"Numerical Analysis in a Thermoacoustic Cryocooler"

A THESIS SUBMITTED IN PARTIAL FULFILLMENT
OF THE REQUIREMENTS FOR THE DEGREE OF

Bachelor of Technology
In
Mechanical Engineering

By
Bibartan Roy
10603068

Under the Guidance of
Dr. R. K. Sahoo

Department of Mechanical Engineering
National Institute of Technology
Rourkela
2010

Acknowledgment

We deem it a privilege to have been the student of Mechanical Engineering stream in National Institute of Technology, ROURKELA.

Our heartfelt thanks to Dr.R.K.Sahoo, our project guide who helped us to bring out this project in good manner with his precious suggestion and rich experience.

We take this opportunity to express our sincere thanks to our project guide for cooperation in accomplishing this project a satisfactory conclusion.

Submitted by:

Bibartan Roy

Roll. 10603068

Department of Mechanical Engineering

National Institute of technology

National Institute of Technology
Rourkela

CERTIFICATE

This is to certify that the thesis entitled **“Numerical Analysis in a Thermoacoustic Cryocooler’’** submitted by **Bibartan Roy**, **Roll No: 10603068** in the partial fulfillment of the requirement for the degree of **Bachelor of Technology** in **Mechanical Engineering**, National Institute of Technology, Rourkela, is being carried out under my supervision.

To the best of my knowledge, the matter embodied in the thesis has not been submitted to any other university/institute for the award of any degree or diploma.

Date:

Professor R.K.Sahoo
Department of Mechanical Engineering
National Institute of Technology
Rourkela-769008

CONTENTS

Contents-- i

Abstract-- ii

	Page No.
1. Introduction	**1-3**
2. Computational Fluid Dynamics	**4-16**
3. Gambit and Fluent	**17-23**
4. CFD Analysis	**24-33**
5. Results and Discussions	**34-40**
6. Conclusion	**41-42**
7. References	**43-44**

ABSTRACT

This project deals with the computational fluid dynamics analysis of flow in a Thermoacoustic Cryocooler showing the comparison between a straight tube and a curved tube resonator. This involves with the two dimensional analysis of flow through of Cryocooler having radial inlet and axial outlet. The software used for this purpose are GAMBIT and FLUENT. The 2 D model of the parts of the Cryocooler are made by GAMBIT and analysis are to be carried out by FLUENT. The models are first generated using the data and then are meshed and then various velocity and pressure contours are to be drawn and graphed in this paper to analyze the flow through the cryocooler. Various graphs indicating the variation of velocity, pressure and temperature along the stream length of the turbine are given.

Keywords: Pressure Inlet, Stack, Resonator, Gambit, Fluent.

Chapter 1

INTRODUCTION

INTRODUCTION

Oxygen, Nitrogen, Helium, Argon etc are industrially important gases. Nature has provided us an abundant supply of these gases in atmosphere and under the earth crust. Oxygen and Nitrogen are available from atmosphere and helium, argon etc are available from the earth crust. The main aim is to harness these gases and use it for important purpose. The production and utilization of these gases form the major part of economy. This can be said as a indicator of technological improvement. These gases have various uses. Oxygen is used for steel manufacturing, rocket propulsion and medical applications. Argon is used in TIG welding and high temperature furnaces. Helium finds its use in superconductivity, nuclear reactors etc. Hydrogen is used as fuel in rocket propulsion systems. Nitrogen is a major input to the fertilizer industry. It is also used in cryosurgery and semiconductor industry. Nitrogen is used as a blanket gas in most chemical processes, and serves as the basic raw material in production of ammonia based fertilizers and chemicals. High purity nitrogen is used as a carrier gas in the electronic industry; and liquid nitrogen provides the most effective cooling medium for many low temperature processes – from shrink fitting to cryosurgery. These important gases are first trapped from the various sources and a low temperature process known as air separation is used to separate them from each other. This air separation is carried out using expanding turbines. Air separation using turbo expanders has several benefits over high pressure (Linde) process. These benefits include low capital cost, better product mix and high operational flexibility. While room temperature processes, based on adsorption and membrane separation, are finding increasing application, particularly for low purity products, cryogenic distillation still remains the predominant method of producing bulk industrial gases. The cryogenic distillation process, operating at temperatures below 100K, offers several advantages over its room temperature counterparts. This process is economical in large scale, delivers both gaseous and liquid products, produces argon and rare gases (such as neon, krypton and xenon), and can respond to variation in demand in product mix. In petrochemical industries, turbo expanders are used for separation of propane and heavier hydrocarbons from natural gas stream. Turbo expanders generate low temperature necessary for recovery of ethane and do it less expensively.

Chapter 2

CFD

COMPUTATIONAL FLUID DYNAMICS

COMPUTER SIMULATION:-

Mathematical representation of the physical phenomena undergone by various components of a system is called simulation. Simulation usually consists of a set of Differential Equations. Computer Simulation involves algorithms of solutions fed into a computer which does the arithmetic operations at a tremendously high speed .The scale of events being simulated by computer simulations has far exceeded anything possible (or perhaps even imaginable) using the traditional paper-and-pencil mathematical modeling.

Advantages

Speed of computing is very high. Hence it is possible to see the simulations of various parameters simultaneously. Cheaper than to setup big and bulky experimental set-ups.Versatile: various problems with different levels of complexity can be solved

Pitfalls

Although sometimes ignored in computer simulations, it is very important to perform sensitivity analysis to ensure that the accuracy of the results are properly understood. If, for instance, one of the key parameters in a simulation is known to only one significant figure, then the result of the simulation might not be more precise than one significant figure, although it might (misleadingly) be presented as having four significant figures.

CFD(Computational fluid dynamics)

CFD is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. The use of CFD can also help in enhancing the quality of the products.The most fundamental consideration in CFD is how one treats a continuous fluid in a discretized fashion on a computer. One method is to discretize the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion (Euler equations for non-viscous and Navier-Stokes equations for viscous flow).

Governing Equations:-

Energy conservation equations are the basic foundation stones for any CFD problem to be solved in a numerical way. Universal laws of conservation of mass, momentum and energy do provide the gateways for any rigorous analysis of a continuous fluid. There are two approaches that are generally followed: 1. Eulerian 2. Lagrangian

Eulerian: Here arbitrary Control Volume (CV) in a stationary frame is used.

Lagrangian: Here the CV is selected in such a way that velocity of the CV always equals the local fluid velocity.

