Eng. & Tech. Journal, Vol.28, No.6, 2010

Numerical Study of Solidification in Cavity with the Presence of Natural Convection

Dr. Wahid S. Mohammad*

Received on: 16/6/2009
Accepted on:3/12/2009

Abstract

A study of Laminar two dimensional transient solidification with time–dependent natural convection in the melt is carried out. The mathematical model for the numerical simulation is based on enthalpy porosity method. The governing equations are descritized on a fixed grid by means of a finite volume technique. The (SIMPLE) procedure was adopted to solve mass, momentum and energy equations for pure phase change material (water) placed in a cooled rectangular cavity. The cavity was cooled from top alone, right side wall alone, and right side wall with top and bottom walls together. In the case of top cooling a parabolic shaped pattern of ice water interface was formed. For the right side wall cooling the effect of density changes has led to an abnormal flow circulation which has moderately modified the heat balance of the freezing interface causing a colliding of cool and warm fluid layers. A density variation was seen in the freezing interface region in most of the cases. The results obtained show good agreement with experimental and numerical results of other researchers for pure convection with small discrepancies in the ice interface. These discrepancies may be attributed to the physical modeling used for water freezing.

Keywords: Two phase flow, solidification, natural convection, CFD

دراسه عدديه لعملية الانجماد في فجوه بوجود تيارات الحمل الطبيعي

الخلاصة

اجريت دراسة طباقية ببعدين لعملية حدوث التصلب (الانجماد) بوجود تيارات الحمل الطبيعي للماء. ولقد اعتمدت طريقة المحتوى الحراري المسامية لنمذجة الموديل الرياضي الذي استخدم في المحاكات الرقمية. وعولجت المعادلات الرياضية الحاكمة باستخدام الشبكة الثابتة وبمساعدة تقنية الحجوم المحددة. وتم اعتماد طريقة (SIMPLE) في حل معادلات الاستمرارية والزخم والطاقة للمادة النقية متغيرة الطور (الماء) الموضوعة في حيز مستطيل مبرد تم تبريد الفجوة من الاعلى فقط او من الجانب الايمن فقط اوالجانب الايمن و الاعلى والاسفل مع بعضهم . في حالة التبريد من الاعلى كان الحد الفاصل للانجماد على شكل قطع مكافى . وفي حالة التبريد من الجانب الايمن ادت تاثيرات تغير الكثافة الى حدوث دوامات غير اعتيادية اثرت على شكل الحد الفاصل للانجماد بسبب تداخل تيارات الماء البارد والساخن. وقد شوهد تغير في الكثافة ضمن حدود الحد الفاصل للانجماد في اغلب الحالات المدروسة. واوضحت النتائج التي تم الحصول عليها تطابق معقول مع نتائج عملية واخرى نظرية لباحثين سابقين مع وجود بعض الفروقات القليلة في الحد الفاصل الثلجي. ويمكن اعزاء هذه الاختلافات الى النمذجة الفيزياوية لعملية انجماد الماء.

*** Mechanical Engineering Department, University of Technology / Baghdad**

https://doi.org/10.30684/etj.28.6.10
2412-0758/University of Technology-Iraq, Baghdad, Iraq

Introduction

The physical phenomena that control the shape of solid liquid interface during solidification has become of increasing importance in many industrial processes. Its main characteristic is that a moving interface separates two phase with different physical properties. Temperature differences in the melt give rise to buoyancy forces that produce significant convection flow. It appears that natural convection can have a large influence on the morphology of the solid – liquid interface, the solidification rate and temperature distribution in the liquid and solid. The location of the interface changes in time as a function of the thermal boundary conditions in a way that is unknown prior to the solution of the problem. The displacement of this interface is responsible for the non linearity of the problem. Due to this problem complexity, direct application of numerical methods to the solidification problem is not a trial task. Error is appeared due to the limited accuracy of different numerical methodology and due to inventible simplification introduced in the models [1]. The thermal boundary conditions are an other case for these errors as identified by Abegg et. al [2]. It seems that numerical model used need several improvements. Hence, the primary target is to employ a method that proved to be efficient and accurate for the analysis transient solidification process with convection dominant [3]. Carlos [4] and Han et.al [5] have used such method to solve the solidification problem of an alloy metal in a square section region. Rattanadecho [6] applied a similar approach to study the solidification process in unsaturated granular packed bed. Alawadhi [7] use the method via ANSYS packaged to solve the solidification process of water in a circular enclosure. Therefore the method is very well suited for the solution of phase transformation problem with fluid flow. In this paper the method will be extended to analyze the solidification in the presence of natural convection.

