Volume 8, No. 6, July 2017 (Special Issue III)
ISSN No. 0976-5697
International Journal of Advanced Research in Computer Science
RESEARCH PAPER
Available Online at www.ijarcs.info

# FLUID FLOW IN A SQUARE CAVITY WITH MOVING HORIZONTAL WALLS - A SIMULATION BY MARKER AND CELL (MAC) METHOD

C Sivarami Reddy
Department of Mathematics
JNTUA University
Anantapuramu-515 002, India

V Ramachandra Prasad
Department of Mathematics
Madanapalle Institute of Technology & Science
Madanapalle-517 325, India

K Jayalakshmi
Department of Mathematics
JNTUA University
Anantapuramu-515 002, India

***Abstract:*** The Marker and Cell (MAC) method with uniform, Cartesian staggered grid has been developed to solve the non-dimensional 2D Navier-Stokes equations. The developed numerical code is validated with the published benchmark problem and compared with the numerical simulation for lid driven cavity flow. Next, the numerical code has been applied for uniformly moving horizontal walls for various Reynolds numbers (Re = 100, 400 and 1000). The formation of vertices with in the cavity strongly depends on Reynolds number.

***Keywords:*** Newtonian; Lid-driven cavity; Laminar flow; MAC Method; Unsteady

## I. INTRODUCTION:

In the last several decades it has been seen rapid advances in the field of Computational Fluid Dynamics because of its wide range of applications. Developments in computer technology & hardware as well as in advanced numerical algorithms have enabled attempts to be made towards analysis and numerical solutions of highly complex fluid flow problems. Flow within an enclosure is more interesting as it includes a number of unique fluid dynamic phenomena and has many applications in numerous engineering systems such as short-dwell coating, drug-reducing riblets in aerodynamics, removal of species from structured surfaces, mixing and flow in drying devices. Such type of flows can be driven by buoyancy effects or by the motion of one or more walls that define the cavity. Many experimental and numerical studies have been conducted to examine the flow field of a single-sided lid-driven cavity flow in the past few decades. Burggarf studied the analytical and numerical solutions for 2D lid driven flow inside a square cavity [1]. He observed that the centre of the main eddy moves with the direction of the lid movement at low Reynolds number (Re) and the vortex is symmetric about the centre of the cavity for high Re. Wright and Gaskell [2], Calhoon and Roach [3], and Sahin et al. [4] carried out the Finite Volume method to study the lid driven flow phenomena inside a cavity at various Re. Ghia et al. [5] analysed with multigrid method of a stream function and a vorticity 2D incompressible Navier–Stokes equations for flow in a square cavity at high Re. Gabriella bognar and Zoltan csati [6] studied time dependent Navier-Stoke equations with the help of Spectral method. They focussed mainly on numerical investigation of Navier-Stoke equations by applying a spectral method and tested as a benchmark problem, two-dimensional incompressible flow in a square cavity of moving top lid with uniform velocity. Numerical solutions of the steady base flow in a square lid-driven cavity for Reynolds numbers (Re) up to 10,000 have been obtained by several investigators [7–11]. Numerical simulations of the 2D lid-driven cavity flow are performed for a wide range of Reynolds numbers. Accurate benchmark results are provided for steady solutions as well as for periodic solutions around the critical Reynolds number by Mazen Saad [12]. Lid-driven cavity flow at moderate Reynolds numbers is studied with mesh-free method known as Smoothed Particle Hydrodynamics (SPH) by Shahab Khorasanizade et al.[13].

Exact solution to the Navier-Stokes equation can be found only in very special cases. The most common numerical methods include finite difference (FDM), finite element (FEM) and finite volume methods (FVM). In this paper is focused only on numerical investigation of the Navier-Stokes equation applying the Marker and Cell (MAC) method.

## II. PROBLEM DESCRIPTION

Fig.1 shows a schematic diagram of the two-dimensional square cavity of width $L$ and height $H$ which is considered in the present numerical study. The two side walls of the square cavity are stationary and top and bottom walls of the cavity are moving with uniform velocity. We assume that the fluid is Newtonian and the flow is laminar. Furthermore, we assume that the flow is isothermal, two-dimensional unsteady incompressible flow with constant fluid properties.

