A FINITE ELEMENT SOLUTION OF DUSTY WILLIAMSON FLUID WITH MARANGONI CONVECTION

A THESIS
submitted in partial fulfillment of the requirement for the award of the degree of
MASTERS OF SCIENCE
In
MATHEMATICS AND COMPUTING

Submitted By

KAJAL BANSAL

Reg.No. 301603012

Under the guidance of

Dr. SAPNA SHARMA

SCHOOL OF MATHEMATICS

THAPAR INSTITUTE OF ENGINEERING AND TECHNOLOGY,

PATIALA-147004

PUNJAB, INDIA

JULY 2018

A grateful heart is beginning of greatness

Dedicated to God, My Parents and My Supervisor.

Acknowledgment

I am submitting my thesis for the fulfilment of my 'M.Sc' degree. This work would not have been accomplished without the help, support and the guidance of many people. I express my deep gratitude and respect to my supervisor **Dr. Sapna Sharma**(Assistant Professor, School of Mathematics) for her strong motivation, trust and constant encouragement during the course of work. I thank her for her great patience, constructive criticism and for giving me the opportunity to undertake this project.

I also express my heartiest gratitude to **Dr. Satish Sharma** (Head of School of Mathematics) and **Dr. A.K Lal** (Professor and Former Head of School of Mathematics) for their support through out the period and all members of School of Mathematics for their help and suggestions at different stages of this work.

My special thanks to **Ms. Madhu Aneja**. This work would not have been completed without support, help and guidance of her.

The meaning of my life and work is incomplete without paying reagards to my respected family whose blessings and continuous encouragement have been shown me the path to achieve my goals.

And above all, I pay my regards to the Almighty for his blessings.

Kajal Bansal

CANDIDATE'S DECLARATION

I hereby certify that the work, which is being presented in the thesis, entitled **"A Finite Element Solution of Dusty Williamson Fluid with Marangoni Convection"** in partial fulfillment of the requirements for the award of the degree of **Masters of Science** in **Mathematics and Computing** and submitted to the School of Mathematics, Thapar Institute of Engineering and Technology, Patiala, is an authentic record of my own work carried out under the supervision of **Dr. Sapna Sharma** Assistant Professor and **M/s Madhu Aneja,** Phd Scholar and other research work which is duly listed in the reference section.

The matter presented in this thesis has not been submitted elsewhere for the award of any other degree or diploma from any institution.

Date: 2/7/18

Kajal Bansal
Roll No. 301603012

This is to certify that the above statement made by the candidate is correct to the best of our knowledge.

Date: 2/7/18

Dr. Sapna Sharma
Assistant Professor
TIET, Patiala

Abstract

The present study aspires us to deal with applications of computational fluid dynamics. The first chapter contains basics of Fluid Dynamics and its mathematical formulation. It includes Navier Stoke's equations on the basis of fundamental laws of Physics which goverened the fluid flow phenomenon. In this chapter some existing methods are discussed to solve the incompressible Navier-Stoke's equation. In Fluid Dynamics, a system of differential equations with heat transfer is considered. With reference to this, the second chapter contains the basics of heat transfer and different modes of heat tranfers. The review of FEM i.e finite element method is given in detail in third chapter. FEM is choosen because of its advantages and compuational applications in field of physics and mathematics over other numerical techiques. After elementary studies, A problem of boundary layer flow and heat transfer is formulated in last chapter. In the given problem Marangoni convection flow of Dusty Williamson Nanofluid is studied. The results under both linear and quadratic variation of surface temperature is accounted. Numerical results are carried out for important parameteres namely Radiation parameter, Williamson parameter and Dust parameter.

List of Figures

Contents

Chapter 1

INTRODUCTION

In the nature, whatever we are seeing, exists in two states either solid or fluid. All materials exhibit deformation under the action of forces. If the deformation in the material increases continually without limit under the action of shearing forces, however small, the material is called fluid. This continuous deformation under the action of forces is manifested in the tendency of fluids to flow. Fluids are usually classified as liquid and gases. A liquid is a substance that flow freely but of a constant volume, and having no definite shape. For most of the purpose it is however sufficient to regard liquids as "incompressible fluids". A gas on the other side, is an air-like fluid substance which expands freely and fill any space available, irrespective of the quantity. A gas has no set shape and volume. A gas can fill any space in which it is placed thats why a gas is called as "compressible fluid"[1]. Fluid plays an important role in human life. Even approximately 70% of the human body is constructed of various types of fluids.

Fluid mechanics is a branch of science, which deals with the behaviour of fluids when subjected to system of forces. It serves as a basis for the study of various fields in engineering such as irrigation engineering, environmental engineering, hydraulic machinery, mechanical engineering, chemical engineering, lubrication aeronautics, meterology, oceanography, biomedicine and plasma physics etc.

There are two branches of fluid mechanics:

(i) **Fluid Statics or hydrostatics** is the study of fluid at rest. The required main equation for this is Newton's second law for nonaccelerating bodies, i.e. $\sum \vec{F} = 0$

(ii) **Fluid Dynamics** is the study of fluids in motion. The main equation required for this is Newton's second law for accelerating bodies, i.e. $.\vec{F} = ma$

The basic law of fluid mechanics is well-known as Newton's law of viscosity, which states that the shearing stress is proportional to velocity gradient i.e.

$$\tau = \mu \frac{du}{dy}$$

where $\frac{du}{dy}$ is velocity gradient, τ is tangential stress and μ is constant term known as coefficient of viscosity. A Newtonian fluid is which obey newton law of viscosity otherwise it is Non-Newtonian. Mineral water, gasoline, alcohol are examples of newtonian fluid whereas casson fluid, williamson fluid, ketchup etc are of non-newtonian.

Fluid Dynamics is the study of fluids in motion. Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyse problems that involve fluid flows.

1.1 Literature Survey

1.1.1 Computational Fluid Dynamics

CFD is Computational fluid dynamics. CFD is the use of applied mathematics, physics and computational software to envisage how the gas and liquid flows – as well as how the gas and liquid affects objects as it flows past. Computational fluid dynamics is based on the famous Navier-Stokes equations. So, CFD mean computational transport phenomena, so which involves heat and mass transfer, or any other process which involve transport phenomenan with it[2].

Objective:- The objective of computational fluid dynamics is to model fluid flow phenomenon with the Partial Differential Equations (P.D.E's) and discretize these PDEs into an algebraic equations (Taylor series), then solve it, validate it and then achieve simulation base design.

There are three approaches to solve this equation i.e. -

1. Experimental Approach
2. Analytical Approach
3. Numerical Approach

We choose computational fluid dyanamics (CFD) because:-

- Experimental modelling is expensive in full scale and often impossible.

- Experment modelling also have important errors which one can not neglect.
- Analytical solution are exact but often very difficult.

APPLICATIONS OF COMPUTATIONAL FLUID DYNAMICS

CFD has important applications in many field. These are:

- Aerospace
- Automobile
- Biomedical (human heart)
- Chemical (separation and mixing)
- Electronics
- Sports

First application is aerospace which is the most common application of computational fluid dynamics. If we consider an aircraft, we can see it has both an interior and exterior. The designing of an aircraft interior includes an air conditioning(ac) and ventilation system. When we have an exterior, we are in general considering the aero file section and combustion chamber which is within the aircraft and all we need to know is the transport phenomenon inside the combustion chamber. In the example of a case study when there is some unintentional incidence i.e, a fire set up in the aircraft, then what will happen. So, this all study is an appliaction of CFD.

Next if, we look into automobile applications, the basic philosophy is very similar. Let us take an example of fluid flow behaviour inside the car, as an example. The complete idea is to make a good design, so that the passenger feels comfortable inside the car, but the other important consideration is the external flow, that is how fluid flow takes place across and outside the car. Because that sort of determines the drag force on the car and that is where most of the clever design goes into. But one should not undermine the importance of design that goes for designing the ventilation system inside the car. Then there is the combustion chamber and combustion chamber design is also very important from the transport phenomena view point. These are some traditional applications of CFD.

In biomedical applications, we can see that we have flow patterns in bronchial tubes whose study is important to treat any disease of lungs. Human heart being modelled as a sort of a pump in CFD applications. Flow through arteries is also an interesting case where fluid flow analysis is very important. Also we have an example of not a fluid flow, but heat transfer. So, the example is like you have a tumour and somebody is trying to have a laser treatment to destroy that tumour. So that healthy cells will not be damaged. So that we can prevent the damage at a critical level only the tumor cells which are designed to be destroyed will be destroyed by the laser treatment.

CFD have a very wide range of applications in field of chemical industries. Injection of streams or separation mixing, these are all important chemical engineering applications. The new generation in present era is using many electronic devices. The size of devices are getting reduced. But if you are still having the same power rating, then what happens? So what should we do we to dissipate a much greater amount of heat per unit volume than what you had to dissipate for larger devices. For this we must have good cooling techniques because no matter what every electronic device needs a cooling strategy.

If we come to some fields of entertainment like sports or may be, there is a huge application of CFD. For example, if there is situation of driving a racing car, then how the fluid flows past that. It is very important because in such applications it is very important idea how to minimize the drag force to improve the performance. And in sports CFD is very extensively used for scientific purposes.

Navier-Stokes Equations

Navier-Stokes equation, in fluid mechanics, a partial differential equation that describes the flow of incompressible fluids. In 1822 French engineer CLAUDE-LOUIS MARIE HENRI NAVIER introduced the element of viscosity for the more realistic and vastly more difficult problem of viscous fluids. In 1845, British physicist and mathematician SIR GEORGE GABRIEL STOKES improved on this work, though complete solutions were obtained only for the case of simple two-dimensional flows. Gabriel gives the famous stoke's equation. So in physics, the Navier–Stokes equations, is named after Claude-Louis Navier and George Gabriel Stokes. These balanced equations comes from applying Newton's second law to fluid motion, along with the assumption that the stress in fluid is sum of a diffusing viscous term and a pressure term—hence describing viscous flow. Navier–Stokes equations are very useful because they explain the physics of many phenomena of scientific and engineering interest. The Navier–Stokes equations has important applications in the design of aircraft and cars, the study of blood flow, the design of power stations, the analysis of pollution, and many other things[4].

