Proceedings of the 4th WSEAS International Conference on Fluid Mechanics and Aerodynamics, Elounda, Greece, August 21-23, 2006 (pp48-53)

A two dimensional unstructured staggered mesh method with special treatment of tangential velocity

MEHMET SHALA, KOULIS PERICLEOUS, MAYUR PATEL
School of Computing and Mathematical Sciences
University of Greenwich
Old Royal Naval College, Park Row, London, SE10 9LS
UNITED KINGDOM

Abstract: - A two dimensional staggered unstructured discretisation scheme for the solution of fluid flow problems has been developed. This scheme stores and solves the velocity vector resolutes normal and parallel to each cell face and other scalar variables (pressure, temperature) are stored at cell centres. The coupled momentum; continuity and energy equations are solved, using the well known pressure correction algorithm SIMPLE. The method is tested for accuracy and convergence behaviour against standard cell-centre solutions in a number of benchmark problems: The Lid-Driven Cavity, Natural Convection in a Cavity and the Melting of Gallium in a rectangular domain.

Key-Words: - Flow solution, momentum, energy, unstructured, staggered mesh, numerical schemes

1 Introduction

There are many branches of engineering science that require solution of fluid flow problems. Some of these flows may involve complex geometrical shapes which are usually modelled using the unstructured mesh discretisation techniques. There are well established cell-centred methods that are used in such simulations. However, these suffer from poor convergence compared to structured staggered techniques.

The aim of this project is to investigate the staggered positioning of variables on an unstructured based context and so inheriting the advantages of complex geometry characterization and staggered mesh stability. A two dimensional staggered unstructured discretisation scheme for the solution of fluid flow and heat transfer problems has been developed for this purpose. Two possible methods are discussed with many ways of dealing with the convection term in the momentum equation.

The new scheme stores and solves the velocity vector resolutes normal and parallel to each cell face and other scalar variables (pressure, temperature) are stored at cell centres. The coupled momentum; continuity and energy equations are solved, using Patankar's [1] pressure correction algorithm SIMPLE.

Standard benchmark test cases such as: the lid driven cavity, natural convection in a cavity and melting of a metal in a rectangular region are simulated and the results compared against benchmark solutions.

2 The staggered mesh

A Cartesian staggered mesh method was originally developed by Harlow & Welch [2] in 1965 for finite difference methods. This method was later developed in a finite volume context by Patankar and Spalding [1].

In the staggered scheme proposed by Harlow and Welch pressure is located at cell centres and velocity is distributed to cell faces in which case only the normal component of the velocity at each cell face is known.

There are many possible staggering schemes possible. A staggered mesh scheme is any numerical scheme where variables are located at different points within the mesh.

The Cartesian staggered mesh method has a number of interesting mathematical properties. In particular the method does not have spurious 'pressure modes' and does not require stabilizations to control unphysical small-scale pressure fluctuations (see [1]).

The staggered mesh approach on unstructured meshes had only been at the attention of researchers in the last decade. This is due to the fact that such approach was numerically difficult.

The covolume method was the first attempt to extend the structured staggered mesh approach of Harlow & Welch to unstructured staggered

approach. The newest unstructured staggered mesh methods were developed by Perot [4] and Wenneker [5, 6]