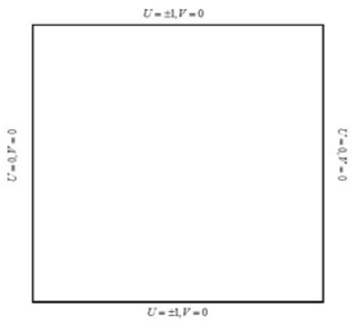



Fig. 1 Schematic diagram of the Square Cavity




CONFERENCE PAPER
National Conference dated 27-28 July 2017 on
Recent Advances in Graph Theory and its Applications (NCRAGTA2017)
Organized by Dept of Applied Mathematics
Sri Padmawati Mahila Vishvavidyalayam (Women's University) Tirupati, A.P., India

The continuity and momentum equations governing the flow in the cavity can be expressed in the form of the Cartesian coordinates as

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (1)$$

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\upsilon\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \qquad (2)$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\upsilon\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \qquad (3)$$

The following dimensionless variables are introduced:

$$\tau=t/H/u_p,(X,Y)=\frac{(x,y)}{H},U=\frac{u}{u_p},V=\frac{v}{u_p},P=\frac{p}{\rho {u_p}^2},\mathrm{Re}=\frac{u_p H}{\upsilon}$$

where $u_p$ is the lid speed

$$\frac{\partial U}{\partial X}+\frac{\partial V}{\partial Y}=0 \qquad (4)$$

$$\frac{\partial U}{\partial \tau}+U\frac{\partial U}{\partial X}+V\frac{\partial U}{\partial Y}=-\frac{\partial P}{\partial X}+\frac{1}{\mathrm{Re}}\left(\frac{\partial^2 U}{\partial X^2}+\frac{\partial^2 U}{\partial Y^2}\right) \qquad (5)$$

$$\frac{\partial V}{\partial \tau}+U\frac{\partial V}{\partial X}+V\frac{\partial V}{\partial Y}=-\frac{\partial P}{\partial X}+\frac{1}{\mathrm{Re}}\left(\frac{\partial^2 V}{\partial X^2}+\frac{\partial^2 V}{\partial Y^2}\right) \qquad (6)$$

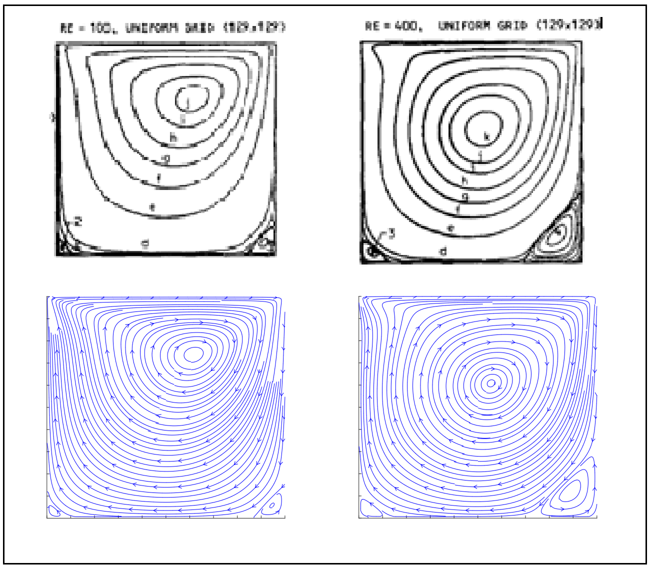



Fig. 2 Comparison of streamlines for our MAC method and results reported by Ghia et al. [5] for various Reynolds numbers

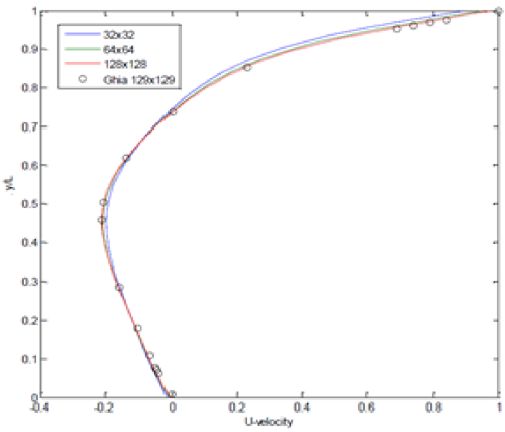



Fig. 3 *U*-velocity profile along the *y*-axis through the geometric center of the square cavity (*Re* = 100) with comparison to Ghia et al. [5].

