Comparison of a pressure correction based solver VoF method in moving floor container

A. RONAK GHANDRIZ, B. MOHSEN GOUDARZI

Abstract: In this paper, free surface flow problems in a 2-D container with a moving floor solved by VoF method. This numerical method (VoF) is used to employ impacted factors on dimensional analysis in laminar and turbulence cases. Comparing laminar and turbulence cases (free surface flows), in laminar flows steady state happened sooner and more symmetric circulation is formed. Then, in turbulence cases both states (with and without surface tension) have been considered and inconsiderable effect of surface tension will be presented. Also, VoF method is utilized in laminar flows (free surface) to study about some impacted factors on dimensional analysis such as different velocity for moving floor, different height of fluid in the container, different fluid properties (density and viscosity), etc. So, in free surface flows, Reynolds number has no magnitude in comparison with flows without free surface and fluid properties should be considered in advance.

***Key-Words*:** Cavity, Free Surface Flow, Laminar Flow, Turbulence Flow, VoF Method

I. INTRODUCTION

Applying numerical models for flows with free surface has less research background and less attention is paid to numerical models. By considering oil industry and vast offshore facility that exist in some countries, it is necessary to believe numerical models in this state of flows.

The application of this problem is in area of survey and investigation the situation of fluid inside the controller (capacity) in transfer pipe line, also in cases, which the bottom wall of oil or fuel capacitor for some reason such as earthquake or unwanted movement are shocked or fluctuated. Research about these flows like other flows is possible in two methods. First method is using laboratory and experimental models; usage of this choice is so difficult and costly. Point of view, it is not reachable by all researchers. Thus, applying numerical models which could simulate the behavior of flow have to recommend such as the second method. Of course it should be mentioned that the numerical research in the field of flow with free surface has less background than numerical research in the field of flow without free surface. Purpose of flow with free surface is that kind of flows that have a common boundary between two different phases. Both of these phases could be liquid or one of them could be gas. Prevalent equations of flows for both phases are Navier- Stocks equations. As the common boundary was unknown and the complexity of forces that exist in common boundary of these two phases make it harder to model these flows in comparison with classic flows without free surface.

Numerical models, which apply for free surface flows are mainly categorized in two parts as follow:

Manuscript received September 21.2009.

F.A. Author is with MAPNA Boiler Company (MAPNA Group), Tehran, Iran. (corresponding author to provide phone: +982188511267; fax:; e-mail: r.ghandriz@gmail.com, ghandriz@mapnaboiler.com).

S. B. Author, Jr., was with Bu- Ali Sina University, Iran, Hamedan. (e-mail: mohsengood@gmail.com).

1. Following the free surface inside a constant grid in order to free surface is always inside the grid. In this method the scalar quantity that introduces the location of free surface is used. Among the applied methods we can cite MaC [1], and VoF [2] methods. These methods are suitable for situation that difference height of free surface at different point is very high or intensive slopes in free surface exist to horizontal surface are suitable.
2. Following the free surface as a solving grid boundary. In these methods, solving grid is exclusive to liquid flow and points of grid do not exist outside of fluid and where free surface is as one of the boundary condition of field should be distinguished continuously. Thus, solving grid in each order of calculation would be regenerated. In this method free surface is nominated like a less thickness surface.

These methods are suitable where difference height of free surface at different points is not very high or sloppily.

Goudarzi and Azimian [3], [4], [5], introduced a numerical method in 2003 and tested this procedure for channel flow. This method is placed in second part and it is practical for free surface flow and also for flow without free surface because in this manner momentum equation has been solved by introducing none hydrostatic pressure term. Also, they have taken advantage of approximate boundary condition instead of exact boundary condition in free surface for some auxiliary channel flows [4], [5]. As their results, because none hydrostatic pressure gradient in direction of channel axis is dominant whereas there is the inconsiderable fluctuations in free surface we can ignore such as these negligible displacements.

