

# A Mesh Free Moving Node Method To Analyze Flow Through Spirals of Orbiting Scroll Pump

I.Banerjee, A.K.Mahendra, T.K.Bera, B.G.Chandresh

**Abstract**—The scroll pump belongs to the category of positive displacement pump can be used for continuous pumping of gases at low pressure apart from general vacuum application. The shape of volume occupied by the gas moves and deforms continuously as the spiral orbits. To capture flow features in such domain where mesh deformation varies with time in a complicated manner, mesh less solver was found to be very useful. Least Squares Kinetic Upwind Method (LSKUM) is a kinetic theory based mesh free Euler solver working on arbitrary distribution of points. Here upwind is enforced in molecular level based on kinetic flux vector splitting scheme (KFVS). In the present study we extended the LSKUM to moving node viscous flow application. This new code LSKUM-NS-MN for moving node viscous flow is validated for standard airfoil pitching test case. Simulation performed for flow through scroll pump using LSKUM-NS-MN code agrees well with the experimental pumping speed data.

**Keywords**—Least Squares, Moving node, Pitching, Spirals.

## I. INTRODUCTION

SCROLL pump involves two intermeshed spirals in which one stationary spiral of involute, Archimedean or logarithmic shape is housed with a moving spiral. Moving spiral executes eccentric orbital motion over stationary spiral via an eccentric crankshaft drive. The gas enters from the periphery of the spirals and gets entrapped in series of gas pockets. In the course of orbital motion, the volume of the pockets gets gradually deformed and reduced resulting in increasing of pressure and temperature of the pockets. Finally gas will be discharged at the centre through discharge gas pockets. Since moving spiral is eccentrically mounted over the stationary spiral and also undergoing an orbital motion the mesh movement in the entire flow domain is complicated. In order to capture flow features in time dependant deformable grid, mesh free solver is found very effective. Least Squares Kinetic Upwind method (LSKUM) is a grid fault tolerant scheme working on cloud of points. Mandal and Deshpande [10] introduced Kinetic Flux Vector Splitting (KFVS) to solve Euler equation for aerodynamic problem. Later Ghosh and Deshpande [4] extended the KFVS for mesh free scheme known as least squares kinetic upwind method (LSKUM) for 2D Euler equation. Mahendra [8] had further expanded

LSKUM 2D/3D for viscous flow for aerodynamic problem and also solved for problems related to strongly rotating flow using LSKUM 2D/3D. Ramesh and Deshpande [11]-[12] extended the LSKUM to solve moving node Euler equation for pitching motion of airfoil known as LSKUM\_MN solver. In the present study we have extended the LSKUM-NS [8] to moving node viscous flow application. This new code LSKUM-NS-MN for moving node viscous flow is validated for standard airfoil pitching [1] test case. The study is carried out with both explicit and implicit schemes. A matrix free implicit scheme [7] with pseudo time marching [12] approach is implemented for the test case validation. In our attempt we have modeled two-dimensional flow through spirals and thus pressure, temperature and velocity distribution of the gas pockets are obtained under un-steady real time flow scenario. The shapes of the spiral wraps under analysis are Archimedean taken from an existing scroll pump.

## II. KINETIC THEORY BASED METHOD ON MOVING NODE

LSKUM is the kinetic theory based solver capable of working with any type of mesh even on arbitrary distribution of points. The basic philosophy of the scheme is to evaluate least square based nodal derivatives via neighborhood connectivity. The upwind is implemented at the Boltzmann level via courant splitting method. Here we describe the formulation of LSKUM moving node method for viscous flow. The Boltzmann transport equation describes molecular velocity distribution function  $f(\vec{x}, \vec{v}, I, t)$  for polyatomic gas as

$$\frac{\partial f}{\partial t} + \nabla_{\vec{x}} \cdot (\vec{v}f) = J(f, f) \quad (1)$$

