HEAT TRANSFER ALONG VERTICAL CHIMNEY

[1] Dr. V. Venugopal [2] Dr. Karthikeyan P, [3] Dr. P. Sengottivel

[1] Associate Professor, Dept. of Mechanical Engineering, Malla Reddy College of Engineering, Sec-100

[2] Associate Professor, Department of Mechanical Engineering, Malla Reddy College of Engineering, Sec-100

[3] Associate Professor, Department of Mechanical Engineering, Malla Reddy College of Engineering, Sec-100

Abstract— Chimney, which form the last component of a system using a flue gas such as boiler, play a vital role in maintaining efficiency, draft, etc, of a system and also in minimizing the atmospheric pollution. Steel chimneys are also known as steel stacks. The steel chimneys are made of steel plates and supported on foundation. The steel chimneys are used to escape and disperse the flue gases to such a height that the gases do not contaminate surrounding atmosphere. The hot gases occupy. For the purpose of the structural design of steel the chimney, the height and diameter of chimney. Chimneys are required larger volume than before. The weight of gases per cubic meter becomes less to carry vertically and discharge, gaseous products of combustion, chemical waste gases, and exhaust air from an industry to the atmosphere. In this thesis, chimney materials (concrete used for the design of the chimney. The chimney was considered as a cantilever beam with annular will be designed considering with insulation and without insulation. The Bureau of Indian Standards (BIS) design codes procedures will be the chimney is done in CREO Parametric software and fluid- structural and thermal analysis is done on the chimney in ANSYS software. A simplified model of chimneys with various insulation cross section. 3D model of and carbon epoxy). Static analysis is to determine the deformation, stress and strain for chimney with insulation and without insulation. Thermal analysis to determine the heat flux of the chimney with different materials to different models. CFD analysis to determine the pressure drop, velocity, heat transfer coefficient, mass flow rate and heat transfer rate.

Keywords: Combustion, insulation, CFD,Heat transfer coefficient

INTRODUCTION

A chimney is a structure that provides ventilation for hot flue gases or smoke from a boiler, stove, furnace or fireplace to the outside atmosphere. Chimneys are typically vertical, or as near as possible to vertical, to ensure that the gases flow smoothly, drawing air into the combustion in what is known as the stack, or chimney effect. The space inside a chimney is called a flue. Chimneys may be found in buildings, steam locomotives and ships. In the United States, the term smokestack (colloquially, stack) is also used when referring to locomotive chimneys or ship chimneys, and the term funnel can also be used. The height of a chimney influences its ability to transfer flue gases to the external environment via stack effect. Additionally, the dispersion of pollutants at higher altitudes can reduce their impact on the immediate surroundings. In the case of chemically aggressive output, a sufficiently tall chimney can allow for partial or complete self-neutralization of airborne chemicals before they reach ground level. The dispersion of pollutants over a greater area can reduce their

concentrations and facilitate compliance with regulatory limits.

RESIDENTIAL FLUE LINERS: A flue liner is a secondary barrier in a chimney that protects the masonry from the acidic products of combustion, helps prevent flue gas from entering the house, and reduces the size of an oversized flue. Newly built chimneys have been required by building codes to have a flue liner in many locations since the 1950s. Chimneys built without a liner can usually have a liner added, but the type of liner needs to match the type of appliance it is servicing. Flue liners may be clay tile, metal, concrete tiles, or poured in place concrete. Clay tile flue liners are very common in the United States. However, this is the only liner which does not meet Underwriters Laboratories 1777 approval and frequently have problems such as cracked tiles and improper installation. Clay tiles are usually about 2 feet (0.61 m) long, various sizes and shapes, and are installed in new construction as the chimney is built. A refractory cement is used between each tile.Metal liners may be stainless steel, aluminum, or galvanized iron and may be flexible or rigid pipes. Stainless steel is made in several types

History:

A smoke hood in the Netherlands. Image: Cultural Heritage Agency of the Netherlands

CHIMNEY POTS, CAPS AND TOPS:

A chimney pot is placed on top of the chimney to expand the length of the chimney inexpensively, and to improve the

chimney's draft. A chimney with more than one pot on it indicates that there is more than one fireplace on different floors sharing the chimney. A chimney cowl is placed on top of the chimney to prevent birds and other animals from nesting in the chimney. They often feature a rain guard to prevent rain or snow from going down the chimney. A metal wire mesh is often used as a spark arrestor to minimize burning debris from rising out of the chimney and making it onto the roof. Although the masonry inside the chimney can absorb a large amount of moisture which later evaporates, rainwater can collect at the base of the chimney. Sometimes weep holes are placed at the bottom of the chimney to drain out collected water.