In this paper we compared the results of one of laminar cases (that has been solved by VoF Method) with this method for free surface flow in moving floor cavity. In this problem unlike the channel flow, none hydrostatic pressure gradient is important in all directions.

II. PROBLEM FORMULATION

Differential equations for a laminar and incompressible fluid flow are Navier- Stocks equations. By choosing Cartesian as reference, dominant equations on 3-D incompressible flows could be introduced. These equations consist of continuous equation, three momentum equation, fraction volume equation and turbulence models equations for turbulence cases. In this Cartesian coordination x-y axises laid on horizontal directions and z axis laid on vertical direction supposed as opposite of gravity acceleration. By considering all above the dominate equation at steady state are written as follow:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0.0 \qquad (1)$$

$$\rho(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z})=-\frac{\partial P}{\partial x} \qquad (2)$$

$$+(\mu+\mu_1)(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2})$$

$$\rho(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z})=-\frac{\partial P}{\partial y} \qquad (3)$$

$$+(\mu+\mu_1)(\frac{\partial^2 v}{\partial x^2}+v\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2})$$

$$\rho(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z})=-\frac{\partial P}{\partial z} \qquad (4)$$

$$-\rho g+(\mu+\mu_1)(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2})$$

Where, u,v,w presented velocity component (m/s) at x,y,z directions, ρ presented fluid density (kg/m^3), g presented gravity intensity (m/s^2) and P presented pressure value (Pa).

It should be noted; in 2-D cavity the third equation in these formulations is unconsidered.

Suppose, two uncombined and incompressible fluids are available, so divergence of the velocity is:

$$\nabla.u=0 \qquad (5)$$

For using the fraction volume, the positions of two fluids are important, so for inside one of the fluid C=1 and for inside another fluid C=0. Cells which are between these two phases are verified as 0<C<1. The first fluid fraction volume is expressed as follow:

$$\frac{\partial c}{\partial t}+\nabla.(uc)=0 \qquad (6)$$

No slip condition is another boundary condition for studding the lid driven cavity flow that surrounded the fluid completely. But in free surface flow problem, one of the boundary condition is free surface. In this boundary, kinematic and dynamic conditions should be met. If we assume that free surface flow is a scheme (z) without any breakdown, it will be possible to consider the height of different points proper to a reference surface (figure 1) as following equation:

$$z=h(x,t) \qquad (7)$$

Where, h denoted liquid height, x presented horizontal direction and t denoted time dependent.

Without any evaporation through free surface, no mass transfer from free surface would be occurred, that is presented as:

$$\frac{Dz}{Dt}=\frac{\partial h}{\partial t}+u_{fs}\frac{\partial h}{\partial x} \qquad (8)$$

Left hand term illustrates free surface velocity (w_{fs}), so we can rewrite it such as:

$$w_{fs}=\frac{\partial h}{\partial t}+u_{fs}\frac{\partial h}{\partial x} \qquad (9)$$

Therefore, kinematic conditions for free surface would be demonstrated by no mass transferring.

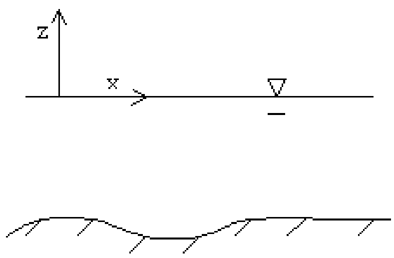


Fig.1 Cartesian coordination and reference surface

In order to dynamic condition the free surface forces should be equal which were consisted of the consequent of the tangent forces such as surface tension and shear stress, and the consequent of the vertical forces affected by atmosphere pressure and fluid pressure have to be zero. Frequently, because of negligible density and viscosity of surrounded permanent, shear stress has to be inconsiderable:

$$\partial u/\partial\zeta=0.0 \qquad (10)$$

Where, ζ denoted vertical vector through the free surface.

Also, surface tension is negligible for extensive free surface flow. Thus, in this case, dynamic condition is equality of atmosphere pressure (in this paper) and fluid pressure.

$$p_{fs}=p_{atm} \qquad (11)$$

For atmospheric and free surface attribute, we use atm, fs subscripts.

