NUMERICAL COMPARISONS BETWEEN UNIFAES, QUICK AND 2ND ORDER UPWIND SCHEMES TO NAVIER-STOKES EQUATIONS

Carlos Alberto de Almeida Vilela, carlosavgeo@yahoo.com
Faculdade de Engenharia Mecânica, Universidade Federal de Goias (UFG)

José Ricardo Figueiredo, jrfigue@fem.unicamp.br
Departamento de Energia, Faculdade de Engenharia Mecânica, Universidade Estadual de Campinas (UNICAMP)

Abstract. *Numerical solution of Navier-Stokes equations is still a hard task when applied more aggressive boundary conditions and complex geometries. To achieve satisfactory results, the discretization schemes face great obstacles that must be overcome, including the numerical stability, convergence rate, dependence on the grid configuration, etc. A widespread practice for checking the quality of a numerical scheme is its application in a wider range of problems with increasingly complexity and where the boundary conditions can be extended to more critical levels, making it possible to identify the spectrum of behavior of the scheme. The classical problem of lid driven cavity is considered here with inclined walls. Using the formulation in generalized coordinates, the objective is to expand the test cases for this geometric configuration and compare UNIFAES to other discretization schemes such as QUICK and 2nd Order Upwind. Results were obtained through algorithm developed by the author using the finite volume method with the SIMPLE method for velocity-pressure coupling and also through ANSYS software based on finite element method with Streamline Upwind Petrov Galerkin (SUPG) element type.*

Keywords*: finite volume method, UNIFAES, lid driven cavity, generalized coordinate*

1. INTRODUCTION

Numerical methods in fluid flow are very powerful tools to solve problems that solutions eventually are not possible to be found in analytical way. The finite volume method is a traditional numerical tool that has been widely used to solve fluid flow problems, is based on a principle similar to finite element method that is the domain division into smaller control volumes and then performed the governing equations integration over this discrete volume. Idelsohn and Onate (1994) has found some equivalent formulations between finite volume and finite element when specific interpolation schemes are used.

An interpolation scheme appears from the need to describe the fluid behavior by simpler equations. Central interpolation scheme assumes that the variable, “u”, “v” velocity components, temperature, etc, at the volume interface is the arithmetic mean between two adjacent nodes. This assumption results in a 2nd order interpolation curve that gives Central scheme some important numerical qualities such as stability and accuracy, but it is not so good in high convective problems.

The success of the method depends on the kind of interpolation is made between the volume interfaces to evaluate interfacial variables. Divergent solutions and numerical instabilities appears from highly convective flows, non linearity in fluid properties and high gradients.

Some traditional schemes like Central and Upwind due its mathematical simplicity to be implemented are good options for some kind of problems but they can fail and generate wrong results if boundary conditions and geometric configurations are more aggressive. It is known that Upwind scheme has a good behavior in high convective problems but it is highly numerically diffusive. Find the ideal discretization scheme is a task that still running and testing cases even more complex is a way to verify the scheme quality. This paper brings some additional results to Navier-Stokes solution obtained with finite volume method and UNIFAES scheme and comparisons with QUICK and 2nd Order Upwind schemes.

2. PROBLEM DESCRIPTION AND SOLUTION METHOD

2.1 Governing equations

The equations used to describe this problem are the two-dimensional continuity equation and the steady state Navier-Stokes equations for incompressible fluid and constant properties. Computations were performed using ANSYS code used as comparative result and FLOW code written by Vilela (2001) and already used to solve convective-diffusive transport problems in generalized coordinates, Vilela and Figueiredo (2002) and fluid flow inside bi-dimensional gradual divergent duct, Vilela (2001). ANSYS is based on finite element method and FLOW code is based on finite volume method with generalized coordinate form of equations as can be seen in eq. (1) and eq. (2)

Continuity equation:

$$\frac{\partial(\rho\tilde{u})}{\partial\xi}+\frac{\partial(\rho\tilde{v})}{\partial\eta}=0 \tag{1}$$

Navier-Stokes equation for ξ direction

$$\frac{\partial}{\partial\xi}(\rho\tilde{u}u)+\frac{\partial}{\partial\eta}(\rho\tilde{v}u)=\frac{\partial}{\partial\xi}\left[\frac{\mu}{J}\left(q_{11}\frac{\partial u}{\partial\xi}+q_{12}\frac{\partial u}{\partial\eta}\right)\right]+\frac{\partial}{\partial\eta}\left[\frac{\mu}{J}\left(q_{21}\frac{\partial u}{\partial\xi}+q_{22}\frac{\partial u}{\partial\eta}\right)\right]-\left[\frac{\partial}{\partial\xi}(f_{11}p)+\frac{\partial}{\partial\eta}(f_{21}p)\right]+g_xJ \tag{2a}$$

Navier-Stokes equation for η direction

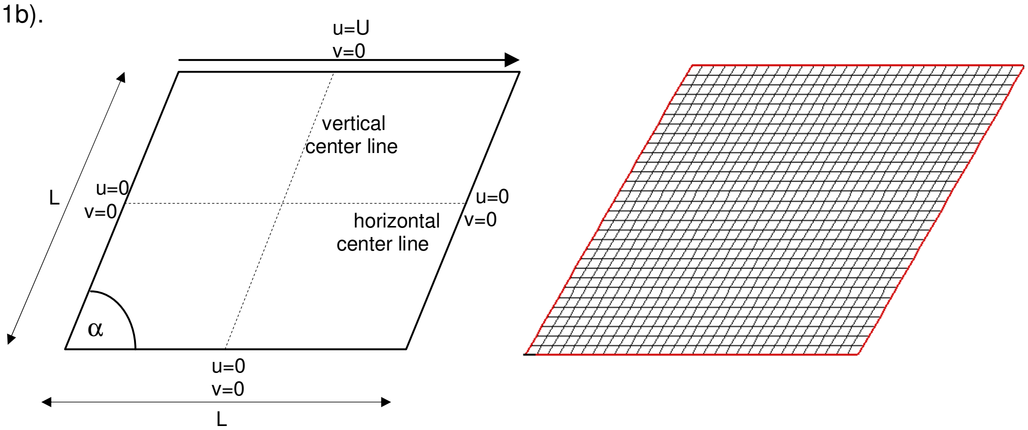
$$\frac{\partial}{\partial\xi}(\rho\tilde{u}v)+\frac{\partial}{\partial\eta}(\rho\tilde{v}v)=\frac{\partial}{\partial\xi}\left[\frac{\mu}{J}\left(q_{11}\frac{\partial v}{\partial\xi}+q_{12}\frac{\partial v}{\partial\eta}\right)\right]+\frac{\partial}{\partial\eta}\left[\frac{\mu}{J}\left(q_{21}\frac{\partial v}{\partial\xi}+q_{22}\frac{\partial v}{\partial\eta}\right)\right]-\left[\frac{\partial}{\partial\xi}(f_{12}p)+\frac{\partial}{\partial\eta}(f_{22}p)\right]+g_yJ \tag{2b}$$