The no-slip and no-penetration boundary conditions are satisfied at all the walls. The dimensionless initial and boundary conditions of the problem are

| | |
|---|---|
| On the left and right wall | $U=V=0$ |
| On top wall | $U=\pm 1, V=0$ |
| On bottom wall | $U=\pm 1, V=0$ |

## III. NUMERICAL SOLUTION & VALIDATION:

The governing non-dimensional partial differential equation along with appropriate boundary conditions are solved numerically by the finite-difference discretization is carried out in a staggered grid, usually known as MAC (marker and cell), this method was initially proposed by Harlow and Welch (1965). Many researchers conducted numerical study of MAC method [14-18] for cavity flows. In this paper the numerical solutions are computed by choosing uniform grid system. To solve the discretized transient equations by marching in time until an asymptotic steady-state solution is reached. The following criterion is employed to check for steady-state solution.

$$\sum_{i,j}\left|\Omega_{i,j}^{k+1}-\Omega_{i,j}^{k}\right|<\varepsilon$$

The generic variable $\Omega$ stands for $U,V,P$ and $k$ denotes the iteration time levels. The value of $\varepsilon$ is chosen as $10^{-8}$. The time step used in the numerical computations is varied between 0.001 and 0.000001 depending on *Re*.

In order to check the accuracy of the MAC method employed for the solution of the problem under consideration, it was validated (after making the necessary modifications) with the problem of High-*Re* Solutions for Incompressible Flow in a lid driven cavity reported earlier by Ghia, et al. [5]. Fig. 2 present comparisons for the streamlines with *Re*=100 & 400 also present comparisons for *U*-velocity profile along the *y*-axis through the geometric center of the square cavity (*Re* = 100) in fig. 3. These comparisons show good agreement between the results and lend confidence in the numerical results to be reported subsequently.

## IV. RESULTS AND DISCUSSION

We now present the computed numerical results of two-dimensional cavity for various values of Reynolds number (*Re*=100,400 and 1000). In this paper the results are organized in two ways based on moving velocities of horizontal walls. The formation of eddies within the cavity strongly depends on Reynolds number. The streamlines are depicted with the movement of horizontal walls, in first case these walls are moving in the same direction and in the other case in opposite directions.

The top and bottom walls of the cavity are moving towards the right direction with uniform velocity and the corresponding streamlines for the Reynolds numbers 100, 400 and 1000 are presented in fig. 4. The formation of fluid circulations within the cavity is varying with increasing the Reynolds number. Two circulations are formed symmetrically with *x*-axis in the square cavity and the fluid circulated along the moving walls. The two eddies occupied horizontally in the whole area of the square cavity. One of

CONFERENCE PAPER
National Conference dated 27-28 July 2017 on
Recent Advances in Graph Theory and its Applications (NCRAGTA2017)
Organized by Dept of Applied Mathematics
Sri Padmawati Mahila Vishvavidyalayam (Women's University) Tirupati, A.P., India

the circulation is occupied in the top half area of the cavity is caused by the top wall moving in right direction and another one occupied in the bottom half area of the cavity is affected by the bottom wall which is also moving with uniform velocity in right direction. The equal size of circulations is observed for the Reynolds numbers 100 and 400. The major circulations are formed in upper half and lower half area about the mid – section of the square cavity for high Reynolds number *Re*=100 and also two small vertices are formed near the middle part of the right wall inside the cavity.

Fig. 5 presents the streamlines for the Reynolds numbers 100,400 and 1000 with top and bottom walls are moving in opposite directions. The mono circulation of the fluid flow changes gradually with increasing of Reynolds number. The top wall is moving in right direction and bottom wall is moving in left direction while both walls are in uniform velocity, this effect generates a mono circulation. The uni-circulation circulates diagonally for *Re*=100 as shown in the fig. 5(a) and the mono diagonal circulations diminish for *Re*=400 as shown in the fig. 5(b). Considerable change is observed in the fluid flow when the Reynolds number is increased to 1000. The fluid rotates circularly at the center of the square cavity and two minor eddies are appeared at the top corner of left wall and bottom corner of right wall.