Mathematical Model

Solidification in the presence of natural convention in a square cavity as in figure (1) will be studied with the momentum field is subjected to no – slip boundary conditions at the walls. The flow will be assumed to be two dimensional laminar and incompressible. The densities of solid and liquid are equal except when utilizing the buoyancy term for free convection. A fixed grid method is used which relies on the enthalpy formulation method [8]. This method defines the liquid mass fraction (γ) as the ratio of the liquid mass to the total mass in a given computation cell. If (T_m)and (h_r)are set to the melting temperature and to the reference enthalpy, the specific enthalpy can be written as [9]:

$$h = gL + C_p T \qquad \text{…..} (1)$$

The specific heat (C_p) may vary with phase change. The liquid mass fraction can be obtained from:

$$g = \left| \begin{array}{lll} 0 & if & h < 0 \\ (\frac{h}{L}) & if & 0 \le h \le L \\ 1 & if & h < L \end{array} \right. \qquad \text{…….} (2)$$

The enthalpy – based governing equations for isothermal phase change are [9]

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad \text{….} (3)$$

$$r\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right)= -\frac{\partial p}{\partial x}+m.\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)+S_u \quad \text{…. (4)}$$

$$r\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}\right)=-\frac{\partial p}{\partial y}+ m\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)+S_v+rgb(T-T_r) \quad \text{…. (5)}$$

$$r\left(\frac{\partial}{\partial t}(c_pT)+u\frac{\partial}{\partial x}(c_pT)+v\frac{\partial}{\partial y}(c_pT)\right)= k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right)-rL\frac{\partial g}{\partial t} \quad \text{… (6)}$$

Where:

$$S_u=-c\frac{(1-g)^2}{g^3+b}u$$

$$S_v=-c\frac{(1-g)^2}{g^3+b}v \quad \text{…….. (7)}$$

In which b is a small constant introduced to avoid division by zero when $(g=0)$ and c is a large number to suppress the velocity as the cell become solid. In the present work c and b given the values of ($c = 1.5 * 10^5$)and ($b = 0.001$).

Numerical Model

A numerical simulation of the problem was performed using finite volume model of the Naveir–Stokes and energy equations. The (SIMPLE) algorithm of Patankar [10] is employed to determine the velocity and pressure field. The discretized equations in the finite volume formulation can be expressed as

$$a_pf_p=\sum_{nb}a_{nb}f_{nb}+S_f-a_p^{\,o}(g_p-g_p^{\,o}) \quad \text{…… (8)}$$

where subscripts p and nb refers to the value of the present neighboring cell respectively superscript o denote the value of previous time step.

The descritized equation (8) is solved for u, v, T and pressure correction. Then the enthalpy and liquid fraction can be obtained from equations 1 and 2 respectively. The domain to be studied is a two dimensional cavity (0.038 m * 0.038m) with boundary conditions for three cases. In case (1) the cavity which is filled with pure water is subjected to free convection only without solidification where the top wall is exposed to a temperature of (T_{cold} = 15.5 ^{0}C) and the bottom wall is maintain at (T_{hot} = 21^{o}C) while the side walls are given a temperature following Abegg et.al [2]

$$T_w=d\left[1.0-e^{-dx}\right](T_{hot}-T_{cold})+T_{cold} \quad \text{…….. (9)}$$

Where (d) is constant specified as required.

In case (2) the temperature of the hot wall is at (T_{hot} = 20 ^{o}C) while the cold wall temperature is reduced suddenly to (-10 ^{o}C) to simulate a solidification case.

In case (3) the problem is solved for free convection by expressing the side walls to cooling and heating effect and the top and bottom wall given a temperature as specified by equation (9). The solution started with right wall at a temperature of (0 ^{o}C) and left wall at (10 ^{o}C) to obtain a free convection problem, followed by a sudden decrease to the right wall temperature to (-10 ^{o}C) to obtain the solidification case with cooled side wall. In all cases the grid used was expanded grid near the walls and uniform in the middle with (42*42) nodes as shown in figure (2). These cases were chosen following cases studied by Abegg et.al [2] and Ginngi et.al [1] for one mean reason is to verify the capability of the present code in comparison with previous experimental and numerical studied.

Results and Discussion

Case 1: Free convection with out solidification .