where:
ξ, η are generalized coordinates
J is the geometric transform Jacobian matrix
qij are transform coefficients
u,v are cartesian velocities
$\tilde{u},\tilde{v}$ are covariant base velocities

Equations (1) and (2) are solve in a rectangular, equally spaced transformed plane ξ, η and the geometric relations between physical and transformed planes are given by Jacobian matrix *J* and q_{ij} coefficients. Even in the 90^o cavity configuration, where the Jacobian matrix gives only a scale transformation and in theory there is no need to use generalized coordinate form, this formulation were completely used. The Jacobian matrix also indicates a measure of volume deformation to the reference volume/element in finite volume and element formulations, and some care must be taken to grid generation to avoid volumes with large deformations.

2.2 Geometry

The geometry and boundary conditions are presented in fig. (1a). It represents a square cavity with left, right and bottom walls with no slip conditions and the superior wall moves with uniform velocity *U* from left to right and the angle α is the cavity deformation. All grid points follows the wall orientation as can be seen in fig. (1b).



(a) (b)

Figure 1. (a) Geometry and boundary conditions, (b) Grid configuration

2.3 Discretization schemes

The finite volume method is use to solve eq. (1) and eq. (2) and three numerical discretization schemes are considered: QUICK, by Leonard (1979), UNIFAES, by Figueiredo (1997) and 2nd Order Upwind, by Leonard (1988). QUICK and 2nd Order Upwind are part of a class of schemes based on polynomial interpolation curves. One of the most traditional and known scheme of this class is the Central scheme. Here the Central scheme is use as a reference to iteration number comparisons. QUICK has been tested for a large range of problems and it has demonstrate to have a very good behavior facing aggressive boundary conditions and flow characteristics, but it still stalls generating unphysical solutions and oscillations in highly convective step profiles and some non linear flows, Leonard (1988). It is based on a quadratic interpolation curve and Leonard is very incisive to it`s quality face to exponential type schemes, Leonard (1995). 2nd Order Upwind is a discretization scheme that came to overcome the numeric diffusion and oscillations problems present in traditional Upwind scheme. Other class of numerical schemes is based on exponential interpolation curve. Some schemes of this class is the traditional Exponential which uses a simple exponential curve, and examples of more complex and robust schemes are LOADS by Wong and Raithby (1979) and more recently UNIFAES, which is the object of this paper. UNIFAES has been tested in cases like natural and mixed convection in porous media Figueiredo and Llagostera(1999), Llagostera and Figueiredo (2000), and it has demonstrated very good behavior in accuracy and numerical stability.

Figure 2 shows the nodes contributions in a one-dimensional computational cell for QUICK, UNIFAES and 2nd Order Upwind schemes. The FLOW code uses only one grid for all variables: velocity component "*u*", velocity component *"v"* and pressure. This characterizes the collocated grid arrangement and as can be seen, special treatment must be done on boundary nodes when using a high order scheme.

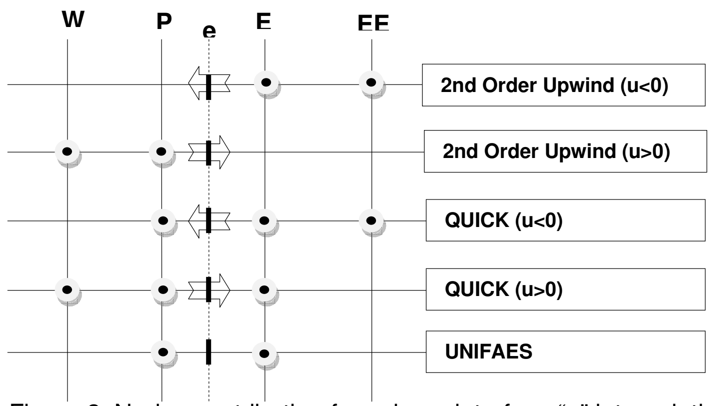


Figure 2. Nodes contribution for volume interface "*e*" interpolation

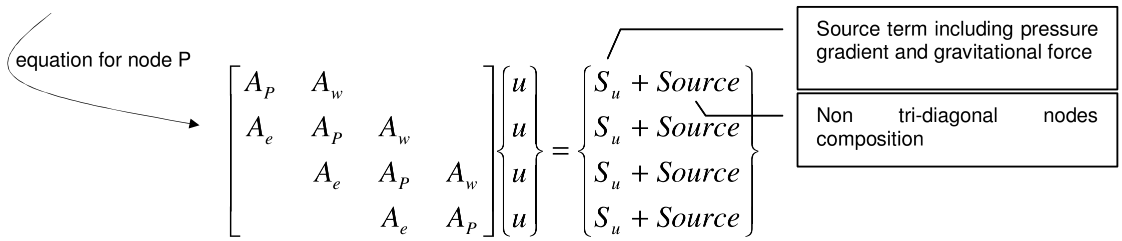
When the complete scheme formulation is applied on boundaries, eventually some cell nodes falls out of the computational domain due to orientation of the considered velocity which is the case of QUICK and 2nd Order Upwind. On boundaries where the complete formulation is possible to be applied, it was done, but on boundaries which it was not possible, only in this particular node was considered Central discretization scheme. Table (1) relates all cases where it happens.