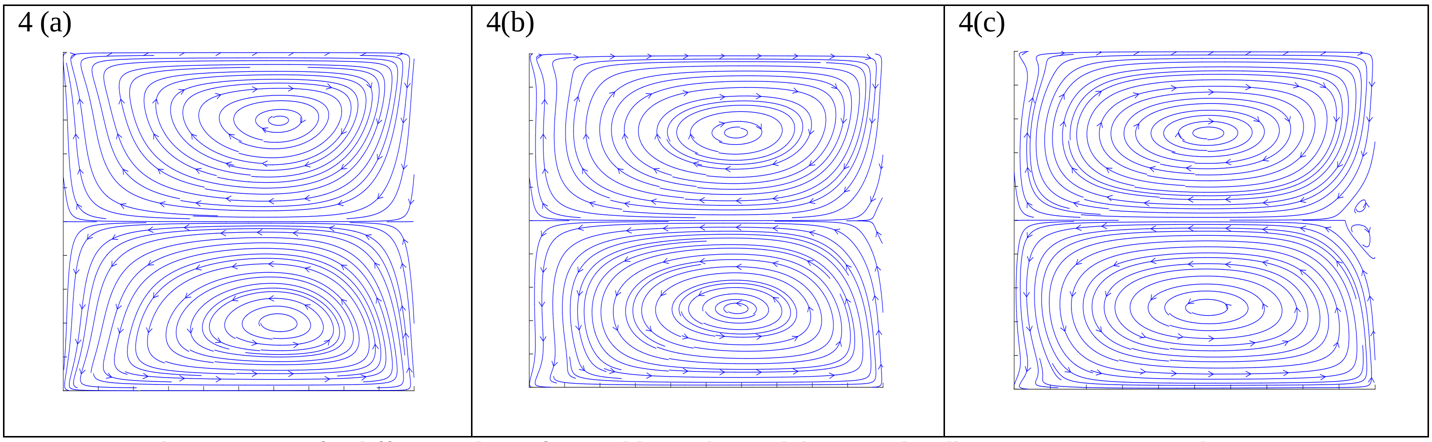



Fig. 4 Streamlines contours for different values of Reynolds number with horizontal walls are moving in same direction (a) Re=100 (b) Re=400 (c) Re= 1000

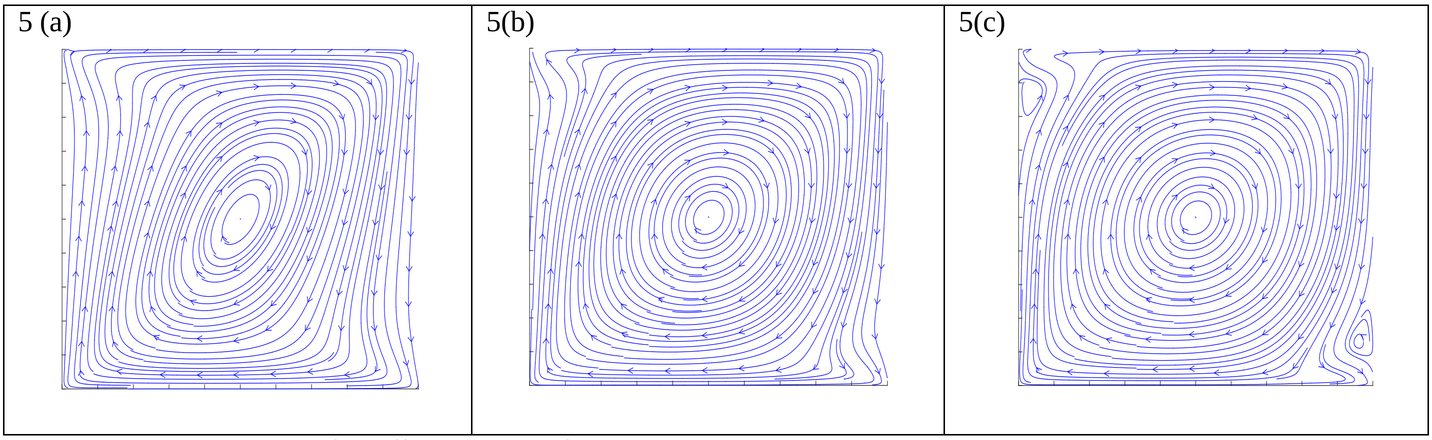
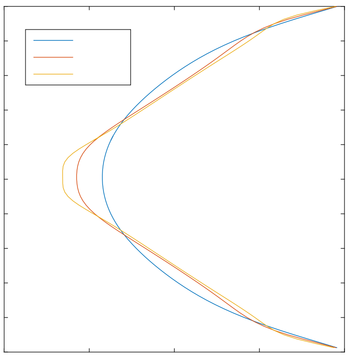
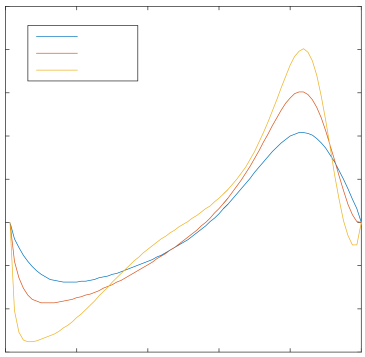