A chimney cowl or wind directional cap is a helmet-shaped chimney cap that rotates to align with the wind and prevent a backdraft of smoke and wind back down the chimney.


A H-style cowl:

An H-style cap (cowl) is a chimney top constructed from chimney pipes shaped like the letter H. (Its image is included in cowl (chimney).) It is an age-old method of regulating draft in situations where prevailing winds or turbulences cause downdraft and backpuffing. Although the H cap has a distinct advantage over most other downdraft caps, it fell out of favor because of its bulky design. It is found mostly in marine use but has been regaining popularity due

to its energy-saving functionality. The H-cap stabilizes the draft rather than increasing it. Other downdraft caps are based on the Venturi effect, solving downdraft problems by increasing the updraft constantly resulting in much higher fuel consumption. A chimney damper is a metal plate that can be positioned to close off the chimney when not in use and prevent outside air from entering the interior space, and can be opened to permit hot gases to exhaust when a fire is burning. A top damper or cap damper is a metal spring door placed at the top of the chimney with a long metal chain that allows one to open and close the damper from the fireplace. A throat damper is a metal plate at the base of the chimney, just above the firebox, that can be opened and closed by a lever, gear, or chain to seal off the fireplace from the chimney. The advantage of a top damper is the tight weatherproof seal that it provides when closed, which prevents cold outside air from flowing down the chimney and into the living space—a feature that can rarely be matched by the metal-on-metal seal afforded by a throat damper.

CHIMNEY DRAUGHT OR DRAFT:
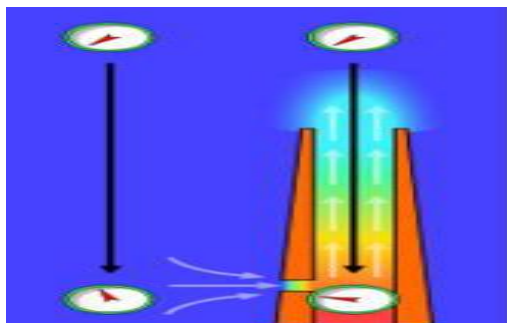
The stack effect in chimneys: the gauges represent absolute air pressure and the airflow is indicated with light grey arrows. The gauge dials move clockwise with increasing pressure.

Main article: Flue gas stack When coal, oil, natural gas, wood, or any other fuel is combusted in a stove, oven, fireplace, hot water boiler, or industrial furnace, the hot combustion product gases that are formed are called flue gases. Those gases are generally exhausted to the ambient outside air through chimneys or industrial flue gas stacks (sometimes referred to

as smokestacks). The combustion flue gases inside the chimneys or stacks are much hotter than the ambient outside air and therefore less dense than the ambient air. That causes the bottom of the vertical column of hot flue gas to have a lower pressure than the pressure at the bottom of a corresponding column of outside air. That higher pressure outside the chimney is the driving force that moves the required combustion air into the combustion zone and also moves the flue gas up and out of the chimney. That movement or flow of combustion air and flue gas is called "natural draught/draft", "natural ventilation", "chimney effect", or "stack effect". The taller the stack, the more draught or draft is created. There can be cases of diminishing returns: if a stack is overly tall in relation to the heat being sent out of the stack, the flue gases may cool before reaching the top of the chimney. This condition can result in poor drafting, and in the case of wood burning appliances, the cooling of the gases before emission can cause creosote to condense near the top of the chimney. The creosote can restrict the exit of flue gases and may pose a fire hazard. Designing chimneys and stacks to provide the correct amount of natural draft involves a number of design factors, many of which require iterative trial-and-error methods. As a "first guess" approximation, the following equation can be used to estimate the natural draught/draft flow rate by assuming that the molecular mass (i.e., molecular weight) of the flue gas and the external air are equal and that the frictional pressure and heat losses are negligible:

where:

Q = chimney draught/draft flow rate, m³/s

A = cross-sectional area of chimney, m² (assuming it
has a constant cross-section)

C = discharge coefficient (usually taken to be from
0.65 to 0.70)

G = gravitational acceleration, 9.807 m/s²

H = height of chimney, m

Ti = average temperature inside the chimney, K

Te = external air temperature, K.