In this case the top wall is given a temperature of (15.5 ^{o}C) while the bottom wall is kept at (21 ^{o}C). A comparison of the flow structure and temperature contour is appeared in figure (3). A hot boundary layer appeared of two upward flowing jets of hot liquid along the side walls. This flow create a single downward cold jet in the center of the cavity and in turn generate several recirculation zones transporting heat from the side walls to the center. The agreement is fairly good in these flows with the results of Abegg et.al [2].

Case 2: Water solidification

The solidification of water has been studied by decreasing the lid temperature to (-10 ^{o}C). A complex flow field is obtained after convection start as shown in figure (4). It was found that the creation of ice layer at the lid has stabilizing effect on the flow. The parabolic shaped pattern of the ice water interface which forms imposes the direction and character of the flow. Again a good agreement is obtained with results of Abegg et.al [2] specially in the primary cells that carry hot fluid up the walls to the ice interface and the small centered rotating flows.

Case 3: Free convection with water freezing .

The effect of temperature dependence of the fluid density on the overall flow structure was examined in this case. A non linear variation of water density is considered in the buoyancy term only. Following Ginngi et.al [1] a fourth order polynomial is used for the density. The left side wall is given a temperature (10 ^{o}C) and the right side wall kept at (0 ^{o}C). The effects of density inversion and of the thermal boundary conditions at bottom and top walls on the flow structure are shown in figure (5). This figure show a good agreement and exhibits two recirculation zones, upper one, where the water density decreases with temperature and lower one with an abnormal density variation. Figure (6) shows another comparison with Ginngi et.al [1] results for water freezing. In this figure the right side wall temperature is reduced to (-10 ^{o}C) while other walls remain as they were before. The results show, when freezing starts an abnormal flow circulation located in the lower part of the cavity transport the cold liquid up the adjacent ice surface and back to the bottom. This cold water circulation only moderately modifies the heat balance at the interface. The convective heat transfer between both lower and upper region is to be limited mainly to the upper high corner of the cavity. The colliding cold and warm fluid layers reproduces the shape of freezing interface in both studies identically.

Figure (7) show a comparison for the predicted temperature contours with that of Ginnigi [1] for case 3 with right wall temperature at (– 10 ^{o}C). The ice inter face in this figure moved strongly outward near the bottom wall boundary giving rise to more ice growth rate there . Again it can be said that both results are in fairly good.

Case 4: Cooling from Three Sides. This case includes the cooling of the cavity from three sides i.e. top, right side and bottom walls as shown in figures (8 to 11).

Figure (8) shows the velocity vector of the flow in case 4. The effect of convection mode of heat transfer plays an important rule in transferring the warm water from the hot side to the freezing zone and reducing the thermal resistance . This effect leads also to the creation of a single liquid packet near the left wall surrounding by ice from the other three walls. A parabolic shape is noticed for the freezing line for top and bottom sides, which is an extension to what have been reported previously for the top wall cooling.

Figure (9) represents the temperature contours for this case. The temperature varied from (-10 ^{o}C) on these cooled walls to (10 ^{o}C) on the warm left side wall. A convection heat transfer is limited near the left wall as indicated previously in the velocity vector. The conduction heat transfer dominated the solid region as given by the shape of the temperature contour of this region.

The density variation of such flow is given in figure (10). There are two main regions, the liquid region and solid region separated by the interface line of a variable density depending on the amount of latent heat of the flow in this region as shown in figure (11).

Finally, it can be concluded that, the results obtained from the three cases show a good agreement of the present prediction with those of other researchers and give the present code the capacity to be used in simulating the two phase problem successfully.

References

[1] Ginngi M, Stella F., Kaowaleuski T. A.: Phase change problem with free convection; fixed grid numerical simulation, compute visual Sc2, 123-130, 1999.

[2] Abegg C., devochl Dans, Hiller W. J., Koch St., Kaowalewski T. A., Leonardi E. and Yeoh G. H.: Experimental and numerical study of three-dimensional natural convection and freezing in water: Heat transfer 4, 1-6 I chem. E. Taylor and Francis London 1994.

[3] Wahid S. M. Zainab H.: Numerical simulation of solidification around staggered tubes arrangement with convection dominant: Engineering and Technology Journal ; Vol.2 No.2, 2008.

[4] Carlos H. S. L.,: Solidification in square section: Chile, University del Bio-Bio, Theoria, vol 10, pp 47-56, 2001.