Table 1. Boundaries with special treatment

Right wall		Left wall		Top		Bottom	
2nd Order Upwind	$u_e<0$	2nd Order Upwind	$u_w>0$	2nd Order Upwind	$v_s>0$	2nd Order Upwind	$v_n<0$
QUICK	$u_e<0$	QUICK	$u_w>0$	QUICK	$v_s>0$	QUICK	$v_n<0$

Analyzing figure (2) sketch for one-dimensional cells is easy to notice that for bi-dimensional cells for each node discretization will be needed more adjacent nodes, depending on the scheme and velocity orientation. The assembly of the global matrix may results in a penta-diagonal linear system. To normalize all schemes, the global matrix is forced to be tri-diagonal and a TDMA algorithm can be use to solve it. The contribution to global system of all extra nodes that do not belong to a tri-diagonal association is consider as second source term. Cho and Chung (1994) explains that if is use a penta-diagonal matrix, specially for pressure correction, the system becomes implicit and instable resulting in very small relaxation coefficients and the use of more robust system solver.

$$A_e u_E + A_P u_P + A_w u_W = A_n u_N + A_s u_S + A_{ne} u_{NE} + A_{nw} u_{NW} + A_{se} u_{SE} + A_{sw} u_{SW} + S_u$$



$$\begin{bmatrix} A_P & A_w & & \\ A_e & A_P & A_w & \\ & A_e & A_P & A_w \\ & & A_e & A_P \end{bmatrix} \begin{Bmatrix} u \\ u \\ u \\ u \end{Bmatrix} = \begin{Bmatrix} S_u + Source \\ S_u + Source \\ S_u + Source \\ S_u + Source \end{Bmatrix}$$

Figure 3. Global matrix assembly

2.4 Computational code fluxogram

Figure (4) shows the FLOW code fluxogram where can be identified some important procedures.

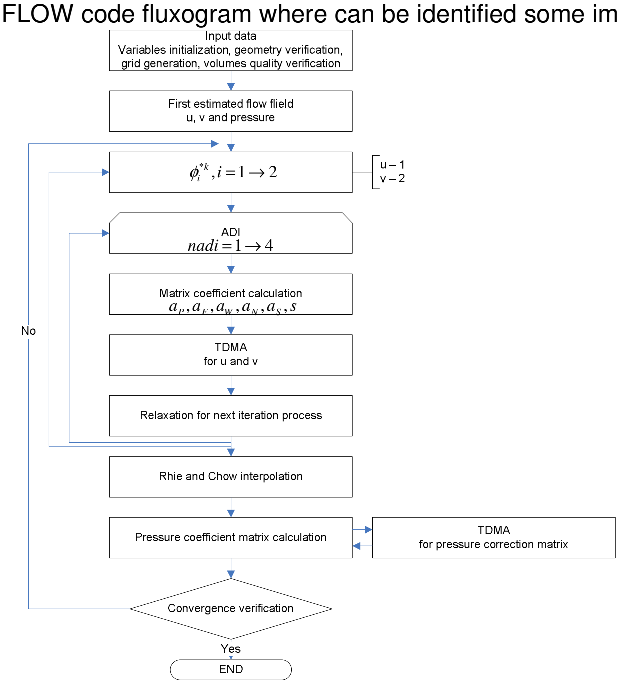


Figure 4. FLOW code fluxogram

The velocity-pressure coupling is provide by SIMPLE scheme and to prevent the checkboard pressure distribution that eventually arise from collocated grid arrangement, Rhie and Chow (1983) interpolation for velocities is used.

3. NUMERICAL RESULTS

Numerical solutions are obtained for primary variables and it's analysis is divided in four stages:

a) convergence speed that shows the scheme velocity to achieve convergence criteria at each test case,

b) velocities solutions that are presented as streamlines plot, numerical solution and extrapolation for center point cavity values, *u* velocity distribution along vertical center line and *v* velocity distribution along horizontal center line,

c) L_2 and L_∞ norm error analysis for *u* and *v* velocities along respective center lines, figure (5)

d) vortex locations, figure (5).

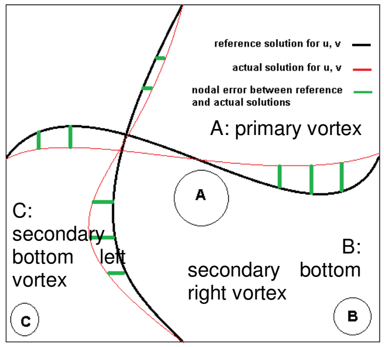


Figure 5. Error distribution for *u* and *v* velocities and vortex locations

The solution is considered satisfied if the convergence criteria described in eq (3) is satisfied for all variables at all nodes.

$$\varepsilon_{max} = \left| \frac{\overset{k}{\underset{i}{\phi}} - \overset{k-1}{\underset{i}{\phi}}}{\overset{k-1}{\underset{i}{\phi}}} \right|_{max} \leq 10^{-6} \qquad (3)$$

where: ϕ represents the velocities components and pressure
k, k-1 are actual and last iteration process
i is the node number, 1 ≤ *i* ≤ npoints

Table (2) relates all flow conditions, cavity configuration and grid refinement considered.

Table 2. Configurations for test cases

Reynolds number (based on top moving wall) {100, 500, 1000	**Grid refinement** (equally spaced) {30x30, 60x60	**Cavity deformation** α {90º, 60º, 45º, 30º

For convergence speed comparisons a reference value considered is the number of iterations for Central scheme discretization, 30x30 divisions grid, 90° cavity deformation and Re=100. The number of iterations for each case is related to reference by coefficient described at eq (4) and are all listed in table (3).

$$\beta = \frac{Iter}{Iter_{ref}} \qquad (4)$$

where:
Iter - iterations for current case
$Iter_{ref}$ - Iterations for reference case: Re=100, 90° cavity, central scheme, 30x30 grid.