1. **Conservation of mass:-**

Terms/quantities used generally are mass fractions rather than molar concentrations, because there is always variations in physical conditions like temperature, pressure and concentration.

From Navier-Stokes equation; mass conservation equations for species ‘k’ can be written as

$$(\partial / \partial t)(\rho m_k) + \nabla.(\rho U m_k) = -\nabla.(j_k) + S_k$$

where: t =time

ρ =fluid density

m_k =mass fraction of species ‘k’

U = fluid velocity

1st term on the left indicates the acceleration of species 'k' in the C.V.

2nd tem indicates the change in species mass fraction due to convection.

1st term on the right gives the change in species mass fraction due to Diffusive fluxes ' j_k '

2nd term on the right (S_k) gives the source of species 'k' .

In general, the diffusive mass flux comprises diffusion due to concentration gradient + diffusion due to thermal effects + diffusion due to pressure & external forces.

$$J_k = -\rho D_{km}.\nabla(m_k) - D_{kT}.(\nabla T / T)$$

where:

D_{km} = Diffusion coefficient for species 'k'

D_{kT} = Thermal mass diffusion coefficient for species 'k'

Here contributions made by thermal mass diffusion are very small. Hence it can be neglected.

So now the mass conservation equation can be written as:

$$(\partial / \partial t)(\rho m_k) + \nabla.(\rho U m_k) = \nabla(\rho.D_{km}.\nabla m_k) + S_k$$

Over all the species;

$$(\partial / \partial t)(\rho) + \nabla.(\rho U) = \sum_k S_k$$

2. Conservation of Momentum:-

The Differential Equations governing the conservation of momentum in a given direction can be formulated similar to the mass conservation equations. But here the complications are more due to both shear and normal stresses which need to be put emphasis on. Application of the basic set of law of conservation of momentum gives us the set of equations which govern the motion of fluids (used to calculate velocity & pressure fields) The governing Equation:-

$$(\partial / \partial t)(\rho U) + \nabla.(\rho UU) = -\nabla.\pi + \rho g + F$$

Where:

$\pi \rightarrow$ Molecular Flux of momentum

g $\rightarrow$ Gravitational acceleration

F $\rightarrow$ External body forces

LHS of Equation:

1st term: rate of increase of momentum per unit volume

2nd term: rate of increase of momentum per unit volume caused by convection.

RHS of Equation:

1st term: Molecular contributions (including the pressure & viscous forces per unit volume.)

2nd term: Gravitational Force per unit volume.

3rd term: External force.

1st term representing the molecular contributions:

$$-\nabla(\pi) = -\nabla(p) + \nabla(z)$$

(z= viscous stress tensor.)

The equations which relate the stress tensor with the motion of the continuous fluid are called 'Constitutive Equations' of state and these equations vary from one fluid material to another.

3. **Conservation Of Energy:-**

Law of conservation of energy can be used to derive the Transport equations for total energy. The difference of the Total Energy & the Mechanical Energy gives us the Internal Energy (I.E). So, to deduce the Governing Equation for I.E., we need to derive the transport equation for the Mechanical Energy.

$$(\partial/\partial t)(\rho h) + \nabla.(\rho U h) = -\nabla.(q) + (Dp/Dt) - (\tau : \nabla U) - \nabla.(\sum_k h_k j_k) + S_h$$

Here 'h' is the enthalpy, which is defined as

$$h = \sum_k m_k h_k \qquad \because h_k = \int_{T_{ref}}^{T} C_{pk}\, dT$$

$T_{ref} \rightarrow$ Reference temp.

$C_{pk} \rightarrow$ Specific Heat of species 'k'

$q \rightarrow$ Flux of enthalpy.

In the equation:

1st term: Accumulation due to convection.

2nd term: change of enthalpy due to convection.

3^{rd} term: change of enthalpy due to conduction.

4^{th} term: Reversible & irreversible change of enthalpy due to pressure

5^{th} term: Reversible & irreversible change of enthalpy due to viscosity.

6^{th} term: Changes in enthalpy due to diffusive mass fluxes

7^{th} term: Volumetric source of enthalpy.

TURBULENT FLOW PROCESS:-

Turbulent flow is difficult to visualize but can be described with the help of its characteristic features. Fluid motion is defined as turbulent if it is irregular, rotational, intermittent, diffusive, dissipative and highly disordered. It is inherently unsteady and three-dimensional. Practical visualization of turbulent flows reveal formation of rotational flow structures called 'eddies'. These eddy motions lead to effective contact between fluid particles which are initially separated by a long distance, resulting in effective exchange of heat, mass and momentum, which are significantly higher than in laminar flows. Turbulent flows are also associated with higher values of friction drag and pressure drop. For practical engineering purposes turbulent flows are absolutely necessary to make certain operations feasible.

Modeling approaches:-

Out of the various approaches like statistical, structural and deterministic approaches, the modeling approach is by far the most reliable one. The basic premise in modeling turbulence is that it can be understood within the continuum assumption of fluid dynamics. The beginning appeared to be dubious as one thought of solving only the momentum equations with appropriate B.C. to predict the desired flow characteristics. But it was soon cleared as, at large Reynolds numbers, the inherent nonlinearity in these equations manifest in terms of turbulence. The main obstacle in solving the basic G.E. under turbulent conditions is the inability to resolve the wide range of spatial and temporal scales simultaneously. It is seen that the distance between the large and small scales grows with increase in Reynolds number. The no. of grid points and smallness of time steps required to resolve all the time and space scales of turbulent motion push the limits of Computation beyond the limits of present computing abilities. The required mesh spacing is different for different estimates of various models.

Even though models can't replace experiments, they can be genuinely useful, as the latest modern computational models allow almost all the flows to be calculated to much higher accuracy than the best possible guess

MATHEMATICAL MODELING OF FLOW PROCESS

Analysis/Simplification of Governing Equation

The Governing Equations (G.E) are needed to be scanned for possibility of simplifications and tailoring them to suits the need of problem under consideration. The G.E. can be made more useful by writing them in the non-dimensional form by representing them in form of fundamental quantities (length, time, velocity, temperature etc.) From here on we get some of the important dimensionless numbers, which make it possible to perform critical appreciation of the possible interactions between various processes (convection, conduction, diffusion etc.)

One such dimensionless number is the 'Reynolds Number (Re)',which can be used for the effective simplification of the G.E.