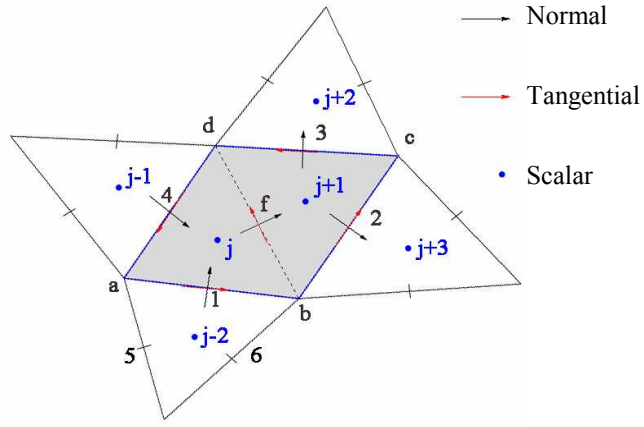

Fig.1 Unstructured staggered mesh

3 Governing Equations

In this section the discretisation of the incompressible Navier-Stokes and continuity equations is presented. The discretisation is based on the method developed by Wenneker [5] in that it uses the same control volume for the momentum equation. Such a control volume consists of two adjacent triangles (the shaded region in Fig.1). This method is generally more attractive than Perot's [4] since it is not restricted to a Delaunay/Voronoi mesh.

The Continuity equation has the form:

$$div(\rho u) = 0 \qquad (1)$$

The incompressible Navier-Stokes equation has the form:

$$\frac{\partial \rho u}{\partial t} + div(\rho \underline{u} u) = div(\mu grad(u)) - \Delta p + S_u \quad (2)$$

where μ is the fluid viscosity and S_u is a momentum source term, u represents the normal velocity u, t is the time step and $\underline{u}$ is the field velocity.

The difficulty arising with flow problems is the role played by the pressure. In the compressible flow case the continuity equation represents an evolution equation for density. The pressure can then be obtained form the equation of state that relates pressure with temperature and density as.

In the incompressible flow the density is constant and hence is not linked to pressure. Coupling of pressure-velocity gives rise to the constraint that if the correct pressure field is applied to the momentum equation (2), then the velocity field should satisfy continuity.

This coupling is achieved using the SIMPLE algorithm developed by Patankar and Spalding [2]. This method transforms the continuity equation in an equation for the pressure correction.

In the method presented by Perot [4] and Wenneker [6], the tangential velocity component is interpolated. In Wenneker [6] the primary variable for momentum is $m=\rho u$. In the staggered mesh methods discussed here the primary variable for momentum is u. Furthermore there are a few choices on how the convection term is evaluated. Those are discussed later in this section.

The energy equation expressed in terms of enthalpy, $h=c_pT$ has the following form:

$$\frac{\partial(\rho h)}{\partial t} + div(\rho \underline{u} h) = div\left(\Gamma grad\left(\frac{h}{c_p}\right)\right) + S_k \quad (3)$$

where ρ is the density, $\underline{u}$ the velocity vector, Γ the effective diffusion coefficient, T the temperature and S_k any other heat source terms. The form taken by the diffusion coefficient Γ depends on whether laminar or turbulent assumptions are being used. In the case of laminar flow the diffusion coefficient is equal to the thermal conductivity, k.

4 Discretisation

The continuity equation (1) is integrated over the control volume (a single triangle) to yield:

$$\int_V div(\rho u)\, dx = \sum_{i=1}^{3} \rho u A = 0 \qquad (4)$$

where u_i is the velocity at face i, A_i is the length of the face i and ρ_i is the face density. The summation runs over three faces of the control volume V.

Note that the velocity u_i is already located at the appropriate place, hence no interpolation is required.

The momentum equation (2) is integrated over the control volume consisting of two adjacent triangles. The shaded area in Fig. 1 represents a control volume for momentum equation at faces i.

Integrating (2) over the control volume V and applying the divergence theorem we get:

$$V_i \frac{(\rho u)_f^{n+1} - (\rho u)_f^n}{\Delta t} + \sum_{i=1}^{4} \rho_i u_i \left(\underline{u}.n_f\right)_i A_i = \sum_{i=1}^{4} \mu_i \left(\frac{\partial u}{\partial n}\right)_i A_i - \left(\frac{p_{j+1} - p_j}{d_i}\right) V_i \qquad (5)$$

The summation runs over four faces of control volumeV_i, A_i is the area of the face, n_f is the normal vector at face f, μ_i is the viscosity coefficient, d_i is the distance between two neighbouring cell centres and ρ_i is the cell face value of density.