Table 3. Iterations for convergence – best values highlighted

Re	α	QUICK 30x30	UNIFAES 30x30	2nd Order Upwind 30x30	QUICK 60x60	UNIFAES 60x60	2nd Order Upwind 60x60
100	90	1,0705	1,0589	1,0648	4,4076	4,4675	4,3456
100	60	1,3784	1,4129	1,4426	5,5775	5,1691	5,9505
100	45	1,7400	1,7328	1,7318	7,4571	6,5323	6,6101
100	30	2,0536	2,1969	2,2268	7,4956	7,6415	7,6951
500	90	1,4284	1,3874	1,5708	5,2264	6,0833	4,8659
500	60	1,3562	1,3153	1,0465	4,2474	3,8531	3,9478
500	45	1,1449	1,1965	1,3199	4,7972	4,8288	5,2857
500	30	1,5989	1,7114	1,9786	6,5040	6,6069	27,6336
1000	90	4,3196	2,6601	1,6603	7,8539	6,2362	9,6858
1000	60	2,2229	0,9713	1,1508	3,8439	3,5525	4,0895
1000	45	1,5707	1,5074	1,3214	18,1977	18,6976	15,6743
1000	30	2,1098	2,2749	2,0304	25,6794	24,7675	25,2525

The first set of results listed in table (3) shows the convergence velocity. In general among 24 test cases, UNIFAES was the best in 9 cases, 2nd Order Upwind was the best in 9 cases and QUICK the best in 6 cases. With the coarse grid size, 30x30, UNIFAES is the best in 3 cases and 6 cases with 60x60 grid size. For Re=1000 grid size 30x30 2nd Order Upwind is the best except for one case and with 60x60 grid size UNIFAES is the best also except for one case. For 30° cavity deformation QUICK is the best for 4 cases, UNIFAES and 2nd Order Upwind are the best for roughest case with higher Reynolds respectively for 60x60 and 30x30 grid size.

Qualitative plots of streamlines distribution are presented in figure (6) for solutions obtained with UNIFAES scheme, 60x60 grid size.

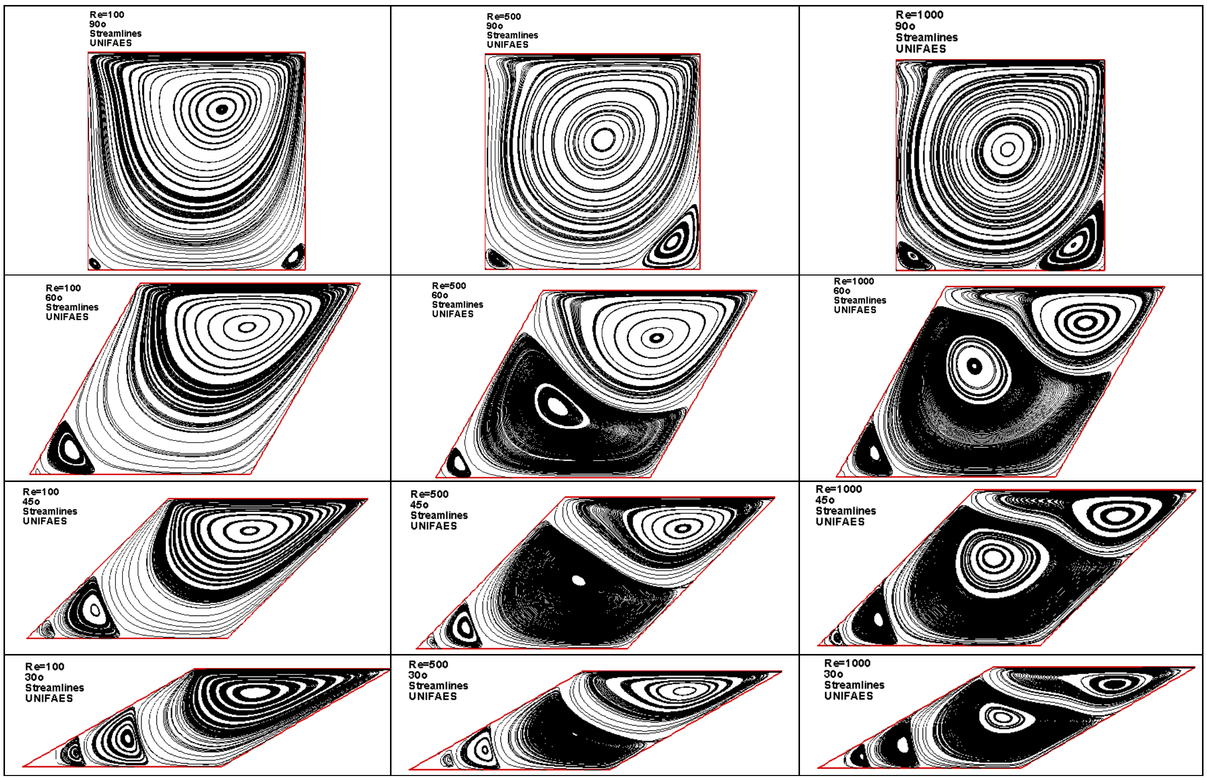


Figure 6. Streamlines for UNIFAES, 60x60 grid

Figure (6) represents the streamline distribution inside the cavity. Results looks to be physically realistic and do not shows wavy solutions for UNIFAES 60x60 grid size. Velocities components *u* and *v* are monitored at cavity center point. Solutions and extrapolated values are listed in table (4).