By variable replacement we have:

$$p_{fs} = \rho g z_{fs} \tag{12}$$

III. PROBLEM SOLUTION

Numerical method is based on grid generation. In free surface flow, the boundary condition of free surface is not obvious; in fact one of the unknown parameter is the configuration and geometry of free surface. First, for solving this problem, we should suppose a model for free surface. Considering it; first grid has to be generated that doesn't confirm to the boundary condition of free surface. Then, by numerical solving and employment of kinematic and dynamic boundary conditions in free surface, the correct boundary geometry and solving grid should be exhibited. Some of the numerically models for solving VoF method were presented by scientist that based on Hert and Nickels' method. This solver estimated the curve of medial surface by vertical and horizontal lines and was structured by upwind and downwind methods. The advantage of upwind method convention is its stability, but this method is much spacious and might be extend in some middle surface cells. Although downwind method is not stable, this able to form the middle surface and it is very useful in that's function. Some of the Vof methods were incremented which are able to offset the stability of upwind method and the advantage for front surface modeling by downwind method [6], [7], [8].

IV. NUMERICAL RESULTS

To confirm the accuracy of VoF Method, we solved numerically a 2-D lid driven cavity. The numerical results in comparison with Ghia's [9] results are shown in figure (2). The numerical accordance obtained from this method and which one explained by Ghia's [9], portended acceptable accuracy of employed this method, so this procedure would be applicable. By assuring this numerical method ability, we have investigated such as 2-D cavity with moving floor and free surface flow instead of upon rigid wall (Figure 3). First, we suppose laminar case and Reynolds Number equal to 1000. Once, a grid mesh 100*100 and second a grid mesh 200*200 have to be used and boundary conditions were defined by steady lateral walls, moving floor by velocity equal to 1 m/s and pressure outlet condition for upon wall. Then, consider the half of this cavity was full of a fluid.

$$\frac{\rho_{gas}}{\mu_{gas}} = \frac{1.22}{1.78e-5} \approx 10^5 s/m^2$$

$$\frac{\rho_{liq}}{\mu_{liq}} = \frac{10}{0.01} = 10^3 s/m^2$$

Where μ presented viscosity value (kg/ms^{-1}).

Consider laminar unsteady flow and time step = 0.001, by solving this problem with VoF Method after 36.28s (CPU time = 259200s) the steady state was achieved. Obviously, in figure 4 (stream function and vorticity contours) the right corner of the surface flow rises and the circulation is closer to the right side and a visible dip springs in the middling of flow. In turbulence case, we consider the physical theorem as same as laminar case except that the fluid in the cavity was liquid water.

$$\frac{\rho_{liq}}{\mu_{liq}} = \frac{998.2}{0.001} \approx 10^6 s/m^2$$

$$\frac{\rho_{gas}}{\mu_{gas}} = \frac{1.22}{1.78e-5} \approx 10^5 s/m^2$$

Hence, the mesh regeneration has to be harmonized to that statue. Figure (5) demonstrate unsteady solution via VoF Method by time step = 0.03 verged to steady state after 88.76s (CPU time = 432000s) and the circulation traversed the same auxiliary orbit and stood close to the right side. Although, the ledge in the right corner have been emerged, the visible dip in the midline of flow was not obvious. Afterwards, this problem was solved by involving surface tension up to 0.07 and considering unsteady solution via VoF Method with time step = 0.03. In this situation, after 118.76s (CPU time = 518400s) that redounded steady state and the same consequences happened such as previous problem (Figure 6). These outcomes are derived from stream function and vorticity contours, pressure contours, free surface profile contour, horizontal velocity vector component in the middle of cavity (Figure 12,13a), velocity vectors diagrams (Figure 10,11a) in the steady state for laminar and turbulence cases.

