Reed Type Valve. Ph.D.Thesis. Departamento de Engenharia Mecanica, Universidade Federal de Santa Catarina. Florianopolis. SC. pp10. Brazil.
- [10]Matos, F, F, S, Prata, A, T, Deschamps, C, J. 2002. Numerical Simulation of the Dynamics of Reed Type Valve. Proc. Compressor Engineering Conference at Purdue. Indiana. pp8. USA.