Where  $\vec{x}$ , the position vector and  $\vec{v}$  is the molecular velocity vector given in  $R^N$ . In right hand side  $J(f, f)$  is the collision integral describes binary collision amongst the molecules which vanishes in the Euler limit. Applying moment method strategy to Boltzmann equation by introducing moment

function  $\psi = \left[ 1, \vec{v}, I + \frac{1}{2} \vec{v}^T \vec{v} \right]^T$  we get

$$[\rho, \rho \vec{u}, \rho E]^T = \langle \psi, f \rangle \equiv \int_{R^N} \int_R \psi f(\vec{x}, \vec{v}, I, t) \quad (2)$$

Where  $E = \frac{RT}{\gamma - 1} + \frac{1}{2} (\vec{u}^T \vec{u})$ ,  $\gamma$  is ratio of specific heat,  $u$  is

the fluid velocity. The Chapman-Enskog expansion of the Boltzmann equation without Knudsen number as parameter yields first order distribution for shear and heat transfer term.

I.Banerjee is with the Chemical Technology Group, Bhabha Atomic Research Center, Mysore, India (phone: +91-821-24065333; e-mail: banerjeei@yahoo.com).

A.K.Mahendra, is with the Chemical Technology Group, Bhabha Atomic Research Center, Mumbai, India (phone: +91-22-25593611; fax: +91 -22-25505151; e-mail: mahendra@barc.gov.in).

First order Chapman-Enskog distribution function can be expressed as  $f_{CE} = F(1 - P_{CE})$  (3)

Where  $F$  is the Maxwellian distribution [4] and  $P_{CE}$  is the first order Chapman-Enskog polynomial associated with shear stress tensor and heat flux vector [8]-[9]. Chapman-Enskog polynomial in two dimensions is covered in APPENDIX.

In the KFVS for viscous flow, upwinding is implemented using courant splitting. The equation for two dimensions in Cartesian coordinate is as follows

$$\frac{\partial f_{CE}}{\partial t} + \frac{v_x \pm |v_x|}{2} \frac{\partial f_{CE}}{\partial x} + \frac{v_y \pm |v_y|}{2} \frac{\partial f_{CE}}{\partial y} = 0 \quad (4)$$

Taking  $\psi$  moment leads to Navier-Stokes equation in split flux form.

$$\frac{\partial U}{\partial t} + \frac{\partial GX^\pm}{\partial x} + \frac{\partial GY^\pm}{\partial y} = 0 \quad (5)$$

Here  $U$  is the conserved vector,  $U = [\rho, \rho \bar{u}, \rho E]^T$ .

In order to solve problems involving moving node, we define any vector  $\vec{\phi}$  along the nodal direction as

$$\left( \frac{\partial \vec{\phi}}{\partial t} \right)_M = \left( \frac{\partial \vec{\phi}}{\partial t} \right)_S + \bar{w} \cdot \nabla \cdot \vec{\phi} \quad (6)$$

Where subscript  $M$  and  $S$  signify moving and stationary frame,  $\bar{w} = [w_x, w_y]^T$   $w_x$  and  $w_y$  represents the Cartesian components of the velocity of a moving node for 2 dimensional geometry. The Boltzmann equation and replacing  $\vec{\phi}$  with Chapman-Enskog distribution function  $f_{CE}$  followed by Courant splitting we get

$$\frac{\partial f_{CE}}{\partial t} + \frac{\bar{v}_x \pm |\bar{v}_x|}{2} \frac{\partial f_{CE}}{\partial x} + \frac{\bar{v}_y \pm |\bar{v}_y|}{2} \frac{\partial f_{CE}}{\partial y} = 0 \quad (7)$$

Where  $\bar{v}_x = (v_x - w_x)$ ,  $\bar{v}_y = (v_y - w_y)$  are the Cartesian components of the particle velocity relative to the moving node. Since  $P_{CE}$  is contributing towards shear stress and heat flux, the coefficient of  $P_{CE}$  is kept un-split. Taking  $\psi$  moment leads to split Navier-Stokes equation for moving node. The state update for Navier-Stokes can be expressed as

$$U^{n+1} = U^n - \Delta t \left( \frac{\partial GX_M^\pm}{\partial x} + \frac{\partial GY_M^\pm}{\partial y} + \frac{\partial GX_{MV}}{\partial x} + \frac{\partial GY_{MV}}{\partial y} \right)^n \quad (8)$$