The velocity gradient at face I is approximated as:

$$\left(\frac{\partial u}{\partial n}\right)_i = \frac{u_j - u_{j-2}}{d_1} \quad (6)$$

With d_1 being the distance between cell j and cell j-2.For the evaluation of the convection term a number of ways is possible. These differ on how the velocity vector $\underline{u}.n_f$ at face i is calculated. The first one would be to use an upwind method similar to the one used by Wenneker [5]. A second way to evaluate the velocity vector at face 1 is by decomposing into its normal and tangential velocity components as follows:

$$\underline{u}_1 = u_1 + v_1 \quad (7)$$

The tangential velocity vector can then be obtained in a number of ways. In the case of interpolation an upwind method similar to Wenneker [5] is used. The normal velocity component u_1 in (7) is given the exact solution value.

Alternatively the tangential velocity component is solved directly using equation (2) as follows

$$\frac{\partial \rho v}{\partial t} + div\left(\rho \underline{u} v\right) = div\left(\mu grad\left(v\right)\right) - \left(\nabla p\right) + S_v \quad (8)$$

Equation (8) represents the transport equation for the tangential momentum in a cell.

Circulation is the line integral of a vector field around a closed path. As such, it is another way to measure the amount of ``swirl'' in a vector field. Often, circulation is defined by a line integral as follows:

$$\oint_C \underline{V}.dr \quad (9)$$

where C is a closed region and $\underline{V}$ is the velocity field. In Fig. 1, the line integral around cell j is defined as:

$$\sum_{i=1}^{3} \int_C \underline{V}.dr \quad (10)$$

Since the normal velocity is perpendicular to the tangential velocity, the normal velocity contribution equals zero. What is left is a relation of the tangential velocities in cell j. Expression (10) will be used in the tangential momentum equation to arrive at an expression for the v_f velocity.

Integrating of (8) for the tangential velocity component with over the control volume V and after applying the divergence theorem gives:

$$V_i \frac{(\rho v)_i^{n+1} - (\rho v)_i^n}{\Delta t} + \sum_{i=1}^{4} \rho_i v_i \left(\underline{u} \cdot n_f\right)_i A_i = \sum_{i=1}^{4} \mu_i \left(\frac{\partial v}{\partial t}\right)_i A_i - \left(\frac{p_d - p_b}{d_{bd}}\right) V_i \quad (11)$$

The summation runs over four faces of control volume V_i, A_i, is the area of the face and p_b and p_d are nodal values of pressure. The gradient of the velocity at face 1 is evaluated as:

$$\left(\frac{\partial v}{\partial t}\right)_1 = \frac{v_b - v_a}{d_1} \quad (12)$$

With d_1 being the distance between nodes b and a.

Diffusion Term

The coefficient of viscosity μ_i is computed using its harmonic mean at the cell face. Approximations for the cell centre velocities are also required. This can be done by the use of a central averaging as:

$$u_j = \frac{1}{3}\left(u_1 + u_f + u_4\right) \quad (13)$$

Where u_1, u_f and u_4 are the face velocities in a given triangle cell j. The same method is used to interpolate the cell values of tangential velocity.

The appropriate direction is taken into account thus for relation (13) velocity vectors at the faces of cell j are resolved into the direction of the velocity n_f being solved as follows:

$$u_j = \frac{1}{3}\left[u_1\left(n_1.n_f\right) + u_f\left(n_f.n_f\right) + u_4\left(n_4.n_f\right)\right] \quad (14)$$

With n_1, n_f and n_4 being the direction in which the normal velocity at each face is solved.The nodal pressure values p_b and p_d are interpolated using the *Inverse Distance Weighting*.In case of tangential velocity gradient, a similar procedure as (14) is used to extrapolate tangential velocity values from cell faces to nodes. Again the averaging tangential velocity components are resolved in the direction of the tangential velocity being solved.

5 Results- model validation

The governing equations were discretised on a two-dimensional unstructured mesh leading to a set of finite algebraic equations that can be solved iteratively using standard linear equation solvers.