- Laminar flow achieved to steady state earlier than turbulence flow.
- Comparing the steady state results expose that formed circulation in laminar case is closer to the right side; also, the right lower corner in stream function and vorticity contours is sharp.
- Although in laminar case a small circulation was formed in right upon corner, which was not appeared in the turbulence mode.
- According to horizontal velocity vector component diagram, in laminar flow the circulation is more symmetric than turbulence flow. So, in laminar case these vectors mitigated from bottom to upon side.
- Because of larger amount of diffusion terms in comparison with inertia terms in laminar case with further density content, free surface velocity vector in the middle of the cavity in turbulence case is -0.2 m/s while in laminar mode it is -0.7 m/s. Also, in laminar case velocity vectors diverge to zero value without continuance; these facts actualized the fluctuated free surface laminar flow profile. (Figure 12, 13a)
- Velocity vector of turbulence flow converged to zero value in higher height.
- More continues velocity vectors in turbulence flow demonstrate steadier free surface profile.
- Considering the left part of the velocity diagram concluded that fluctuation near the floor in laminar case is more continuous and further than turbulence mode in the

same distance, this reason was occasion of sharpness near the right bottom corner (Figure 4).

- Obviously, the vortex flow is the dominant flow in cavity. In this flow, pressure gradient is important in all of directions and there is not any horizontal or vertical conqueror pressure gradient in the fluid flow. According to pressure counters, the difference between minimum and maximum pressure value in laminar and turbulence cases have been 38.15 k Pa, 2774 k Pa. This fact illustrated a little free surface fluctuation is able to vary pressure dominate completely. Therefore, the pressure value reduces in turbulence case more than in laminar case.
- Velocity vectors for both cases in the cavity conceded the previous results. Also, the descent gradient in figure (11a), near the floor is less in laminar flow than turbulence flow, hence velocity vectors is more continuous.

Next, for investigating about free surface flow, which is closer to the moving floor, we suppose the same physical problem and the same property for liquid and gas, by considering ¼ height of liquid in the cavity and floor velocity is equal to 2 m/s (Re= 2000) and solving this problem via VoF Method, after 26.11 s (CPU time= 172800 s), the steady state was occurred. This moment is lower than previous case. According to figure (7), there are many circulations and the initial one is closer to the right side corner. Comparing this result and another one solved numerically by Goudarzi an Azimian [3], [4], [5] (Figure 9), verified this method for free surface flow. By examining the results from stream function and velocity contours, pressure contours, phase contours, horizontal velocity component vector in the midline of the cavity (Figures 13a, b) and the velocity vectors (Figures 11a, b), from outset to steady state, in the same time for both laminar cases, we mentioned such as:

- Laminar flow in the second case (which the height of liquid was ¼ and the floor velocity was 2 m/s) attained to the steady state rather than the first case, because of lower height of liquid.
- In the second case, more circulations have been formed, because the velocity of moving floor increased.
- Comparing the circulation promenade to achieve the steady state in both cases, we are able to represent the circulation in the second case has not grow and has not orbit completely opposite the first one.
- Real circulation in the first case of laminar flow was more symmetric than another one, which was derived by the horizontal velocity component diagram. (In the further section, you would follow by receding from floor, velocity vectors in the first case opposite of second one decreased gently up to achieved to free surface.)

Comparing the horizontal velocity component in the midline of the cavity (Figures 13 a, b), these results could be decelerated:

- In the first case, after circulation grew and promoted completely, according to C.C.W revolution, horizontal velocity components descended up to free surface, though the descend course in the second one changed in the middle of fluid and incepted the ascendant course earlier than another one, that explained the center of the circulation was closer to free surface in the first case.
- Minimum velocity in the first case occurred on free surface (-0.7 m/s), though in the second one because of forming the adverse circulation, minimum velocity happened in the middle of fluid height (-0.65 m/s).
- According to the phase contours, free surface fluctuated more in the first case that is able to describe by attending to the horizontal velocity component diagram.
- Obviously, the vortex flow is the dominant flow in cavity. In this flow, pressure gradient is important in all of directions and there is not a conqueror pressure gradient in the fluid flow. Due to quicker moving floor, pressure value decreased more in the second laminar case than the first laminar case which is able to consent by pressure difference (38.15 k Pa in the first case and 99.56 k Pa in the second one).
- According to the horizontal velocity component diagram, the descendent gradient of the curve in the first laminar case was much less than another one which proved the continuous variation in velocity vectors.