[5] Han Z.Q., Lewis R. W., and Liu B. C.: Modeling of the thermosolutal convection and macro segregation in the solidification of an Fe-C binary alloy: International Journal of Numerical Methods for Heat & Fluid

Flow Vol. 17 No. 3, 2007 pp. 313-321.
[6] Rattanadecho P.: Experimental and Numerical Study of Solidification Process in Unsaturated Granular Packed Bed: JOURNAL OF THERMOPHYSICS AND HEAT TRANSFER Vol. 18, No. 1, January–March 2004 pp. 87-93.
[7] Alawadhi E. M.: Solidification Process with Free Convection in a Circular Enclosure: Kuwait University College of Engineering & Petroleum Mechanical Engineering Dept,2002.
[8] Voller V. R., Cross M., Markatos N. C.,: An enthalpy method for convection-diffusion phase change : Int. J. Num. Mech. Eng. 24, 271-284, 1987.
[9] Viswannth R., Jaluria Y.: A comparison of different solution methodologies for melting and solidification problem in enclosures: Num. Heat transfer B 24, 77-105, 1993.
[10].Patankar S. V.: Numerical heat transfer and fluid flow, Hemisphere, Washington, 1980.

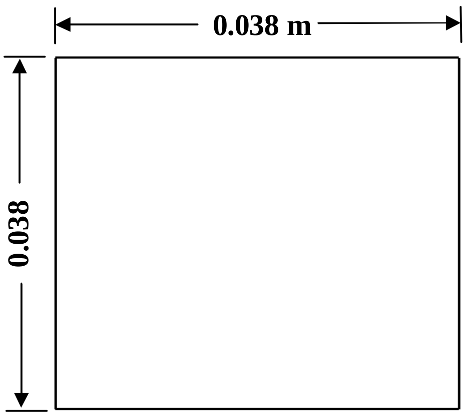


Figure (1) The numerical domain and Cavity geometry (0.038 m * 0.038 m).

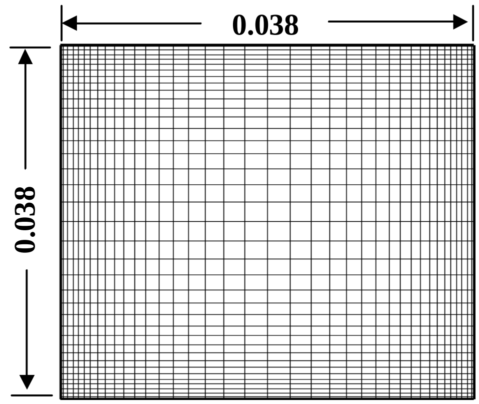
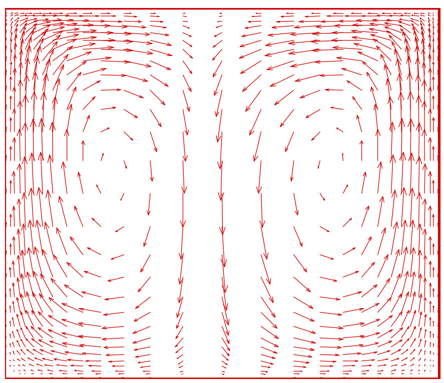
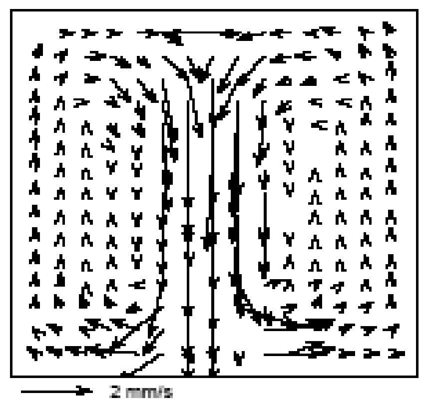


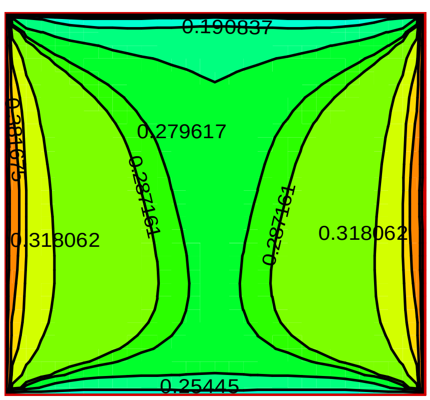
Figure (2) The numerical expanding mesh and cavity geometry (0.038 m * 0.038 m).