This section will assess the accuracy of the method by comparing the numerical results with:

- The cell-centred method that uses the Rhie-Chow interpolation method and
- The same unstructured staggered method but with interpolation of tangential velocity.

Three test cases are given. For each case three methods are compared to each other. They are the cell-centred approach, the staggered approach with interpolated tangential velocities and the third one is the staggered approach with solved tangential velocities.

For the Cell-Centred method the momentum and pressure equations are solved using the SOR (successive-over relaxation) method and JOR (Jacobi-over relaxed) methods respectively.

For both Staggered methods the matrix that is obtained for momentum is asymmetric and therefore solved using a Gauss-Jordan direct solver. This method may not be useful when dealing with very large matrices. Other iterative methods like Bi-CGSTAB and GMRES are the ideal methods for solving large, sparse and asymmetric matrices. All simulations run with unstructured staggered mesh methods used the Bi-CGSTAB solver.

5.1 Lid Driven Cavity

A 2D square cavity (1mx1m) is chosen. The top wall of the cavity is the moving wall. All other walls are stationary. The velocity on the moving wall is varied to arrive at the desired Reynolds number.

For this test case, the results are compared against those obtained by Ghia et al. [7] but the mesh used in our case is very different from that they used, ours being an orthogonal mesh consisting of 1462 triangles and 2233 faces.

5.1.1 Results for Re = 100

The velocities of the moving wall are u = 0[m/s] and v = 1[m/s]. The other boundaries are set to 0.0. The density is set to 1 kg m^{-3} and the laminar viscosity is set to 0.01 kg sec^{-1}. The specified u and v velocities are the normal and tangential velocity components and not the Cartesian components.

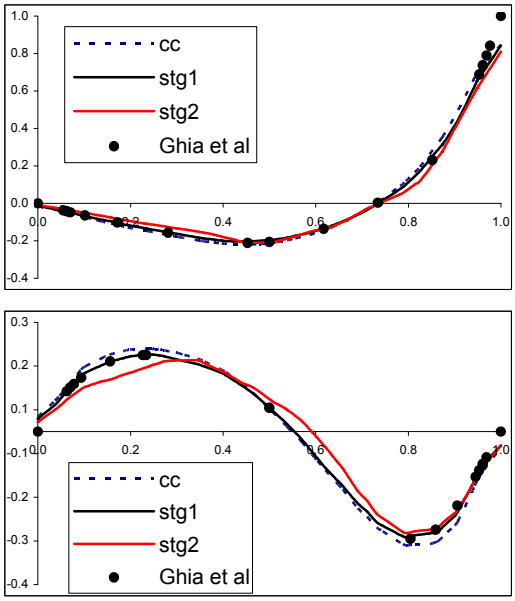

Fig. 2 Comparison of u and v velocities

For a more appropriate comparison a line is plotted along the centre of cavity. This is done for both u and v velocities. The lines obtained are plotted on the Graphs given in Fig. 2.

For Re=100, the solutions seem to be in good agreement with benchmark solutions of Ghia et al [7] apart form the staggered method with solved v-velocity. Results obtained with this method are a little different due to the fact that there is no way to ensure that circulation is satisfied for every given cell in a domain. It must be noted however that the results obtained with the staggered mesh method by interpolation v- velocities are better than results obtained using the cell-centred method.

	cc	stg1	stg2
Time(sec.)	32	302	507
Iterations	1028	884	1011
Memory(bytes)	81454	108730	108730

Table1. Summary of simulation properties

In Table 1 a summary of run- time information is gathered. The time taken to run this test case using the cell-centred method is much smaller then for both the other methods. This is due to the fact that the cell-centred method utilises solvers such as SOR and JOR whereas the other two methods utilise the Bi-CGSTAB method. Also in terms of memory requirements the cell-centred method uses less memory than the staggered methods. The reason behind that is that the code used to run simulations was explicitly written to be able to run the cell-centred method. The added routines to the code required more memory assignment. On the positive side, the staggered methods take fewer iterations to converge. Note that the staggered method with solved tangential velocity does not give the desired results for larger values of Re. This is subject to further research.