For laminar flow study in this cavity, when the difference between liquid density and gas density was much enough to ignore gas density, first we consider the same physical problem (the height of liquid is equal to ¼ and the velocity of moving floor is equal to 2 m/s), if properties of liquid and gas change such as below, it would be approached by these properties and density difference, the results must be unusual. That was because of ignorable gas density. After 235.39 s (CPU time= 777600 s), we get the steady state.

$$\frac{\rho_{liq}}{\mu_{liq}} = \frac{1}{0.001} = 10^3 s/m^2$$

$$\frac{\rho_{gas}}{\mu_{gas}} = \frac{0.01}{1.42e-7} \approx 10^5 s/m^2$$

According to figure (8) the result is so different from other laminar cases and the circulation was more extended because of high distinctive between gas viscosity and liquid viscosity. Next, by examining the results of second laminar flow and third laminar flow, which consist of stream function and velocity contours (Figure 7, 8), phase contours, horizontal velocity component distribution diagram in the midline of cavity (Figure 13b, c), velocity vectors diagram (Figures 11b, c) for flow in the cavity.

- Second flow achieved to the steady state (26.11 s) rather than third flow.
- The scale and the location of circulation in the second flow are more adjacent than the scale and location of circulation in the third one and the circulation is closer to the right side.
- The circulation of the second flow is more symmetric than another one that is also arising from parabolic zone of horizontal velocity component (Figure 13).

By ignoring the density of air opposite of water we were not able to persuade the accurate results that should be attended in solving problems by VoF method.

Comparing horizontal velocity component distribution diagram in the midline of cavity with moving floor velocity equal to 2 m/s (Figures 13b, c), in the laminar flows following results were stated.

- Due to opposite revolving circulation from the first circulation in the second flow, the minimum velocity occurred almost in the middle of liquid height (-0.65 m/s), whereas the minimum velocity of third flow happened near free surface and it is because of homogenous revolving circulations and it is more than minimum velocity in the second flow (-1.25 m/s).
- Parabolic zone in the horizontal velocity component diagram in the second flow is more symmetric. So, free surface in this flow fluctuated less and the circulation is more symmetric than the third one.
- In the second flow, the value of horizontal velocity component is positive because the powerful circulation near free surface was formed
- According to figures (7, 8), it is obvious that free surface in the second one is steady and the formed dip is in the reduced region. That could be obtained by observing the horizontal velocity component distribution diagram.
- Because of ignorable gas and liquid density in the third case opposite of second one, the pressure decrease in the third case must be so little. The pressure value reduction in the second one is 99.56 k Pa whereas in the third case, it is 11.53 k Pa. The maximum pressure reduction occurred in free surface.
- Due to less downward slope of parabolic curve in the horizontal velocity component distribution diagram (Figure 13b, c), we expected continuous variation for velocity vectors near the moving floor in the third case (Figure 11b, c). Certainly this anticipation realized by comparing these figures, near the lateral walls discontinues distribution of velocity vectors was formed admitted by different assumption in fluid properties.

V. CONCLUSION

As we knew, laminar flows achieve to the steady state rather than turbulence flows. Although, there was the same pressure gradient, the free surface fluctuation gauge in laminar case was further than turbulence case that occurred because of less fluid density value in laminar flow. Also, because of the ignorable surface tension between air and difference between upper and lower part that offered such Benarth case in fluids.

Also, in free surface flows opposite of other flows, Reynolds number has no effect on problem analyses and viscosity and density parameters are important in solving this problem exclusively, by ignoring gas density versus from liquid density, unusual results were achieved. Also, we have to examine considerately in these cases for Reynolds number, as, each steady turbulent in free surface flows could not result steady state in the problem.

liquid water there was no discrepancy in both turbulence cases.