Where  $(GX_M^\pm, GY_M^\pm)$  are the Cartesian components of the inviscid moving split fluxes; and  $(GX_{MV}, GY_{MV})$  are the viscous moving fluxes. The details of fluxes are covered in [5], [12].

### III. BOUNDARY TREATMENT OF MOVING SOLID WALL

Boundary conditions at the moving solid surfaces can be obtained treating Boltzmann equation without splitting in moving coordinates and therefore can be expressed as

$$\left( \frac{\partial f_{CE}}{\partial t} \right)_M + \bar{v}_x \frac{\partial f_{CE}}{\partial x} + \bar{v}_y \frac{\partial f_{CE}}{\partial y} = 0 \quad (9)$$

Using moment method strategy the state update for moving boundary can be obtained as

$$U_b^{n+1} = U_b^n - \Delta t \left( \frac{\partial GX_M}{\partial x} + \frac{\partial GY_M}{\partial y} \right)^n \quad (10)$$

Where  $(GX_M, GY_M)$  are combined inviscid and viscous moving fluxes without splitting. The subscript  $b$  for state update equation represents moving solid boundary. In the matrix vector form the state update can be represented in two dimensions as

$$U_b^{n+1} = \begin{bmatrix} U_b^n(1) - \Delta t \nabla \cdot \vec{G}_M \\ w_1 U_b^{n+1}(1) \\ w_2 U_b^{n+1}(1) \\ U_b^n(4) - \Delta t \nabla \cdot \vec{G}_M \end{bmatrix} \quad (11)$$

## IV. EXPLICIT AND IMPLICIT APPROACH

### A. Explicit Scheme

Least squares approximation of various flux derivatives results in advancement of solution vector in time marching fashion. An explicit time accurate higher order Runge-Kutta scheme has been used to march the solution under real time scenario. Higher order accuracy in space is achieved via two step defect correction technique suggested by Ghosh and Deshpande [4]. Though higher order accuracy in both space and time is achieved through explicit approach, severe time step limitation on account of stability requirement makes the scheme awfully slow.

### B. Implicit Scheme

In order to get rid of time step restriction semi implicit dual time stepping approach by Ramesh and Deshpande [12] is adopted. The solution vector is advanced through an iterative pseudo time marching process. However accuracy in real time step is second order due to semi implicit approach.

## V. PITCHING AIRFOIL TEST CASE

### A. Description

We first present the case about flow passing through a pitching airfoil. The airfoil undergoes pitching oscillations around a point on the chord with one-quarter length from the leading edge. In the current case, a NACA 0012 airfoil is used and the flow conditions are defined as per standard AGARD [1] test case. The free stream velocity  $U_\infty$  is parallel to the X axis with a mach number  $M=0.755$ . The oscillation cycle is defined by

$$\alpha(t) = \alpha_m + \alpha_0 \sin(\omega t) \text{ where } \alpha_m = 0.016^\circ, \alpha_0 = 2.51$$

Where  $\alpha(t)$  represents the instantaneous angle of attack and  $\alpha_m$  represents the mean angle of attack. Reduced frequency based on chord length  $C$  of the airfoil is given by

$\frac{\omega C}{2U_\infty} = 0.0814$ , Where  $\omega$  is circular pitch frequency. In many computations mesh is fixed and the free stream velocity  $U_\infty$  is rotated in the opposite direction to the pitching direction of the airfoil [11]. However in the present computation, the whole mesh around the airfoil is moving with the airfoil and the node velocities are calculated according to the angular velocity of the pitching airfoil.