5.2 Natural Convection in a Cavity

De Vahl Davis and Jones [8-9] suggested that buoyancy-driven flow in a square cavity would be a suitable test case for validation of CFD codes. The flow is driven by differing temperature along two opposite walls that leads to a thermal gradient across the solution domain. The flow is assumed incompressible and the Boussinesq method is used to approximate the buoyancy forces. The approximation results in a source per unit volume of the form:

$$S_i = -\rho\beta g_i \left(T - T_{ref}\right) \tag{15}$$

Where β is the thermal coefficient of volumetric expansion, g_i is the component of gravity in the i'th direction and T_{ref} is the reference temperature. This source term is added to the momentum equation. In

the cell-centred method the source term is added in the y- direction. In the method with the tangential velocity interpolation the source term is resolved in the direction of the solved normal velocity. When the tangential velocity is also solved, then an extra term is added and resolved in the direction of the solved tangential velocity. The material properties are those for air at 300K. The normalised distances and velocities are used. The desired Rayleigh numbers can be obtained by varying the temperature on each of the vertical walls. Three different meshes are used for the simulations with Ra = 10^3, 10^4, 10^5 and 10^6. The first mesh (M1) had 20 divisions on each side with 1462 triangles and 2233 faces. The second mesh (M2) had 30 divisions on each side with 3294 triangles and 5001 faces. The third mesh (M3) had 40 divisions on each side with 5890 triangles and 8915 faces.

	Ra=10^3				Ra=10^4			
	cc	stg1	stg2	De Vahl	cc	stg1	stg2	De Vahl
U	3.595	3.487	3.768	(3.649)	15.681	15.695	16.259	(16.178)
Y	0.194	0.194	0.194	(0.187)	0.175	0.175	0.194	(0.177)
V	3.640	3.481	3.554	(3.692)	19.262	18.551	18.696	(19.617)
X	0.184	0.155	0.155	(0.167)	0.105	0.137	0.105	(0.119)
	Ra=10^5				Ra=10^6			
	cc	stg1	stg2	De Vahl	cc	stg1	stg2	De Vahl
U	33.832	36.058	40.835	(34.73)	67.973	100.136	82.684	(64.63)
Y	0.134	0.158	0.126	(0.145)	0.158	0.175	0.126	(0.15)
V	63.397	62.383	65.479	(68.59)	205.388	211.791	188.138	(219.36)
X	0.063	0.063	0.057	(0.066)	0.037	0.037	0.037	(0.0379)

Table2. Maximum values for mesh M1

	Ra=10^3				Ra=10^4			
	cc	stg1	stg2	De Vahl	cc	stg1	stg2	De Vahl
U	3.648	3.472	3.664	(3.649)	15.767	15.783	16.792	(16.178)
Y	0.202	0.176	0.176	(0.187)	0.176	0.172	0.172	(0.177)
V	3.700	3.475	3.685	(3.692)	19.241	18.588	18.640	(19.617)
X	0.179	0.179	0.179	(0.167)	0.105	0.105	0.134	(0.119)
	Ra=10^5				Ra=10^6			
	cc	stg1	stg2	De Vahl	cc	stg1	stg2	De Vahl
U	33.854	35.931	40.269	(34.73)	65.729	77.299	38.811	(64.63)
Y	0.148	0.148	0.148	(0.145)	0.123	0.148	0.054	(0.15)
V	67.272	66.443	68.732	(68.59)	211.699	222.248	214.216	(219.36)
X	0.069	0.069	0.069	(0.066)	0.044	0.044	0.046	(0.0379)