Table 4. Richardson second-order extrapolation for *u* and *v* velocities at center point position
QUICK

	u		*v*		Test case
Grid	Solution	Extrap.	Solution	Extrap.	
30x30	-0,00307073		0,0008472		Re100, 90°
60x60	-0,00322367	-0,00327548	0,0008832	0,0008954	Re100, 90°
30x30	-0,00187283		0,0013959		Re100, 30°
60x60	-0,00195389	-0,00198135	0,0014929	0,0015257	Re100, 30°
30x30	-0,00790846		0,0073137		Re1000, 90°
60x60	-0,00553893	-0,00473623	0,0061939	0,0058146	Re1000, 90°
30x30	-0,00081448		0,0002720		Re1000, 30°
60x60	-0,00003168	0,0002335	0,0003545	0,0003825	Re1000, 30°

UNIFAES

Grid	u Solution	u Extrap.	v Solution	v Extrap.	Test case
30x30	-0,00306331		0,0008504		Re100, 90°
60x60	-0,00322159	-0,00327521	0,0008837	0,0008949	Re100, 90°
30x30	-0,00187033		0,0013986		Re100, 30°
60x60	-0,00195319	-0,00198126	0,0014941	0,0015265	Re100, 30°
30x30	-0,01036427		0,0038747		Re1000, 90°
60x60	-0,01033611	-0,01032657	0,0034299	0,0032792	Re1000, 90°
30x30	-0,00083456		0,0002249		Re1000, 30°
60x60	-0,00001364	0,00026445	0,0003477	0,0003893	Re1000, 30°

2nd Order Upwind

Grid	u Solution	u Extrap.	v Solution	v Extrap.	Test case
30x30	-0,00314596		0,0008359		Re100, 90°
60x60	-0,00324218	-0,00327478	0,0008807	0,0008959	Re100, 90°
30x30	-0,00189308		0,0013660		Re100, 30°
60x60	-0,00195952	-0,00198203	0,0014800	0,0015186	Re100, 30°
30x30	-0,01745494		0,0039668		Re1000, 90°
60x60	-0,01238579	-0,01066857	0,0028698	0,0024981	Re1000, 90°
30x30	-0,00196373		0,0004170		Re1000, 30°
60x60	-0,00050121	-0,00000576	0,0003845	0,0003735	Re1000, 30°

Velocity comparisons are presented as distribution along center lines in figure 7. Numerical solutions obtained with QUICK, UNIFAES and 2nd Order Upwind with 30x30 grid size are compared to solutions obtained by Ghia *et al* (1982), 256x256 grid size and 90° cavity, Mariano *et al* (2010) IMERSPEC code using a Fourier pseudo spectral method with a 128x128 grid size for 90° cavity, and a finite element with SUPG (Streamline Upwind Petrov Galerkin) element type obtained with ANSYS with 200x200 grid size for all cases.