The present test case  $\alpha_{\min} = -2.494^\circ$  and  $\alpha_{\max} = -2.526^\circ$ , assuming free stream density of air is  $\rho_\infty = 1.228 \text{ kg/m}^3$  and the temperature  $T_\infty = 298 \text{ K}$ , we get the time period of oscillation as  $T = 0.14773 \text{ s}$ .

As the body oscillates the points in the interior also move. Therefore ideally one should generate connectivity for all the points at every time step or adapt the mesh after a certain number of iteration. However for this particular problem we first generate connectivity for all the points at the beginning corresponding to  $\alpha(t) = \alpha_m$ . For any other position the connectivity remains same even though there is slight variation in the coordinates of the points. This is because maximum angular deflection is about  $2.526^\circ$  about mean position.

### B. Result and Discussion

Unsteady aerodynamics and gas dynamics require time accuracy. Third order time accurate Runge-Kutta (RK-3) scheme is employed for explicit time marching. In semi implicit approach, pseudo time marching is associated with (RK-3) scheme. However accuracy in global time stepping is second order, higher order time accuracy calls for larger storage space and therefore avoided. Approximately 2000 to 3000 inner iterations are required to achieve one real time step. If larger time step is chosen the number of inner iteration should be more for convergence. In the present approach 20 and 40 time step per cycle is chosen. Fig. 1 shows comparison of lift coefficient with angle of attack for explicit, implicit and experimental data.

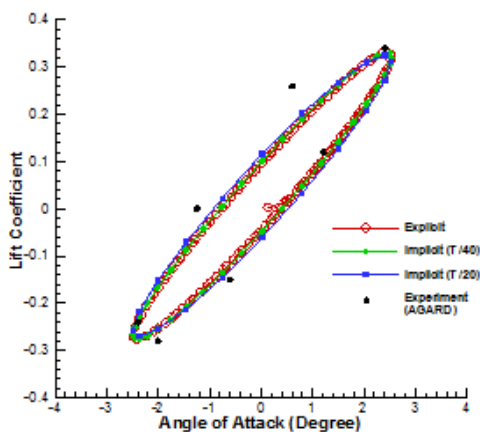


Fig. 1 Experimental comparison of Lift coefficient for both explicit and implicit approach

It is evident that implicit scheme with smaller time step matches nearly with the explicit scheme and implicit scheme with 20 time step per cycle slightly deviates from the explicit scheme.

## VI. DESCRIPTION OF SCROLL PUMP

Scroll pump consists of series of gas pockets or volumes. These pockets will travel from the periphery to the centre. In the course of travelling the volume of the pockets gets gradually reduced resulting in increasing of pressure and temperature of the pockets. Finally gas will be discharged at the centre through discharge gas pockets.

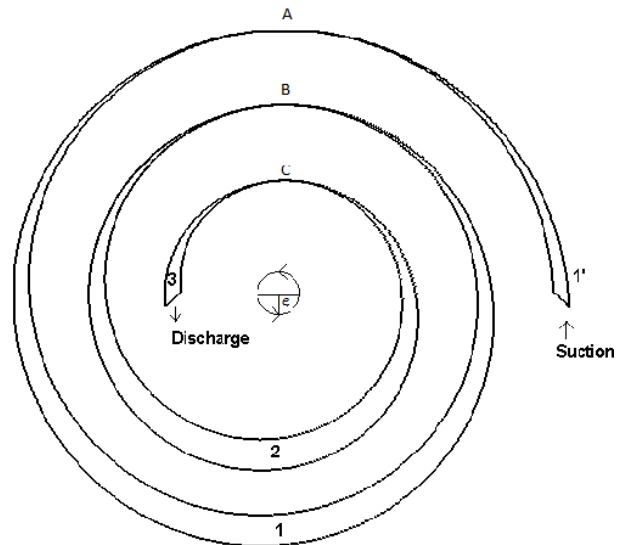


Fig. 2 Relative position of two spirals for an intermediate crank angle showing various gas pockets