Table3. Maximum values for mesh M2

	Ra=10^3				Ra=10^4			
	cc	stg1	stg2	De Vahl	cc	stg1	stg2	De Vahl
U	3.677	3.517	3.737	(3.649)	16.026	15.793	16.803	(16.178)
Y	0.181	0.181	0.190	(0.187)	0.181	0.181	0.181	(0.177)
V	3.731	3.520	3.651	(3.692)	19.474	18.776	19.216	(19.617)
X	0.177	0.174	0.174	(0.167)	0.119	0.119	0.108	(0.119)
	Ra=10^5				Ra=10^6			
	cc	stg1	stg2	De Vahl	cc	stg1	stg2	De Vahl
U	34.148	33.108	38.976	(34.73)	67.833	83.939	88.713	(64.63)
Y	0.151	0.175	0.181	(0.145)	0.181	0.216	0.181	(0.15)
V	67.313	66.504	73.334	(68.59)	215.634	223.080	221.672	(219.36)
X	0.073	0.073	0.073	(0.066)	0.044	0.044	0.044	(0.0379)

Table4. Maximum values for mesh M3

A comparison is made between three CFD solvers and the benchmark solutions in Table 2 - 4. Generally speaking as it is the case in CFD the finer mesh M3 gives better results than the other two meshes. There is a good agreement with benchmark solutions throughout the range of Rayleigh Numbers, up to a Rayleigh number of 10^6, the staggered mesh methods give very poor results which are not comparable to the cell-centred method. The cell-centred method in most of the cases gives better results than the staggered mesh methods.

5.3 Melting of pure gallium

In this part the melting of pure gallium in a rectangular domain is discussed. As in the previous case in 5.2 this case also deals with natural-convection fluid flow and heat transfer. The problem is transient, highly non-linear and coupled physics phenomena. A source term that needs to be evaluated in addition to the Buoyancy source is the solidification source term that goes in the energy equation (3). The algorithm for discretisation of the solidification source term is the enthalpy-based approach of Voller et al [10]. The solidification source term is:

$$S_k = -\frac{\partial(L\rho f)}{\partial t} - div(L\rho \underline{u} f) \qquad (16)$$

Where where L is the latent heat of solidification, ρ is the density and $\underline{u}$ is the velocity vector, f is the liquid fraction.

In order to be able to monitor the material as it undergoes a phase change from liquid state to solid state and vice-versa a Darcy Source term is used to suppress or initiate the velocity components.

The domain of interest is rectangular and where by the top and bottom walls are perfectly insulated and the two vertical walls are one cold and the other hot. The cold wall is below the melting temperature of gallium and the hot wall is above the melting temperature. The experimental study of this case was conducted by Gau and Viskanta [11]. The results they published were the melting fronts of the metal at various times. This makes it possible to compare against results form numerical methods. The dimension of the rectangular region is that given by Gau and Viskanta [11] which is 8.89cm in the x-direction and 6.35cm in the y-direction. The cold wall temperature (right vertical wall) is 28.3^0C and the hot wall temperature (left vertical wall) is 38^0C. The physical properties of pure gallium are those used in [11]. The mesh used is an orthogonal mesh consisting of 2042 triangles and 3111 faces. The time step used is 5 seconds. Melting fronts for various time levels and for three different results are given in Fig.3. The staggered mesh method is in good agreement with experimental results and so is

the cell-centred method. The dotted lines represent the experimental results of Gau and Viskanta [11], the full line represents the staggered results and the interrupted line is the cell-centred method.

Runtime comparison was monitored to determine which method runs faster. In Fig.4 runtime lines are plotted against nine time levels. The popular cell-centred method again runs faster.