'Re' is interpreted as the ratio of Convective transport to Molecular transport of momentum or as the ratio of the inertial to viscous forces:

$$Re= (L_r U_r)/(\mu_r / \rho_r)$$

Now considerations: for very high speed flows, the inverse of the Reynolds number tends to zero & hence we can ignore the viscous stress terms in the momentum conservation equations. At very low speeds, the Reynolds number is small. Hence the convective or inertial terms in the Navier-Stokes equations can be neglected. This leads to the creeping flow equations.

Boundary Conditions:-

Specification of appropriate Initial conditions and Boundary conditions is necessary to solve the closed set of governing model equations. Selection of the appropriate solution domain is the basic step in model formulation. The B.C. requirements are determined by the solution domain, the

Inlet:-

It is the interface through which the solution domain is communicated by the surrounding environment. In general the properties of the incoming fluid stream (i.e. temperature, velocity/pressure, composition etc.) are assumed to be known.

For laminar flow through a cylindrical inlet pipe , we can assume to have a parabolic velocity profile as a B.C. at the inlet. But for complex shapes additional models are necessary for the formulation of the stating of appropriate velocity inlet B.C.

When the velocity components at the inlet boundary are not known, pressure specification at the inlet is mandatory and by virtue of Bernoulli's equation we can have the velocity profile at the inlet boundary.

Surface of the solution domain through which the flow exits is defined as outlet. Normalized conditions imply gradients normal to the outlet boundary except pressure are all zero. Taking the direction normal to the outlet boundary as 'z'

So, it is not necessary to specify the pressure at the outlet boundary. Note: When the gradients of the outlet boundary conditions are not zero or when conditions downstream of the outlet boundary may have some influence over the flow within the solution domain, it's not appropriate to use the outlet B.C.

Walls:-

The walls are generally assumed to be impermeable. So, a 'No-Slip' B.C. is used. This is achieved by setting the transverse fluid velocity

NUMERICAL SOLUTION OF MODEL EQUATIONS:-

Mathematical models of flow processes are non-linear, coupled D.E. generally; numerical solution of the governing transport equation replaces continuous information carried by the exact solution of partial D.E. by discrete information derived at the grid points. At the grid points values of the dependent variables are considered as basic unknowns. We assume that the pattern in which the unknown dependent variables change between the grid points is known. Generally piecewise profiles are assumed, which describe variation over a small region around the grid point. The solution domain is further divided into no. of sub-domains and computational cells.

For a particular D.E, there can be ways and means to derive the discretized equations (finite volume, finite element and finite difference)

Here we first consider a steady state condition. So the outer integral over time and the 1st term vanishes. The 2nd term is the net convective flux, 3rd term is the net diffusive flux, 4th term represents the volumetric source or the sink.

For calculation of the diffusive n convective fluxes, we need to know about all the parameters present in the transport equation at all the points in the CV. But these values are known to us at only the computational nodes. So, we need to make approximations. Actually two levels of approximations need to be invoked. The 1st level includes the surface integral being approximated in terms of variable values at one or more locations on the cell face.

At the 2nd level, the variable values at these locations are approximated in terms of the values at the computational nodes.

Several attempts have been made to employ higher order interpolation schemes. Of them QUICK (Quadratic Upstream Interpolation for Convective Kinetics) is the most popular. Here the face value of ϕ is obtained by a quadratic function passing through two bracketing nodes (on each side of the face) and one on the upstream side. For the uniform grid, coefficients of the three nodal values become 3/8 for the downstream point, 6/8 for the first upstream node and -1/8 for the second upstream node. The Transportiveness property is built into the scheme by building two upstream and one downstream node. But the main coefficients of the discretized equations are not guaranteed to be positive. This may lead to instability and may lead to unbounded (wiggles) solutions under certain conditions

Solution of Algebraic Equations:- After the implementation of B.C's, we derive one algebraic equation per node, which relates the variable value at the node to that of the neighboring nodes. The equation for any CV has the form:

$$a_p \Phi_p = \sum_l a_{nb} \Phi_{nb} + S_{\Phi C}$$

Where: P= node at which the values are approximated.

nb=neighboring nodes.

l= index covering all the neighboring nodes.

S= source term estimated using the initial and the guess values.

The **under-relaxation** factor needs to be considered before we can put forth the solution of the algebraic equation. The overall iterative procedure is repetitive in nature and the procedure is susceptible to divergence. So, α_{Φ} is introduced, which takes a value between zero and one. Lower values of under relaxation lead to better stability in results but lead to slower convergence. Generally, small values are used during early iterations, in which gradually increased convergence is approached.

Finite Volume Method for calculation of the Flow Field:-

Generally the velocity field is calculated as a part of the overall solution procedure by solving the momentum conservation equations.

The momentum equations are vector equations, with convective and diffusive equations being more complicated than the previous generic transport equations. All the extra non-zero terms can be combined to form a single source term. It must also be noted that all the three momentum equations are strongly coupled because each velocity component occurs in all the three momentum equations. The unique feature of momentum equations, which distinguishes them from the generic transport equations, is the role of pressure. But there is no obvious way to calculate pressure. The pressure field is indirectly specified using the continuity equation. It is therefore mandatory to calculate the pressure field in such a way that the resulting velocity satisfies the continuity equation.

Implementation of Boundary Conditions:-

The BC's may be implemented numerically within the finite volume framework by expressing the flux at the boundary as a combination of interior values and boundary data. Usually BC's enter the discretized equations by suppression of the link to the boundary side and modification of the source term. Appropriate coefficient of discretized eqn. being set to zero, the boundary side flux is introduced through the linearized source terms. Since there are no nodes outside the solution domain, the approximations of the boundary side flux are based on one-sided difference or extrapolations.

Convective fluxes are usually specified at the inlet boundaries (and set to zero at the impermeable walls and symmetric axes). Upwind schemes are generally used. For staggered grid, velocity can be directly specified as velocity node is situated at the boundary surface. But for a co-located grid, specified inlet velocity is required is used to calculate the convective flux. For a staggered grid, knowledge of pressure is not required at the boundaries on which normal velocities are specified. At such boundaries, zero gradient BC's should be used for pressure correction and pressure equations. At outlet, extrapolation of the velocity to the boundary can be used. At impermeable walls, normal velocity is set to zero. The wall shear stress can then be included in source terms.

Chapter 3

GAMBIT & FLUENT

The CFD Process

In the last century, various modeling softwares have been developed like PHOENICS, FLUENT, SRAT-CD, CFX, FLOW -3D and COMPACT. These softwares are based on finite volume approach. FLUENT is one of the major softwares which has helped a lot in modeling fluid and heat transfer problems. Complex geometries can be meshed in 2D triangular or quadrilateral, 3D tetrahedral, hexahedral, pyramid, wedge and mixed meshes. Fluent thus helps in simulating the unstructured mesh geometries.