Fig. 2 describes the pumping mechanism in which outer spiral is an orbiting spiral. Inner spiral is stationary element. Outer spiral executes orbital motion via an eccentric crankshaft in such a way that centre of the moving spiral is eccentric to the centre of the stationary spiral. Hence gap between the spirals gradually varies from minimum to the maximum. Minimum gap between the spirals determines the extent of peripheral back travelling of the gas i.e. sealing against peripheral leakage. The minimum gap between the spirals therefore called sealing zone. The gas entrapped between two such sealing zones is called a gas pockets. In Fig. 2 gas entrapped between sealing zone A and B is named as gas pocket 1. Similarly gas pocket 2 entraps gas between sealing zone B and C. Number of gas pockets decides existence of number of sealing zone which in turn is the function of spiral wrap or Roll angle. For a perfectly meshed spiral this sealing zones does not permit any peripheral back leakage as the spirals are in dynamic contact with each other. This arrangement invites problem like excessive heat generation, abrasion. Such design may only be suitable for low capacity general pumping application. In practice some volume of gas

is permitted to flow through sealing zones that in turn reduces volumetric efficiency of the pump. Inter pockets transmission of gas through sealing zones also dilutes effective compression of the gas in respective pockets. This adverse effect can only be mitigated by increasing number of sealing zones i.e. number of gas pockets.

As the outer spiral orbits, sealing zone A, B, C rotates along the periphery to the centre creating suction gas pocket 1', compression gas pockets 1, 2 and discharge pocket 3. With time volume of suction pocket increases till the closing of suction port and discharge pocket volume gets reduced till discharge port fully closed.

Many of the prior works [3],[6] involving refrigeration scroll for high pressure application have considered working process of scroll is a transient flow process involving energy and mass change with time. The temporal variation of flow properties in spiral domain is continuously progressing through a series of time intervals defined by crank angle. However the present work is mainly governed by low pressure transport of fluid by the squish motion of the orbiting spiral. This is a complicated time varying flow domain problem involving moving and deforming curvilinear boundary where flow properties are influenced by leakage flow through sealing zone and inter pocket gas transportation. Pumping speed characteristics are strongly influenced by leakage through suction pocket. The temporal pressure and temperature variation on various gas pockets are equally influenced by inter pocket gas transportation through sealing zone. Due to all the above reasons dynamic variation of suction volume and corresponding back flow through suction pockets must be evaluated to generate pumping characteristics. Geometric modelling clubbing with computational fluid dynamics (CFD) approach can solve the present issue.

## VII. METHODOLOGY

### A. Geometric Modeling

Geometric suction volume for different spiral wrap can be evaluated by volume integration. The details are expressed in [5]. Suction volume of Archimedean spiral can be evaluated as

$$V_s = \pi H (c_1 + c_1' - 2c_2\pi) D \quad (12)$$

And logarithmic spiral can be given as

$$V_s = \left[ \frac{b_1' H}{b_2'} \left( e^{(2b_2' \pi - 1)} - \pi H^2 \right) \right] D \quad (13)$$

Where  $V_s$  is geometric suction volume. Where H is radial offset between two spirals at zero eccentricity and D is the depth of the pump.  $c_1, c_1', c_2$  are the geometric constants for Archimedean spiral.  $b_1'$  and  $b_2'$  are the geometric constants for logarithmic spiral. Dynamic variation of suction volume with crank angle can be expressed as

$$V_s = \left( \frac{c_3 H}{2} \theta - \frac{c_2 H}{2} \theta^2 - \frac{c_3 e}{2} \sin \theta \right) D \quad (14)$$

In the present studies we have confined our analysis into Archimedean spiral wrap scroll pump for low pressure application.