The fundamental equations of flow of viscous incompressible fluids are

1. **Equation of continuity- Conservation of mass**

   The Law of Conservation of Mass states that mass can be neither created nor be destroyed. Using the Mass Conservation Law on a steady flow process where the flow rate do not change over time through a control volume where the stored mass in the control volume do not change it implements that inflow equals outflow. This statement is called the Equation of Continuity. Common application where the Equation of Continuity are used are pipes, tubes and ducts with flowing fluids or gases, rivers, semiconductor technology and more. The equation of continuity is:

$$\frac{\partial v_j}{\partial x_j} = 0 \tag{1.1}$$

2. **Equation of motion- Conservation of momentum**

   Conservation of momentum is a fundamental law of physics which states that the momentum of a system is constant if there are no external forces acting on the system. The equation of motion are derived from newton second law of motion which states that rate of change of linear momentum is equal to the sum of the forces[1].

$$\rho(\frac{D}{Dt}v_i) = \rho f_i - \frac{\partial p}{\partial x_i} + \mu\frac{\partial^2 v_i}{\partial x_j \partial x_j} \tag{1.2}$$

3. **Equation of energy- Conservation of energy**

   In physics, the law of conservation of energy states that the total energy of an isolated system remains constant, it is said to be conserved over time. This law means that energy can neither be created nor destroyed; rather, it can only be transformed from one form to another. The equation of energy is:-

$$\rho c_v \frac{DT}{Dt} = \frac{\partial Q}{\partial t} + k\frac{\partial}{\partial x_i}(\frac{\partial T}{\partial x_i}) + \phi \tag{1.3}$$

where,

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + v_j \frac{\partial}{\partial x_j}, \tag{1.4}$$

and,

$$\phi = \mu(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i})\frac{\partial v_i}{\partial x_j} \tag{1.5}$$

Where, v is the velocity, ρ is fluid density, P is fluid pressure, f is body force, μ is dynamic viscosity, Q is heat, c_v is specific heat, T is temperature, $i = (1, 2, 3)$ One of the essential difference between the compressible flow and incompressible flow is that, in compressible fluid the equation of motion and energy are coupled and in incompressible flow, with the constant fluid property, they are uncoupled and therefore in this case the equation of continuity and equation of motion are first solved for the velocity component and the pressure thereafter equation of energy can be solved for temperature field.

The solution of the above equations become fully determined physically when boundary and initial condition are those required by geometrical considerations[3].

Problem in solution of Navier-Stokes equations

1. Continuity equation consists of a constraint on the velocity field which must be divergence free.

2. No direct link for the pressure.

3. For compressible flow, pressure and velocity can be coupled with the Equation of State. But for incompressible flow, there is no obvious way to couple pressure and velocity. So, their are INDIRECT METHODS.

Proposed important strategy

There are many formulations that have been proposed in the literature to deal with the difficulty of incompressibility condition.

- Pressure Poisson Equation
- Stream function vorticity

- Projection Method

1. **Pressure Poisson Equation:-** We take the divergence of the momentum equation and use the continuity equation to get a Poisson equation for pressure.

$$-\frac{1}{\rho}(\frac{\partial^2 p}{\partial x^2} + \frac{\partial^2 p}{\partial y^2}) = (\frac{\partial u}{\partial x})^2 + 2\frac{\partial u}{\partial x}\frac{\partial v}{\partial x} + (\frac{\partial v}{\partial y})^2 \tag{1.6}$$

where ρ is density, p is fluid pressure, u and v are components of velocity. A Poisson Equation for pressure, which ensures that continuity, is satisfied. Now pressure and velocity are coupled in the continuous domain.

2. **Stream function vorticity:-** For incompressible 2D flows with constant fluid properties, the Navier Stokes Equation can be simplified by introducing the stream function ψ and vorticity ω as dependent variables.
The vorticity vector at a point is defined as twice the angular velocity and is $\omega = \nabla \times u$.
Vorticity is mathematically defined as the curl of the velocity field and is hence measure of local rotation of fluid. For two-dimensional, incompressible flows, a scalar function may be defined in such a way that the continuity equation is identically satisfied[7].
Such a function is known as the streamfunction, and is given by:-

$$U = \nabla \times \psi\hat{k} \tag{1.7}$$

where, $U = u_i + v_j$

$$u = \frac{\partial \psi}{\partial y}, \quad v = -\frac{\partial \psi}{\partial x} \tag{1.8}$$

The PPE (pressure poisson equation) can also be written in terms of stream function using the relations in:-

$$\nabla^2 p = 2\rho(\frac{\partial^2 \psi}{\partial x^2}.\frac{\partial^2 \psi}{\partial y^2} - [\frac{\partial^2 \psi}{\partial x \partial y}^2]) \tag{1.9}$$

Poisson equation for pressure is an elliptic equation, showing the elliptic nature of pressure in incompressible flows. For a steady flow problem, the PPE is solved only once, i.e., after the steady state values of ω and ψ have been computed.

3. **Projection Method:-** The projection method is an effective means of numerically solving time-dependent incompressible fluid-flow problems. They achieve this efficiency by: Interpreting the pressure as effecting a projection of the flow velocity evolution into the set of incompressible fields. It was originally introduced by ALEXANDRE CHORIN in 1967 as an efficient means of solving the incompressible Navier-Stokes equations[3, 5]. The key advantage of the projection method is that the computation of the velocity and the pressure fields are decoupled.

   The algorithm of projection method is based on the Helmholtz decomposition of any vector field into a solenoidal part and an irrotational part. Typically, the algorithm consists of two stages.

   - In the first stage, an intermediate velocity that does not satisfy the incompressibility constraint is computed at each time step.
   - In the second, the pressure is used to project the intermediate velocity onto a space of divergence-free velocity field to get the next update of velocity and pressure.

   3(a) CHORIN PROJECTION METHOD

   Navier stoke equation for incompressible flow:-

$$\frac{\partial u}{\partial t}u \ + (u \cdot \nabla)u = -\frac{\nabla P}{\rho} + \nu \Delta^2 u \tag{1.10}$$

   - In Chorin projection method find u^*, explicitly using the momentum equation by ignoring pressure gradient term[3].

$$\frac{u^* - u^n}{\Delta t} = -(u^n.\nabla)u^n + \vartheta \Delta^2 u^n \tag{1.11}$$

$$\frac{u^{n+1} - u^*}{\Delta t} = -\frac{1}{\rho}\nabla P^{(n+1)} \tag{1.12}$$

   Pressure Poisson equation for $P^{(n+1)}$:-

$$\nabla^2.P^{(n+1)} = \frac{\rho}{\Delta t}\nabla.u^* \tag{1.13}$$

   Knowing the value of $P^{(n+1)}$ we can calculate $u^{(n+1)}$.

1.1.2 Two phase fluid

In fluid mechanics, two-phase flow is a particular example of multiphase flow. It can occur in various forms, such as flows changing from pure liquid to vapour by any source like external heating, dispersed two-phase flows where one phase is present in the form of particles, droplets, or bubbles in a continuous carrier phase.

The method to categorize the two-phase flows is to consider the velocity of each phase as other phases is not available. Examples of two-phase flow include bubbles, rain, waves on the sea. This fluid system has numerous application in various natural processes like blood flow in arteries, dust in gas cooling systems, movement of inert solid particles in atmosphere, sand or other suspended particle in sea beaches.

Dusty fluid model flows have been a subject of special interest in recent studies. This phenomenon occurs in fluid flows containing a distribution of solid particles. For example, motion of the dusty air in fluidization problems and the chemical process in which raindrops are formed by coalescence of small dust particles. Cosmic dust, which is formed due to the mixing of dust particles and gas, is primary precursor for planetary systems. The production of tails of comet 238 is due to emission of ionized gas and the dust particles from the comet body. Heat transfer of dusty nanofluids have wide applications in engineering and sciences. The application of the dusty fluid can also be visualized in the processes such as nuclear reactor cooling, atmospheric fallout, powder technology, dust collection, acoustics, paint spray, rain erosion, sedimentation, performance of solid fuel rock nozzles, and guided missiles. These facts have expedited the consideration of modeling, solving, and analyzing the flow of dusty fluids.

Saffman [12] was the first to develop expressions for viscous liquid flow with the insertion of immiscible inert solid particles. Chakrabarti [13] investigated the flow of dusty gas in the boundary layer region. Singleton[16] initiated the study of boundary layer problem of impact of dust particle suspended in liquid. Flow and heat transfer behavior of dusty fluid over a long annular pipe was studied by Datta and Dalal (1995). With the use of Legendre transformation, Siddabasappa et. al. [20] solved a coupled partial differential equations arises in the flow of dusty viscous fluid. It was observed that as the density of the dust particle increases the velocity of the dust phase decreses. Non newtonian fluid suspended with dust particle theory extremely accomodates in understanding the physical phenomenan of engineering situations. In the category of non-Newtonian fluids, the Williamson fluid has distinct features. Williamson fluid model describes the flow of shear thinning non-newtonian fluid. It is used in modeling of biological fluids in small passages and have many other applications in chemical and biological science, geophysics and petroleum industries. Dusty casson fluid, dusty williamson nanofluid, dusty nanofluid are examples of two phase model. Williamson et al. [9] reported theory of pseudo plastic fluids. He explained practical significance of plastic flows that plastic flows are very different from viscous flows. The williamson nanofluid is better that williamson fluid due to thermal conductivity. Dusty williamson fluid is a new approach to understand the phenomenon of heat transfer. It is studied that rate of heat transfer can be enhanced by suspension of dust particle in base fluid.

Chapter 2

HEAT TRANSFER

2.1 Introduction

Heat transfer deals with the rate of heat transfer between different bodies. Heat transfer analysis permits a calculation of the heat loss from a building surface to the surroundings for a given building size, window area and wall design, e.g. the level of insulation in the wall cavity. The comfort conditions for occupants in a room is determined by a balance of heat transfer from the person to the air surrounding him or her as well as the heat transfer to the walls of the interior. The size and cost of a heat exchanger is also determined by considering the heat transfer between the fluid streams in the exchanger.

In other fields, heat transfer plays a key role as well. The design of integrated microprocessors which contain very closely spaced elements, each with a finite amount of heat generation, is limited by the requirement for adequate cooling so that the operating temperature of the electronic components is not exceeded. Reentry of the space shuttle in the earth's atmosphere must be carefully programmed so that temperature extremes due to air friction are confined to the insulating tiles on the shuttle's surface.

2.2 Difference between heat and temperature

In heat transfer problems, we often interchangeably use the terms heat and temperature. Actually, there is a difference between the heat and temperature. Temperature is a measure of the amount of energy possessed by the molecules of a substance. It manifests itself as a degree of hotness, and can be used to predict the direction of heat transfer. The usual symbol for temperature is T. The scales for measuring temperature in SI units are the Celsius and Kelvin temperature scales. Heat, on the other hand, is energy in transit. Spontaneously, heat flows from a hotter body to a colder one. The usual symbol for heat is Q. In the SI system, common units for measuring heat are the Joule and calorie.

2.3 Thermodynamics and Heat Transfer

The science of thermodynamics deals with the amount of heat transfer when a system undergoes a process from one equilibrium state to another, and it make no refernce to how long the process will take. But rate of heat tranfer i.e how much heat is tranferring is subject of concern. In this text the area of interest is in heat, which is form of energy that can be transferred from one system to another as a result of difference. The science that deals with the determination of the rates of such energy transfers is heat transfer. Heat transfer obey the laws of thermodynamics.

2.3.1 The four laws of thermodynamics

- **Zeroth law:** If two bodies both are in thermal equilibrium with a third body, they are also in thermal equilibrium with each other, and they then are said to have the same temperature.