FLUENT consists of two parts, GAMBIT and FLUENT solver. Gambit is used for designing purposes and fluent solver is used for simulation. Gambit helps to mesh the geometries and coarse and fine meshing can be done as per the flow solutions. Once the meshing is done in Gambit, further modifications like setting up boundary condition and executing the solution is done in Fluent.

There are essentially three stages to every CFD simulation process: preprocessing, solving and post processing.

Preprocessing

This is the first step in building and analyzing a flow model, including building the model within a computer-aided design (CAD) package, creating and applying a suitable computational mesh, and entering the flow boundary conditions and fluid materials properties. The major preprocessing tools are Gambit, TGrid and G/Turbo.

CAD geometries are easily imported and adapted for CFD solutions in GAMBIT, Fluent's own preprocessor. 3D solid modeling options in GAMBIT allow for straightforward geometry construction as well as high quality geometry translation. Among a wide range of geometry tools, Boolean operators provide a simple way of getting from a CAD solid to a fluid domain. A state-of-the-art set of cleanup and conditioning tools prepares the model for meshing. GAMBIT's unique curvature and proximity based "size function" produces a correct and smooth CFD-type mesh throughout the model. Together with the boundary layer technology, a number of volumetric meshing schemes produce the right mesh for one application. Parametric variations are also inherent to the process.

Fluent's solvers also couple with other meshing tools such as ANSA, Harpoon, Sculptor and YAMS.

GAMBIT:

On starting the designer care should be taken that you are in the correct directory. Then select the Fluent 5/6 solver. Give a filename to save your project to the above name and then get started with the geometry. First you create a vertex from which you start an edge. Assembling the edges give the face which in turn gives the volume. Once we obtain the face or volume we can mesh the geometry and can export it to fluent for analysis.

In the upper right portion we have four heads which are geometry, mesh, zones and tool. Click on the geometry which gives the lead to vertex, edge, face, volume and group. Click on the vertex which leads to create vertex which leads to create real vertex button. Enter the coordinates in global form and then click on the Graphics/window control button to get the vertex.

Once all the vertexes are created we can create the edge by joining the vertex. Click on the edge button under geometry. There it asks for the vertex between which the edges have to be drawn. Selecting the edges click on the create edge button to get the edge. Under the geometry button, click on the face button . A floating window called Create Face from Wireframe will appear. Select the edges by holding the shift and clicking the edges from which the face has to be obtained. Click on the create face option and thus we obtain the face.

Once the face is created we need to mesh the file. Under the option command select mesh which give five results. From them select the face option. Click on the top left button in the FACE menu area, the button is called Mesh Faces, and then you will see the mesh in yellow

Then the next work is to set the boundary condition. Under the operation window, zones button is clicked. Under the word ZONES two buttons will appear: Boundary Types and Continuum are specified. Specify Boundary Types button is clicked. Make sure that at the top of this window the solver name 'Fluent 5/6' appears. Then the edge is selected and named, then the proper type of condition is selected in which we want to do the simulation.

After the work is over the file is saved and then exported for further analysis of the design.

Solving

The CFD solver does the flow calculations and produces the results. FLUENT is used in most industries. FloWizard is the first general-purpose rapid flow modeling tool for design and process engineers built by Fluent. POLYFLOW (and FIDAP) are also used in a wide range of fields, with emphasis on the materials processing industries.

The FLUENT CFD code has extensive interactivity, so one can make changes to the analysis at any time during the process. This saves one time and enables one to refine designs more efficiently. The graphical user interface (GUI) is intuitive, which helps to shorten the learning curve and make the modeling process faster. It is also easy to customize physics and interface functions to one specific need. In addition, FLUENT's adaptive and dynamic mesh capability is unique among CFD vendors and works with a wide range of physical models. This capability makes it possible and simple to model complex moving objects in relation to flow.

FLUENT provides the broadest range of rigorous physical models that have been validated against industrial scale applications, so one can accurately simulate real-world conditions, including:

- multiphase flows
- reacting flows
- rotating equipment
- moving and deforming objects
- turbulence
- radiation
- acoustics, and
- dynamic meshing

The FLUENT solver has repeatedly proven to be fast and reliable for a wide range of CFD applications. The speed to solution is faster because our suite of software enables one to stay within one interface from geometry building through the solution process, to postprocessing and final output. FLUENT's performance has been tried and proven on a variety of multi-platform clusters. The parallel computing capability is flexible and enables one to solve larger problems faster.

Postprocessing

This is the final step in CFD analysis, and it involves the organization and interpretation of the predicted flow data and the production of CFD images and animations. All of Fluent's software products include full postprocessing capabilities. Our postprocessing tools enable one to provide several levels of reporting, so one can satisfy the needs and interests of all the stakeholders in one's design process. Quantitative data analysis can be as sophisticated as one require. High-resolution images and animations help one to tell the story in a quick and impactful manner.

FLUENT

Fluent in general solve the governing equation of conservation of mass and momentum. It chooses two numerical methods, one is the segregated solver and the other is the couple solver. In the segregated solver the governing equations are solved sequentially while the couple solver solves the governing equation and the transport functions simultaneously.

While solving in fluent, first the mesh file is import and then the grid check is done. Then the solver is selected. Then the energy equation is checked. The material is set and the operating condition is fixed. The boundary condition is set for both inlet and outlet. The monitors are set and the flow field is initialized. Then iteration is done to obtain the converged results.

PROBLEM SOLVING STEPS

After determining the important features of the problem following procedural steps are followed for solving it

1. The geometry model was created and meshing was done.

2. The appropriate solver was started for 2D or 3D modeling.

3. The grid was checked.

4. The solver formulation was selected.

5. The basic equation to solve: laminar or turbulent was selected.

6. The material properties were specified.

7. The boundary properties were specified.

8. The solution control parameter was adjusted.

9. The flow field was initialized.

10. The solution was calculated.

11. The results were examined.

12. The results were saved.

13. If necessary, refine the grid or consider revisions to the numerical or physical model

Chapter 4

CFD ANALYSIS

CFD MODEL:

A two dimensional axisymmetric model was developed using GAMBIT.