TABLE I  
SCROLL PUMP GEOMETRIC PARAMETERS

Parameters	Values
Nominal volume	$1.785 \times 10^{-3} m^3$
Base circle Radius	0.18 m
Wrap Depth	0.1 m
Number of wraps	2.5-3.0
Vane thickness	0.02 m

### B. Assumption

Study was carried out with the following assumptions:

- Since the working chamber has a large peripheral length to depth ratio, the main flow direction is assumed to be along the spiral orbiting direction i.e. the flow can be simplified as a two dimensional problem.
- The working gas is assumed to be in a single phase and behaves as an ideal Gas.
- Gravity forces are negligible as compared to tangential pressure gradient.
- Continuum hypothesis prevails.

### C. Cloud of point generation

One of the greatest advantages of LSKUM is its ability to work on cloud of points obtained from any type of grid (structured, unstructured, prismatic, Cartesian, chimera, hybrid, dragon etc.). This advantage of the scheme is suitably utilized to get point distribution containing

- 1) 7200 points obtained from a structure grid with 900 points on the spiral body in theta direction and 8 points in the radial direction.
- 2) 21600 points obtained from a structure grid. There are 1800 points on the spiral body in theta direction and 12 points in the radial direction.

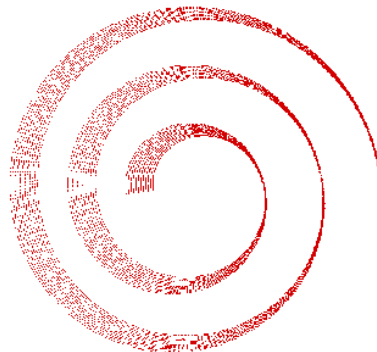


Fig. 3 Cloud of points in computational domain

#### D. Results and Discussions

Two types of test cases are studied here;

- 1) Pumping of gas through scroll pump under inflow and outflow boundary condition in order to check the ideal volumetric pumping capacity of the pump. Heat flux boundary condition is implemented.
- 2) Pumping of gas through scroll pump under actual operating pressure condition i.e. inlet with inflow open boundary condition outlet with outflow back pressure boundary condition are prevailed in order to validate the solver for volumetric pumping and gas temperature distribution. Average heat flux boundary condition is implemented based on experimental data.

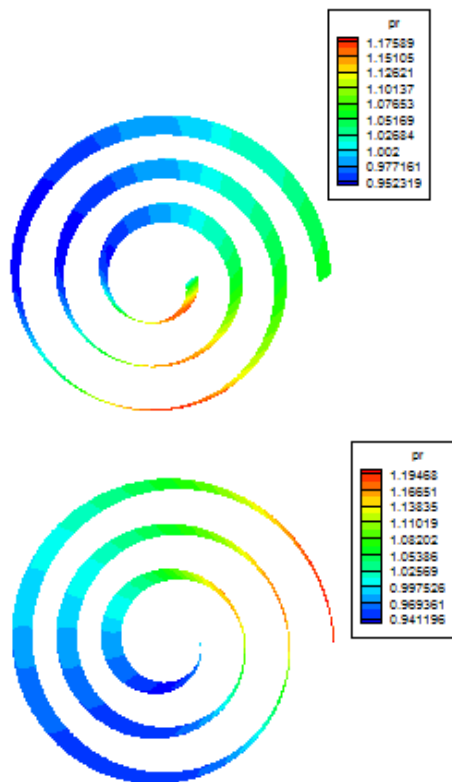


Fig. 4 Non dimensional Pressure contour for 90 degree and 180 degree crank location

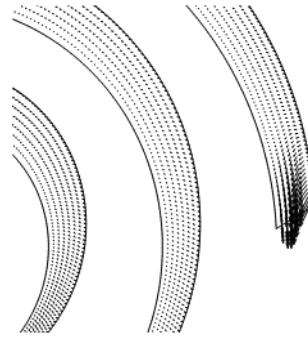


Fig. 5 Streamline plot shows leakage from suction pocket

As the crank angle moves from zero degree, the mesh gets distorted and therefore mesh regeneration and interpolation is required. However the mesh free nature of LSKUM that works on arbitrary distribution of points only calls for regeneration of connectivity after certain crank angle movement.