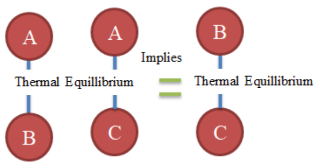

Figure 2.1: Zeroth law of thermodynamics

- **First law:** (Energy principle) Energy cannot be generated or destroyed, only converted to different forms. ("Energy consumption" is the transfer of "prime" energy to thermal energy in the surrounding)

- **Second law:** Heat cannot by itself pass from one body to another body with higher temperature. (Entropy [disorder] strives to a maximum in a closed system.)

- **Third law:** The entropy of a pure, crystalline material takes its lowest value at absolute zero temperature, where it is 0. (There is a lowest limit to the temperature, 0K = -273.15°C. World record: 0.00000017 K = 170) nanokelvin).

2.4 Modes of Heat Transfer

There are main three types of heat transfer

1. Conduction
2. Convection
3. Radiation

2.4.1 Conduction

Conduction is the heat transfer of energy from more energetic particles of a substance to adjacent, less energetic ones as a result of interaction between the particles. An example is a copper rod. If one end is heated, thermal energy moves along the rod in a "daisy chain" from atom to atom because those atoms are "against" each other. In a paper published in 1822, Theorie Analytique de la Chaleur, Jean Baptiste Joseph Fourier set the mathematical theory of heat conduction.

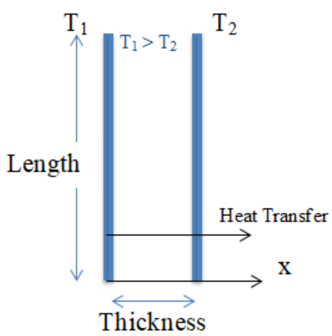

Figure 2.2: Conduction

Fourier's law is the defining equation for the thermal conductivity.

$$q = -k \cdot A \cdot \frac{\delta T}{\delta x}$$

where, q = heat flow (W), A = area perpendicular to heat flow (m^2), $\frac{\delta T}{\delta x}$ = temperature gradient in the direction of heat flow(°C/m) and k = thermal conductivity (W/(m °C)).

For one-dimensional heat transfer (a plane wall,) with constant thermal conductivity, Fourier's law is simplified to

$$q = k \cdot A \cdot \Delta T/\delta$$

where ΔT = temperature difference (°C), δ = distance or thickness (m).

2.4.2 Convection

Convection is the mode of heat transfer between a solid surface and adjacent liquid or gas that is in motion and it involves the combined effects of conduction and fluid motion. If we look at a hot stove in a kitchen, that appliance will heat air that is against its surfaces. The hot air will rise away from the unit, and cooler air will take its place and become heated in turn. This continues with the air carrying off the heat and warming the whole kitchen. Many people are familiar with convection from heating their food or their houses, and it also plays a pivotal role in creating the weather conditions on the planet.
The convective heat transfer coefficient is defined as,

$$h = \frac{q}{A\ (T_s - T_a)}$$

where T_a is temperature of air, T_s is temperature of surface, q is heat flow from surface, A is surface area from which convection is occurring.

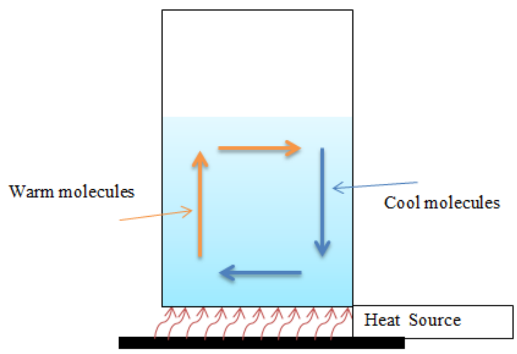

Figure 2.3: Convection when heat is given to water.

There are two main types of convection: forced convection and natural convection.

- **Natural convection**:- Natural convection occurs when the medium transferring the heat is being inspired to move by the heat itself. This happens both because the medium expands as it heats up, as in the case of gases, and also because buoyancy causes the warmer fluid to rise. Natural convection is also sometimes referred to as free convection.

- **Forced convection:-** Forced convection is a mechanism, or type of transport in which fluid motion is generated by an external source (like a pump, fan, suction device, etc.). It should be considered as one of the main methods of useful heat transfer as significant amounts of heat energy can be transported very efficiently.

2.4.3 Radiation

Radiation is the energy emitted by matter in the form of electromagnetic waves or photons as a result of the changes in the electronic configuration of the atoms and molecules. An individual who has had a sunburn on his face might look at a red hot electric range element and feel his face burning. The sunburn will have made his skin extremely sensitive to heat, particularly since it has been damaged by heat. And the infrared electromagnetic waves from the burner are reaching the individual's face and it burns. Just holding a hand palm out toward a heat source (but not above it to avoid convection) can have a dramatic result. The heat can be felt, and there is no conduction or convection taking place immediately to get the thermal energy from the source to the hand of the observer. That heat must be arriving via an "energy transfer" of some kind.
For a black body at a uniform temperature T, the rate of radiation heat transfer emitted by the body and leaving the surface, over all wavelengths, is given as

$$q_{emitted} = A\sigma T^4$$

where T is the absolute temperature and σ is the Stefan-Boltzmann constant.

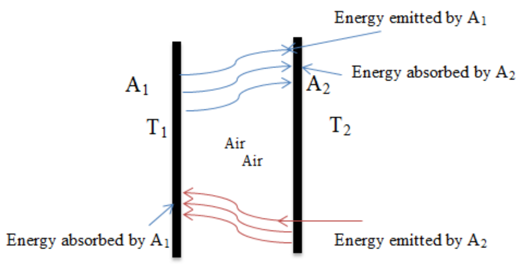

Figure 2.4: Radiation between parallel black plates

2.5 Marangoni Convection

The Marangoni effect refers to the variation of surface tension of a liquid with temperature (thermocapillarity) or with the concentration of a surfactant (solutal Marangoni effect). The variation of surface tension in turn leads to convective motion of the fluid: Marangoni convection. This motion along the surface of the liquid layer then leads to flow in the bulk and may be used to transport fluids in microfluidic devices.

Solutal Marangoni convection is caused by variation in the concentration of a surfactant at the interface between two liquids or a liquid and a gas. The surface tension being a function of the local surface concentration of surfactant, a variation in the latter will affect the value of surface tension locally and therefore give rise to stresses along the interface. The relation between the surface concentration of surfactant and the surface tension is a complex one which will be addressed.

The surface tension typically changes due to variations in solute concentration, surfactant concentration, and temperature variations along the interface. In day today life we can see the example of Marangoni convection in tear of wine.

The alcohol concentration gradient, caused by differences in the transport rate of alcohol from the bulk to the flat surface and from the bulk to the meniscus, results in surface tension gradients and the formation of tears of wine.

Figure 2.5: Tears of wine

The phenomenon called tears of wine is manifested as a ring of clear liquid, near the top of a glass of wine, from which droplets continuously form and drop back into the wine. It is most readily observed in a wine which has a high alcohol content. It is also referred to as wine legs.

This is because alcohol has a lower surface tension than water. If alcohol is mixed with water, a region with a lower concentration of alcohol will pull on the surrounding fluid more strongly than a region with a higher alcohol concentration. The result is that the liquid tends to flow away from regions with higher alcohol concentration. This can be easily demonstrated by spreading a thin film of water on a smooth surface and then allowing a drop of alcohol to fall on the center of the film. The liquid will rush out of the region where the drop of alcohol fell.

Wine is mostly a mixture of alcohol and water, with dissolved sugars, acids, colourants and flavourants, where the surface of the wine meets the side of the glass, capillary action makes the liquid climb the side of the glass. As it does so, both alcohol and water evapourate from the rising film, but the alcohol evapourates faster, due to its higher vapour pressure. The resulting decrease in the concentration of alcohol causes the surface tension of the liquid to increase, and this causes more liquid to be drawn up from the bulk of the wine, which has a lower surface tension because of its higher alcohol content. The wine moves up the side of the glass and forms droplets that fall back under their own weight.

The Marangoni number is the non-dimensional number that gives the ratio between the thermocapillary effect and the viscous forces.

2.5.1 Impact of Marangoni Effect

First noted in the phenomenon of tears of wine, the Marangoni effect has been observed in various surface chemistry and fluid flow processes.

1. **Welding**
   Welding is a fabrication process where the Marangoni effect has to be accounted for. When the base metal during welding reaches its melting point, a weld pool forms. Marangoni forces within these pools can affect the flow and temperature distribution and modify the molten pool extension. This can potentially result in stresses within the material as well as deformation.

2. **Crystal Growth**
   Semiconductors are usually comprised of crystal lattice structures. The

processes for developing pure crystals (e.g., silicon) consist of purifying the metal, which begins with melting the solid. During purification, convection in the liquid phase must be allowed so that impurities like oxides, which are often lighter than metal, have time to separate. Additionally, heat transfer must be regulated so that the shape of the solidification front is controlled.

Forces from the Marangoni effect can impact crystal growth, causing faults within the structure. These faults can inhibit the material's semiconducting capabilities and result in defects within the device.

3. **Electron Beam Melting**
Using an electron beam as its power source, metal powder can be melted to manufacture mechanical parts. This method of additive manufacturing has gained popularity in the development of medical implants as well as the aerospace industry. During the melting process, large thermal gradients can generate Marangoni forces within the melt. Such forces have the potential to negatively affect the quality of the material.

Chapter 3

FINITE ELEMENT METHOD

Finite element method (FEM) is a mathematical tool with which we solve boundary-value problems (BVP) or initial value problems (IVP) of mathematical physics.

3.1 An Introduction to the Finite Element Method

The description of the law of physics for time and space dependent problems are generally expressed in terms of PDEs (partial differential equations). For the wide majority of problems and geometries, these PDEs can't be solved using analytical methods. Instead, an approximation solution of these equations can be drawn, based upon the disparate types of discretizations. These discretization method approximates the PDEs with the numerical model equations, which can further be solved using given numerical methods. The solution to the numerical model equations are an approximation of real solution to partial differential equations. The finite element method (FEM) is used to compute such approximations.

The finite element method involves basically four steps:

1. First step is discretization. The solution region is discretize into finite number of element or subregion.

2. Derive the governing equations for every element related to either a variational approach or any other method like Galerkin method.

3. Assemble all the elements in the solution space together.

4. Now last step is to solve the system of equations.

Discretization is dividing a model body into an equivalent systems of many smaller finite elements/bodies which are connected at points common to two or more nodes/ nodal points and/or surfaces and/or boundary lines. The elements or nodes are numbered.

3.2 One-Dimentional Domain

In one dimensional problem, the domain can only be divided into line element. A typical element $\Omega^e = (\eta_e, \eta_{e+1})$, whose end points have the coordinates $\eta = \eta_e$ and $\eta = \eta_{e+1}$, is isolated from mesh.