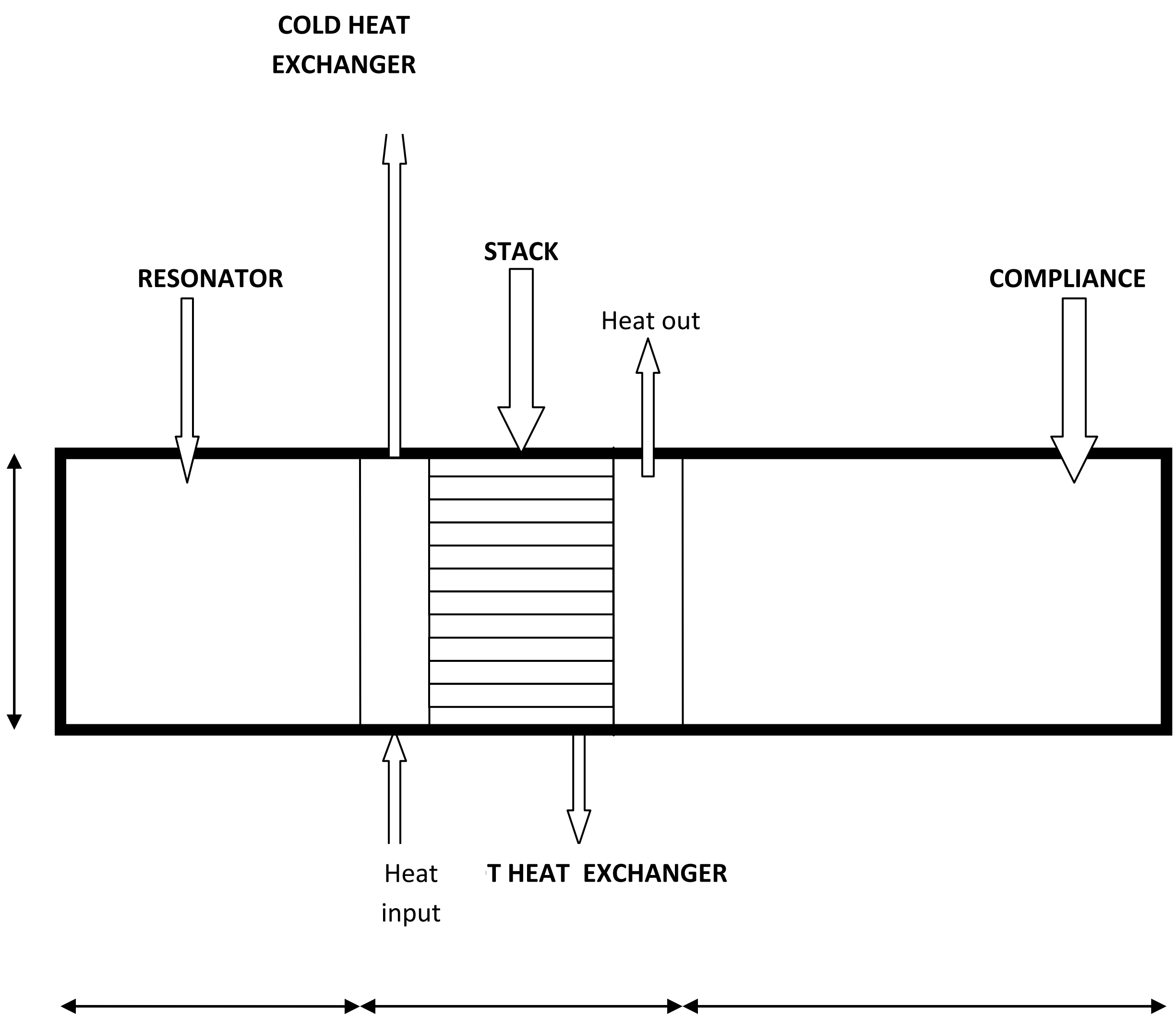


Fig. shows the 2D CFD model of a thermoacoustic cryocooler

The above model was designed in Gambit with the proper dimensions. The base of the model is the base line or the centre line of the tube. The length of the tube is of 170mm. The total tube consists of mainly 5 parts: Resonator, Cold Heat Exchanger (CHX), Stack, Hot Heat Exchanger (HHX) and the Compliance. Geometry goes as: the resonator tube is 30 mm long, then CHX elongates upto 40 mm i.e. 10mm length, stack is of 20 mm length, the HHX elongates upto 70 mm and the compliance elongates upto 170 mm. width of the tube is taken as 12 mm. The left inlet has an acoustic loudspeaker which acts as a generator of longitudinal sound waves. Air is used as the material to be used in the vortex tube.

PROCEDURE FOR MODELING

STEP 1:

Specify that the mesh to be created is for use with FLUENT 6.0:

Main Menu > Solver > FLUENT 5/6

Verify this has been done by looking in the *Transcript Window* where you should see. The boundary types that you will be able to select in the third step depends on the solver selected.

STEP 2:

Select vertex as tool geometry

TOOL> GEOMETRY> VOLUME

Vertexes at various points are located. The vertex were created at

A= (0, 0),

B= (0, 12),

C= (30, 12),

D= (40, 12),

E= (60, 12),

F= (70, 12),

G= (170, 12),

H= (170, 0),

J= (70, 0),

K= (60, 0),

L= (40, 0),

M= (30, 0),

TOOL> GEOMETRY >EDGE

Then the edges were created using the vertexes. The vertexes were selected using shift and left clicking on the vertexes. Then the points A,B,C,D,E,F,G,H,J,K,L and M are joined to get the required geometry.

TOOL> GEOMETRY >FACE

Then using the edges the face was formed by selecting all the edges together.

STEP 3: MESH

The element is of Tri and the type is pave.

Number of triangular cells= 1309

STEP 4 :(set boundary types)

ZONES > SPECIFY BOUNDARY TYPES.

The edges are selected to assign them particular boundary conditions. The edge AB is the inlet which is of the type presure inlet.

The edge CM is the CHX boundary which is of the type INTERIOR

The edge DL & EK are the stack boundaries of the type INTERIOR.

The edge FJ is the HHX boundary which is of the type INTERIOR.

Rest all are walls.

STEP 5: (Export the mesh and save the session)

FILE > EXPORT > MESH

File name was entered for the file to be exported. Accept was clicked for 2-D.

Gambit session was saved and exit was clicked.

FILE >EXIT

ANALYSIS IN FLUENT

PROCEDURE:

STEP 1: (GRID)

FILE > **READ** > **CASE**

The file channel mesh is selected by clicking on it under files and

Then ok is clicked.

The grid is checked.

GRID > CHECK

The grid was displayed.