Fig. 4 shows non dimensional pressure contour at 90 degree and 180 degree crank angle rotation. Fig. 5 exhibits back flow through suction gas pockets. After subtracting leakage volume from suction volume we get effective suction at any particular pressure.

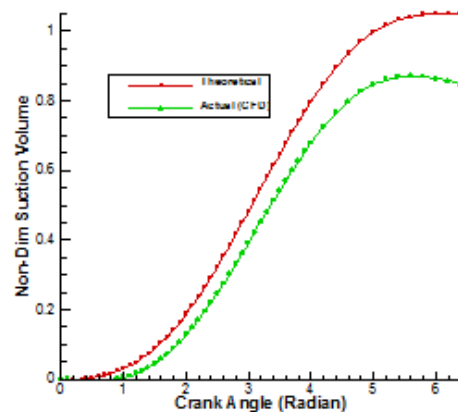


Fig. 6 Variation of theoretical and actual suction volume with crank angle

Fig. 6 depicts dynamic variation of theoretical suction volume and actual volume change via numerical calculation with crank angle movement. Initially from  $0^\circ$  when crank starts moving suction volume change is not significant. After certain crank angle ( $\sim 90^\circ$ ) volumetric expansion of suction pocket begins. At the end ( $\sim 360^\circ$ ) suction volume variation with crank angle becomes in significant.

#### VIII. EXPERIMENTAL SET UP AND FLOW MEASUREMENT

Experimental set up consists of a surge tank, piping with suitable connector for gauges and flow meter, vacuum tight on/off valve, flow control valve, indigenous mass flow meter and pumps under testing. Over all leak tightness of the system

should be at least  $1 \times 10^{-6}$  mbar-lit/sec Helium. Individual connectors, flanges and valve should have leak tightness of the order  $10^{-9}$  mbar-lit/sec Helium. Directly we get flow measurement using flow meter at various suction pressure. In order to keep validity of continuum hypothesis 10 mbar and higher suction pressure results are used for pumping speed prediction. Results are presented in non dimensional form where volumetric flow is normalized using nominal pump volume.

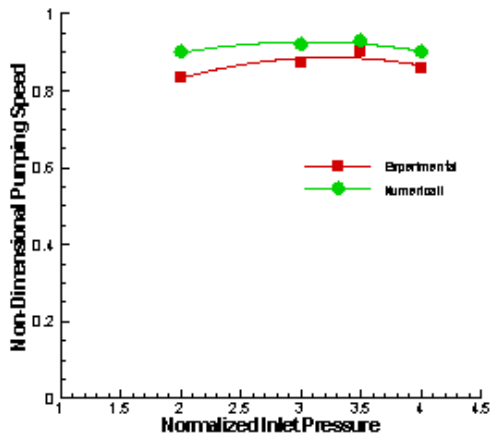


Fig. 7 Graphical comparison of experimental and theoretical Non-dimensional pumping speed with Non-dimensional inlet pressure

Fig. 7 shows comparative studies of theoretical and experimental pumping speed. At higher pressure theoretical prediction is keeping good agreement with experimental result. However at low pressure range experimental result starts deviating from the theoretical prediction.

IX. CONCLUSION

Mesh free nature of LSKUM is found very effective in working on complicated time varying mesh since it is capable of working with cloud of points and their associated connectivity. Pumping speed characteristics of pump using newly developed LSKUM-NS-MN solver is found effective at higher pressure zone. The analysis may be improved by axial leakage flow modeling and at low pressure zone it can be further improved by introducing local slip flow through Split stencil. However the advantage of present method is avoidance of grid related complicity viz. regeneration and interpolation by suitably altering neighborhood connectivity. Implementation of implicit scheme has further helped in terms of saving computational time.