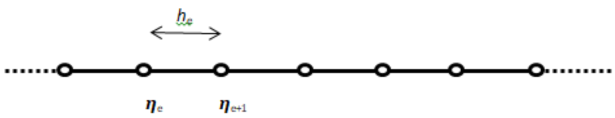

Figure 3.1: One dimentional domain

The shape function Ω^e over a typical line element is given by:

$$\phi_1^e = \frac{\eta_{e+1} - \eta}{\eta_{e+1} - \eta_e}, \qquad \phi_2^e = \frac{\eta - \eta_e}{\eta_{e+1} - \eta_e}, \qquad where\ \eta_e \leq \eta \leq \eta_{e+1} \tag{3.1}$$

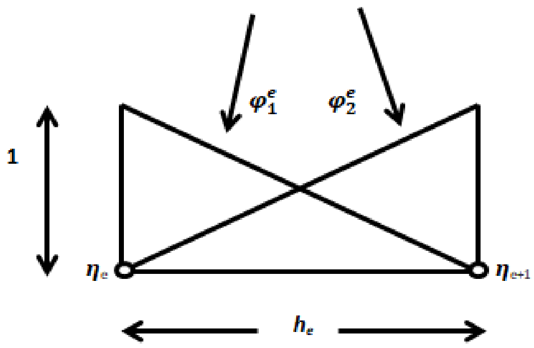

Figure 3.2: Domain

3.3 Interpolation function

The finite element approximation over an element Ω^e is $U^e(x, y)$ of $u(x, y)$ must satisfy the following conditions so that approximate solution to be convergent to the true solution:

1. U^e must be differentiable, as in weak form of problem.
2. The polynomial used to represent U^e must be complete.
3. All term in polynomials must be L.I i.e, linearly independent.

The number of L.I terms in representation of U^e give the shape and number of degrees of freedom for that element. For example, the polynomial

$$U^e(x, y) = c_1 + c_2x + c_3y,$$

contains three L.I terms, and it is linear in both x and y. To write the constants $c_i(i = 1, 2, 3)$ in the terms of the nodal values U^e, it must identify three points or nodes in the elements Ω^e. These 3 nodes also uniquely define the geometry of the element. The geometric shape define by 3 points in a 2-D domain is a triangle. On the other side, the polynomial:

$$U^e(x, y) = c_1 + c_2x + c_3y + c_4xy,$$

it contains four L.I terms and is linear in x, y with a bilinear term in x and y. The element require an element with 4 nodes. There are 2 possible geometric shapes - a rectangle with the node at vertices or a triangle with fourth node at its centre. The linear rectangular element is a compatible element. A triangle with fourth node at its centre will not provide a single valued variation of u at interelement boundary, which results in incompatible variations in u, and is therefore not admissible.
A polynomial with five constants is incomplete quadratic polynomials.

$$U^e(x, y) = c_1 + c_2x + c_3y + c_5(x^2 + y^2)$$

which can be used to form an element of five nodes which is a rectangle with node at corner and one node at its midpoint. However, the element will not give single valued variation of u. Also the terms x^2 and y^2 cannot be varied seperately from each other.
The quadratic polynomial

$$U^e(x, y) = c_1 + c_2x + c_3y + c_5x^2 + c_6y^2$$

with six constants can be used to construct the element with 6 nodes. For example, a triangle with a node at each vertex and a node at midpoint at each side is admissible, it is known as quadratic triangular.

If the degree of approximation that is used to describe the geometry is equal to degree of approximation of solution $U^e(x, y)$ then that element will known as isoparametric elements.

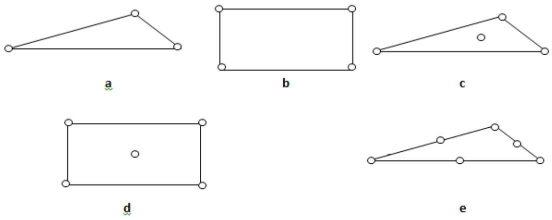

Figure 3.3: Nodes

3.4 Derivation of element equation

1. A typical element is isolated from the mesh and variational formulation of the required problem over the typical element is build up.
2. The approximate solution of the variational problem is assumed, by substituting the values in domain equations, the element equation is constructed.
3. The element matrix known as stiffness matrix is bulid up by using the element interpolation functions. The interpolation function depends on the type of element i.e number of nodes, geometry, and number of primary unknowns per node. The function have to be selected. [18]

3.5 Assembly of Element Equation

The assembly of FEM is based on two principles that is:

1. Continuity of primary variables
2. Equilibrium of secondary variables

The process by considering a finite element mesh consisting of a quadrilateral element and a triangular element is shown in fig 3.4. Let $K^1_{ij}(i = 1, 2, 3)$ denotes the coefficient matrix corresponding to the triangular element, and let $K^1_{ij}(i = 1, 2, 3, 4)$ denotes the coefficient matrix corresponding to quadrilateral element.

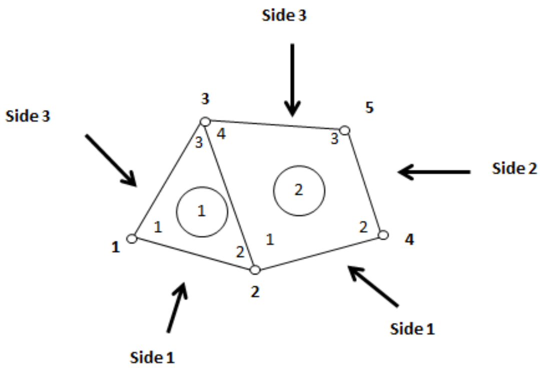

Figure 3.4: Assembly of nodes

Global	*Local*
K_{11}	K^1_{11}
K_{12}	K^1_{12}
K_{22}	$K^1_{22} + K^2_{11}$
K_{14}	0
K_{15}	0
K_{23}	$K^1_{23} + K^2_{14}$

For finite element mesh that is shown in figure assembly of nodes. It is to be noted following connectivity relations between global and element nodes:

$$[B] = \begin{bmatrix} 1 & 2 & 3 & \times \\ 2 & 4 & 5 & 3 \end{bmatrix}$$

where $\times$ denotes that there is no entry. The correspondance between global and local nodes can be seen in fig 3.4.

3.6 Imposition of boundary conditions

The natural and essential boundary conditions are imposed on the assembled equations.

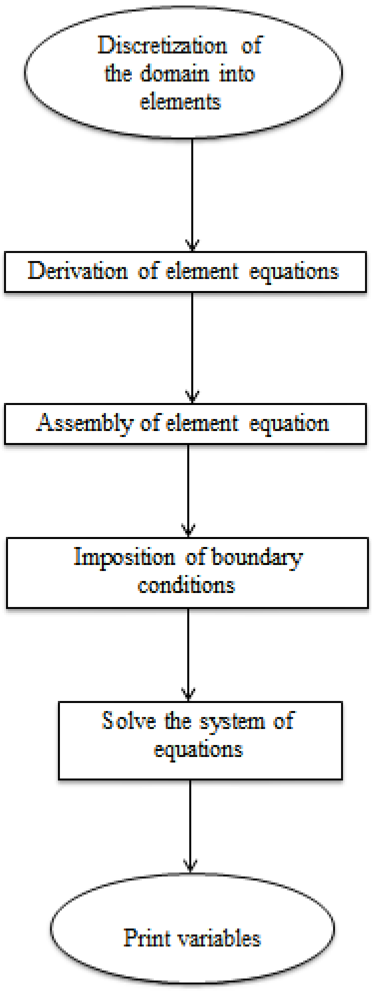

Figure 3.5: Flow chart of FEM

3.7 Solution of assembled equations

The coming assembled equations can be solved by any numerical method that can be Gauss- Elimination method, L-U decomposition, Gauss-Seidel etc

3.8 Advantages of Finite Element Method

1. FEM be used on complex geometries. (in contrary to finite difference method)

2. The weak form automatically include "natural boundary conditions".

3. Powerful computational method for a large number of different applications.

4. Finite element method scales well give calculations more accurate without changing your mesh.

5. FEM tends to capture small details in solutions better than finite volume methods.

6. FEM is superior for multiphysics problems, where there is a high degree of coupling and non-linearity.

Common FEM Applications

- Mechanical
- Aerospace
- Civil/Automotive Engineering
- Structural/Stress Analysis
- Fluid Flow
- Heat Transfer
- Electromagnetic Fields
- Soil Mechanics
- Acoustics
- Biomechanics

Chapter 4

2-PHASE DUSTY WILLAMSON NANOFLUID FLOW

4.1 Introduction

Fluid flow which contain identical non-miscible inert solid particles is known as a two-phase fluid system. The method to categorize the two-phase flows is to consider the velocity of each phase as other phases available. Examples of two-phase flow include bubbles, rain, waves on the sea. This fluid system has numerous application in various natural processes like blood flow in arteries, dust in gas cooling systems, movement of inert solid particles in atmosphere, sand or other suspended particle in sea beaches. Dusty fluid model flows have been a subject of special interest in recent studies. From past few decades, researchers have been focusing on analyzing the heat and mass transfer characteristics of dusty nanofluids through different channels. The present study is about the heat transfer characteristics of a dusty williamson nanofluids.

The Marangoni effect refers to the variation of surface tension of a liquid with temperature (thermocapillarity) or with the concentration of a surfactant (solutal Marangoni effect). Marangoni effect is divided into the thermal Marangoni effect (EMT) and the solute Marangoni effect (EMS). The basic mechanism of EMT and EMS has been extensively investigated. Theory of EMT is used in present work. In EMT, Pearson[10] created the initial model and criterion. Experiments showed that the surface tension of fluid is connected to the temperature, in most of the cases the surface decreases as the temperature increases[11].

Nano-fluid is a fluid that contains nanosized particles well known as nanoparticles. The nano particles used to form nanofluid are carbon nanorods, metal, non-metals and other metal oxides. Nanofluids have a wide applications in geophysics, heat transfer (due to their strong convection process), fluid flow, fuel cell, pharmaceutical process. Choi (1995)[6] was the first person who used the term nano fluid due to suspension of nano sized particle in fluid. He discussed about that the nano particles are much better as compared to micro sized particles. Nanofluid have higher thermal conductivity in a base fluid. Base fluid can be water or oil. It is to be noted that these nanoparticles must be ultrafine i.e length of order 1-50 nm, so that nanofluids behaves more like a single-phase fluid than a solid-liquid suspension. Non-newtonian nanofluid has better heat trasfer property as compared to newtonian fluid e.g liquid metals such as mercury, francium, rubidium, this is the reason non-newtonian fluid are prefered.

The theory of multi/two phase fluids extremely accommodates in understanding the physical phenomena of engineering situations such as the cement production, steel manufacturing industry, waste water treatment, combustion, dust in gas cooling systems and atmospheric fallout. To explain some of the recent studies in past on the flow and heat transfer analysis, Saffman [12] was the first to develop expressions for viscous liquid flow with the insertion of immiscible inert solid particles. Chakrabarti [13] investigated the flow of dusty gas in the boundary layer region. Singleton[16] initiated the study of boundary layer problem of impact of dust particle suspended in liquid. Flow and heat transfer behavior of dusty fluid over a long annular pipe was studied by Datta and Dalal (1995). With the use of Legendre transformation, Siddabasappa et. al. [20] solved a coupled partial differential equations arises in the flow of dusty viscous fluid. It was observed that as the density of the dust particle increases the velocity of the dust phase decreses, and then after many researches where taken into account for two phase dusty fluids under different physical circumstances[19, 22, 23].