DISPLAY > GRID

Grid was copied in ms-word file.

STEP 2 :(Models)

The solver was specified.

DEFINE > MODELS > SOLVER

Solver is segregated

Implicit formulation

Time unsteady

DEFINE > MODEL >ENERGY

Energy equation is clicked on.

DEFINE > MODELS > VISCOUS

The standard k-ε turbulence model was turned on.

K-ε model (2-equation) - Standard

No viscosity

STEP 3: (Materials)

By default the material selected was air with properties.

Viscosity, μ = 1.7894 x 10-5 kg/ms

Density, r= 1.225 kg/ m3.

Thermal Conductivity, K=0.0242 W/mK

Specific heat, Cp= 1.00643kJ/kg K

Molecular weight= 28.96

STEP 4: (Operating conditions)

Operating pressure= 101.325 KPa

No Gravity.

STEP 5: (Boundary conditions)

DEFINE > **BOUNDARY CONDITIONS**

Cold heat exchanger:

Walls: adiabatic; material: aluminium

Porous medium:

Porosity: 0.69 ; viscous resistance : 9.433e+09 1/m^2; inertial resistance: 76090 1/m; material aluminium; internal fluid : air

Hot heat exchanger:

Walls: isothermal maintaining temperature of 300K; material : aluminium

Porous medium:

Porosity: 0.69; viscous resistance: 9.433e+09 1/m^2; inertial resistance: 76090 1/m; material aluminium; internal fluid: air

Compliance:

Wall: adiabatic; material: aluminium

Internal fluid: air

Resonator:

Pressure inlet – pressure defined by UDF generating sinusoid pressure wave

Resonator walls: aluminium

Stack:

Walls: adiabatic; material: aluminium

Porous medium:

Porosity: 0.69; viscous resistance : 9.433e+09 1/m^2; inertial resistance: 76090 1/m; material aluminium; internal fluid : air

STEP 6: (Solution)

SOLVE > CONTROLS>SOLUTIONS

All flow, turbulent and energy equation used.

Under relaxation factors

Pressure= 0.2

Density= 1

Body Force= 1

Momentum= 0.4

Swirl Velocity= 0.9

Turbulence Kinetic energy= 0.8

Turbulence Dissipation Rate= 0.8

SOLVE >**INTIALIZE**

Click **INIT**

~Initial pressure was chosen to be 15 atm

SOLVE>MONITOR>SURFACE

SOLVE>ITERATE

Time step= 0.005 s

Number of Time step= 5000

Maximum number of iteration per time step=20

REPORT>SURFACE INTEGRALS

Temperature at cold heat exchanger, pressure at resonator was noted by taking the area weighted average.

The data was saved.

FILE>WRITE>CASE & DATA

STEP 7: (Displaying the preliminary solution)

Display of filled contours of velocity magnitude

DISPLAY>CONTOURS

~ Velocity was selected and then velocity magnitude in the Drop down list was selected.

~filled under option was selected.

~Display was clicked.

Display of filled contours of temperature

DISPLAY>CONTOURS

~Temperature was selected and then Static temperature in the drop down list was selected..

~filled under option was selected.

~Display was clicked.

Display of filled contours of pressure

DISPLAY>CONTOURS

~ Pressure was selected along various planes

~filled under option was selected.

~Display was clicked.

Display velocity vector.

DISPLAY>VECTOR

~Display was clicked to plot the velocity vectors.

PLOT>XY PLOT

~ Three graphs were plotted.

~ One was Static temperature v/s position, another was static pressure v/s position and the third one was velocity magnitude v/s position.

~ The graphs give the magnitude of each at inlet, hot exit and cold exit.

Chapter 5

Result and Discussion

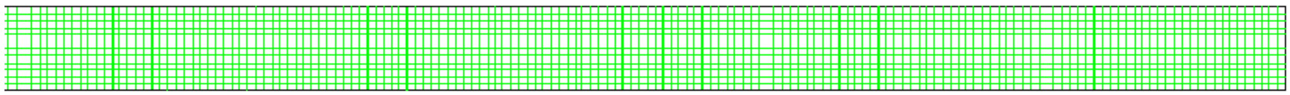


Fig. 4.1 shows the grid pattern of the 2D model.

The figure shows the pressure inlet, the walls and the compliance. Left hand side boundary is of the type pressure inlet. This is where the acoustic loudspeaker is used to generate longitudinal waves. The fluid flowing in the tube is air. Local temperature of the gas particles changes due to compression and rarefaction caused by the standing wave.