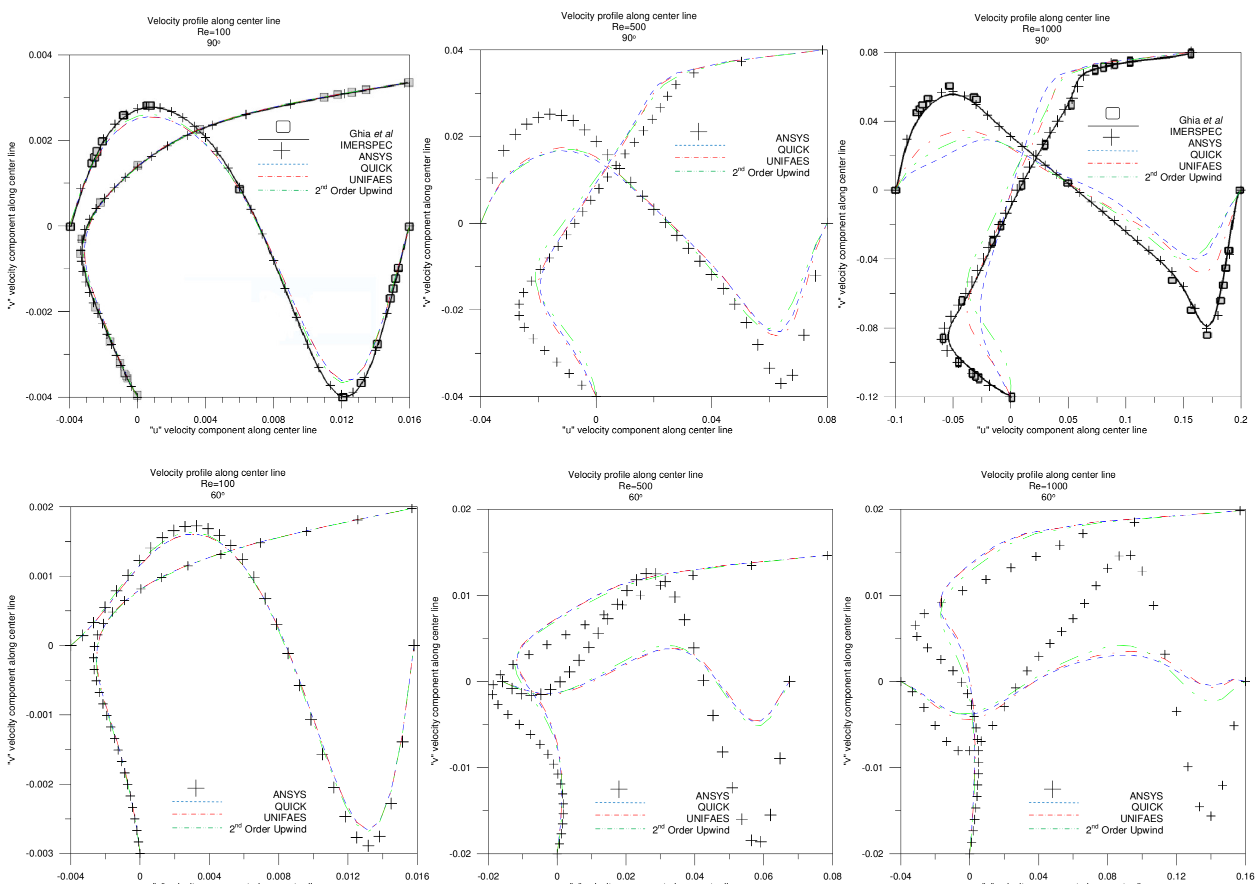


Figure 7. Velocity distribution along center lines

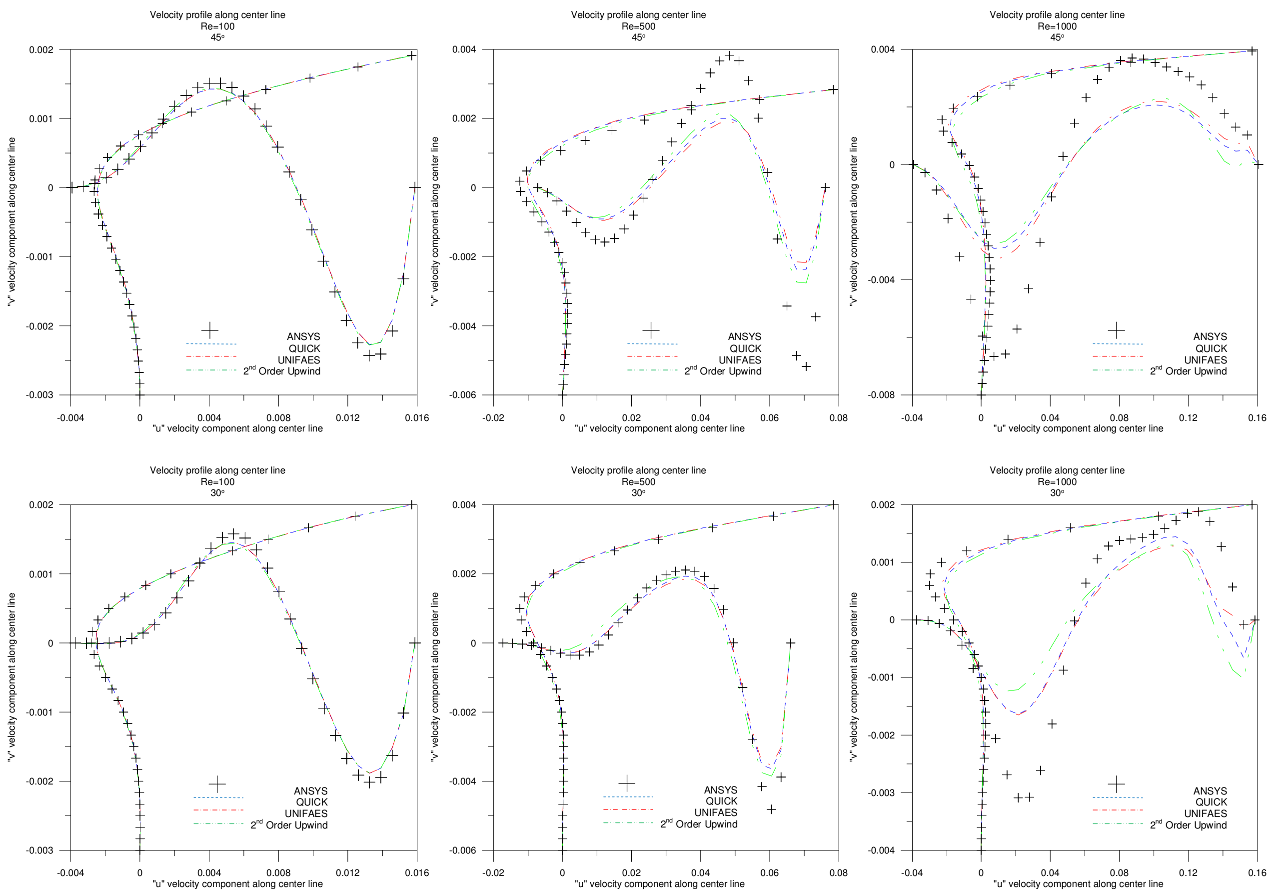


Figure 7 *(cont)*. Velocity distribution along center lines

Figure (7) shows the good agreement of velocity distribution along vertical and horizontal center lines to references solutions by Ghia *et al* (1982) and ANSYS. Velocity distribution along center lines is also compared to ANSYS® solutions considering L_2 and L_∞ norms as defined in eq (5) and eq (6) and described in figure (5). Table (5) presents the error calculations and the best value for each case is highlighted.

$$L_2 = \sqrt{\sum_{i=1}^{n} \left(\phi_i - \phi_{i,ref}\right)^2} \qquad (5)$$

$$L_\infty = \max_{1 \le i \le n} \left|\phi_i - \phi_{i,ref}\right| \qquad (6)$$

where:

ϕ is *"u"* velocity along vertical center line and *"v"* velocity along horizontal center line
i is the nodal solution for all nodes along vertical and horizontal center lines
n is the number of nodes along vertical and horizontal center lines

Table 5. L_2 error for *"u"* and *"v"* velocity component along center lines
Best values highlighted

Re	α	QUICK 30x30	UNIFAES 30x30	2nd Order Upwind 30x30	QUICK 60x60	UNIFAES 60x60	2nd Order Upwind 60x60
100	90	1,354E-03	1,382E-03	1,045E-03	3,098E-04	3,169E-04	2,477E-04
100	60	9,085E-04	9,250E-04	7,592E-04	2,143E-04	2,171E-04	1,979E-04
100	45	7,006E-02	7,064E-04	6,770E-04	1,647E-04	1,644E-04	1,781E-04
100	30	6,834E-04	6,798E-04	7,443E-04	1,783E-04	1,763E-04	2,056E-04
500	90	4,695E-02	4,231E-02	5,097E-02	1,394E-02	1,407E-02	1,331E-02
500	60	5,702E-02	5,786E-02	5,260E-02	2,868E-02	2,887E-02	3,125E-02
500	45	1,634E-02	1,767E-02	1,268E-02	7,877E-03	8,192E-03	6,995E-03
500	30	6,146E-03	6,451E-03	5,751E-03	2,954E-03	3,065E-03	2,403E-03
1000	90	1,578E-01	1,154E-01	1,553E-01	5,280E-02	4,369E-02	5,207E-02
1000	60	9,179E-02	9,078E-02	7,525E-02	9,432E-02	9,570E-02	1,022E-01
1000	45	2,039E-02	2,262E-02	2,516E-02	2,757E-02	3,070E-02	2,955E-02
1000	30	2,226E-02	1,840E-02	2,897E-02	5,696E-03	6,087E-03	5,386E-03