Some of the non-newtonian fluids such as blood, honey, starch, molten plastics, artificial fibers, foodstuffs and slurries exhibit shear-stress-strain relationships which are distinct from the Newtonian model. Mostly non-Newtonian models involve some form of modification to the momentum equations. In the category of non-Newtonian fluids, the Williamson fluid has distinct features. Williamson fluid model describes the flow of shear thinning non-newtonian fluid. It is used in modeling of biological fluids in small passages and have many other applications in chemical and biological science, geophysics and petroleum industries. Williamson et al. [9] reported theory of pseudo plastic fluids. He explained practical significance of plastic flows that plastic flows are very different from viscous flows. The williamson nanofluid is better that williamson fluid due to thermal conductivity[14]. In this context, many authors [14, 15, 17] have studied the heat transfer flow of Willamson nanofluid with distinct physical aspects.

The aim of present work is to investigate the flow and heat transfer phenomenon of dusty williamson fluid with marangoni convection and radiation. In section 4.2 governing non-linear partial differential equation converted into ordinary differential equations using similarity transformation. Those are solved by FEM in section 4.3. Effects of important physical parameters that are radiation parameter, dust particle mass concentration parameter, thermal dust parameter have been shown graphically. To the best of author's knowledge no work has been reported on Dusty Williamson Nanofluid flow with marangoni convection.

4.2 Mathematical Analysis

Let us consider a steady two-dimensional viscous incompressible dusty fluid in a region $y > 0$ driven by a plane surface located at $y = 0$ with a fixed end at $x = 0$. Let the x, y axes are taken along the surface and perpendicular to the surface shown in figure 4.1. The surface temperature is supposed to be power law variation. The surface tension σ fluctuates with temperature in linear form. That is,
$\sigma = \sigma_o \left[1 - \varrho(t - t_o)\right]$
where σ_0 is surface tension and t_o is temperature charcterstics. ϱ is positive

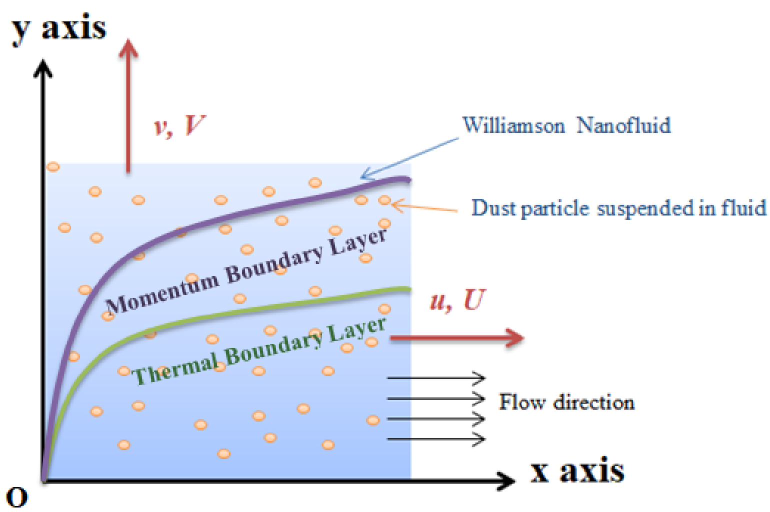

Figure 4.1: Schematic diagram of the problem

fluid property. Let $t = t_o$.

Marangoni convective radiated flow driven by thermal gradient above a planar interface is considered. With the mentioned assumptions and under the usual boundary layer approximation, the governing equation for both williamson fluid and dust particle can be written as follows:

Continuity Equation:-

$$u_x + v_y = 0 \tag{4.1}$$

$$U_x + V_y = 0 \tag{4.2}$$

Momentum Equation:-

$$u\,\frac{\partial u}{\partial x} + v\,\frac{\partial u}{\partial y} = \vartheta\,\frac{\partial^2 u}{\partial y^2} + \sqrt{2}\,\vartheta\,\Gamma\,\frac{\partial u}{\partial y}\,\frac{\partial^2 u}{\partial y^2} + \sqrt{2}\,\frac{\rho_P}{\rho_f \tau_m}\,(U - u) \tag{4.3}$$

$$UU_x + VV_y = \frac{-1}{T_m}(u - U) \tag{4.4}$$

Energy Equation:-

$$(\rho c)_f\,(ut_x + vt_y) = (K_f + \frac{16\sigma^* t_\infty^3}{3K*})t_{yy} + \frac{\rho_P C_P}{\tau_t}(T - t) \tag{4.5}$$

$$UT_x + VT_y = \frac{-1}{\tau_t}(t - T) \tag{4.6}$$

subjected to boundary condition as:

at $y = 0$, $\quad \mu_f u_y = \frac{d\sigma}{dt}\frac{dt}{dx}$, $\quad v = 0$, $\quad t = t_o + Ax^{m+1}$.

at $y \to \infty$, $\quad u \to 0$, $\quad U \to 0$, $\quad V \to v$, $\quad t \to t_\infty$, $\quad T \to t_\infty$

In which (u, v) and (U, V) denotes the velocities of the williamson nanofluid and dust particles respectively. m is constant exponent of temperature, k is thermal conductivity, ϑ is kinematic viscosity, Γ is time constant, $\tau_t = \tau_t^* x^{\frac{2(m-1)}{3}}$, τ_t^* is thermal relaxation time of dust particles $= m_P c_P / 4\pi r_P$, $\tau_m = \tau_m^* x^{\frac{2(m-1)}{3}}$, τ_m^* is momentum relaxation time of dust particles $= m_P / 6\pi\mu r_P$, c and c_P are specific heat of williamson nanofluid and dust particles, m_P and c_P

are the mass and radius of dust particles, σ_* is Stefan- Boltzmann Constant. The subscript (x, y) denotes the partial derivatives and f and P denotes the nanofluid and nanoparticle respectively.

Using the following similarity transformation:-

$$\psi = C_1 x^{\frac{2+m}{3}} f(\eta), \quad \theta(\eta) = \frac{t - t_\infty}{A x^{m+1}}, \quad \eta = C_2 x^{\frac{m-1}{3}} y,$$

$$\Psi = C_1 x^{\frac{2+m}{3}} F(\eta), \quad \Theta(\eta) = \frac{T - t_\infty}{A x^{m+1}},$$

Where ψ and Ψ are stream function such that $u = \frac{\partial \psi}{\partial y}$, $U = \frac{\partial \Psi}{\partial y}$, and $v = \frac{-\partial \psi}{\partial x}$, $V = \frac{-\partial \Psi}{\partial x}$, further m, A, C_1, C_2 are constants with A, C_1, C_2 are given by:-

$$A = \frac{\delta t}{L^(m+1)}, \quad C_1 = \sqrt[3]{\frac{\sigma_T A \mu_f}{\rho_f^2}}, \quad C_2 = \sqrt[3]{\frac{\sigma_T A \rho_f}{\mu_f}^2}$$

δt is the constant chracterstic temperature and L is the surface length.

Substituting these values in eqs. (4.1) (4.2) (4.3) (4.4) (4.5) (4.6). Our net dimensionless equations are:-

$$f^{'''} + \lambda f^{''} f^{'''} + l\beta_v [F^{'} - f^{'}] + \beta[f f^{''}] - \alpha f^{'2} = 0 \tag{4.7}$$

$$\beta_v [f^{'} - F^{'}] + \beta[F F^{''}] - \alpha F^{'2} = 0 \tag{4.8}$$

$$\frac{1+R}{Pr} \theta^{''} + l\gamma\beta_t[\Theta - \theta] + \beta[\theta^{'} f] - (m+1) f^{'} \theta = 0 \tag{4.9}$$

$$\beta_t[\Theta - \theta] + \beta[\Theta^{'} F] - (m+1) F^{'} \Theta = 0 \tag{4.10}$$

Along with the boundary condition:-

$f^{''}(0) = -2$, $\theta(0) = 1$, $f(0) = 0$, $\theta(\infty) = 0$, $\Theta(\infty) = 0$, $F(\infty) \to f(\infty)$, $f^{'}(\infty) = 0$, $F^{'}(\infty) = 0$

where prime signifies the derivative with respect to η, $\beta_v = \frac{1}{\tau_m^* C_1 C_2}$, momentum dust parameter, $\beta_t = \frac{1}{\tau_t^* C_1 C_2}$, thermal dust parameter, $l = \frac{\rho_P}{\rho_f}$, dust particle mass concentration parameter, $\gamma = \frac{c_f}{c_P}$ is specific heat parameter, $Pr = \frac{(\mu c)_f}{k_f}$ is Prandtl number, $\lambda = \Gamma x^m \sqrt{2} C_1 C_2^2$ is non newtonian williamson parameter and $R = \frac{16\sigma^* t^3}{3k_f k^*}$ is radiation parameter, $\alpha = \frac{2m+1}{3}, \beta = \frac{m+1}{3}$ where α and β are constants.

In order to simplify the problem assuming:-

$$f^{'} = h, \tag{4.11}$$

$$F^{'} = H, \tag{4.12}$$

the system of equation (4.7)(4.8)(4.9)(4.10) (4.11) (4.12) with boundary condition, converted into:-

$$h^{''} + \lambda \bar{h}^{'} h^{''} + lB_v[H - h] + \beta[\bar{f} h^{'}] - \alpha[\bar{h} h] = 0, \tag{4.13}$$

$$B_v[h - H] + \beta[\bar{F} H^{'}] - \alpha[\bar{H} H] = 0, \tag{4.14}$$

$$\frac{1+R}{Pr}\theta^{''} + l\gamma B_t[\Theta - \theta] + \beta[\theta^{'} \bar{f}] - (m+1)\bar{h}\theta = 0 \tag{4.15}$$

$$B_t[\Theta - \theta] + \beta[\Theta^{'} \bar{F}] - (m+1)\bar{H}\Theta = 0 \tag{4.16}$$

4.3 Numerical modelling

The dimesionless equations (4.13)(4.14)(4.15)(4.16), obtained are non linear in nature, therefore the above system of equation cannot be solved analytically. So FEM is used for numerical solution.

4.3.1 Variation Formulation

The variation formulation over a typical linear element (η_e, η_{e+1}) is devloped as follows:-

$$\int_{\eta_e}^{\eta_{e+1}} w_1(f^{'} - h) d\eta = 0 \tag{4.17}$$

$$\int_{\eta_e}^{\eta_{e+1}} w_2(F^{'} - H)d\eta = 0 \tag{4.18}$$

$$\int_{\eta_e}^{\eta_{e+1}} w_3(h^{''} + \lambda\bar{h}^{'}h^{''} + lB_v[H - h] + \beta[\bar{f}h^{'}] - \alpha[\bar{h}h])d\eta = 0 \tag{4.19}$$

$$\int_{\eta_e}^{\eta_{e+1}} w_4(B_v[h - H] + \beta[\bar{F}H^{'}] - \alpha[\bar{H}H])d\eta = 0 \tag{4.20}$$

$$\int_{\eta_e}^{\eta_{e+1}} w_5(\frac{1+R}{Pr}\theta^{''} + l\gamma B_t[\Theta - \theta] + \beta[\theta^{'}\bar{f}] - (m+1)\bar{h}\theta)d\eta = 0 \tag{4.21}$$

$$\int_{\eta_e}^{\eta_{e+1}} w_6(B_t[\Theta - \theta] + \beta[\Theta^{'}\bar{F}] - (m+1)\bar{H}\Theta)d\eta = 0 \tag{4.22}$$

where $w_1, w_2, w_3, w_4, w_5, w_6$ are test functions and may be viewed as the variation in $f, F, h, H, \theta, \Theta$ respectively.