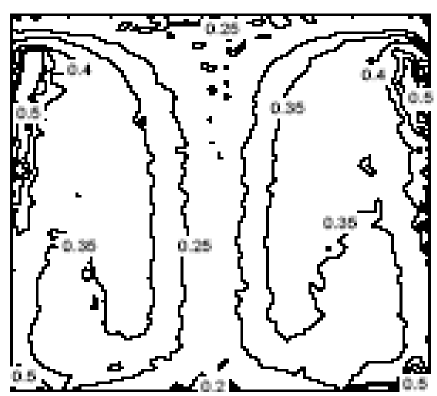
Present results



Abegg et.al [2]

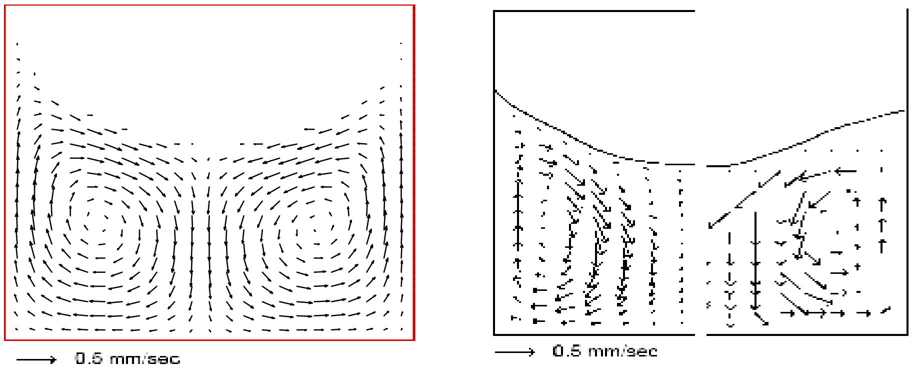


Present results



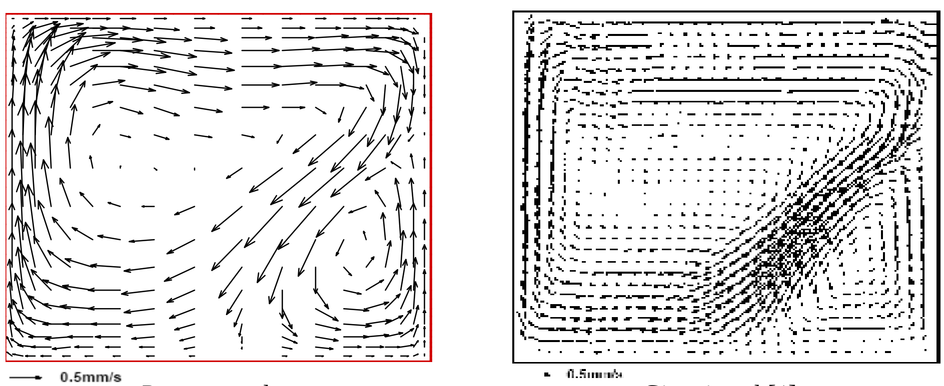
Abegg et.al [2]

Figure (3) Comparison of the velocity vector and temperature contour between the present work and that of Abegg et.al [2]



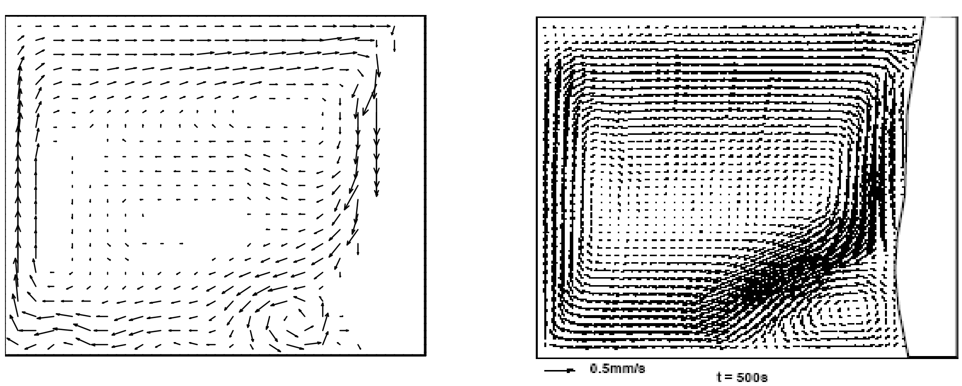
Present results Abegg et.al [2]