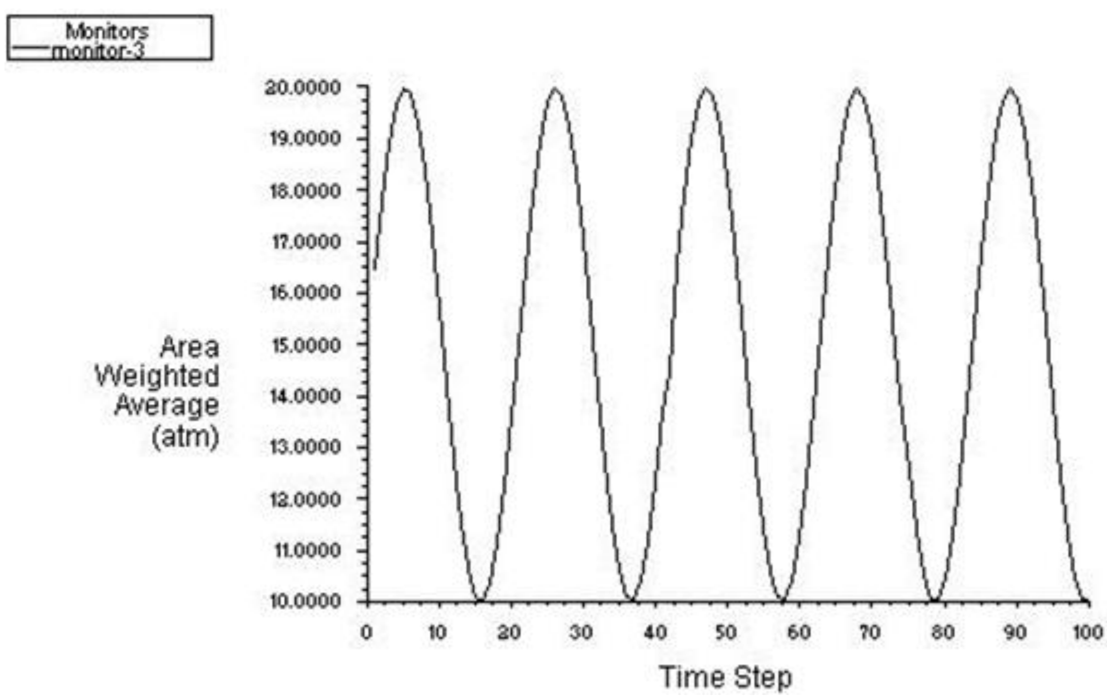
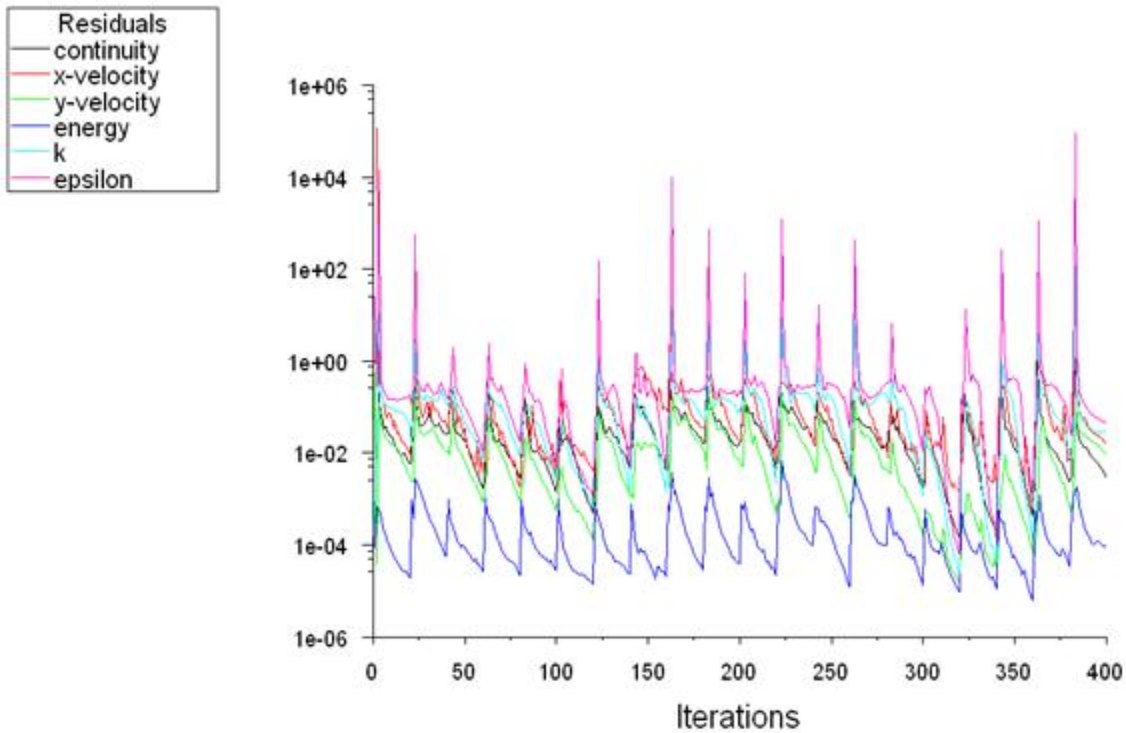


Fig 4.2 shows the sinusoidal pressure udf that is fed from the acoustic loud speaker.



The **Fig 4.3 shows the iteration curve. The curve shows convergence of various scaled residuals consisting of continuity, x velocity, y velocity, swirl, energy, K and epsilon.**

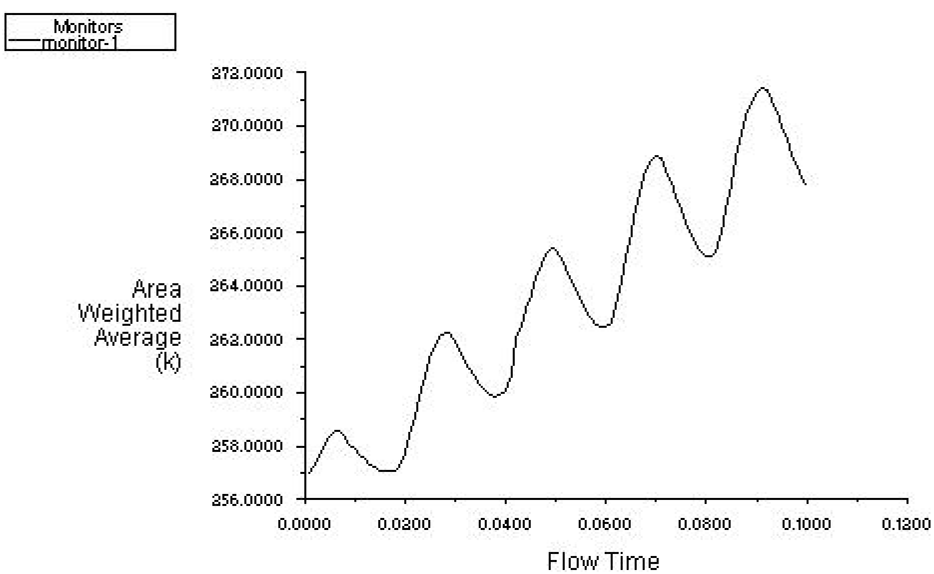


Fig 4.4 shows the variation of temperature along the CHX walls at the time 1.0000e-01