4.3.2 Finite element formulation

The finite element model for equations obtain after variation formulation by substituting finite element approximation of the following form,

$$f = \sum_{j=1}^{2} f_j\phi_j, \quad F = \sum_{j=1}^{2} F_j\phi_j, \quad h = \sum_{j=1}^{2} h_j\phi_j, \quad H = \sum_{j=1}^{2} H_j\phi_j, \tag{4.23}$$

The shape function for the line element (η_e, η_{e+1}) are taken as:

$$\phi_1^e = \frac{\eta_{e+1} - \eta}{\eta_{e+1} - \eta_e}, \qquad \phi_2^e = \frac{\eta - \eta_e}{\eta_{e+1} - \eta_e}, \qquad where\ \eta_e \leq \eta \leq \eta_{e+1} \tag{4.24}$$

The model of finite element of equations is given by:-

$$\begin{bmatrix} [K^{11}] & [K^{12}] & [K^{13}] & [K^{14}] & [K^{15}] & [K^{16}] \\ [K^{21}] & [K^{22}] & [K^{23}] & [K^{24}] & [K^{25}] & [K^{26}] \\ [K^{31}] & [K^{32}] & [K^{33}] & [K^{34}] & [K^{35}] & [K^{36}] \\ [K^{41}] & [K^{42}] & [K^{43}] & [K^{44}] & [K^{45}] & [K^{46}] \\ [K^{51}] & [K^{52}] & [K^{53}] & [K^{54}] & [K^{55}] & [K^{56}] \\ [K^{61}] & [K^{62}] & [K^{63}] & [K^{64}] & [K^{65}] & [K^{66}] \end{bmatrix} \begin{bmatrix} \{f\} \\ \{F\} \\ \{h\} \\ \{H\} \\ \{\theta\} \\ \{\Theta\} \end{bmatrix} = \begin{bmatrix} \{r^1\} \\ \{r^2\} \\ \{r^3\} \\ \{r^4\} \\ \{r^5\} \\ \{r^6\} \end{bmatrix} \tag{4.25}$$

Here each $[K^{\mathrm{mn}}]$ and $[r^m]$ $(m, n = 1, 2, 3, 4, 5, 6)$ are the submatrices of order (2×2) and (2×1), respectively. These matrices are defined as:-

$$K_{ij}^{11} = \int_{\eta_e}^{\eta_{e+1}} \phi_i \frac{d\phi_j}{d\eta} d\eta,$$

$$K_{ij}^{13} = -\int_{\eta_e}^{\eta_{e+1}} \phi_i \phi_j d\eta$$

$$K_{ij}^{12} = 0, \qquad K_{ij}^{14} = 0 \qquad , K_{ij}^{15} = 0, \qquad K_{ij}^{16} = 0;$$

$$K_{ij}^{22} = \int_{\eta_e}^{\eta_{e+1}} \phi_i \frac{d\phi_j}{d\eta} d\eta,$$

$$K_{ij}^{24} = -\int_{\eta_e}^{\eta_{e+1}} \phi_i \phi_j d\eta,$$

$$K_{ij}^{21} = 0, \qquad K_{ij}^{23} = 0 \qquad , K_{ij}^{25} = 0, \qquad K_{ij}^{26} = 0;$$

$$K_{ij}^{33} = -\int_{\eta_e}^{\eta_{e+1}} \frac{d\phi_i}{d\eta}\frac{d\phi_j}{d\eta}d\eta - lB_v\int_{\eta_e}^{\eta_{e+1}} \phi_i\phi_j d\eta - \lambda\bar{h}'\int_{\eta_e}^{\eta_{e+1}} \frac{d\phi_i}{d\eta}\frac{d\phi_j}{d\eta}d\eta$$
$$+\beta\bar{f}\int_{\eta_e}^{\eta_{e+1}} \frac{d\phi_i}{d\eta}\phi_j d\eta - \alpha\bar{h}\int_{\eta_e}^{\eta_{e+1}} \phi_i\phi_j d\eta$$

$$K_{ij}^{34} = lB_v\int_{\eta_e}^{\eta_{e+1}} \phi_i\phi_j d\eta$$

$$K_{ij}^{31} = 0, \qquad K_{ij}^{32} = 0 \qquad , K_{ij}^{35} = 0, \qquad K_{ij}^{36} = 0;$$

$$K_{ij}^{43} = B_v\int_{\eta_e}^{\eta_{e+1}} \phi_i\phi_j d\eta$$

$$K_{ij}^{44} = -B_v\int_{\eta_e}^{\eta_{e+1}} \phi_i\phi_j d\eta - \alpha\bar{H}\int_{\eta_e}^{\eta_{e+1}} \phi_i\phi_j d\eta + \beta\bar{F}\int_{\eta_e}^{\eta_{e+1}} \frac{d\phi_i}{d\eta}\phi_j d\eta,$$

$$K_{ij}^{41} = 0, \qquad K_{ij}^{42} = 0 \qquad , K_{ij}^{45} = 0, \qquad K_{ij}^{46} = 0;$$

$$K_{ij}^{55} = -\frac{1+R}{Pr}\int_{\eta_e}^{\eta_{e+1}} \frac{d\phi_i}{d\eta}\frac{d\phi_j}{d\eta}d\eta - l\gamma B_t\int_{\eta_e}^{\eta_{e+1}} \phi_i\phi_j d\eta + \beta\bar{f}\int_{\eta_e}^{\eta_{e+1}} \frac{d\phi_i}{d\eta}\phi_j d\eta - (m+1)\bar{h}\int_{\eta_e}^{\eta_{e+1}} \phi_i\phi_j d\eta,$$

$$K_{ij}^{56} = l\gamma B_t\int_{\eta_e}^{\eta_{e+1}} \phi_i\phi_j d\eta,$$

$$K_{ij}^{51} = 0, \qquad K_{ij}^{52} = 0 \qquad , K_{ij}^{53} = 0, \qquad K_{ij}^{54} = 0;$$

$$K_{ij}^{65} = B_t\int_{\eta_e}^{\eta_{e+1}} \phi_i\phi_j d\eta,$$

$$K_{ij}^{66} = -B_t\int_{\eta_e}^{\eta_{e+1}} \phi_i\phi_j d\eta + \beta\bar{F}\int_{\eta_e}^{\eta_{e+1}} \frac{d\phi_i}{d\eta}\phi_j d\eta - (m+1)\bar{H}\int_{\eta_e}^{\eta_{e+1}} \phi_i\phi_j d\eta,$$

$$K_{ij}^{61} = 0, \qquad K_{ij}^{62} = 0 \qquad , K_{ij}^{63} = 0, \qquad K_{ij}^{64} = 0;$$

$$r_i^1 = 0, \qquad r_i^2 = 0, \qquad r_i^3 = -(\phi_i \frac{dh}{d\eta})_{\eta_e}^{\eta_{e+1}} - \lambda(\frac{(h_2 - h_1)}{he})(\phi_i \frac{dh}{d\eta})_{\eta_e}^{\eta_{e+1}},$$

$$r_i^4 = 0, \qquad r_i^5 = -\frac{1+R}{Pr}(\phi_i \frac{d\theta}{d\eta})_{\eta_e}^{\eta_{e+1}}, \qquad r_i^6 = 0,$$

where, $\bar{f} = \sum_{j=1}^{2} f_j \phi_j, \quad \bar{F} = \sum_{j=1}^{2} F_j \phi_j, \quad \bar{h} = \sum_{j=1}^{2} h_j \phi_j, \quad \bar{H} = \sum_{j=1}^{2} H_j \phi_j.$

The average Nusselt number i.e u_L, is based on Marangoni number.

$$Nu_L = -\frac{3m+6}{4m+5} Ma^{\frac{1}{3}} Pr^{\frac{-1}{3}} \theta'(0),$$

where $Ma = \frac{\varrho c_f A L^{2+m}}{\nu k_f}$ is Marangoni number.

The complete domain is divided into a set of 80 lines elements. The system of equation obtained after assembly of equation of element are non-linear, therefore an iterative scheme is used to solve it.

4.4 Results and discussions

To study the behaviour of the two phase flow phenomenan, curves are drawn for different values of important parameters named as R- Radiation parameter, l- dust parameter, Ma- Marangoni number and λ- williiamson parameter where, $m = 0, 1$ represents the linear and quadratic variation of surface temperature and expressed by solid lines dotted lines respectively. Other parameters namely Prandtl number, thermal dust parameter, thermal mass parameter, specific heat parameter are taken to be constant at 6.2, 1.2, 1.2 and 0.1 respectively[14, 23].

Figure (4.2) to Figure (4.5) illustrates the behaviour of velocities and temperatures with respect to l i.e dust parameter. Here, both the temperatures and velocities decrease as the concentration of dust particle increases. It is because of relative drag force between the nanofluid and the dust phases. With the increse of dust particles in fluid, the fluid become more viscous and it affect the heat transfer that is why velocity decreses. Thus, it conclude that the use of dust particle suspensions in working liquids is a smart technique to control the flow and heat transfer.

Figure (4.6) and Figure (4.7) reveals the variation of velocities with λ i.e Williamson parameter. Both the velocities of dust particle and nanofluid decreases when parameter λ increased. This reveals the fact that with the increase of Williamson parameter viscosity decreases. As the viscosity decreases, increase in velocities observed. Figure (4.8) and Figure (4.9) shows the variation of temperatures with λ i.e Williamson parameter. Both the temperatures of dust particle and nanofluid increase when parameter λ increased.

Figure (4.10) and Figure (4.11) are depicting the variation of temperature of nanofluid and dust particle with radiation parameter. It is anticipated that $\theta(\eta)$ and $\Theta(\eta)$ increased when the thermal radiation strength is boosted, because the radiation phenomena act as sources of energy to the fluid system. It provide more heat reults to increase in temperatures.

Figure (4.12) depicts the change in average Nusselt number with the impact of Marangoni number and dust parameter. Here the heat transfer rate increases with Ma and decreases with l. Thus Marangoni convection is used to enhances the heat transfer phenomenon.

From the curves of velocities of nanofluid and dust particle and for temperature of nanofluid, linear variation of surface temperature dominates the quadratic variation of surface temperature is also observed. But for temperature of dustparticle the quadratic variation dominates.