L_∞ error for *"u"* and *"v"* velocity component center lines
Best values highlighted

Re	α	QUICK 30x30	UNIFAES 30x30	2nd Order Upwind 30x30	QUICK 60x60	UNIFAES 60x60	2nd Order Upwind 60x60
100	90	3,197E-04	3,260E-04	2,793E-04	7,644E-05	7,751E-05	6,946E-05
100	60	2,226E-04	2,237E-04	2,011E-04	5,293E-05	5,315E-05	5,214E-05
100	45	1,954E-04	1,943E-04	2,119E-04	5,702E-05	5,635E-05	6,530E-05
100	30	2,618E-04	2,600E-04	2,805E-04	7,788E-05	7,687E-05	8,808E-05
500	90	1,364E-02	1,169E-02	1,669E-02	4,400E-03	4,397E-03	5,101E-03
500	60	1,405E-02	1,403E-02	1,349E-02	7,012E-03	7,014E-03	7,910E-03
500	45	6,935E-03	7,455E-03	5,164E-03	3,107E-03	3,222E-03	2,673E-03
500	30	2,639E-03	2,731E-03	2,148E-03	1,270E-03	1,312E-03	1,026E-03
1000	90	4,776E-02	3,284E-02	5,019E-02	1,918E-02	1,684E-02	2,310E-02
1000	60	3,682E-02	3,517E-02	3,047E-02	3,233E-02	3,293E-02	3,718E-02
1000	45	8,913E-03	1,167E-02	8,381E-03	1,553E-02	1,730E-02	1,712E-02
1000	30	1,083E-02	8,583E-03	1,426E-02	3,059E-03	3,389E-03	2,456E-03

Table (5) shows the L_2 and L_∞ errors for *u* and *v* velocities along center lines. From 24 test cases, for L_2 error measure UNIFAES is the best in 7 cases, 2nd Order Upwind is the best in 13 cases and QUICK is the best in 4 cases and for L_∞ error measure UNIFAES is the best in 9 cases, 2nd Order Upwind is the best in 12 cases and QUICK is the best in 3 cases. L_2 error means that all curve is near the reference and the L_∞ error means the most distant value from reference. At this point of view, the best scheme is the one which both errors are smaller. This happens in 11 cases for 2nd Order Upwind, 6 cases for UNIFAES and 3 cases for QUICK. All data in table (5) is calculated considering ANSYS solutions as reference.

The final comparison to be done is the vortex identification. As described in figure (5) there are three recirculation zones that arise with flow distribution. Table (6) shows the center position of each vortex, if detected. As FLOW code solves the governing equations for primary variables and for grid refinement as described before, the vortex center is identified using TECPLOT® software with a 10^{-8} tolerance. Computed values are compared to four different references for 90° cavity.

Table 6. Vortex locations for 90º cavity flow – best values highlighted
(x,y positions are normalized)

Re	Reference	Central vortex X	y	Dist.	Left vortex x	y	Dist.	Right vortex x	Y	Dist.
100	Ghia *et al* (1982)	0,6172	0,7344		0,0313	0,0391		0,9453	0,0625	
	Hou *et al* (1995)	0,6196	0,7373		0,0353	0,0350		0,9451	0,0627	
	Arruda (2004)	0,6200	0,7800		na	na		na	na	
	Mariano *et al* (2010)	0,6172	0,7365		0,022	0,017		0,9525	0,0634	
	UNIFAES 30x30	0,6213	0,7421	0.0087	nd	nd	nd	0,9518	0,0535	0.0111
	QUICK 30x30	0,6209	0,7418	0.0083	nd	nd	nd	0,9515	0,0538	0.0106
	2nd Order Upwind 30x30	0,6186	0,7388	0.0046	nd	nd	nd	0,9491	0,0570	0.0067
	UNIFAES 60x60	0,6174	0,7389	0.0045	0,0334	0,0334	0.0061	0,9441	0,0609	0.0020
	QUICK 60x60	0,6178	0,7389	0.0045	0,0339	0,0332	0.0064	0,9440	0,0610	0.0020
	2nd Order Upwind 60x60	0,6169	0,7380	0.0044	0,0337	0,0331	0.0064	0,9435	0,0618	0.0019
1000	Ghia *et al* (1982)	0,5313	0,5625		0,0859	0,0781		0,8594	0,1094	
	Hou *et al* (1995)	0,5333	0,5647		0,0902	0,0784		0,8667	0,1137	
	Arruda (2004)	0,5300	0,5800		0,0900	0,0800		0,8400	0,1300	
	Mariano *et al* (2010)	0,5269	0,5612		0,1535	0,0699		0,8361	0,1151	
	UNIFAES 30x30	0,5485	0,5952	0.0369	0,0790	0,0639	0.0158	0,8454	0,1334	0.0240
	QUICK 30x30	0,5863	0,5862	0.0599	nd	nd	nd	0,8505	0,1262	0.0190
	2nd Order Upwind 30x30	0,5495	0,6324	0.0722	0,1297	0,1030	0.0504	0,8099	0,1452	0.0610
	UNIFAES 60x60	0,5332	0,5766	0.0142	0,0799	0,0714	0.0089	0,8493	0,1227	0.0167
	QUICK 60x60	0,5499	0,5525	0.0211	0,0850	0,0812	0.0032	0,8623	0,1209	0.0119
	2nd Order Upwind 60x60	0,5298	0,5845	0.0220	0,0835	0,0750	0.0039	0,8386	0,1263	0.0268

Grid size: Mariano *et al* (2010) - 128x128, Ghia *et al* (1982) – 256x256, Arruda (2004) – , Hou *et al* (1995) – 256x256
nd – not detected na – not available best values compared to Ghia *et al* (1982)

Table (6) shows the vortex center position x, y which ones are plotted in figure (6). For Re=100, 30x30 grid size none of the schemes could predict vortex on the left side of cavity, and for Re=1000, 30x30 grid size QUICK is the one that could not predict vortex on the left side of cavity. All the other test cases all vortex region could be predicted by all schemes.