MAINTENANCE AND PROBLEMS:

Chimneys on the Parliamentary Library in Wellington, New Zealand.

A characteristic problem of chimneys is they develop deposits of creosote on the walls of the structure when used with wood as a fuel. Deposits of this substance can interfere with the airflow and more importantly, they are combustible and can cause dangerous chimney fires if the deposits ignite in the chimney. Heaters that burn natural gas drastically reduce the amount of creosote buildup due to natural gas burning much cleaner and more efficiently than traditional solid fuels. While in most cases there is no need to clean a gas chimney on an annual basis that does not mean that other parts of the chimney cannot fall into disrepair. It is now possible to buy "faux-brick" facades to cover these modern chimney structures. Other potential problems include: "spalling" brick, in which moisture seeps into the brick and then freezes, cracking and flaking the brick and loosening mortar seals .shifting foundations, which may degrade integrity of chimney masonry nesting or infestation by unwanted animals such as squirrels, or chimney swifts chimney leaks drafting issues, which may allow smoke inside building issues with fireplace or heating appliance may cause unwanted degradation or hazards to chimney

Cooling tower used as an industrial chimney

At some power stations, which are equipped with plants for the removal of sulfur dioxide and nitrogen oxides, it is possible to use the cooling tower as a chimney. Such cooling towers can be seen in Germany at the Power Station Staudinger Grosskrotzenburg and at the Power Station Rostock. At power stations that are not equipped for removing sulfur dioxide, such usage of cooling towers could result in serious corrosion problems which are not easy to prevent.

LITERATURE REVIEW:
1.Seismic Analysis And Design Of Industrial

Chimneys This paper describes a simplified method that allow obtaining the fundamental period of vibration, lateral displacement, shear force and bending moment through a set of equations, obtaining for all cases studied an error below 10%. The results obtained in this study were applied to a total of 9 real chimneys (4 of steel and 5 of reinforced concrete) built in Chile, with the objective of calibrating founded expressions. During the stage of the analysis, it was verified that the criterion of consistent masses provide better results than the criterion of

lumped masses, and as a very important conclusion a discrete analysis of the model in twenty segments of the beam is satisfactory. The most representative variables that define the model with which it is possible to carry out a parametric analysis of the chimney. As important parameters we could refer to: slenderness ratio H/Dinf, radius ratio Rsup/Rinf, thickness ratio Esup/Einf and thickness diameter ratio Dinf/Einf. Later, by varying each one of the chosen parameters several analysis of representative chimneys of this great family, could be carried out. As seismic loads, the spectrums of accelerations recommended by the code of seismic design for structures and industrial installations in Chile, have been considered. Modal responses were combined using the combination rule CQC. In all the cases studied in this investigation, the influence of the P-Δ effect, the soil structure interaction, and the

influence on responses that provoke the inclusion of lining, have been disregarded.

2.Analysis Of Self Supported Steel Chimney As Per Indian Standard

Most of the industrial steel chimneys are tall structures with circular cross-sections. Such slender, lightly damped structures are prone to wind-exited vibration. Geometry of a self supporting steel chimney plays an important role in its structural behaviour under lateral dynamic loading. This is because geometry is primarily responsible for the stiffness parameters of the chimney. However, basic dimensions of industrial self supporting steel chimney, such as height, diameter at exit, etc., are generally derived from the associated environmental conditions. To ensure a desired failure mode design code (IS-6533: 1989 Part 2) imposes several criteria on the geometry (top-to-base diameter ratio and height-tobase diameter ratio) of steel chimneys. The objective of the present study is to justify the code criteria with regard to basic dimensions of industrial steel chimney.