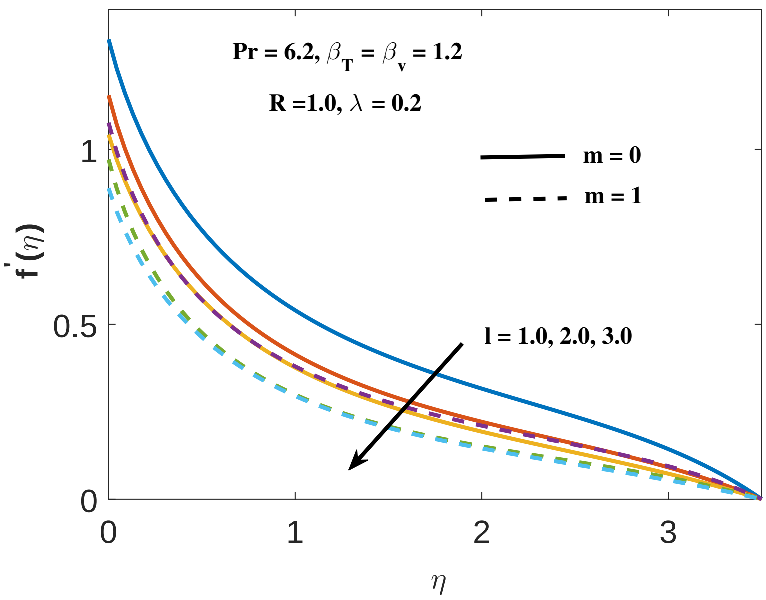

Figure 4.2: Velocity of Williamson fluid for different value of dust parameter

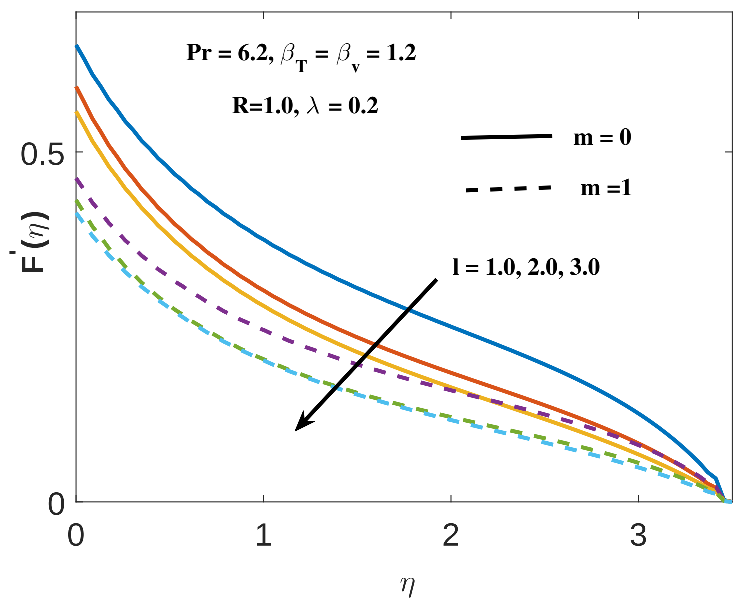

Figure 4.3: Velocity of dust particles for different value of dust parameter

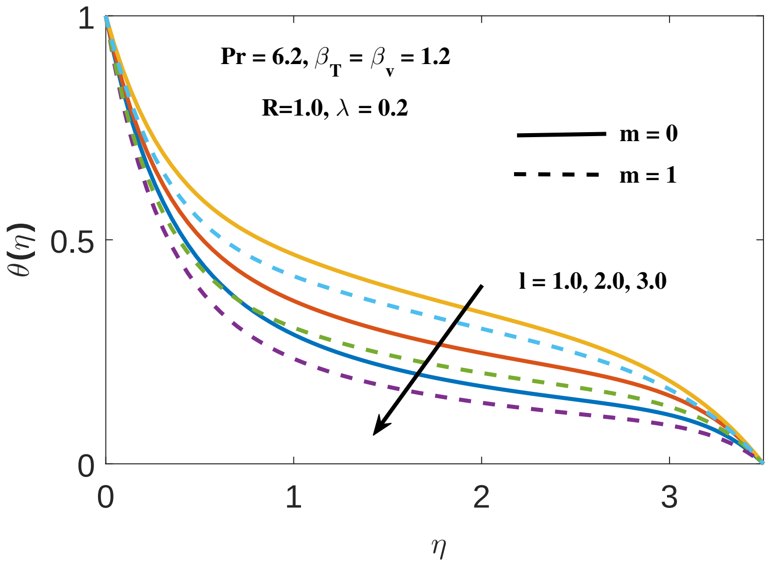

Figure 4.4: Temperature of Williamson fluid for different value of dust parameter