4. CONCLUSION

All numerical solutions performed have a single purpose; evaluate the scheme behavior in problems with some kind of special aspect. Here were used the finite element based code ANSYS to generate results for comparative issues, IMERSPEC code results obtained by Mariano *et al* (2010) were used as comparative values as well. These two set of results were obtained with refined grids 200x200 and 128x128 respectively. FLOW code results were obtained for 30x30 and 60x60 grid size.

The main objective here is compare UNIFAES to QUICK and 2nd Order Upwind schemes. These two last schemes are older ones and already have proved its quality and today they are incorporated to FLUENT® software that is a very large and complete simulation tool. UNIFAES has been tested for some cases considering the whole universe of fluid flow problems and always had a very good behavior in convergence and accuracy, not different in this particular study.

Again UNIFAES shows to be a very robust scheme compared to two traditional ones, QUICK and 2nd Order Upwind. Results demonstrated good behavior in convergence and accuracy to all cases with coarser and finer grids, lower and higher Reynolds.

Among 24 comparisons UNIFAES is the best in 7 cases, 2nd Order Upwind is the best in 6 cases, QUICK is the best in 8 cases and QUICK and 2nd Order Upwind are tied in 1 case. These comparisons are separated for "x" and "y" coordinates, the best approximation for each dimension. Considering the distance vector from center point (calculated)-to-center point (reference) from 11 cases UNIFAES is the best in 4 cases mostly when Re=1000, 2nd Order Upwind is the best in 4 cases all when Re=100 and QUICK is the best in 3 cases all when Re=1000.

5. REFERENCES

Arruda, J.; "Modelagem Matemática de Escoamentos Internos Forçados Utilizando o Método da Fronteira Imersa e o Modelo Físico Virtual", Tese (Doutorado em Mecânica) - Faculdade de Engenheira Mecânica, Universidade Federal de Uberlândia, Uberlândia, 153 f., 2004.

Cho, M. J., Chung, M. K.; "New Treatment of Non-orthogonal Terms in the Pressure-Correction Equation", Numerical Heat Transfer, Part B, vol. 26, pp. 133-145, 1994.

Figueiredo, J. R., Llagostera, J.; "Comparative Study of the Unified Finite Approaches Exponential-Type Scheme (UNIFAES) and Its Application to Natural Convection in Porous Cavity", Numerical Heat Transfer, part. B, vol. 35, pp. 347-367, 1999.

Figueiredo, J. R.; "A Unified Finite-Volume Finite-Differencing Exponential-Type Scheme for Convective-Diffusive Fluid Transport Equations", Journal of Brazilian Society of Mechanical Sciences, vol. 19, n. 03, pp. 371-391, 1997.

Ghia, U., Ghia, K. N., Shin, C. T.; "High Resolutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method", Journal of Computational Physics, vol. 48, pp. 387–411, 1982.

Hou, S., Zou, Q., Chen, S., Doolen, G., Cogley, A.; "Simulation of Cavity Flow by the Lattice Boltzmann Method", Journal of Computational Physics, vol. 118, pp. 329–347, 1995.

Idelsohn, S.R., Onate, E.; "Finite Volumes and Finite Elements: Two `Good Friends*",* International Journal for Numerical Methods in Engineering*,* vol. 37, pp 3323-3341, 1994.

Leonard, B. P., Drummond, J. E.; "Why Should You Not Use Hybrid, Power-Law or Related Exponential Schemes for Convective Modeling - There Are Much Better Alternatives", pp. 421-442, 1995.

Leonard, B. P.; "A Stable and Accurate Convective Modelling Procedure Based on Quadratic Upstream Interpolation", Computational Methods Applied to Mechanical Engineering, Vol. 19, pp. 59-98, 1979.

Leonard, B. P.; "Simple High-Accuracy Resolution Program for Convective Modelling of Discontinuities", International Journal for Numerical Methods in Fluids, Vol. 8, pp. 1291-1318, 1988.

Llagostera, J., Figueiredo, J. R.; "Application of the UNIFAES Discretization Scheme to Mixed Convection in a Porous Layer with a Cavity Using the Darcy Model", Journal of Porous Media, vol. 3, pp. 139-154, 2000.

Mariano, F. P., Moreira, L. Q., Silveira-Neto, A., Silva, C. B., Pereira, J. C. F.; "A New Incompressible Navier-Stokes Solver Combining Fourier Pseudo-Spectral and Immersed Boundary Methods", CMES, vol.59, n..2, pp.181-216, 2010.

Rhie, C. M., Chow, W. L.; "Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation", AIAA Journal, vol. 21, pp. 1525-1532, 1983.

Vilela, C. A. A., Figueiredo, J. R.; "Avaliação de Esquemas de Discretização Numérica para Equação de Transporte Convectivo-Difusivo em Coordenadas Generalizadas", II Congresso Nacional de Engenharia Mecânica (CONEM), João Pessoa – PB – Brasil, 2002.

Vilela, C. A. A.; "Simulação de Escoamento em Geometrias Complexas Utilizando o Método dos Volumes Finitos com o Esquema UNIFAES de Discretização", Tese de doutorado apresentada em 2001 no Departamento de Energia da Faculdade de Engenharia Mecânica da Universidade Estadual de Campinas, 2001.

Wong, H. H., Raithby, G. D.; "Improved Finite-Difference Methods Based on a Critical Evaluation on the Approximation Errors", Numerical Heat Transfer, vol. 2, pp. 139–163, 1979.