3.Analysis of Tall RC Chimney as per Indian Standard Code

Reinforced chimneys are used in Power plants to take the hot and poisonous flue gas to a great height. They are tall and slender structures, designed mainly to resist the lateral forces like wind and earthquake as well as the thermal stressesof the flue gas. An attempt is done to understand the variation of lateral deflection at the top of the chimney, by varying the height of chimney above 275 m. CED 38:7892 Code of practice for design of reinforced concrete chimney (Third revision of IS 4998:1992 [Part I]) is used for the analysis. The location selected for the study is Bellary in Karnataka. Along wind and temperature are only considered for this study. Sufficient amount of reinforcement is provided to resist the bending moment in the vertical direction and horizontal loops are provided to cater for the horizontal shear and temperature gradient. A totalof five models are selected for five different heights and the analysis and design are done. ANSYS software was used to do the analysis. All the models were analyzed and the lateral deflection was calculated.

4.Analysis and Computational Study of a High Chimney Tower for Solar Energy

Abstract: A Solar Chimney Power Plant consists of central chimney that is surrounded by a transparent canopy locateda few meters above ground level. An analysis of solar chimneys has been developed, aimed particularly to study stability and structural strength of a model of cylindrical reinforced concrete tower with 500m in tall and 50m in diameter. The design of this tower has several technical challenges. This model is subjected to his own weight, the effects of wind and the pressure due to the flow of air inside the chimney. In this study, the effect of these loads on the stability and strength of the chimney has been examined. The ringsstiffened are necessary to prevent ovalisation of the structure. In addition, the influence of various designs of rings-stiffened is taken into consideration in the mechanical behaviour of this tower. Numerical simulation modeling method based on finite element method is adopted using the "Autodesk Robot structural analysis professional" software.

5.Nonlinear Dynamic Analysis Of Chimney-Like

Towers In this study the most important problem i.e. earthquake behaviour of the structures, hysteric behaviour of material and section properties are studied. The significance of this study is mainly concentrated on model simplification that provides sufficient accuracy based on a nonlinear discrete model. Tous power plant chimney is investigated numerically as an example. The nonlinear dynamic analysis essentially needed for seismic assessment in evaluation of actual performance of complicated structures during earthquakes than the damage indices of structure had to be calculated using appropriate damage models.

6. Dynamic Soil-Structure Interaction Analysis of Tall Multi-Flue Chimneys Under Aerodynamic And Seismic Force.

The present paper proposes a semi analytic mathematical model based on which both seismic and aerodynamic response of such a tall chimneys are studied for various soil stiffness and are compared with fixed base conventional method as per UBC 97(for seismic load) and CICIND (for wind loading). Soil Structure interaction also has an important effect on seismic forces of tall chimneys. Although for tall chimneys rested on firm soil, earthquake loads decreased as a result of increasing in period values, seismic forces may amplify by using different response spectra in calculation. This means that the soil structure interaction effects are reliant on characteristic of the seismic excitation in addition to chimneys properties. JEEVAN T, SOWJANYA G. V (2014)

INTRODUCTION TO CREO :

PTC CREO, formerly known as Pro/ENGINEER, is 3D modeling software used in mechanical engineering, design, manufacturing, and in CAD drafting service firms. It was one of the first 3D CAD modeling applications that used a rule-based parametric system. Using parameters, dimensions and features to capture the behavior of the product, it can optimize the development product as well as the design itself. The name was changed in 2010 from Pro/ENGINEER Wildfire to CREO. It was announced by the company who developed it, Parametric Technology Company (PTC), during the launch of its suite of design products that includes applications such as assembly modeling, 2D orthographic views for technical drawing, finite element analysis and more. PTC CREO says it can offer a more efficient design experience than other modeling software because of its unique features including the integration of parametric and direct modeling in one platform. The complete suite of applications spans the spectrum of product development, giving designers options to use in each step of the process. The software also has a more user friendly interface that provides a better experience for designers. It also has collaborative capacities that make it easy to share designs and make changes. There are countless benefits to using PTC CREO. We'll take a look at them in this two-part series. First up, the biggest advantage is increased productivity because of its efficient and flexible design capabilities. It was designed to be easier to use and have features that allow for design processes to move more quickly, making a designer's productivity level increase. The time saved by using PTC CREO isn't the only advantage. It has many ways of saving costs. For instance, the cost of creating a new product can be lowered because the development process is shortened due to the automation of the generation of associative manufacturing and service deliverables. PTC also offers comprehensive training on how to use the software. This can save businesses by eliminating the need to hire new employees. Their training program is available online and in-person, but materials are available to access anytime. A unique feature is that the software is available in 10 languages. PTC knows they have people from all over the world using their software, so they offer it in multiple languages so nearly anyone who wants to use it is able to do so.