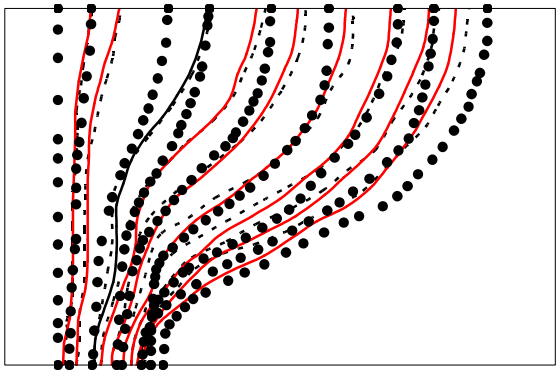

Fig.3 Melting fronts of Gallium

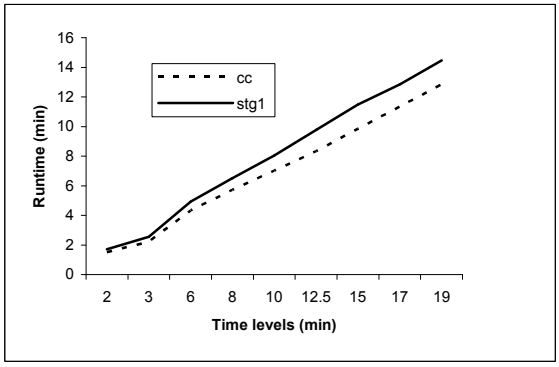

Fig.4 Comparison of runtime

6 Conclusion

The unstructured staggered mesh methods presented in this paper have advantages and disadvantages. The main advantage is that the velocity on the face need not be interpolated; it is located at the appropriate position.

Results were presented in section 5 for three different cases that are used as standard benchmark cases to test the validity of CFD codes. Those results were compared against standard benchmark results provided. The solutions obtained by the method presented here are comparable to the results obtained with the well established cell-centred method.

In terms of speed the cell-centred method has the edge over the unstructured mesh method presented here since it is explicit; the matrix is symmetric, diagonally dominant and can be solved by standard linear solvers. On the other hand the unstructured staggered mesh method is not diagonally dominant if the transient term is omitted, therefore a false time step relaxation must be used. The matrix is asymmetric and can only be solved by linear solvers that are suitable for asymmetric matrices.

In terms of iterations the unstructured staggered mesh methods generally require fewer iterations to converge than the cell-centred method.

References:

[1] S.V. Patankar and D.B. Spalding, A calculation procedure for heat and mass transfer in three-dimensional parabolic flows, *Int. J. Heat and Mass Transfer*, vol.15, 1972, pp.1787-1806.

[2] F.H. Harlow and J.E. Welch, Numerical calculation of time-dependent viscous incompressible flow of fluid with a free surface, *The Physics of Fluids*,vol. 8, 1965, pp.2182-2189.

[3] C.M. Rhie and W.L. Chow, Numerical study of the turbulent flow past an airfoil with training edge separation, *AIAA Journal*, 21:1525-1532, 1983

[4] J.B. Perot, Conservation properties of unstructured staggered mesh schemes, *J. Comp. Physics*, 159:58-89, 2000

[5] I. Wenneker, Computation of flows using unstructured staggered grids, PhD thesis, Delft University of Technology, the Netherlands, 2002

[6] I. Wenneker, A. Segal, P. Wesseling. Conservation properties of a new unstructured staggered scheme, *Computers & Fluids*, 32:139--147, 2003

[7] U. Ghia, K.N. Ghia and C.T. Shin, High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method, *Journal of Computational Physics*, 48, 387-411, 1982

[8] I.P. Jones, A comparison Problem for Numerical Methods in Fluid Dynamics, The 'Double-Glazing' Problem, *Numerical Methods in Thermal Problems, Proceedings of the First International Conference*, 338-348, 1979

[9] G. de Vahl Davis and I.P. Jones, Natural Convection in a Square Cavity: A Comparison Exercise, *International Journal for Numerical Methods in Fluids*, 3, 227-248, 1983

[10] V.R. Voller, M. Cross, N.C. Markatos, An Enthalpy Method for Convection/ Diffusion Phase Change, *International Journal of Numerical Methods in Engineering*, vol.24, 271-284, 1987

[11] C. Gau and R. Viskanta, Melting and Solidification of a Metal on a Vertical Wall, *Journal of Heat Transfer*, vol.108, 174-181, 1986.