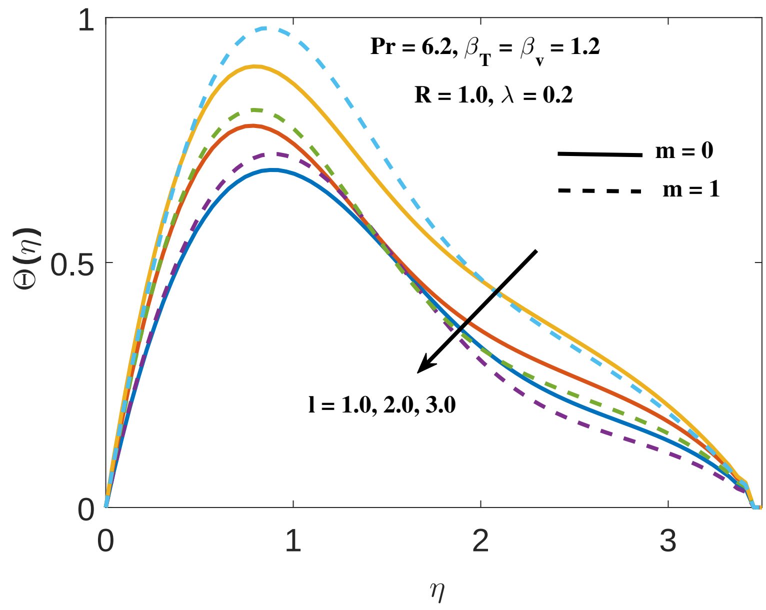

Figure 4.5: Temperature of dust particles for different values of dust parameter

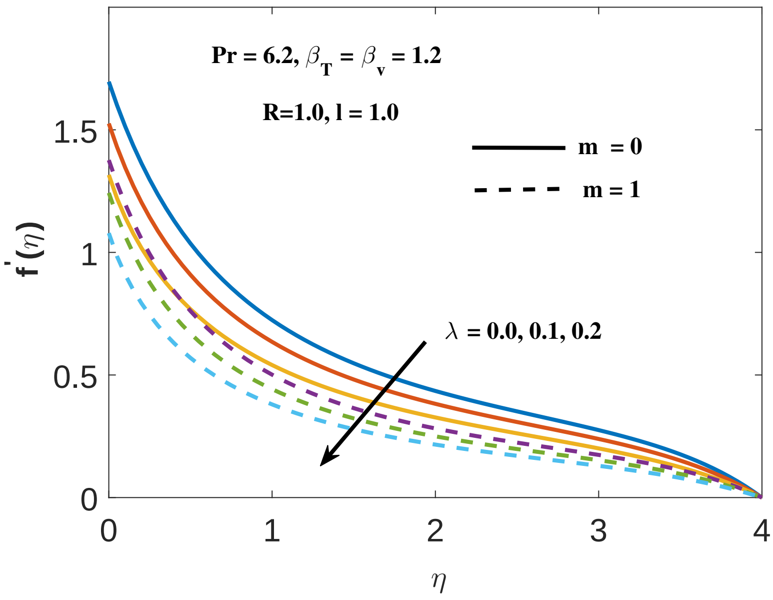

Figure 4.6: Velocity of Williamson fluid for different λ

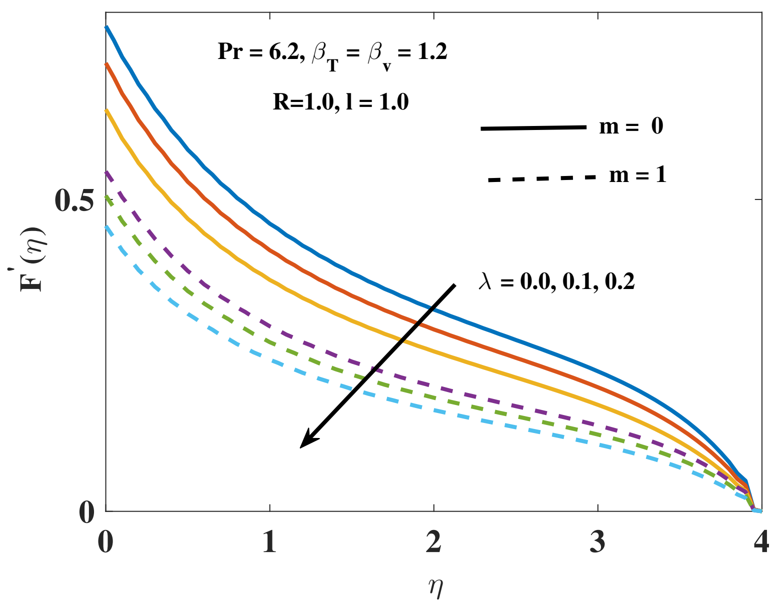

Figure 4.7: Velocity of dust particles for different λ

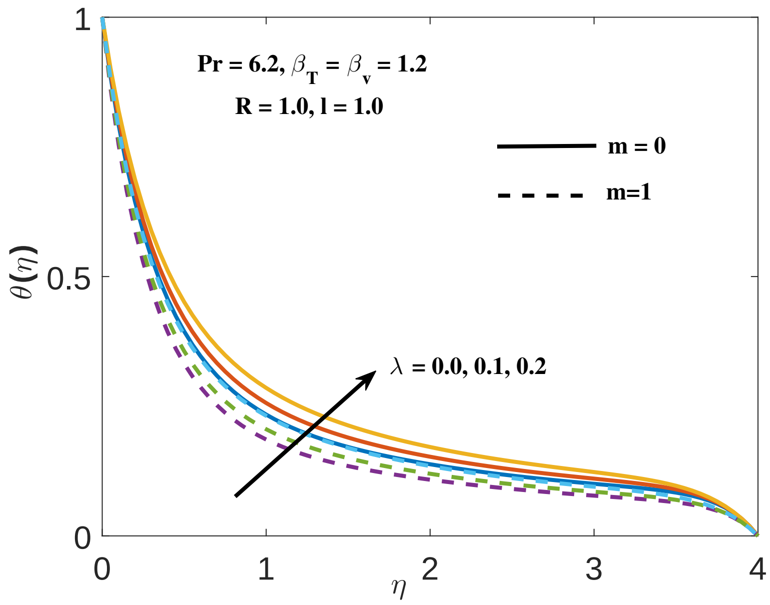

Figure 4.8: Temperature of Williamson fluid for different λ

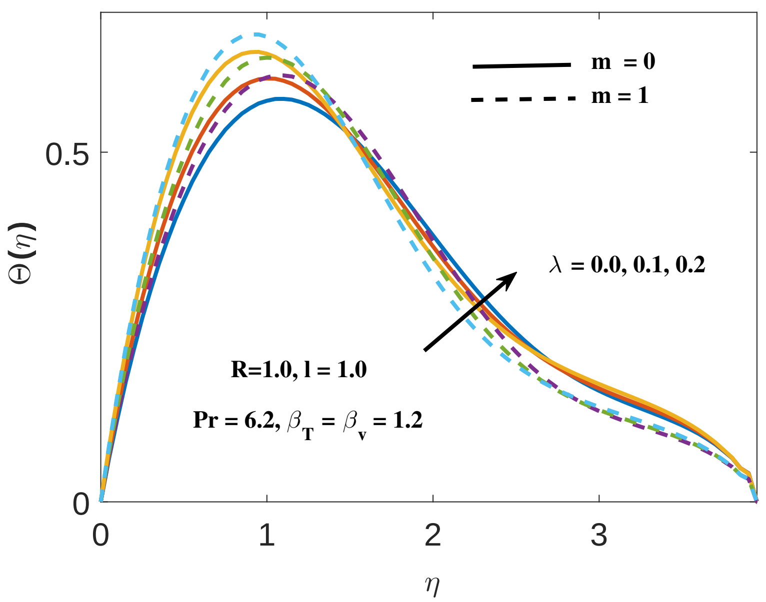

Figure 4.9: Temperature of dust particles for different λ

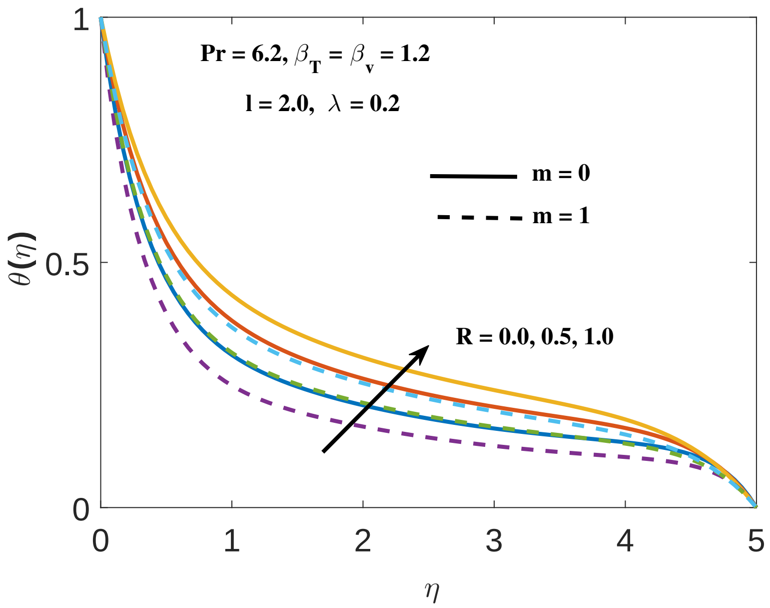

Figure 4.10: Temperature of Williamson fluid for different R

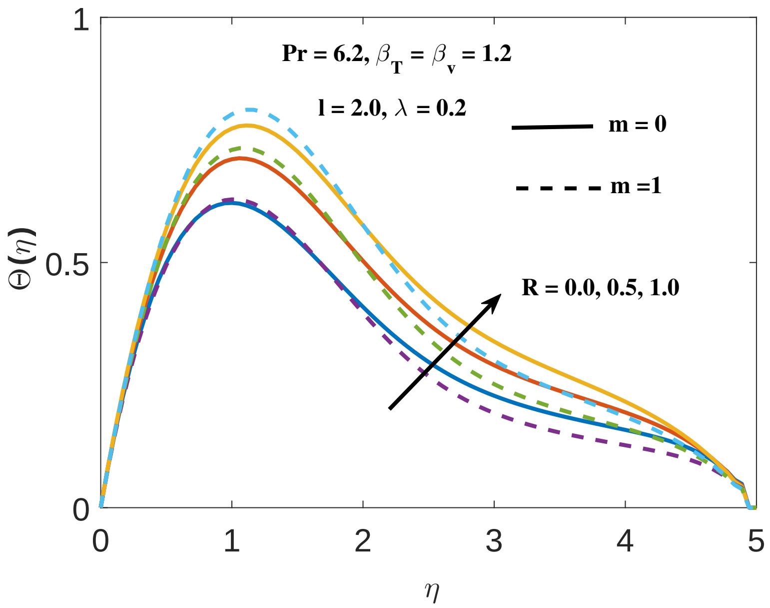

Figure 4.11: Temperature of dust particles for different R

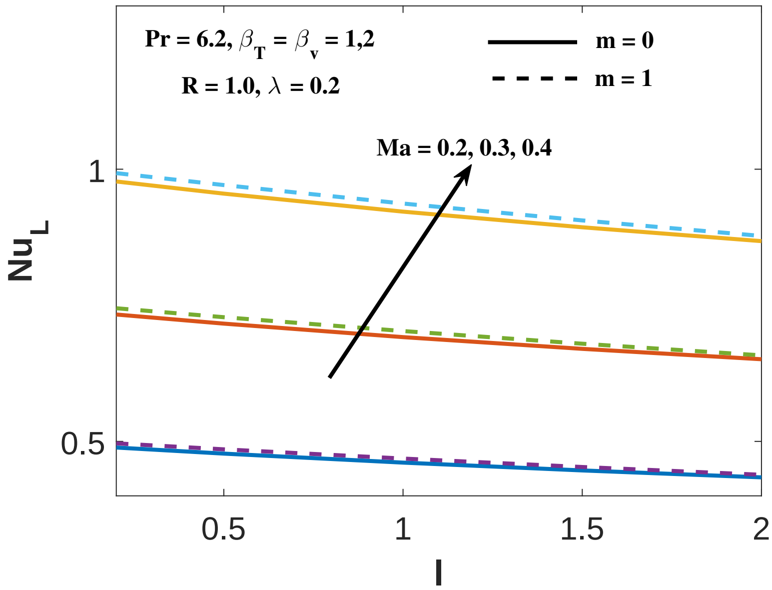

Figure 4.12: Impact of Ma and l on average Nusselt number

4.5 Conclusion

- The rate of heat transfer can be intesify by suspending dust particles in the fluid.
- The velocities and temperatures decreased via dust particle concentration parameter. Thus, it can be used as a main aspect to control the two-phase flow fields.
- Heat transfer rate is improved for R.
- Linear variation of surface temperature dominates over non-linear variation for Williamson nanofluid.
- The Marangoni effect constructive for heat transfer.

Bibliography

[1] J.L. Bansal, *"Viscous Fluid Dynamics."* Oxford and IBH publishing co. pvt. ltd. Fifth edition, 1995.

[2] Dr. Suman Chakraborty, *"Computational Fluid Dynamics."* Department of Mechanical Engineering Indian Institute of Technology, Kharagpur, 2012.

[3] Chorin, A. J, *"The numerical solution of the Navier-Stokes equations for an incompressible fluid"*, 1967.

[4] Fefferman, Charles L., *"Existence and smoothness of the Navier–Stokes equation".*
Clay Mathematics Institute, Retrieved 2017-04-02.

[5] Chorin, A. J. *"Numerical Solution of the Navier-Stokes Equations".*
Math. Comp. 22: 745–762, 1968.

[6] S. Choi, *Enhancing thermal conductivity of fluids with nanoparticles*
Department of Aerospace Engineering
Indian Institute of Space Science and Technology, Thiruvananthapuram, March 2013.

[7] A.Salih *"Streamfunction-VorticityFormulation".*
Math. Comp. 22: 745–762, 1968.

[8] Wärme- und Stoffübertragung, *"Heat and mass transfer",* Springer(Germany), 1995.

[9] R. V. Williamson, *"The flow of pseudoplastic materials",* Int. J. Industrial and Engineering Chemistry Vol 21, pp. 1108-1111, 1929.

[10] Pearson, J. R. A., *"On Convection Cells Induced by Surface Tension",* J. Fluid Mech., 4(5), pp. 489–500, 1958.

[11] Scriven, L. E., and Sternling, C. V., *"The Marangoni Effects",* Int. J. Nature, 187, pp. 186–188.

[12] P.G. Saffman, *"The stability of laminar flow of a dusty gas",* J Fluid Mech, Vol 13, pp. 120-128, 1962.

[13] K. M. Chakrabarti, *"Note on boundary layer in a dusty gas",* AIAA Journal, vol. 12, no. 8, pp. 1136–1137, 1974.

[14] S. Nadeem and S. T. Hussain, *"Analysis of MHD Williamson Nano Fluid Flow over a Heated Surface",* Journal of Applied Fluid Mechanics, Vol. 9, No. 2, pp. 729-739, 2016.

[15] Najeeb Alam Khan, Sidra Khan and Fatima Riaz, *"Boundary Layer Flow of Williamson Fluid with Chemically Reactive Species using Scaling Transformation and Homotopy Analysis Method",* Math. Sci. Lett, Vol 3, No. 3, 199-205, 2014.

[16] R.E. Singleton, *"Fluid mechanics of gas-solid particle flow in boundary layers"*
[Ph.D. Thesis]
California Institute of Technology (1964)

[17] M. Y. Malik, S. Bilal, T. Salahuddin and Khalil Ur Rehman, *"Three-Dimensional Williamson Fluid Flow over a Linear Stretching Surface",* Mathematical Sciences Letters, An International Journal, Vol 6, No. 1, 53-61, 2017.

[18] Sapna Sharma, *"Finite element modelling to heat and mass transfer in micropolar fluid flow"*
[Ph.D. Thesis]
Indian Institute of Technology (2006)

[19] B.Mahanthesha, B.J.Gireeshab, *"Thermal Marangoni convection in two-phase flow of dusty Casson fluid",* Results in Physics, Vol 8, Pages 537-544, March 2018.

[20] Siddabasappa, Y.Venkateshappa, B.Rudraswamy, B.J.Gireesha and K.R.Gopinath, *"Viscous Dusty Fluid Flow with Constant Velocity Magnitude",* Electronic J. Theo. Phy., 5 237-252, 2008.

[21] S. Kumar, N. Natrajan and A. K Pani, Numer, *"Methods for Partial Diff. Eqs."* ,25, 1402-1424, 2006.

[22] Sadia Siddiqaa, Naheed Begumb, M.A. Hossainc, Rama Subba Reddy Gorlad, Abdullah A.A.A.Al-Rashede, *"Two-phase natural convection dusty nanofluid flow",* International Journal of Heat and Mass Transfer, Vol 118, Pages 66-74, March 2018.

[23] Basavarajappa Mahanthesh, Bijjanal Jayanna Gireesha, Ballajja Chandra Prasanna Kumara, Nagavangala Shankarappa Shashikumar, *"Marangoni convection radiative flow of dusty nanoliquid with exponential space dependent heat"*, Nuclear Engineering and Technology, Vol 49, Issue 8, Pages 1660-1668, December 2017.