ADVANTAGES OF CREO PARAMETRIC SOFTWARE :

1. Optimized for model-based enterprises

2. Increased engineer productivity

3. Better enabled concept design

4. Increased engineering capabilities

5. Increased manufacturing capabilities

6. Better simulation

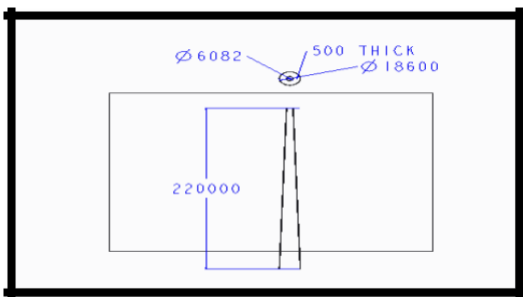7. Design capabilities for additive manufacturing

CREO PARAMETRIC MODULES:

• Sketcher

• Part modeling

• Assembly

• Drafting

2D MODEL OF CHIMNEY

INTRODUCTION TO FEAL:

Finite element analysis is a method of solving, usually approximately, certain problems in engineering and science. It is used mainly for problems for which no exact solution, expressible in some mathematical form, is available. As such, it is a numerical rather than an analytical method. Methods of this type are needed because analytical methods cannot cope with the real, complicated problems that are met with in engineering. For example, engineering strength of materials or the mathematical theory of elasticity can be used to calculate analytically the stresses and strains in a bent beam, but neither will be very successful in finding out what is happening in part of a car suspension system during cornering. One of the first applications of FEA was, indeed, to find the stresses and strains in engineering components under load. FEA, when applied to any realistic model of an engineering component, requires an enormous amount of computation and the development of the method has depended on the availability of suitable digital computers for it to run on. The method is now applied to problems involving a wide range of phenomena, including vibrations, heat conduction, fluid mechanics and electrostatics, and a wide range of material properties, such as linear-elastic (Hookean) behavior and behavior involving deviation from Hooke's law (for example, plasticity or rubber-elasticity). Many comprehensive general-purpose computer packages are now available that can deal with a wide range of phenomena, together with more specialized packages for particular applications, for example, for the study of dynamic phenomena or large-scale plastic flow. Depending on the type and complexity of the analysis, such packages may run on a microcomputer or, at the other extreme, on a supercomputer. FEA is essentially a piece-wise process. It can be applied to one-dimensional problems, but more usually there is an area or volume within which the solution is required. This is split up into a number of smaller areas or volumes, which are called finite elements. Figure 1 shows a two-dimensional model of a spanner that has been so divided: the process is called discretisation, and the assembly of elements is called a mesh.

INTRODUCTION TO ANSYS:

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behaviour of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated, or graphical forms.

Generic Steps to Solving any Problem in ANSYS:

Like solving any problem analytically, you need to define (1) your solution domain, (2) the physical model, (3) boundary conditions and (4) the physical properties. You then solve the problem and present the results. In numerical methods, the main difference is an extra step called mesh generation. This is the step that divides the complex model into small elements that become solvable in an otherwise too

complex situation. Below describes the processes in terminology slightly more attune to the software.

Build Geometry:Construct a two or three dimensional representation of the object to be modeled and tested using the work plane coordinate system within ANSYS.

Define Material Properties:Now that the part exists, define a library of the necessary materials that compose the object (or project) being modeled. This includes thermal and mechanical properties.

Generate Mesh: At this point ANSYS understands the makeup of the part. Now define how the modeled system should be broken down into finite pieces.

Apply Loads: Once the system is fully designed, the last task is to burden the system with constraints, such as physical loadings or boundary conditions.

Present the Results:After the solution has been obtained, there are many ways to present ANSYS' results, choose from many options such as tables, graphs, and contour plots.

SPECIFIC CAPABILITIES OF ANSYS:

Structural

Static Analysis - Used to determine displacements, stresses, etc. under static loading conditions. ANSYS can compute both linear and nonlinear static analyses. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep. Transient Dynamic Analysis - Used to determine the response of a structure to arbitrarily time-varying loads. All nonlinearities mentioned under Static Analysis above are allowed. Buckling Analysis - Used to calculate the buckling loads and determine the buckling mode shape. Both linear (eigenvalue) buckling

and nonlinear buckling analyses are possible.