By this investigation, we can observe this problem in Taylor fluids is similar to Benarth problem in convection branch, which occurred because of the temperature

REFERENCES

[1] F. H. Harlow, and J. E. Welch, “Numerical Calculation of Time-dependent Viscous Incompressible Flow of Fluid with Free Surface”, *Physics of Fluid*, 8 (12): pp.2182-2189, 1965.

[2] C. W. Hirt, and B. D. Nichols, “Volume of Fluid (VoF) Method for the Dynamics of Free Boundaries”, *Journal of Computational Physics*, 39: pp. 201- 225, 1981.

[3] M. Goudarzi, *“Investigation about Diffusion Phenomena in Incorporate Stead of Two Channels”.* Doctoral Thesis, Isfehan Industrial University, 2003.

[4] M. Goudarzi, A. R. Azimian, *“Numerical Flow Modeling in Open Channel with Free Surface Following Method”,* Engineering Department of Tabriz University, 2004.

[5] M. Goodarzi, and A. R. Azimian, " Mixing of Two Parallel Flows with Free Surfaces ", *Proceeding of 9ACFM,* 27-31 *May,* 2002.

[6] J. Farmer, L. Martinelli, and A. Jameson, “ Fast Multigrid Method for Solving Incompressible Hydrodynamic Problems with Free Surface “, *AIAA J.*, 32(6): pp. 1175-1182, 1994.

[7] S. E. Norris, *“A Parallel Navier-Stokes Solver for Natural Convection and Free Surface Flow”,* Ph.D. Thesis, University of Sydney, pp. 195-215, 2000.

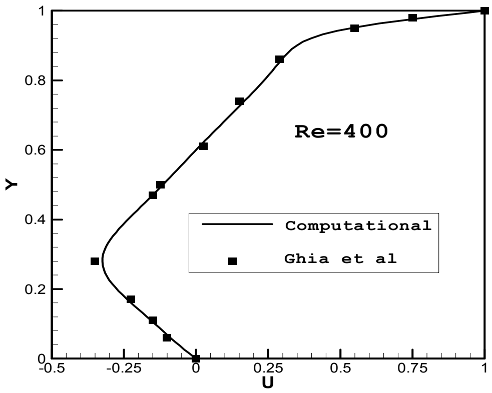
[8] J. H. Ferziger, and M. Peric, “*Computational Methods for Fluid Dynamics*”, Corrected 2nd printing, Springer, pp. 333-343, 1997.

[9] U. Ghia, K. Ghia, and C. Shin, “High-re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method”, *J. of Computational Physics*, Vol. 48, pp. 387-411, 1982.

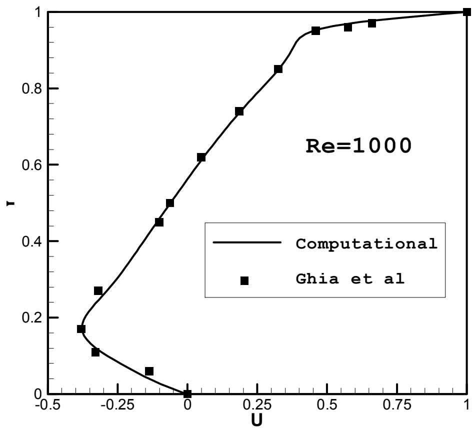
[10] Siavash H. Sohrab, ”Universality of a scale invariant model of turbulence and its quantum mechanical foundation”, *Proceeding of WSEAS,134-141, 20-22 August 2009.*

[11] A. Majid Bahari, Kourosh Hejazi, “Development and investigation of a non- isotropic k-e turbulence model for horizontal gravity currents applications”, *Proceeding of WSEAS,134-141, 20-22 August 2009.*

[12] Bo Wan, Friedrich K. Benra, Siavash H. Sohrab, “A comparative numerical study of the modified versus the Navier-Stocks of motion”, *Proceeding of WSEAS,134-141, 20-22 August 2009.*