Fig. 5 Streamlines contours for different values of Reynolds number with horizontal walls are moving in opposite direction (a) Re=100 (b) Re=400 (c) Re= 1000



(a)

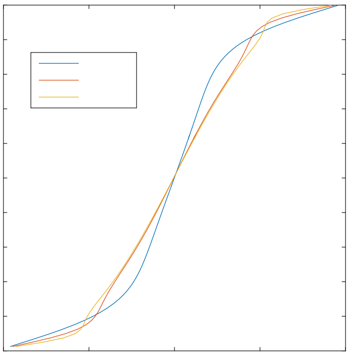


(b)

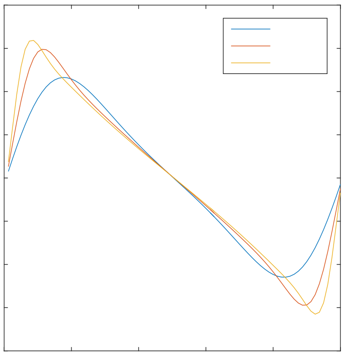



CONFERENCE PAPER
National Conference dated 27-28 July 2017 on
Recent Advances in Graph Theory and its Applications (NCRAGTA2017)
Organized by Dept of Applied Mathematics
Sri Padmawati Mahila Vishvavidyalayam (Women's University) Tirupati, A.P., India

Fig. 6 Velocity profiles at mid-section for different values *Re* with top and bottom walls are moving in same (right) direction



(a)



(b)

Fig. 7 Velocity profiles at mid-section for different values *Re* with top and bottom walls are moving in opposite directions

The mid-section *U* velocity profiles at *x*=0.5 have been shown in fig. 6(a) for different values of Reynolds number. Horizontal velocity profiles change gradually and for each of the *Re* and *u*-velocity is symmetric about *y* = 0.5. From fig. 6(a) it is observed that *u*-velocity is gradually decreases with enhancing of Reynolds number. Fig. 6(b) depicts the change in v-velocity profile along the *x*-axis at *y*=0.5. It is found that the v-velocity is increased with the increase of Reynolds number. Fig. 7(a) illustrates the horizontal velocity component at *x*=0.5 along the *y*-axis for various Reynolds numbers with the top and bottom walls are moving in opposite directions. The velocity of *u* is decreases with increasing of Reynolds number. The vertical velocity profile along the *x*-axis is shown in the fig. 7(b) the velocity v is increasing symmetrically with Reynolds number.

## V. CONCLUSION:

The present study is involved with the numerical simulation of two - dimensional square cavity with the movement of horizontal and the vertical walls with zero velocity. The numerical computations are performed in two cases when the horizontal walls are moving in positive direction (first case) and horizontal walls are moving in opposite direction (second case). MAC method is employed for conservation of momentum equations. The major conclusions of the present analysis are summarized as follows:

- The formation of eddies in the square cavity strongly depends on moving direction of the horizontal walls
- The formation of vertices within driven cavity strongly depends on Reynolds number
- Two eddies are formed in the cavity when horizontal walls are moving in same directions and the mono circulation is appeared in the cavity when horizontal walls are moving in opposite direction.
- The flow circulates circularly at centre of the square cavity for high Reynolds number *Re*=1000
- The vertical velocity profile v is gradually increases with enhancing the Reynolds number
- The horizontal velocity profile *u* is gradually decreases symmetrically with increasing of the Reynolds number
- The vertices formed in more than one in driven cavity for Reynolds number *Re*=1000 with both cases.