APPENDIX

Chapman-Enskog polynomial can be written in two dimension

$$\begin{aligned}
 P_{CE} = & 2\beta \frac{\tau_{xy}}{P} c_x c_y + \frac{\tau_{xx}}{P} \left( \beta c_x^2 + \frac{2\gamma-3}{4-2\gamma} - \frac{I}{4\beta I_0^2} \right) + \\
 & \frac{\tau_{yy}}{P} \left( \beta c_y^2 + \frac{2\gamma-3}{4-2\gamma} - \frac{I}{4\beta I_0^2} \right) \\
 & - \frac{q_x}{PTC_P} \left( -3c_x + \beta c_x^3 + \beta c_x c_y^2 + \frac{I}{I_0} c_x \right) \\
 & - \frac{q_y}{PTC_P} \left( -3c_y + \beta c_y^3 + \beta c_y c_x^2 + \frac{I}{I_0} c_y \right)
 \end{aligned}
 \tag{15}$$

Where  $c_x = v_x - u_x, c_y = v_y - u_y, v_i, u_i$  are the Cartesian components of molecular velocity and fluid velocity respectively.

$$\beta = \frac{1}{2RT}, I_0 = \frac{2-\gamma}{\gamma-1} RT, P = (\gamma-1)\rho \left( E - \frac{u_x^2 + u_y^2}{2} \right)$$

$T$  is static temperature,  $\rho$  is fluid density,

$E$  is total energy,  $\gamma$  is the ratio of specific heat

$\tau_{xy}, \tau_{xx}, \tau_{yy}$  are the stress tensor

$q_x, q_y$  are the heat flux vector in two dimension.

ACKNOWLEDGMENT

Authors are highly grateful to Shri M.R.Srikanthan, Project manager (M) and Shri S.Sarkar, Project Manager (P) of Bhabha Atomic Research Center (B.A.R.C), Mysore to perform experimental trial on Scroll pump. Authors are also grateful to Shri G.Gouthaman, Head MDD, Chemical Technology Group, B.A.R.C to encourage computational work for the present project.

REFERENCES

- [1] AGARD-R-702. "Compendium of unsteady aerodynamic measurements." August 1982
- [2] Batina JT. "Unsteady airfoil solutions using unstructured dynamic meshes" AIAA Journal, Vol 28, PP 1381-1388 (1990)
- [3] Cui. M. M. "Numerical Study of Unsteady Flows in a Scroll Compressor" International Journal of Fluids Engineering, Vol. 128 / (947-955), September (2006)
- [4] Ghosh A.K., Deshpande S.M., "A robust least square kinetic upwind scheme for Euler equation" Int. Conference on Numerical Method (ICNMFD-14) (1994)
- [5] I.Banerjee, "CFD simulation of flow through single and multi vane spiral pump using moving node LSKUM-NS" M.Sc. thesis, HBNI, Mumbai, (2012)
- [6] Kim Tiow Ooi, Jiang Zhu, "Convective heat transfer in a scroll compressor chamber: a 2-D simulation", International Journal of Thermal Sciences 43, 677- 688, (2004)
- [7] Luo Hong, Joseph D Baum, Lo " hner Rainald. "An accurate, fast, matrix-free implicit method for computing unsteady flows on unstructured grids." Comput Fluids;30:137-59, (2001)

- [8] Mahendra A.K., “application of least squares kinetic upwind method to strongly rotating viscous flow” , M.Sc. thesis, IISC, Bangalore, (2003)
- [9] Mahendra A.K, R.K.Singh, G.Gouthaman, “Meshless Kinetic Upwind method for compressible, viscous rotating flows”, computers and fluids (2011).
- [10] Mandal J.C., Deshpande S.M, “Kinetic flux vector splitting for Euler equations”, computers and fluids 23(2), 447-478(1994)
- [11] Ramesh V, Deshpande SM. Least squares kinetic upwind method on moving grids for unsteady Euler computations. Comput Fluids J;30(5):621–41, (2001)
- [12] Ramesh V, Deshpande SM. “Unsteady flow computations for flow past multiple moving boundaries using LSKUM” Comput Fluids J 36,1592–1608, (2007)