Thermal:

ANSYS is capable of both steady state and transient analysis of any solid with thermal boundary conditions. Steady-state thermal analyses calculate the effects of steady thermal loads on a system or component. Users often perform a steady-state analysis before doing a transient thermal analysis, to help establish initial conditions. A steady-state analysis also can be the last step of a transient thermal analysis; performed after all transient effects have diminished. ANSYS can be used to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. Such loads include the following:

• Convection

• Radiation

• Heat flow rates

• Heat fluxes (heat flow per unit area)

• Heat generation rates (heat flow per unit volume)

Fluid Flow:

The ANSYS/FLOTRAN CFD (Computational Fluid Dynamics) offers comprehensive tools for analyzing two-dimensional and three-dimensional fluid flow fields. ANSYS is capable of modeling a vast range of analysis types such as: airfoils for pressure analysis of airplane wings (lift and drag), flow in supersonic nozzles, and complex, three-dimensional

6

flow patterns in a pipe bend. In addition, ANSYS/FLOTRAN could be used to perform tasks including:

• Calculating the gas pressure and temperature distributions in an engine exhaust manifold

• Studying the thermal stratification and breakup in piping systems

• Using flow mixing studies to evaluate potential for thermal shock

• Doing natural convection analyses to evaluate the thermal performance of chips in electronic enclosures

• Conducting heat exchanger studies involving different fluids separated by solid regions

COUPLED FIELD:

A coupled-field analysis is an analysis that takes into account the interaction (coupling) between two or more disciplines (fields) of engineering. A piezoelectric analysis, for example, handles the interaction between the structural and electric fields: it solves for the voltage distribution due to applied displacements, or vice versa. Other examples of coupled-field analysis are thermal-stress analysis, thermal-electric analysis, and fluid-structure analysis. Some of the applications in which coupled-field analysis may be required are pressure vessels (thermal-stress analysis), fluid flow constrictions (fluid-structure analysis), induction heating

(magnetic-thermal analysis), ultrasonic transducers (piezoelectric analysis), magnetic forming (magneto-structural analysis), and micro-electro mechanical systems (MEMS).

Modal Analysis - A modal analysis is typically used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a harmonic response or full transient dynamic analysis. Modal analyses, while being one of the most basic dynamic analysis types available in ANSYS, can also be more computationally time consuming than a typical static analysis. A reduced solver, utilizing automatically or manually selected master degrees of freedom is used to drastically reduce the problem size and solution time.

Harmonic Analysis - Used extensively by companies who produce rotating machinery, ANSYS Harmonic analysis is used to predict the sustained dynamic behavior of structures to consistent cyclic loading. Examples of rotating machines which produced or are subjected to harmonic loading are:

• Turbines

• Gas Turbines for Aircraft and Power Generation

• Steam Turbines

• Wind Turbine

• Water Turbines

• Turbopumps

• Internal Combustion engines

• Electric motors and generators

• Gas and fluid pumps

• Disc drives

A harmonic analysis can be used to verify whether or not a machine design will successfully overcome resonance, fatigue, and other harmful effects of forced vibrations.

INTRODUCTION TO CFD:

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the

interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

METHODOLOGY:

In all of these approaches the same basic procedure is followed.

• During preprocessing

• The geometry (physical bounds) of the problem is defined.

• The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform.

• The physical modeling is defined – for example, the equations of motion + enthalpy + radiation + species conservation

• Boundary conditions are defined. This involves specifying the fluid behaviour and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined.

• The simulation is started and the equations are solved iteratively as a steady-state or transient.

• Finally a postprocessor is used for the analysis and visualization of the resulting solution.

CONCLUSION:

Chimney, which form the last component of a system using a flue gas such as boiler, play a vital role in maintaining efficiency, draft, etc, of a system and also in minimizing the atmospheric pollution. The ease of thermal control by means of air natural convection stimulates the investigation of configurations with the aim at

improving the thermal performance. The steel chimneys are used to escape and disperse the flue gases to such a height that the gases do not contaminate surrounding atmosphere. The hot gases occupy.