## VI. REFERENCES


[1] O.R. Burggarf, Analytical and numerical studies of the structure of steady separated flows, J. Fluid. Mech. 24 (1966) 113–151.

[2] N.G. Wright, P.H. Gaskell, An efficient multigrid approach to solving highly recirculating flows, Comput. Fluids 24 (1995) 63–79.

[3] W.H. Calhoon, R.L. Roach, A naturally up winded conservative procedure for the incompressible Navier–Stokes equations on non-staggered grids, Comput. Fluids 26 (1997) 525–545

[4] M. Sahin, R.G. Owens, A novel fully-implicit finite volume method applied to the lid-driven cavity problem. Part I. High Reynolds number flow calculations, Int. J. Numer. Methods Fluids 42 (2003) 57–77.

[5] U. Ghia, K.N. Ghia, C.T. Shin, High-Re solution for incompressible flow using the Navier–Stokes equations and a multigrid method, J. Comput. Phys. 48 (1982) 387–411.

[6] Gabriella Bognar and Zoltan Csati , Spectral Method for Time Dependent Navier-Stokes Equations, Miskolc Mathematical Notes. 17 (2016) 43–56.

[7] Schreiber R, Keller HB. Driven cavity flows by efficient numerical techniques. J Computat Phys 49 (1983) 310–33.

[8] Felice G, Denaro FM, Meola C, Multidimensional single-step vector upwind schemes for highly convective transport problems. Numer Heat Transf, Part B 23 (1993) 425–60.

[9] Thompson MC, Ferziger JH. An adaptive multigrid technique for the incompressible Navier–Stokes equations, J Computat Phys 82 (1989) 94–121.

[10] Hou S, Zou Q, Chen S, Doolen G, Cogley AC. Simulation of cavity flow by the lattice Boltzmann method.J Computat Phys 1995;118:329–47.

[11] Botella O, Peyret R. Benchmark spectral results on the lid-driven cavity flow. Comput Fluids 1998;27:421–33.

[12] Charles-Henri Bruneau, Mazen Saad The 2D lid-driven cavity problem revisited, Computers & Fluids 35 (2006) 326–348.

[13] Shahab Khorasanizade and Joao M. M. Sousa, A detailed study of lid-driven cavity flow at moderate Reynolds numbers using Incompressible SPH, Int. J. Numer. Meth. Fluids 2014; 76:653–668.

[14] Md. A. Ikbal, S. Chakravarty , Sarifuddin & P. K. Mandal,Unsteady Analysis Of Viscoelastic Blood Flow Through





CONFERENCE PAPER
National Conference dated 27-28 July 2017 on
Recent Advances in Graph Theory and its Applications (NCRAGTA2017)
Organized by Dept of Applied Mathematics
Sri Padmawati Mahila Vishvavidyalayam (Women's University) Tirupati, A.P., India

Arterialstenosis, Chemical Engineering ommunications, 2012 199:40–62.

[15] Harlow, F. H., and Welch, J. E, Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface, 1965, Phy. Fluids, 8, 2182–2189

[16] Amsden, A. A., and Harlow, F. H, The SMAC Method: A numerical technique for calculating incompressible fluid flow, Los Alamos Scientific Lab. Report LA-4370, 1970.

[17] Welch, J. E., Harlow, F. H., Shannon, J. P., and Daly, B. J, The MAC method, Los Alamos Scientific Lab. Report LA-3425 1996.

[18] V. Ambethkar, Mohit Kumar Srivastava, Numerical Study Of An Unsteady 2-D Incompressible Viscous Flow With Heat Transfer At Moderate Reynolds Number With Slip Boundary Conditions, 25 No. 6 2012, 883-908.





CONFERENCE PAPER
National Conference dated 27-28 July 2017 on
Recent Advances in Graph Theory and its Applications (NCRAGTA2017)
Organized by Dept of Applied Mathematics
Sri Padmawati Mahila Vishvavidyalayam (Women's University) Tirupati, A.P., India