(a)



(b)

Fig.2 Horizontal velocity component diagram in the middle of lid driven cavity. a) Re=400. b) Re=1000

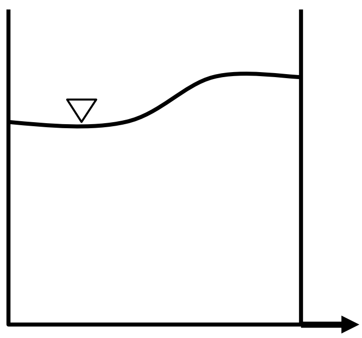


Fig.3 Geometry of the moving floor cavity with free surface

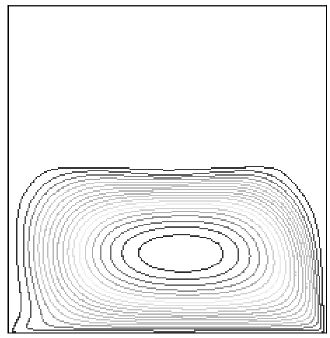


Fig.4 Laminar flow in the moving floor cavity with free surface.

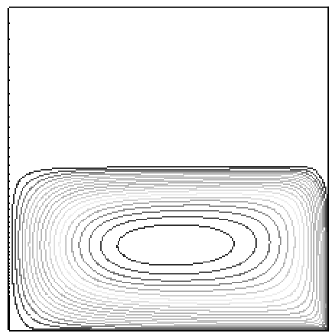


Fig.5 Turbulence flow in the moving floor cavity with free surface and without surface tension

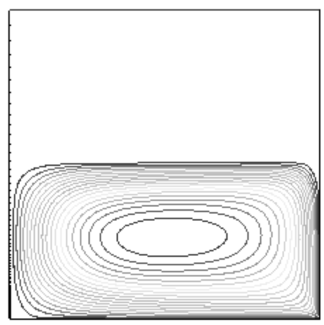


Fig.6 Turbulence flow in the moving floor cavity with free surface and surface tension

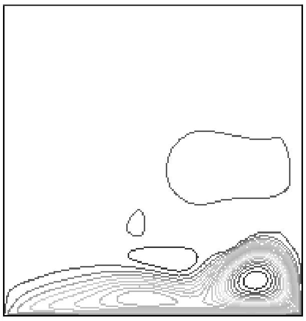


Fig.7 Laminar flow in moving floor cavity with free surface and the height of liquid was 1/4

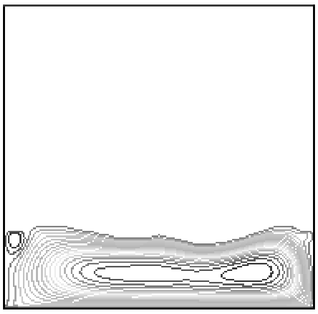


Fig.8 Laminar flow in the moving floor cavity with free surface, the height of liquid was ¼, by ignoring gas density versus from liquid density

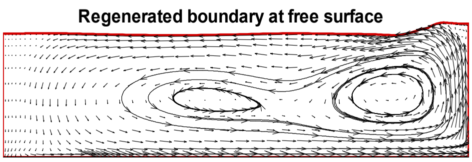


Fig.9 Stream function contours for laminar flow in the moving floor cavity with free surface and the height of liquid was 1/4. (Solving numerically by Goudarzi and Azimian [3], [4], [5].