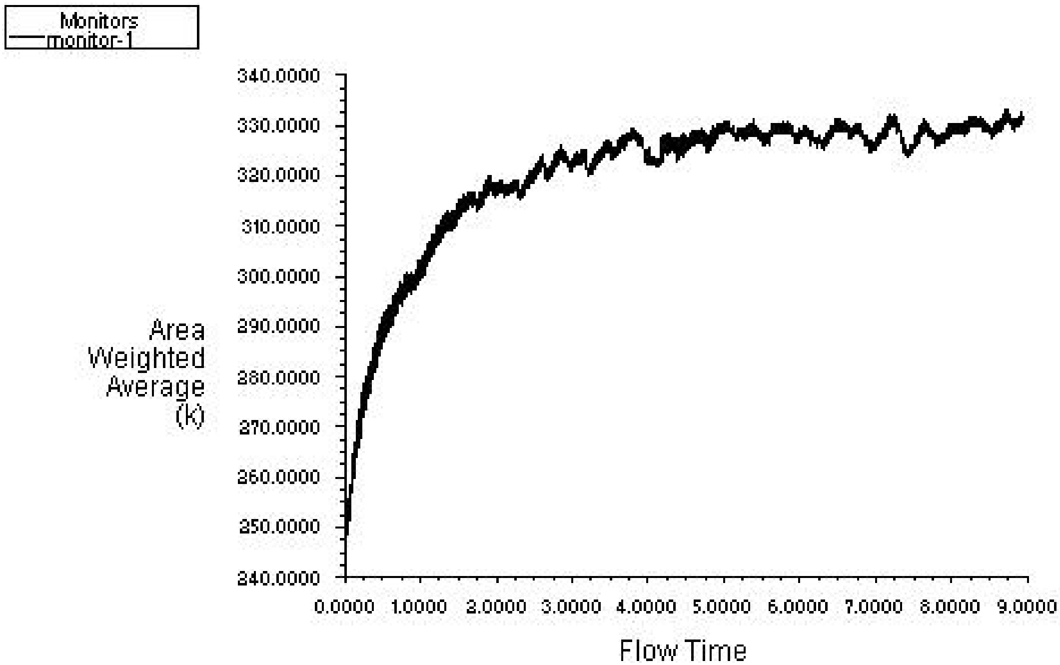


Fig 4.5 shows the temperature variation along the CHX walls at the time period of 3.6800e+00 sec.

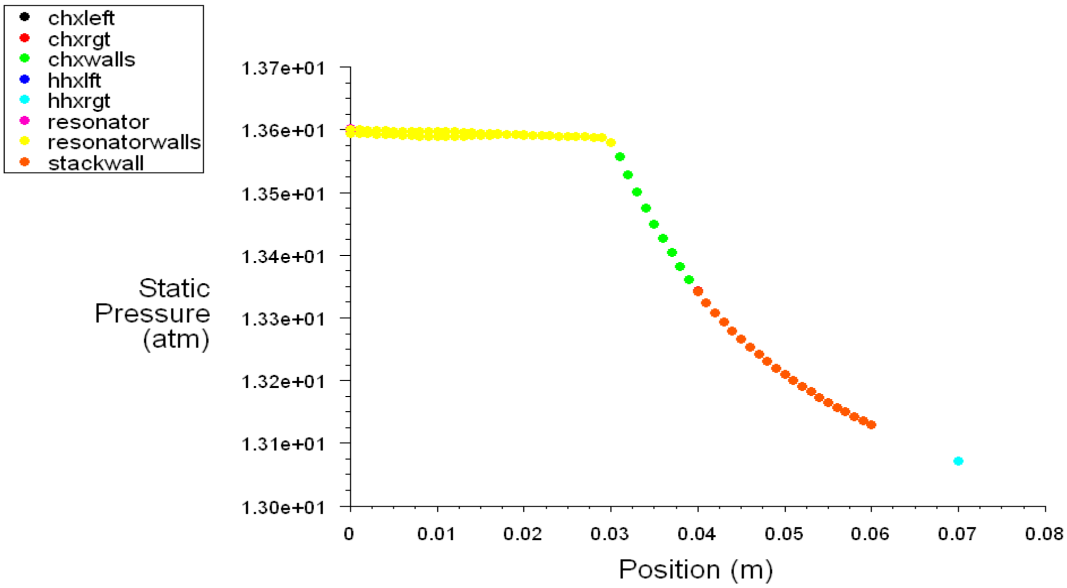


Fig 4.6 shows the variation of static pressure along the resonator walls, chx walls, stack walls and hhx right wall.

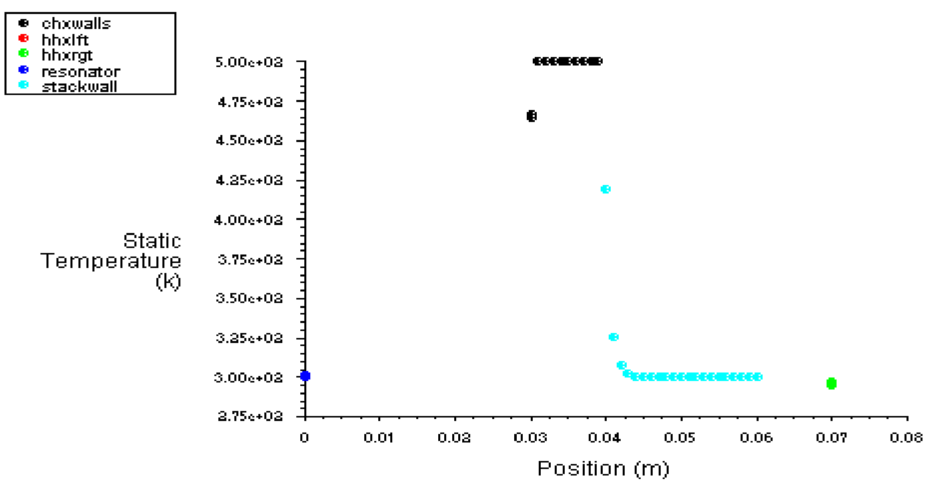


Fig 4.7 shows the vatiation of static pressure along with the chx walls, stack walls and hhx right wall.

Chapter 6

CONCLUSION

CONCLUSION

A 2D model of a thermoacoustic cryocooler was developed and its numerical analysis was done using GAMBIT and FLUENT. Various results were plotted, which would play a major role in understanding the characteristics and behavior of the cryocooler in specified conditions. Some of the results plotted were pressure plot at (… names of different zones….), temperature behavior at cold heat exchanger, pressure behavior at resonator etc. The temperature at the cold heat exchanger was observed to increase with the given initial conditions. By varying the input parameters and their values, a number of similar analysis can be done and hence the optimum operating conditions could be determined.

Chapter 7

REFERENCE

REFERENCES

[1] M.E.H. Tijani, J.C.H. Zeegers, A.T.A.M. de Waele, Construction and performance of a thermoacoustic refrigerator, Department of Applied Physics, Eindhoven University of Technology.

[2]Swift GW. Thermoacoustic engines. J Acoust Soc Am 1988;84:1146–80.

[3]Tijani MEH, Zeegers JCH, de Waele ATAM. Design of thermoacoustic refrigerators. Cryogenics [submitted].

[4]The performance measurements are all measured with a Dynaudio loudspeaker type D54 AF.

[5]Tijani MEH. Loudspeaker-driven thermo-acoustic refrigeration. PhD Thesis, Unpublished, Eindhoven University of Technology

[6] Fluent.com