Figure (4) velocity vector for solidification of water in squared cavity cooled from top wall a comparison of present results and the experimental and theoretical of Abegg et al [2]



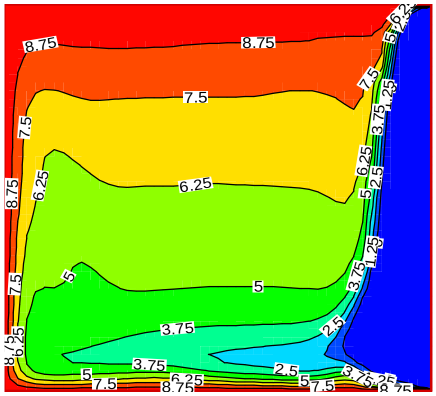
Present results Ginngi et.al [1]

Figure (5) Comparison of the velocity vector between the present work and that of Ginngi et.al [1] for squared cavity cooled from the right wall

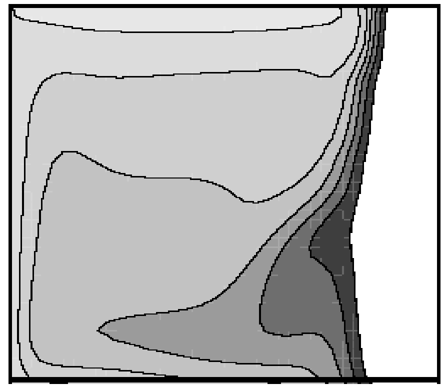


Present results Ginngi et.al [1]

Figure (6) Comparison of the velocity vector between the present work and that of Ginngi et.al [1] for squared cavity cooled from the right wall after 500 second from the start of cooling .



Present results



Ginngi et.al [1]

Figure (7) Comparison of temperature contours between the present work and that of Ginngi et.al [1] for squared cavity with the right wall at (- 10°C) after 500 second from the start of cooling .

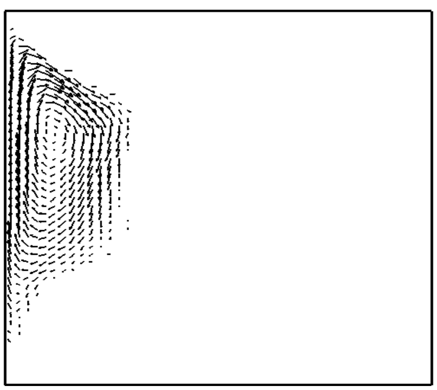


Figure (8) Velocity vector for case 4.

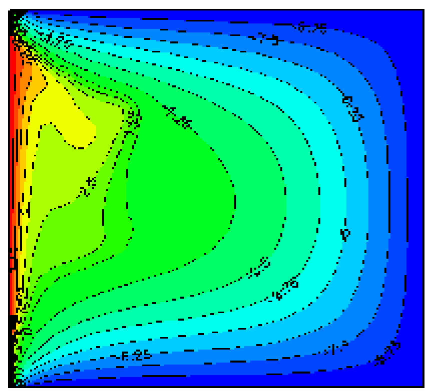


Figure (9) Temperature contour for case

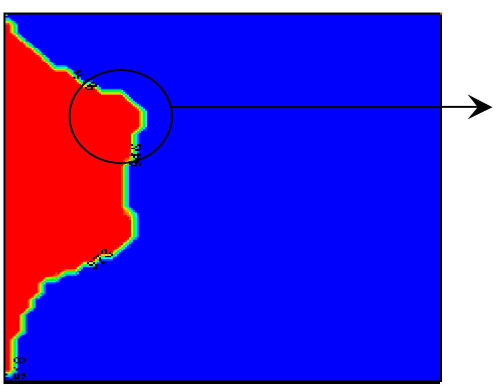


Figure (10) Density variation for case 4

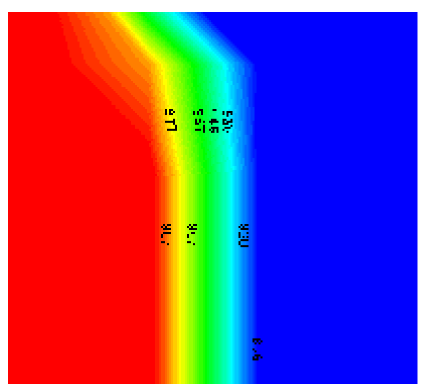


Figure (11) Density variation in the interface.