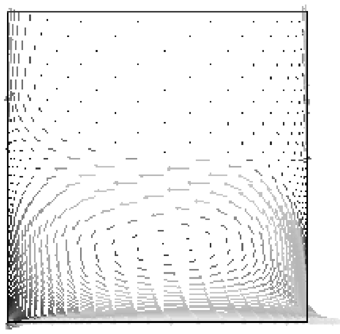
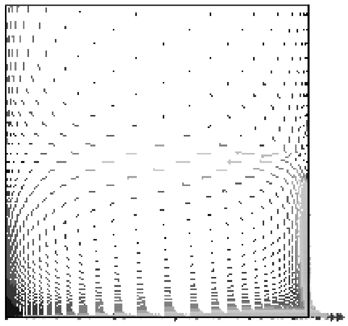
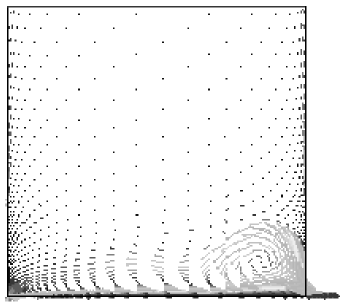


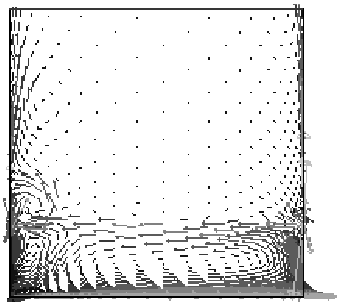
Fig.10 Velocity vectors for turbulence flow in the moving floor cavity with free surface



(a)

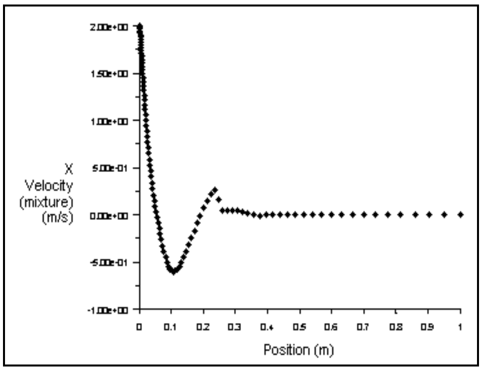


(b)



(c)

Fig.11 Velocity vectors for laminar flows in the moving floor cavity with free surface. a) First case. b) Second case. c) Third case.



(b)

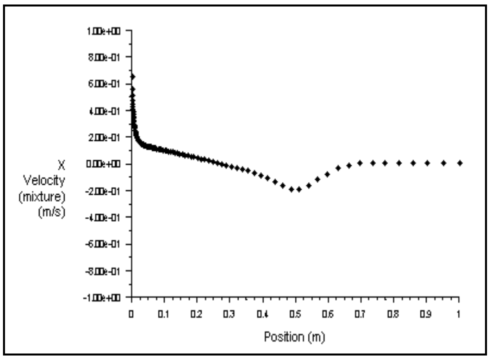
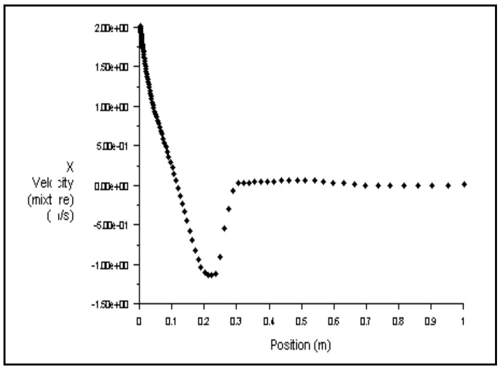
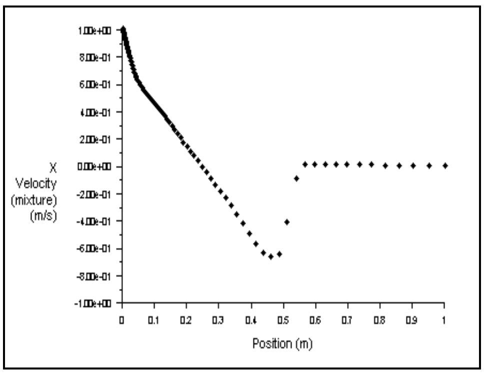


Fig.12 Horizontal velocity component distribution in the midline of the cavity for turbulence flow



(c)

Fig.13 Horizontal velocity component distribution in midline of the cavity. a) First case. b) Second case. c) Third case



(a)